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

HFSS™ Help

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

Release 2025 R1
January 2025

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

Copyright and Trademark Information

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

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

Disclaimer Notice

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

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

U.S. Government Rights

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

Third-Party Software

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

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

Table of Contents

Table of Contents **Contents-1**

1 - Welcome to HFSS Help **1-1**

Modeling Quick Links 1-2

Materials Quick Links 1-2

Meshing Quick Links 1-3

Analysis Quick Links 1-3

Optimetrics Quick Links 1-4

Results Quick Links 1-5

HFSS Getting Started Guides 1-6

Example Projects 1-20

HFSS Examples 1-21

ADAS Examples (Advanced Driver Assistance Systems) 1-22

Antenna Examples 1-23

EMI EMC Examples (Electromagnetic Interference, Electromagnetic Compatibility) 1-24

Filter Examples 1-24

Metamaterial Examples 1-25

RCS Examples (Radar Cross Section) 1-25

RF Microwave Examples 1-26

Signal Integrity Examples 1-26

Transmission Line Examples 1-27

HFSS Examples Associated with Getting Started Guides 1-27

Dielectric Resonator Antenna 1-28

Johanson 2450AT18D0100-EB1SMA 1-30

Abstract 1-30

Model and Setup Details 1-32

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

Postprocessing 1-34
Monostatic Trihedral RCS Example Project 1-41
Tee and OptimTee Waveguide Projects 1-42
Waveguide Combiner 1-44
Abstract 1-44
Setup Description: 1-45
Postprocessing 1-46
ADAS Examples (Advanced Driver Assistance Systems) 1-49
Autonomous Vehicular Radar ADP (SBR+) 1-50
Antenna Examples 1-51
5G Antenna Using HFSS 3D Components 1-51
Introduction 1-51
Description 1-52
Creation of the 3D Component: 1-59
Postprocessing 1-64
References 1-69
Antenna Radome SBR+ Using Fresnel Boundary 1-69
Antennas on Mast 1-72
Determining Phase Center Using Optimetrics 1-73
Define the Relative Coordinate System using a Post Processing Variable 1-75
Use the Relative Coordinate System in a Far Field Setup Calculation 1-76
Create an Optimization Setup to Search for the Phase Center 1-77
Analyze and View the Optimization Results 1-80
Finite Array Radome - Multi Cell Array 1-81
Helical Antenna 1-83
Abstract 1-83
Description 1-84
Postprocessing 1-86

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

HFSS SBR+ Hybrid Reflector Antenna Simulation Example 1-97
Parabolic Dish with Horn 1-101
Abstract 1-101
Postprocessing 1-104
Johanson 2450AT18B100-EB1SMA 1-105
Abstract 1-105
Model and Setup Details 1-106
Postprocessing 1-107
Potter Horn Antenna 1-115
Abstract 1-115
Model and Setup Details 1-116
Postprocessing 1-119
Appendix 1: 1-121
References: 1-125
Small Array of Planar Flared Dipole Antennas 1-125
Transient Simulation of Ground Penetrating Radar 1-128
Tune a Coax Fed Patch Antenna 1-130
Unit Cell of a Phased Array 1-132
EMI EMC Examples (Electromagnetic Interference, Electromagnetic Compatibility) 1-134
BCI ISO11452-2 Calibration Test 1-135
CISPR25 Conducted Emissions Example 1-136
CISPR25 Radiated Emissions Example 1-137
EMP Electromagnetic Pulse (HFSS Transient) Example 1-139
ESD Enclosure with Slot (HFSS Transient) Example 1-141
ESD IEC 6100000-4-2 VCP (HFSS Transient) Example 1-143
Radiation Hazards MIL STD461C ICNIRP Example 1-144
Integrated Circuits in HFSS 1-146
Filter Examples 1-147

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

Metamaterial Examples 1-147
Cylindrical Hyperlens 1-147
Abstract 1-147
Introduction 1-148
Model Construction and Setup Details 1-149
Viewing Results with the Hyperlens 1-152
Comparing Results with and without the Hyperlens 1-156
References: 1-158
Split Ring Resonator 1-159
Abstract 1-159
Introduction 1-159
Model Construction and Setup Details 1-160
Effective Material Parameter Retrieval in HFSS 1-162
Constructing an Equivalent Material from the Retrieved Properties: 1-171
Post Processing 1-174
References: 1-176
RCS Examples (Radar Cross Section) 1-176
Monostatic RCS of a PEC Sphere Using SBR Solver with Creeping Waves 1-177
Design “0_VRT_creeping_waves” 1-177
1_Monostatic_RCS_Creeping_Waves 1-179
NASA Almond Parametric 1-181
Abstract 1-181
Model and Setup Details 1-182
Postprocessing 1-184
References 1-189
RF Microwave Examples 1-189
Coax Bend 1-190
Abstract 1-190

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

Description ... 1-190
Postprocessing ... 1-192
Coax to Waveguide ... 1-194
Abstract ... 1-194
Model and Setup Details ... 1-195
Postprocessing ... 1-199
Coaxial Resonator ... 1-202
Abstract ... 1-202
Description ... 1-203
Simulation Summary ... 1-203
Postprocessing ... 1-203
Corporate Feed ... 1-205
Ferrite Circulator ... 1-206
Ring Hybrid ... 1-209
Abstract ... 1-209
Description ... 1-210
Simulation Summary ... 1-211
Postprocessing ... 1-211
Waveguide FEM versus Theory ... 1-215
Abstract ... 1-215
Model and Setup Details ... 1-216
Postprocessing ... 1-219
Interpolating Sweep Setup ... 1-224
Signal Integrity Examples ... 1-225
Connector - Terminal Example ... 1-226
Package Section ... 1-229
Transmission Line Examples ... 1-231
Co-Planar Waveguide (Driven Terminal) ... 1-231

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

Coplanar Waveguide with Ground (Driven Terminal) 1-236
Differential Pair Microstrip 1-238
Differential Pair Stripline 1-244
Stripline (Driven Terminal) 1-248
Twinaxial Cable 1-252
Finding Information in the Online Help 1-256
Using the Search Function in the Help 1-257
Help Conventions 1-261
Getting Help from Ansys Technical Support 1-263
2 - Getting Started with Ansys Electronics Desktop 2-1
Ansys Electronics Desktop Student 2-3
HFSS Student Limitations 2-4
Q3D Extractor and 2D Extractor Student Limitations 2-4
Circuit Limitations 2-4
Maxwell and RMxprt Student Limitations 2-4
Icepak Student Limitations 2-5
System Requirements 2-5
Launching Ansys Electronics Desktop 2-5
Ansys Electronics Desktop Overview 2-7
Choosing a Color Scheme [Beta] 2-8
Working with Ribbons 2-10
IC Mode Layout Ribbon 2-13
Ansys Electronics Desktop Windows 2-13
Showing and Hiding Windows 2-14
Auto Hiding Windows 2-15
Moving and Resizing Windows 2-16
Project Manager Window 2-18
Working with the Project Tree 2-19

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

Setting the Project Tree to Expand Automatically 2-19
Viewing Material Definitions 2-20
Viewing Ansys Electronics Desktop Design Details 2-21
Message Manager Window 2-22
Setting the Message Manager to Open Automatically 2-23
Showing New Messages 2-23
Automatically Expanding the Message Manager Tree 2-24
Action Messages 2-24
Clearing Messages 2-25
Hiding the Message Manager Window until Messages Appear 2-25
Progress Window 2-25
Stopping or Aborting Simulation Progress 2-26
Properties Window 2-27
Opening the Properties Window 2-28
Setting the Properties Window to Open Automatically 2-28
Showing and Hiding the Properties Window 2-29
Modifying Object Attributes Using the Properties Window 2-29
Modifying Object Command Properties Using the Properties Window 2-29
Property Window for Schematic and Layout Editors 2-30
Param Values Tab 2-30
Launching Help from the Property Window 2-31
General Tab 2-31
Symbol Tab 2-32
Closing the Property Window 2-34
Properties Dialog Box 2-34
Opening the Properties Dialog Box 2-34
Parameter Values Tab 2-35
General and Symbol Tabs 2-36

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

Property Displays Tab 2-36
Working with the Components Window 2-37
Working with the Nets Window 2-38
Nets Tab 2-39
Classification Pane 2-39
Nets Pane 2-40
Differential Pairs 2-42
Edit Differential Pairs Window 2-45
Extended Nets Tab 2-46
Extended Nets Pane 2-47
Auto Identify Settings window 2-50
Nets Pane 2-51
ACT Extensions Window 2-52
Ansys Electronics Desktop Menus 2-57
Top Menu Bar 2-58
File Menu 2-59
Edit Menu 2-60
Edit Menu for 3D Modeler 2-60
Edit Menu for Netlist Editor 2-63
Edit Menu for Report Window 2-64
View Menu 2-65
Basic View Menu 2-65
3D Modeler View Menu 2-66
Project Menu 2-71
Editor and Design Specific Menus 2-72
Layout Editor Draw Menu 2-73
Tools Menu 2-74
Adding External Tools to the Tools Menu 2-77

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

Window Menu 2-79
Help Menu 2-82
Context-Sensitive Help 2-83
Obtaining Information about the Software and Release 2-83
Shortcut Menus 2-83
Shortcut Menu in the 3D Modeler Window 2-84
Shortcut Menus in the Project Manager Window 2-85
Customizing Ansys Electronics Desktop Menus 2-87
Ansys Electronics Desktop Design Area 2-97
3D Modeler Window 2-98
Layout Window 2-98
Report Window 2-101
Ansys Electronics Desktop History Tree 2-101
Organizing the History Tree 2-104
Controlling the View of Objects in the History Tree 2-105
Group Commands for Modeler Objects 2-105
Selecting Objects in the History Tree 2-110
Viewing and Editing Commands on History Tree Objects 2-112
Upgrading Version in the History Tree 2-113
Ansys Electronics Desktop Status Bar 2-114
Keyboard Shortcuts 2-115
Desktop Shortcuts 2-115
3D Modeler Window Shortcuts 2-116
Schematic Shortcuts 2-117
Layout Shortcuts 2-118
Report Shortcuts 2-119
Netlist Editor Shortcuts 2-120
Choosing the View Navigation Options 2-120

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

Custom Keyboard Shortcuts 2-122
Auto-Complete for Variables and Properties in Electronics Desktop 2-124
Using the Password Manager to Control Access to Resources 2-128
Running Ansys Electronics Desktop from a Command Line 2-129
Command-line Syntax 2-129
Run Commands 2-130
Job Management from the Command Line 2-135
Run Command Examples 2-136
Specifying Project Files 2-136
Options 2-137
-Batchoptions 2-138
-Batchoptions Examples 2-139
Export Options Files 2-140
Examples and Further Explanations of -batchoptions Use 2-140
For -batchoptions Use: Project Directory and Lib Paths 2-143
For -batchoptions Use: TempDirectory 2-143
For -batchoptions Use: Various Desktop Settings 2-143
For -batchoptions Use: HFSS Options 2-146
Batchoptions: Override Registry Entry 2-149
Running Ansys Electronics Desktop from a Windows Remote Terminal 2-150
Using Windows HPC Commands 2-150
Customizing Ansys Electronics Desktop with Ansys ACT 2-151
Installing PyAEDT (Beta) 2-151
Debug Logging 2-152
Antennas: Application Specific Modeling Guide 2-153
Spiral Inductors on Silicon Substrate: Application Specific Modeling Guide 2-154
3 - Working with Ansys Electronics Desktop Projects 3-1
Ansys Electronics Desktop Files 3-1

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

HFSS Files ... 3-2
Setting Project Options ... 3-2
Setting General Options ... 3-4
General Options: Desktop Configuration ... 3-6
General Options: Project Options ... 3-8
General Options: Miscellaneous Options ... 3-9
General Options: User Interface Options ... 3-10
General Options: Directories Options ... 3-14
General Options: Desktop Performance ... 3-15
General Options: Default Units ... 3-18
General Options: Remote Analysis Options ... 3-19
General Options: Component Libraries Options ... 3-20
Setting 3D Modeler Options ... 3-22
3D Modeler Options: Drawing Options ... 3-23
3D Modeler Options: Operation Options ... 3-24
3D Modeler Options: Snap Options ... 3-25
3D Modeler Options: Display Options ... 3-27
Display Options: General Options ... 3-28
Display Options: Rendering Options ... 3-30
Display Options: History Tree Options ... 3-32
3D Modeler Options: Group Options ... 3-33
3D Modeler Options: Discovery Link Options ... 3-33
3D Modeler Options: Advanced Options ... 3-35
Setting Reporter Options ... 3-36
Reporter Options: Report Setup Options ... 3-37
Reporter Options: Fields Reporter Options ... 3-38
Reporter Options: Report2D Options ... 3-38
Report2D Options: Curve ... 3-39

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

Report2D Options: Axis ... 3-39
Report2D Options: Grid ... 3-40
Report2D Options: Header ... 3-40
Report2D Options: Note ... 3-40
Report2D Options: Legend ... 3-41
Report2D Options: Marker ... 3-41
Report2D Options: Marker Table ... 3-42
Report2D Options: X-Y Markers ... 3-42
Report2D Options: Stacked ... 3-43
Report2D Options: Digital ... 3-44
Report2D Options: General ... 3-44
Curve Tooltip ... 3-44
Clipboard Option ... 3-45
Format ... 3-45
Report2D Options: Table ... 3-45
Header Row ... 3-45
Format ... 3-46
Copy to Clipboard ... 3-46
Setting HPC and Analysis Options ... 3-46
Configurations Tab ... 3-47
Selecting an Available Configuration ... 3-48
Options Tab ... 3-49
Specifying the Remote Spawn Command (Linux) ... 3-53
Licensing Settings Tool ... 3-55
Setting Options via Configuration Files ... 3-57
Behavior Examples ... 3-58
Rules for Modifying Option Settings ... 3-58
Configuration File Locations ... 3-58

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

Products with Multiple Desktop Versions 3-59
Table of Directories and Files 3-59
Setting or Removing Option Values in Configuration Files: UpdateRegistry Command 3-60
UpdateRegistry -Set Command 3-61
UpdateRegistry -Get Command 3-61
UpdateRegistry -GetKeys Command 3-62
UpdateRegistry -Delete Command 3-63
UpdateRegistry -FromFile Command 3-64
UpdateRegistry File Format 3-64
Workaround for HFSS Design Setting for Power S 3-65
Example Uses for Export Options Features 3-66
Options that Apply to All Users 3-66
Example: Searching for a Registry Key Pathname 3-67
Example: Setting an Installation Default Value 3-67
Example: Setting a Host-Dependent Default Value 3-68
Example: Reverting from a User-Defined Option Value to the Administrator Default 3-68
User Options and the UpdateRegistry Tool 3-69
Example: Removing a Host-Dependent User Option Setting 3-70
Example: Getting a Value from a Specific Configuration File 3-70
Example: Getting a Value Using Precedence Rules 3-71
Example: Adding a Host-Independent User Option Setting 3-71
Example: Setting a RegistryKey not defined in ElectronicsDesktopRegistrySyntax.xml 3-71
Example: Change a Newly Set Registry Key 3-71
Setting the Temporary Directory 3-72
Temporary Directory Configuration File Format 3-74

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

UpdateRegistry: Setting or Removing Temporary Directory Values in Configuration Files ... 3-75
Batchoptions Command Line Examples ... 3-76
Batchoptions File Format ... 3-76
Example -BatchOptions with -Remote (Windows) ... 3-77
Example -Batchsolve with -Machinelist (Windows) ... 3-78
Example -Batchsolve with -Machinelist (Linux) ... 3-78
Example -Batchsolve for Local (Windows) ... 3-79
Batch Options and Analysis Configurations in the Registry ... 3-80
Setting HFSS Options ... 3-84
Wave Port Wizard or Multi-tab Dialog Menu ... 3-87
Managing Projects and Designs ... 3-88
Opening Projects ... 3-89
Opening Recent Projects ... 3-91
Opening Example Projects ... 3-92
Opening Legacy HFSS-IE Projects ... 3-92
Opening Legacy Projects ... 3-93
Legacy Project Translation ... 3-94
Creating Projects ... 3-96
Creating a Target Design from 3D Designs ... 3-97
Icepak Designs ... 3-98
Mechanical Designs ... 3-99
Saving Projects ... 3-100
Path Name Length Issues for Windows ... 3-100
Changing Auto-Save Settings ... 3-102
Save Before Solving Option ... 3-103
Recovering Project Data from an Auto-Save File ... 3-103
Importing and Exporting Projects and Data ... 3-103

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

Importing Files ... 3-103
Importing 2D Model (GDSII) Files ... 3-105
Creating GDSII Layer Mapping Files ... 3-108
Creating GDSII Layer Mapping Files ... 3-109
Tech File Colors with RGB Values ... 3-110
Importing 3D Model Files ... 3-124
Lightweight Geometry for HFSS SBR+ Designs ... 3-126
Initial Mesh Operations for Lightweight Geometries ... 3-129
Permitted Modeler Operations on Faceted Objects ... 3-129
Other Characteristics of Faceted Objects ... 3-129
Importing DXF and DWG Format Files ... 3-129
Importing Solution Data ... 3-132
Importing Data Tables ... 3-133
Importing Ansys Electronics Desktop Plot Data ... 3-134
Importing Plot Data ... 3-134
Exporting Files ... 3-135
Exporting 2D Model Files ... 3-136
Exporting 3D Model Files ... 3-136
GDSII Files ... 3-137
Parasolid Files ... 3-138
Exporting Graphics Files ... 3-139
Exporting Array Metadata ... 3-141
Export File Specifications ... 3-143
Exporting Metadata ... 3-146
Export File Specifications ... 3-148
Exporting Reports ... 3-151
Exporting Options Files ... 3-153
Exporting Options Files Using the Desktop UI ... 3-153

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

Exporting Options Files Using a Script ... 3-153
Validating Projects ... 3-154
Modeler Validation Settings ... 3-156
Copying and Pasting a Project or Design ... 3-157
Renaming Projects ... 3-157
Archiving Projects ... 3-158
Restoring an Archived Project ... 3-161
Deleting a Project or Design ... 3-165
Setting Read Only Designs ... 3-165
Updating Design Components ... 3-166
Undoing and Redoing Commands ... 3-167
Inserting a Documentation File ... 3-168
Working with Design Notes ... 3-168
Working with Design Notes ... 3-169
Printing ... 3-170
Getting Started with HFSS ... 3-171
Setting up an HFSS Design ... 3-172
Inserting an HFSS Design ... 3-173
Selecting the Solution Type ... 3-174
Composite Excitation Option for HFSS Driven Solution Types ... 3-180
Using Auto-Open Region for the HFSS Driven Solution Type for Antenna Designs ... 3-182
Design Flow for SBR+ Solution Type ... 3-187
Using SBR+ Solution Type for HFSS ... 3-189
3D Components for and from SBR+ Designs ... 3-213
Configure Simulation with SBR+ Antenna Geometry Blockage (SBR+ Solution Type) ... 3-213
Workflow of an HFSS 3D Layout Design ... 3-216

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

Inserting an HFSS 3D Layout Design ... 3-218
Example Projects ... 3-219
HFSS Examples ... 3-220
ADAS Examples (Advanced Driver Assistance Systems) ... 3-220
Antenna Examples ... 3-221
EMI EMC Examples (Electromagnetic Interference, Electromagnetic Compatibility) ... 3-222
Filter Examples ... 3-222
Metamaterial Examples ... 3-223
RCS Examples (Radar Cross Section) ... 3-223
RF Microwave Examples ... 3-224
Signal Integrity Examples ... 3-224
Transmission Line Examples ... 3-225
Working with Variables ... 3-225
Working with Project Variables ... 3-227
Adding a Project Variable ... 3-227
Importing and Exporting Project Variables ... 3-230
Deleting a Project Variable ... 3-231
Editing a Project Variable ... 3-231
Adding a Project Variable Array ... 3-231
List of Intrinsic Variables ... 3-236
List of Constants ... 3-237
Working with Design Variables ... 3-238
Defining Local Variables at the Design Level ... 3-239
Adding a Local Variable ... 3-239
Deleting a Local Variable ... 3-242
Editing a Local Variable ... 3-243
Overriding a Local Variable ... 3-243

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

Defining Parameter Defaults at the Design Level ... 3-243
Adding a Parameter Default ... 3-243
Deleting a Parameter Default ... 3-245
Editing a Parameter Default ... 3-245
Overriding a Parameter Default ... 3-245
Adding a Design Variable Array ... 3-246
Converting Variables and Parameter Defaults ... 3-251
Adding Variables and Parameter Defaults from Components ... 3-251
Fixed vs. Non-Fixed Variables ... 3-254
Non-Fixed Variable Sweeps ... 3-255
Orphaned Sweeps ... 3-255
Choosing a Variable to Optimize ... 3-256
Choosing a Variable to Tune ... 3-257
Including a Variable in a Sensitivity Analysis ... 3-258
Including a Variable in a Statistical Analysis ... 3-259
Copying and Pasting a Variables List ... 3-260
Working with Datasets ... 3-261
Adding Datasets ... 3-262
Adding 3D Datasets ... 3-266
Importing Datasets ... 3-271
Editing Datasets ... 3-273
Cloning Datasets ... 3-275
Exporting Datasets ... 3-278
Removing Datasets ... 3-279
Changing Dataset Plot Properties ... 3-280
Using SheetScan ... 3-283
SheetScan Menus and Settings ... 3-284
SheetScan View Options ... 3-284

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

SheetScan Right-click Menu ... 3-285
SheetScan Settings ... 3-285
Loading a Datasheet Picture into SheetScan ... 3-286
Loading a Datasheet Picture Directly ... 3-286
Loading a Datasheet Picture Using the HTML Viewer ... 3-286
Defining a SheetScan Coordinate System ... 3-287
Defining a Characteristic Curve in SheetScan ... 3-288
Performing Operations on SheetScan Curves ... 3-289
Checking for Monotonicity in X ... 3-290
Importing Characteristic Data into SheetScan ... 3-291
Exporting SheetScan Data ... 3-293
Defining an Expression ... 3-294
Valid Operators for Expressions ... 3-294
Using Intrinsic Functions in Expressions ... 3-295
clp Formula ... 3-296
Using Piecewise Linear Functions in Expressions ... 3-297
Using Dataset Expressions ... 3-298
Handling Delta Temperature Units in Expressions ... 3-300
Defining Mathematical Functions ... 3-302
Meshing Quick Links ... 3-302
Modeling Quick Links ... 3-303
4 - Minerva Remote Storage Environment ... 4-1
5 - Assigning Materials ... 5-1
Solving Inside or on the Surface ... 5-7
Assigning DC Thickness ... 5-8
Searching for Materials ... 5-10
Searching by Material Property ... 5-13
Adding New Materials ... 5-15

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

Assigning Material Property Types ... 5-19
Defining Anisotropic Tensors ... 5-20
Assigning Anisotropic Tensors ... 5-21
Cylindrical Anisotropic Material Properties: Tensor Conversion from Cylindrical to Cartesian CS ... 5-24
Spherical Anisotropic Material Properties: Tensor Conversion from Spherical to Cartesian CS ... 5-28
Defining Variable Material Properties ... 5-32
Defining Frequency-Dependent Material Properties ... 5-32
Assigning Frequency-Dependent Properties ... 5-32
Frequency Dependence Visualization ... 5-34
Saved Input Data Invalidation For Frequency-Dependent Setup ... 5-35
Assigning Frequency Dependent Material: Piecewise Linear Input ... 5-36
Assigning Frequency Dependent Material: Debye Model Input ... 5-38
Assigning Frequency Dependent Material: Multipole Debye Model Input ... 5-39
Assigning Frequency Dependent Material: Djordjevic-Sarkar Model Input ... 5-42
Assigning Frequency Dependent Material: Nonlinear Drude Input ... 5-43
Advanced Settings ... 5-44
Advanced Settings: Plasma Parameters ... 5-45
Advanced Settings: Gas and Ionization Parameters ... 5-45
Plasma Density Plot ... 5-46
Assigning Frequency Dependent Material: Enter Frequency Dependent Data Points ... 5-49
Spatially Dependent Materials in HFSS ... 5-52
Support for 3D Spatial Modifier Dataset Material Property in HFSS ... 5-54
Advanced Complex Material Properties and Models ... 5-61
Complex Material Properties ... 5-62
Drude Model For Dispersive Metals ... 5-62
Generalized Superconductive Model ... 5-65

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

Specifying Thermal Modifiers 5-68
Defining Material Properties as Expressions 5-75
Defining Functional Material Properties 5-76
Assigning Materials from the Object Properties Window 5-76
Viewing and Modifying Material Attributes 5-76
Validating Materials 5-79
Copying Materials 5-79
Removing Materials 5-79
Exporting Materials to a Library 5-80
Sorting Materials 5-80
Filtering Materials 5-81
Water (Temperature- and Frequency-Dependent Properties) 5-82
Application Procedure 5-83
Relative Permittivity Calculation 5-86
Reference: 5-87
Working with Material Libraries 5-88
Editing Libraries 5-88
6 - Drawing a Model 6-1
Setting Units of Measurement for the Model 6-2
Drawing Objects 6-3
Drawing a Line Segment 6-5
Drawing a Three-Point Arc Line 6-10
Drawing a Center-Point Arc Line 6-12
Drawing a Spline 6-16
Drawing a Polyline 6-19
Assigning a Cross Section and Dimensions to a Polyline 6-21
Inserting Line Segments 6-23
Drawing an Equation-Based Curve 6-24

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

Functions, Operators and Quantities for the Edit Equation Dialog Box 6-28
Drawing a Circle 6-29
Drawing an Ellipse 6-32
Drawing a Rectangle 6-36
Drawing a Regular Polygon 6-39
Drawing an Equation-Based Surface 6-42
Drawing a Sphere 6-46
Drawing a Cylinder 6-50
Drawing a Box 6-54
Drawing a Regular Polyhedron 6-58
Drawing a Cone 6-61
Drawing a Torus 6-64
Drawing a Helix 6-67
Drawing a Segmented Helix with Polygon Cross-Section Using a User Defined Primitive 6-71
Drawing a Segmented Helix with Rectangular Cross-Section Using a User Defined Primitive 6-72
Drawing a Spiral 6-73
Drawing a Spiral Using User Defined Primitives 6-75
Drawing a Bondwire 6-76
Drawing a Point 6-78
Drawing a Plane 6-78
Creating Segmented Geometry 6-79
Segmented Objects 6-80
Drawing Non-Model Objects 6-81
Selecting Non-Model Drawing Mode 6-82
Changing an Object to Non Model 6-82
Drawing a Region 6-82

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

Drawing a SubRegion 6-85
Reassigning a SubRegion 6-91
Group Commands for Modeler Objects 6-94
Model Analysis 6-99
Analysis Options Dialog Box 6-100
Analyzing the Surface Mesh 6-101
Model Analysis Dialog Box 6-101
Objects Tab 6-102
Object Misalignment Tab 6-104
Surface Mesh (Single/Pairs) Tab 6-104
Mesh Feedback Tab 6-105
Design Settings 6-110
Design Settings for HFSS 6-111
Set Material Override Tab 6-113
Lossy Dielectrics Tab 6-114
DC Extrapolation Tab 6-114
Setting the Temperature of Objects 6-123
Model Preparation Commands 6-125
Simplify Command 6-125
Heal 6-132
Validation and Healing Options for Import File 6-134
Healing Non-Manifold Objects 6-134
Setting the Healing Options 6-136
Stitch Sheets 6-137
Align Selected Faces 6-138
Remove Selected Faces 6-139
Remove Selected Edges 6-140
Perform Explicit Subtractions 6-140

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

Creating a User-Defined Primitive 6-141
User Customization through User Defined Primitives (UDPs) 6-143
User Defined Model (UDM) 6-144
UDM Compared to User Defined Primitives 6-145
Insert UDM Command on Draw Menu 6-146
UDM Properties 6-146
UDM Parameters 6-147
UDM Part Edits 6-148
Library of Models for CAD Integration 6-148
3D Modeler Discovery Link 6-150
Importing Discovery Geometry into Ansys Electronics Desktop 6-150
Connecting to an Active Discovery Session 6-152
Property Window of Discovery Link UDM 6-153
Archive a Project Containing Discovery Link 6-156
Break Connection with Discovery Link 6-156
SpaceClaim Integration 6-156
3D Component Library 6-156
Creating a 3D Component from an Existing Model 6-158
Create 3D Component: Model Tab 6-161
Create 3D Component Boundaries Tab 6-162
Create 3D Component Excitations Tab 6-163
Create 3D Component Circuit Elements Tab 6-164
Create 3D Component Hybrid Regions Tab 6-165
Create 3D Component Mesh Tab 6-166
Create 3D Component Coordinate Systems Tab 6-167
Create 3D Component Parameters Tab 6-169
Non-Geometry Variables in 3D Components 6-170
Dependent Variables in 3D Components 6-170

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

Create 3D Component Encryption Tab 6-171
Set Passwords 6-174
Hiding Component Content 6-175
Inserting Encrypted Components 6-176
Editing Encrypted Components 6-176
Opening a 3D Component File 6-177
Copying and Pasting Encrypted Components 6-178
Create 3D Component Licensing Tab 6-178
Create 3D Component Image Tab 6-178
Create 3D Component Info Tab 6-181
Save 3D Component File 6-183
Replace Selection by 3D Component 6-184
Using the Component Libraries Window 6-189
Component file organization on the disk 6-192
Inserting a 3D Component into a Design 6-192
Making Component Instances Independent 6-199
Project Manager 6-199
History Tree 6-199
Viewing Properties of 3D Components 6-200
3D Components in the Project Manager Window 6-203
3D Components in the History Tree 6-204
Editing 3D Component Properties 6-206
Editing a 3D Component Definition 6-210
Verifying Updates to Components 6-213
Updating Definitions 6-213
Updating a 3D Component Definition 6-214
Legacy Projects with 3D Components 6-215
Layout Components [Beta] – Topic Index by Product 6-215

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

Layout Component in HFSS 3D (Beta) 6-216
Layout Component in HFSS 3D Process Flow 6-226
Using Layout Custom CS for Placement in HFSS 3D 6-235
Editing the Placement of a Layout Component 6-240
Visualization for Layout Components in HFSS 3D Modeler 6-241
Using Show Layout for Layout Components 6-241
Object Attributes Control of Visualization for Layout Components 6-243
Visualization for RLC boundaries and Component Groups Categorized as Circuit Elements 6-249
Parameter Support for Layout Components 6-252
Show Nets for Layout Components in HFSS 3D Geometries 6-254
Editing a Layout Component Definition 6-265
Replacing or Editing Layout Components 6-272
Edit/Replace Procedure 6-272
Flex PCB Component Support in HFSS 3D 6-273
Field Plots on Layout Components Layer or Net 6-282
Using Fields Calculator with Layout Component in a Design 6-287
Cable Modeling in HFSS (Beta) 6-290
HFSS Cable Modeling Editor Window 6-292
Beginning with a Straight Wire Cable 6-293
Adding Twisted Pair Cable 6-294
Adding Bundles 6-295
Define Route as Polyline 6-302
Add New Cable Harness 6-302
Defining and Editing Transient Sources 6-311
Parametric Antennas for SBR+ Solution Type 6-317
Adding Parametric Antennas 6-320
Parametric Antenna Descriptions 6-328

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

By File Antenna for SBR+ 6-329
From External Near Field Data 6-331
Link to Source Design 6-337
Conical Horn 6-338
Cross Dipole 6-340
Half-Wave Dipole 6-342
Horizontal Dipole 6-343
Parametric Beam 6-345
Parametric Slot 6-345
Pyramidal Horn 6-347
Quarter-Wave Monopole 6-349
Short Dipole 6-350
Small Loop 6-350
Wire Dipole 6-351
Wire Monopole 6-352
Modifying Objects 6-353
Object and History Editing 6-354
Modify Object Appearance 6-354
Modify Object Dimensions 6-354
Modify Object Location or Orientation 6-354
Surface or Edge Operations for Objects 6-354
Boolean Operations on Objects 6-355
Assigning Color to Objects 6-355
Setting the Default Color of Objects 6-356
Assigning Transparency to an Object 6-357
Setting the Default Transparency of Objects 6-357
Copying and Pasting Objects 6-357
Import a Model from the Clipboard 6-358

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

Copy Image 6-359
Deleting Objects 6-360
Deleting Start Points and Endpoints 6-360
Delete Last Operation 6-361
Cutting Objects 6-362
Moving Objects 6-362
Rotating Objects 6-363
Changing the Orientation of an Object 6-365
Mirroring Objects 6-366
Offsetting Objects 6-368
Duplicating Objects 6-368
Duplicating Objects Along a Line 6-370
Duplicating Objects Around an Axis 6-371
Duplicating and Mirroring Objects 6-372
Scaling Objects 6-373
Sweeping Objects 6-375
Sweeping Around an Axis 6-375
Sweeping Along a Vector 6-377
Sweeping Along a Path 6-379
Best Practices 6-379
Sweeping Faces Along Normal 6-381
Draft Type Options 6-381
Thicken Sheet 6-384
Wrap Sheet Command 6-385
Covering Lines 6-390
Covering Faces 6-390
Uncovering Faces 6-391
Detaching Faces 6-391

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

Detaching Edges ... 6-391
Creating a Cross-Section ... 6-392
Connecting Objects ... 6-392
Project Sheet Object ... 6-393
Moving Faces or Edges ... 6-397
Moving Faces Along the Normal ... 6-397
Moving Faces Along a Vector ... 6-398
Moving Edges Along Normal ... 6-400
Uniting Objects ... 6-401
Subtracting Objects ... 6-401
Creating Objects from Intersections ... 6-403
Creating an Object from a Face ... 6-404
Creating an Object from an Edge ... 6-405
Splitting Objects ... 6-406
Separating Bodies ... 6-410
Converting Polyline Segments ... 6-410
Rounding the Edge of Objects (Fillet Command) ... 6-411
Flattening the Edge of Objects (Chamfer Command) ... 6-413
Imprinting an Object ... 6-416
Imprint Projection Commands ... 6-419
Purging Object History ... 6-422
Generate History to Reproduce Portions of Model ... 6-422
Selecting Items in the 3D Modeler Window ... 6-423
Selecting Objects ... 6-425
Selecting Several Objects ... 6-426
Selecting Objects by Name or Material ... 6-427
Selection by Object Name ... 6-428
Selection by Object Material ... 6-428

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

Select By Area 6-429
Setting the Default Color and Transparency of Selected Objects 6-431
Setting the Default Color of Highlighted Objects 6-431
Creating an Object List 6-432
Reassigning Objects to Another Object List 6-433
Using or Viewing List Objects 6-433
Selecting Faces 6-434
Selecting All Faces of an Object 6-435
Selecting Faces by Name 6-436
Selecting Faces by Plane 6-437
Creating a Face List 6-438
Selecting Edges 6-439
Selecting All Edges of an Object or Face 6-440
Selecting Vertices 6-441
Selecting Multi 6-443
Controlling the Selection in Multi Mode 6-444
Selecting Groups and Submodels 6-445
Clearing a Selection 6-445
Selecting the Face, Edge, Vertex, or Object Behind 6-446
Selecting Objects and Surfaces That Lie Inside Other Objects 6-446
Selecting Objects by Variable 6-449
Assigning Coordinates to New Objects 6-449
Setting the Reference Point 6-450
Defining Cartesian Coordinates 6-450
Defining Cylindrical Coordinates 6-451
Defining Spherical Coordinates 6-451
Using Absolute Coordinates 6-452
Using Relative Coordinates 6-453

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

Choosing the Movement Mode ... 6-453
Moving the Cursor In Plane ... 6-454
Moving the Cursor Out of Plane ... 6-456
Moving the Cursor in 3D Space ... 6-457
Moving the Cursor Along the X-Axis ... 6-458
Moving the Cursor Along the Y-Axis ... 6-459
Moving the Cursor Along the Z-Axis ... 6-460
Measure Modes ... 6-460
Setting Coordinate Systems ... 6-464
Setting the Working Coordinate System ... 6-466
Creating a Relative Coordinate System ... 6-466
Creating an Offset Relative CS ... 6-467
Creating a Rotated Relative CS ... 6-469
Creating an Offset and Rotated Relative CS ... 6-471
Creating a Face Coordinate System ... 6-473
Automatically Creating Face Coordinate Systems ... 6-476
Creating an Object Coordinate System ... 6-476
Creating an Offset Object CS ... 6-477
Creating a Rotated Object CS ... 6-478
Creating an Object CS that is Both Offset and Rotated ... 6-479
Move CS to End Command for History Tree ... 6-480
Modifying Coordinate Systems ... 6-481
Expressing Cartesian, Cylindrical, or Spherical Coordinates ... 6-482
Deleting Coordinate Systems ... 6-483
Setting the Drawing Plane ... 6-483
Cleaning Up Model History ... 6-484
7 - Modifying the 3D Model View ... 7-1
Change the View ... 7-1

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

Show or Hide Design Objects 7-2
Change Object Visualization 7-2
Change the Background 7-3
View Options: 3D UI Options 7-3
Panning the View 7-8
Zooming in and Out 7-9
Zoom In and Zoom Out Commands 7-11
Zoom Previous Command 7-12
Zooming into a Rectangular Area 7-14
Zooming in or out Using a Mouse Wheel 7-15
Zoom to a Selected Boundary or Excitation 7-15
Modifying the View Orientation 7-16
Apply an Orientation to the Current View 7-18
Using the Orientation Gadget 7-20
Controlling Visibility of the Orientation Gadget 7-24
Rotating or Spinning the View 7-25
Changing the Model View with Alt+Double-Click Areas 7-32
Applying a Default View Orientation 7-33
Applying a New View Orientation 7-35
Removing an Orientation 7-36
Fitting Objects in the View Window 7-37
Fitting All Objects in a View Window 7-38
Fitting Selected Objects in a View Window 7-38
Hiding Objects from View 7-39
Showing Objects 7-41
Showing Only Selected Objects in All or Active Views 7-44
Active View Visibility Dialog Box 7-46
Rendering Objects as Wireframes or Solids 7-48

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

Setting the Default View Rendering Mode 7-48
Setting the Enhanced Display of Material Color and Transparency 7-49
Setting the Curved Surface Visualization 7-51
Using Clip Planes 7-52
Modifying the Lighting 7-56
Setting the Projection View 7-58
Setting the Background Color 7-59
Modifying the Coordinate System Axes View 7-60
Showing or Hiding the Axes 7-61
Show the Axes for Selected Objects 7-61
Enlarging or Shrinking the Axes 7-62
Showing or Hiding the Triad Axes 7-62
Choosing Grid Settings 7-62
Setting the Grid Type 7-63
Setting the Grid Style 7-64
Setting the Grid Density and Spacing 7-65
Setting the Grid's Visibility 7-67
Setting the Grid Plane 7-67
8 - Optimetrics 8-1
optiSLang Integration with Electronics Desktop 8-2
Prerequisites 8-3
optiSLang User Workflow 8-4
Parametrization for optiSLang Integration 8-6
Inputs 8-6
Outputs 8-7
Parameterization with optiSLang Analysis Goals in Mind 8-7
Results and Reports for optiSLang Integration 8-8
Creating an optiSLang Setup in AEDT 8-10

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

Setting up Calculations ... 8-11
Setting Options ... 8-13
Finishing Setup ... 8-14
Example optiSLang Setups ... 8-14
Example 1: Solving a User-created, Pre-existing optiSLang Setup ... 8-15
Example 2: Increasing Performance by Avoiding Wasteful Solution of the Nominal Design ... 8-17
Example 3: Avoiding Redundant Computations while Exploiting AEDT Features around Postprocessing Variables ... 8-18
Example 4: Parametrization of an AEDT Design for optiSLang Integration ... 8-18
Solving an optiSLang Setup ... 8-18
Considerations for Using the Analyze All Command ... 8-19
Considerations for Distributed Analysis ... 8-20
optiSLang Menu Options in AEDT ... 8-21
Viewing optiSLang Postprocessing Results ... 8-21
Parametric Overview ... 8-23
Setting Up a Parametric Analysis ... 8-24
Adding a Variable Sweep Definition ... 8-25
Specifying Variable Values for a Sweep Definition ... 8-26
Synchronizing Variable Sweep Definitions ... 8-27
Modifying a Variable Sweep Definition Manually ... 8-28
Overriding a Variable's Current Value in a Parametric Setup ... 8-29
Specifying a Solution Setup for a Parametric Setup ... 8-29
Specifying the Solution Quantity to Evaluate for Parametric Analysis ... 8-30
Setting up Calculations for Optimetrics ... 8-30
Specifying a Solution Quantity's Calculation Range ... 8-32
Viewing Results for Parametric Solution Quantities ... 8-32
Using Distributed Analysis ... 8-33

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

Solving a Parametric Analysis on Another Machine ... 8-34
Adding a Parametric Sweep from a File ... 8-35
Validating a Parametric Setup ... 8-36
Optimization Overview ... 8-38
Choosing an Optimizer ... 8-38
Legacy Optimizers ... 8-40
Quasi Newton (Gradient) ... 8-40
Pattern Search (Search-Based) ... 8-44
Sequential Nonlinear Programming (Gradient) ... 8-46
Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) ... 8-47
Genetic Algorithm (Random Search) ... 8-48
MATLAB Optimizer ... 8-49
Multi-Objective Genetic Algorithm (MOGA) ... 8-54
MOGA Workflow ... 8-55
MOGA Steps ... 8-56
MOGA Steps to Generate a New Population ... 8-57
Convergence Rate % and Initial Finite Difference Delta % in NLPQ and MISQP ... 8-60
Mixed-Integer Sequential Quadratic Programming ... 8-62
Adaptive Multiple Objective Optimization ... 8-63
AMO Workflow ... 8-63
AMO Steps ... 8-64
Adaptive Single Objective Optimization ... 8-66
ASO Workflow ... 8-66
AMO Steps ... 8-68
Optimization Variables and the Design Space ... 8-69
Setting Up an Optimization Analysis ... 8-70
Optimization Setup for the Quasi Newton (Gradient) Optimizer ... 8-72
Optimization Setup for the Pattern Search (Search-based) Optimizer ... 8-74

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

Setting Up Merit-based Sequential Quadratic Programming(Gradient) Optimizer 8-76
Optimization Setup for the Sequential Nonlinear Programming (Gradient) Optimizer 8-77
Optimization Setup for the Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) Optimizer 8-79
Optimization Setup for the Genetic Algorithm (Random search) Optimizer 8-81
Optimization Setup for the MATLAB Optimizer 8-82
Setting Up Screening (Search Based) Optimizer 8-84
Setting Up Multi-Objective Genetic Algorithm(Random Search) Optimizer 8-87
Setting Up Nonlinear Programming by Quadratic Lagrangian (Gradient) Optimizer 8-92
Setting Up Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete) Optimizer 8-97
Setting Up Adaptive Multiple-Objective (Random Search) Optimizer 8-102
Setting Up Adaptive Single-Objective (Gradient) Optimizer 8-108
Setting the Maximum Iterations for an Optimization Analysis 8-113
Cost Function 8-113
Acceptable Cost 8-114
Cost Function Noise 8-114
Adding a Cost Function 8-115
Specifying a Solution Quantity for a Cost Function Goal 8-118
Setting the Calculation Range of a Cost Function Goal 8-118
Adding or Editing a Cost Function Calculation 8-119
Setting a Goal Value 8-121
Specifying a Single Goal Value 8-121
Specifying an Expression as a Goal Value 8-121
Specifying a Variable-Dependent Goal Value 8-122
Goal Weight 8-122
Modifying the Starting Variable Value for Optimization 8-124
Setting the Min. and Max. Variable Values for Optimization 8-124

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

Edit Calculation Range 8-125
Overriding the Min. and Max. Variable Values for a Single Optimization Setup 8-127
Changing the Min. and Max. Variable Values for Every Optimization Setup 8-128
Step Size 8-128
Setting the Min. and Max. Step Sizes 8-129
Setting the Min and Max Focus 8-130
Equalizing the Influence of Different Optimization Variables 8-130
Setting the Min and Max Focus Values 8-131
Solving a Parametric Setup Before an Optimization 8-131
Solving a Parametric Setup During an Optimization 8-131
Automatically Updating a Variable's Value after Optimization 8-132
Changing the Cost Function Norm 8-132
Explanation of L1, L2, and Max Norms in Optimization 8-132
Example of a More Complex Cost Function 8-135
Advanced Genetic Algorithm Optimizer Options 8-136
Sensitivity Analysis Overview 8-139
Selecting a Primary Output 8-139
Setting Up a Sensitivity Analysis 8-139
Setting the Maximum Iterations Per Variable 8-141
Setting Up an Output Parameter 8-142
Specifying a Solution Quantity for an Output Parameter 8-142
Setting the Calculation Range of an Output Parameter 8-144
Modifying the Starting Variable Value for Sensitivity Analysis 8-144
Setting the Min. and Max. Variable Values 8-145
Overriding the Min. and Max. Variable Values for a Single Sensitivity Setup 8-145
Changing the Min. and Max. Variable Values for Every Sensitivity Setup 8-146
Setting the Initial Displacement 8-146
Solving a Parametric Setup before a Sensitivity Analysis 8-146

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

Solving a Parametric Setup during a Sensitivity Analysis ... 8-147
Performing Worst Case Analysis ... 8-147
Statistical Analysis Overview ... 8-151
Setting Up a Statistical Analysis ... 8-152
Setting the Maximum Iterations for a Statistical Analysis ... 8-153
Specifying the Solution Quantity to Evaluate for Statistical Analysis ... 8-154
Setting the Solution Quantity's Calculation Range ... 8-155
Setting the Distribution Criteria ... 8-156
Overriding the Distribution Criteria for a Single Statistical Setup ... 8-156
Changing the Distribution Criteria for Every Statistical Setup ... 8-157
Statistical Cutoff Probability ... 8-158
Edit Distribution ... 8-159
Modifying the Starting Variable Value for Statistical Analysis ... 8-160
Solving a Parametric Setup During a Statistical Analysis ... 8-161
Using Design of Experiments ... 8-161
Setting Up Design of Experiments in Ansys Electronics Desktop ... 8-164
Add the Design of Experiments Setup ... 8-164
Design of Experiments Tab ... 8-168
Optimal Space Filling Design (OSF) ... 8-170
Central Composite Design (CCD) ... 8-172
Box Behnken Design (CCD) ... 8-173
Custom DOE Type ... 8-173
Latin Hypercube Sampling ... 8-174
Table Tab for Design of Experiments ... 8-175
Response Surface Tab for Design of Experiments ... 8-176
Variables Tab for Design of Experiments ... 8-178
Viewing Analysis Result for Design of Experiments ... 8-181
Min-Max Search View for Design of Experiments Result ... 8-184

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

Refinement Points Table ... 8-184
Performing a Manual Refinement ... 8-185
Response Points Table ... 8-185
Response Curve ... 8-186
Response Curve (2D Slices) ... 8-189
Verification Points Table ... 8-191
Goodness of Fit (Predicted vs Observed Chart) ... 8-192
Response Surface Results ... 8-193
Local Sensitivity Charts ... 8-196
Local Sensitivity Curves ... 8-197
Tuning Overview ... 8-199
Tuning a Variable ... 8-199
Applying a Tuned State to a Design ... 8-202
Saving a Tuned State ... 8-202
Reverting to a Saved Tuned State ... 8-203
Resetting Variable Values after Tuning ... 8-203
Saving Field Solutions for Optimetrics Analyses ... 8-203
Saving Field Solutions for a Parametric Setup ... 8-204
Saving Field Solutions for an Optimization Setup ... 8-205
Saving Field Solutions for a Sensitivity Setup ... 8-206
Saving Field Solutions for a Statistical Setup ... 8-207
Saving Field Solutions for a Design of Experiments Setup ... 8-207
Copying Meshes in Optimetrics Sweeps ... 8-208
Adding an Expression in the Output Variables Window ... 8-209
Excluding a Variable from an Optimetrics Analysis ... 8-209
Modifying the Value of a Fixed Variable ... 8-210
Linear Constraints ... 8-211
Setting a Linear Constraint ... 8-211

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

Modifying a Linear Constraint ... 8-212
Deleting a Linear Constraint ... 8-212
Running an Optimetrics Analysis ... 8-213
Viewing Analysis Results for Optimetrics Solutions ... 8-213
Viewing Solution Data for an Optimetrics Design Variation ... 8-214
Viewing an Optimetrics Solution's Profile Data ... 8-214
Viewing Results for Parametric Solution Quantities ... 8-215
Plotting Solution Quantity Results vs. a Swept Variable ... 8-215
Viewing Cost Results for an Optimization Analysis ... 8-216
Plotting Cost Results for an Optimization Analysis ... 8-217
Viewing Output Parameter Results for a Sensitivity Analysis ... 8-217
Plotting Output Parameter Results for a Sensitivity Analysis ... 8-217
Viewing Distribution Results for a Statistical Analysis ... 8-218
Plotting Distribution Results for a Statistical Analysis ... 8-219
Link to DesignXplorer ... 8-219
9 - High Performance Computing ... 9-1
Remote Analysis ... 9-1
Prerequisites for Remote and Distributed Analysis ... 9-2
Configuring the Local Machine to Solve Remotely ... 9-3
Remote Analysis Options ... 9-3
Running Remote Analysis ... 9-4
Troubleshooting ... 9-5
Remote Solve Node = Windows ... 9-7
Remote Solve Node = Linux ... 9-8
Distributed Analysis ... 9-9
Beta Feature: Parallel Component Mesh Adapt for 3D Component Array ... 9-9
Configure a Distributed Analysis ... 9-10
Distributed Analysis Configuration ... 9-10

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

Distributed Analysis Configuration - Machines Tab ... 9-13
Manually Adding a Machine ... 9-14
Importing a Machine List ... 9-14
Machine List Details ... 9-15
Testing Machines ... 9-16
Distributed Analysis Configuration - Job Distribution Tab ... 9-16
Distribution Levels ... 9-20
Single-level Distributions ... 9-20
Two-Level Distributions ... 9-21
HFSS Frequency Distribution ... 9-23
Distributed Analysis Configuration - Options Tab ... 9-26
Relation to Batchoptions ... 9-27
Adding Configurations or Accepting Edits ... 9-28
Enabling GPU ... 9-28
Selecting Optimal Configurations for Distributed Analysis ... 9-32
Two-Level Distribution Guidelines ... 9-32
Domain Decomposition Solver HPC Configuration Guidelines ... 9-33
Distributed Direct Solver Guidelines for HPC Configuration ... 9-34
Distributed Iterative Solver Guidelines for HPC Configuration ... 9-34
Distributed HFSS-IE Guidelines for HPC Configuration ... 9-35
Distributed Mesh Assembly Guidelines for HPC Configuration ... 9-35
Large Scale DSO for Parametric Analysis ... 9-35
Prerequisites for Large Scale DSO ... 9-37
DSO Behavior in HFSS ... 9-38
Job Management Interface for Large Scale DSO ... 9-40
Large Scale DSO Command Line Syntax ... 9-46
Large Scale DSO Job Outputs ... 9-50
Output Location and Organization ... 9-50

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

CSV File Contents 9-50
Postprocessing Large Scale DSO Dataset Solutions 9-51
Importing a Large Scale DSO Solution 9-51
Viewing Imported Solutions 9-52
Creating a Dataset Report 9-53
Cloning a Dataset Solution 9-53
Large Scale DSO Job Monitoring 9-53
Additional Resources for Large Scale DSO Monitoring 9-54
Large Scale DSO Deployment/Configuration 9-54
Large Scale DSO Tutorial Example 9-56
Prepare the Model for Large Scale DSO Analysis 9-56
Submit the Large Scale DSO Job: Examples 9-61
Large Scale DSO Example: Post Process the Results 9-63
Process Limits for Large Scale DSO on Windows 9-65
Large Scale DSO Known Issues/Troubleshooting 9-65
Node Order 9-65
Parallel Task Limitation for LS-DSO Parametric Variations 9-66
Job Restart 9-66
Linux-Only Issues 9-67
Interactive Scheduler Jobs 9-67
Specifying Options for Interactive Scheduler Jobs 9-68
Batchoptions for Interactive Scheduler Jobs 9-68
DSO Configuration for Interactive Scheduler Jobs 9-70
Analysis Configuration - Machines Tab 9-71
Analysis Configuration - Job Distribution Tab 9-72
Analysis Configuration - Options Tab 9-73
Design Type Options for Interactive Scheduler Jobs 9-74
High Performance Computing (HPC) Integration 9-75

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

Scheduler Terminology 9-75
What a Scheduler Does 9-76
Configuring Electronics Installation for HPC 9-77
Firewall Configuration 9-78
Installation Directory Examples 9-78
Integration with Microsoft Windows® HPC Scheduler 9-78
Submitting and Monitoring Ansys EM HPC Jobs 9-79
Submitting and Monitoring Jobs for Windows HPC 9-81
Windows® HPC Job Templates 9-99
Selecting Computation Resource Units (Job Unit Type) 9-99
Windows® HPC Non-Exclusive Jobs 9-100
Windows® HPC Job Credentials 9-112
Integration with Grid Engine (GE) 9-113
GE Job Management 9-113
Installation of Ansys EM Tools on GE 9-115
GE Commands for Information about Jobs and Cluster Configuration 9-117
Monitoring GE Serial and Parallel Batch Jobs 9-120
Ansys EM Desktop -monitor Command Line Option for SGE 9-120
Example SGE qsub Command Lines 9-120
Recommended Practices for GE Clusters 9-122
Issue with qrsh (SGE) 9-127
Issue with MainWin Core Services for SGE 9-127
Integration with Platform's Load Sharing Facility (LSF) 9-129
LSF Job Management 9-129
LSF Job Submission Guidelines 9-130
Integration of Ansys EM Products with LSF 9-130
Installing Ansys EM Tools on LSF Cluster 9-131
Using the bsub Command to Submit Batch Jobs 9-133

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

bsub Arguments ... 9-134
Example LSF bsub Command Lines (Linux Only) ... 9-135
Serial Job ... 9-135
Serial Job Requiring a Minimum of 4GB ... 9-135
Multi-processing Job using 4 Cores ... 9-135
Distributed Processing Job using 4 Engines ... 9-135
Distributed Processing and Multi-processing Job using 4 Cores, with 2 Cores for Multi-processing ... 9-136
Shell Script (~/projects/OptimTee.csh): ... 9-136
Monitoring LSF Batch Jobs ... 9-136
Terminating LSF Batch Jobs ... 9-137
LSF Known Issues and Workarounds ... 9-138
LSF Troubleshooting ... 9-138
Integration with PBS (Portable Batch System) ... 9-139
PBS Job Management ... 9-139
Non Standard Installations for PBS ... 9-140
PBS Limitations ... 9-140
Submitting Ansys EM PBS Batch Jobs ... 9-142
Monitoring PBS Batch Jobs ... 9-143
qsub Arguments ... 9-144
Example PBS qsub Command Lines ... 9-145
Integration with Ansys Remote Simulation Management (RSM) ... 9-147
Changing the AnsoftRSMService Listening Port ... 9-161
AnsoftRSMService Configuration ... 9-162
Ansoft Electromagnetics Desktop Configuration ... 9-163
Integration with SLURM Scheduler (Linux Utility) ... 9-165
Configuration of the generic scheduler with SLURM ... 9-165
Environment Variables ... 9-166

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

Configuration File ... 9-167
Persistence Mode ... 9-167
Remote RSM Service Support ... 9-168
Using the generic scheduler with SLURM ... 9-168
Using MPI Tight Integration ... 9-169
Using the Command Line to Submit HPC Jobs ... 9-169
Distributed Jobs ... 9-169
Serial Job on a Single Processor ... 9-170
Distributed Job Using Four Processors ... 9-170
Multiprocessing Job Using Four Cores ... 9-170
Distributed Analysis and Multi-Processing in the Same Job ... 9-171
Integrating Ansys EM Tools with Third-Party Schedulers ... 9-171
Introduction ... 9-171
Serial Jobs ... 9-171
Parallel Jobs ... 9-172
Common Requirements for Running Jobs ... 9-172
Using a Shared Library (Linux) or a DLL (Microsoft Windows) ... 9-173
Build Information for Scheduler Proxy Library ... 9-173
Implementation Details for Custom Scheduler Integration ... 9-174
IsProductLaunchedInYourEnvironment ... 9-175
GetTempDirectory ... 9-175
GetMachineListAvailableForDistribution ... 9-176
GetMessageStringToRegisterForSigTerm ... 9-177
LaunchProcess ... 9-178
GetUseRsmForEngineLaunch ... 9-179
GetThisJobID ... 9-180
GetSchedulerDisplayName ... 9-181
Scheduler Proxy Interfaces ... 9-182

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

Extending Scheduler Integration with Tight-Integration for MPI 9-187
Support for Open MPI on Linux [Beta] 9-190
Testing Scheduler Integration 9-194
Testing IsProductLaunchedInYourEnvironment 9-195
Testing GetSchedulerDisplayName and GetThisJobID 9-195
Testing GetTempDirectory 9-195
Testing GetMachineListAvailableForDistribution 9-196
Testing LaunchProcess 9-196
Testing GetUseRsmForEngineLaunch 9-197
Troubleshooting Custom Scheduler Integration 9-197
None of the Proxy Functions are Called 9-197
Troubleshooting IsProductLaunchedInYourEnvironment Function 9-198
Troubleshooting GetSchedulerDisplayName 9-198
Troubleshooting GetThisJobID 9-198
Troubleshooting GetTempDirectory 9-199
Troubleshooting GetMachineListAvailableForDistribution 9-199
Troubleshooting LaunchProcess 9-199
Troubleshooting GetUseRsmForEngineLaunch 9-200
Using an IronPython Program for Integration with a Scheduler 9-200
GetName [IronPython] 9-202
GetDescription [IronPython] 9-202
IsProductLaunchedInYourEnvironment [IronPython] 9-203
GetSchedulerDisplayName [IronPython] 9-203
GetThisJobID [IronPython] 9-204
GetUseRsmForEngineLaunch [IronPython] 9-204
GetTempDirectory [IronPython] 9-205
GetMessageStringToRegisterForSigTerm [IronPython] 9-205
GetMachineListAvailableForDistribution [IronPython] 9-206

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

LaunchProcess [IronPython] 9-207
Selecting a Scheduler 9-208
Submitting a Job 9-210
Using Advanced Job Submission Options 9-215
Batchoptions 9-216
Environment Variables 9-217
Batch Extract 9-218
Customize Job Submission 9-219
Using the Command Line to Submit HPC Jobs 9-219
Distributed Jobs 9-219
Serial Job on a Single Processor 9-220
Distributed Job Using Four Processors 9-220
Multiprocessing Job Using Four Cores 9-220
Distributed Analysis and Multi-Processing in the Same Job 9-221
Job Import and Export 9-221
Scripting 9-221
Multi-Step Job Submission 9-221
Windows to Linux Job Submission 9-227
Prerequisites for Job Submission 9-228
Prerequisites for Job Monitoring 9-229
Supported Schedulers 9-229
Job Submission Scripting 9-235
Monitoring Jobs 9-236
Monitor Job Window 9-236
Monitoring Ansys Cloud Direct Jobs 9-239
Monitoring Ansys Cloud Burst Jobs 9-241
Monitoring Large Scale DSO Jobs 9-242
Web Client for Batch Solve Monitoring and Reporting 9-242

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

Aborting an Analysis ... 9-248
Distribution Command Line Options ... 9-249
Ansys Cloud Direct Support for HPC Job Management ... 9-252
Prerequisites for Using Ansys Cloud Direct with Ansys Electronics Desktop ... 9-252
Job Management UI for Ansys Cloud Direct ... 9-253
Environment Variable Settings for Ansys Cloud Direct ... 9-263
Troubleshooting and Debugging Environment Variables ... 9-263
Option Settings for Ansys Cloud Direct ... 9-264
Automatic Batchoption Settings ... 9-264
Ansys Cloud Burst Support for HPC Job Management ... 9-264
Prerequisites for Using Ansys Cloud Burst Compute with Ansys Electronics Desktop ... 9-265
Ansys Cloud Burst Compute-Specific Settings ... 9-266
Ansys HPC Platform Services Support for HPC Job Management ... 9-268
Prerequisites for Using Ansys HPC Platform Services with Ansys Electronics Desktop ... 9-269
Job Management UI for Ansys HPC Platform Services ... 9-270
Ansys HPC Platform Services-Specific Settings: ... 9-270
Aborting an Analysis ... 9-276
Cluster Specific Considerations ... 9-276
Option Settings for Ansys HPC Platform Services ... 9-277
GPU Acceleration ... 9-277
License Options ... 9-277
Enable/Disable GPU Acceleration from the Ansys Electronics Desktop Solver User Interface ... 9-277
Supported GPUs by Solver ... 9-282
GPU Installation and Setup ... 9-283
Integration with Ansys Remote Simulation Management (RSM) ... 9-290
Changing the AnsoftRSMService Listening Port ... 9-304

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

AnsoftRSMService Configuration 9-305
Ansoft Electromagnetics Desktop Configuration 9-306
Running HPC Diagnostics 9-308
Site-Specific Diagnostics Job 9-311
Changing Solution Priority for System Resources 9-313
desktopproxy Utility to Abort or Cleanstop a BatchSolve 9-314
10 - Defining Mesh Operations 10-1
Length Based Mesh Refinement 10-2
Assigning Length-Based Mesh Refinement on Object Faces 10-4
Assigning Length-Based Mesh Refinement Inside Objects 10-5
Assigning Skin Depth-Based Mesh Refinement on Object Faces 10-6
Visualization for Skin Depth Refinement Mesh Results 10-11
Modifying Surface Approximation Settings 10-13
Applying Curvilinear Elements 10-17
SBR+ Curvature Extraction for Faceted Surfaces 10-18
Specifying Surface Priority for TAU Mesher 10-19
Specifying the Model Resolution 10-20
Reverting to the Initial Mesh 10-22
Generating Mesh without Solving 10-23
Specifying Initial Mesh Settings 10-24
TAU Flex Meshing 10-30
Steps for Using TAU Flex 10-31
TAU Flex Warning Messages 10-35
Reviewing TAU Flex Mesh Warnings 10-35
Addressing TAU Flex Mesh Issues 10-37
Deleting Previously Assigned Mesh Refinements 10-37
Viewing List of Mesh Operations 10-38
Reassigning a Mesh Operation 10-39

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

Viewing Mesh Properties 10-40
11 - Example Toolkits **11-1**
HFSS ACT 5G Wizard 11-2
5G Wizard: Setup 11-4
5G Wizard: Power Density Settings 11-5
Power Density Calculation Overview 11-5
Beam ID Variable Select 11-7
Power Density Surface 11-7
Step Size (mm) 11-8
Averaging Area (cm2) 11-8
Power Density Worst Case 11-8
5G Wizard: Codebook 11-8
Beam ID Codebook 11-9
Include Beam Pair 11-11
Update Beam ID Codebook (Applies immediately) 11-11
5G Wizard: Cumulative Distribution Function 11-11
Beam ID Variable Select 11-12
Quantity 11-12
5G Wizard: Example Reports 11-12
HFSS Antenna Design Toolkit 11-15
Using the HFSS Antenna Wizard 11-17
HFSS-EMA3D Datalink Toolkit 11-50
HFSS RADARpre ACT Toolkit 11-56
Using the RADARpre Wizard 11-59
HFSS RADARpost ACT Toolkit 11-62
Using the RADARpost Wizard 11-64
Cable Modeling - Automotive Toolkit 11-68
Cable Modeling - Oil and Gas Toolkit 11-72

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

Finite Array Beam Angle Calculator Toolkit 11-75
Hearing Aid Compliance Test 11-93
Using the Hearing Aid Compliance Test 11-94
User Defined Solution for MIMO Calculations 11-99
Wavelength Calculator 11-105
12 - HFSS Transient **12-1**
Materials for Transient Solution Types 12-2
Diagonal Anisotropic Material Support for the Implicit Transient Solver 12-3
Adding a Solution Setup for Transient Solutions 12-6
Transient Network Analysis Input Signal Tab: a Time Profile 12-7
Duration Tab for Transient Solution Setup 12-10
Saved Fields Tab for Transient Solution Setup 12-11
Multimode Waveguide Simulations Transient 12-12
Active or Passive Excitations in HFSS Transient 12-14
Setting Magnitude and Delay for Transient Composite Solution Excitations 12-17
Differential Pairs in HFSS Transient Network 12-18
Design Settings for HFSS Transient 12-19
Transient Solution Data 12-21
Selecting the Report Type in HFSS Transient 12-23
Reports for Transient Network with Differential Pairs 12-24
Procedure for Viewing Transient Radiated Fields 12-25
13 - HFSS with SBR+ **13-1**
Using SBR+ with HFSS 13-2
Design Flow for SBR+ Solution Type 13-2
Using SBR+ Solution Type for HFSS 13-3
3D Components for and from SBR+ Designs 13-27
Configure Simulation with SBR+ Antenna Geometry Blockage (SBR+ Solution Type) 13-28
Current Limitations on PTD/UTD Wedge Generation/Editing 13-31

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

SBR+ Range-Doppler Solution Setup 13-35
Using SBR+ Solution Type for HFSS SBR+ Range Doppler Designs 13-37
Chirp Sequence FMCW for SBR+ Range Doppler Solution Setup 13-74
Surface Roughness for SBR+ 13-79
Lightweight Geometry for HFSS SBR+ Designs 13-83
Initial Mesh Operations for Lightweight Geometries 13-85
Permitted Modeler Operations on Faceted Objects 13-85
Other Characteristics of Faceted Objects 13-85
SBR+ Curvature Extraction for Faceted Surfaces 13-85
Assigning SBR+ Hybrid Regions 13-86
Configure Simulation with SBR+ Antenna Geometry Blockage (HFSS with Hybrid and Array Solution Type): 13-105
Assigning (SBR+) Fresnel Table Boundaries 13-108
Assigning (SBR+) Perfect Absorber Boundaries 13-113
Assigning a Dielectric or Conducting Material to Volumetric Region 13-114
Parametric Antennas for SBR+ Solution Type 13-117
Adding Parametric Antennas 13-120
Parametric Antenna Descriptions 13-128
By File Antenna for SBR+ 13-129
From External Near Field Data 13-131
Link to Source Design 13-137
Conical Horn 13-138
Cross Dipole 13-140
Half-Wave Dipole 13-142
Horizontal Dipole 13-143
Parametric Beam 13-145
Parametric Slot 13-145
Pyramidal Horn 13-147

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

Quarter-Wave Monopole 13-149
Short Dipole 13-150
Small Loop 13-150
Wire Dipole 13-151
Wire Monopole 13-152
Spherical Wave Expansion Source (*.swef) 13-153
AB Coefficients 13-155
Q Coefficients 13-157
Parametric Array Antennas for SBR+ Solution Type 13-160
Using with Other SBR+ Features 13-166
Primary Workflows for Parametric Antenna Arrays 13-168
Create and Use Parametric Array in SBR+ Simulation 13-168
Summary Workflow for Parametric Antenna Arrays 13-168
Detailed Workflow for using Parametric Antenna Arrays in SBR+ Simulation 13-169
Create and Use Custom Array By File in SBR+ Simulation 13-187
Summary Workflow for Custom Array By File Antenna Arrays 13-187
Detailed Workflow for using Custom Array by File... in SBR+ Simulation 13-189
Array Element List (*.sarr) Format 13-200
Header Keys 13-201
Link from Driven Modal or Terminal Antenna to SBR+ Design 13-204
Creating a Link from Driven Modal or Driven Terminal Source to SBR+ Target 13-208
Simulation Setup for Driven Modal Antenna to SBR+ Design Link 13-225
Viewing Results in the SBR+ Simulation 13-227
Requirements for Creating a Gain Report for SBR+ Design with Driven Modal or Terminal Source 13-228
Link as Composite Port 13-229
Linked Antenna Variable Support 13-232
Set SBR+ Source Regions 13-233

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

Set Current Source Option 13-239
NF Accuracy Settings SBR+ Advanced Option 13-241
Select Tx/Rx Antennas for SBR+ Solutions 13-243
Prerequisites 13-243
To Assign Transmit and Receive Assignments for Antennas in SBR+ 13-244
Visual Ray Trace (VRT) for SBR+ Simulation 13-251
Prerequisites 13-252
Using Visual Ray Trace (VRT) in Ansys Electronics Desktop 13-252
Create VRT (Visual Ray Trace) Plot with Geometry Blockage 13-256
VRT Plot Properties 13-258
Modifying VRT Plots 13-260
Design Edits that Invalidate the Mesh Invalidate VRT Plots 13-261
Visual Ray Trace SBR Folder Properties 13-262
Rendering Properties Tab 13-262
Filters Properties Tab 13-264
Rx Filter Properties Tab 13-268
SBR+ with PTD and UTD 13-269
Parametric Antennas and Parametric Array Antennas for VRT Plots 13-270
Summary Workflow 13-270
Detailed Workflow 13-271
Creeping Wave VRT Plots 13-276
Using Visual Ray Trace (VRT) with Creeping Wave for RCS 13-276
Creeping Wave Plot Properties 13-281
Modifying Creeping Wave Plots 13-283
Design Edits that Invalidate the Mesh Invalidate Creeping Ray VRT Plots 13-284
Visual Ray Trace CW Folder Properties 13-285
Rendering Properties Tab 13-285
Filter Properties Tab 13-288

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

Using Visual Ray Trace (VRT) with Creeping Wave for Antenna Placement 13-293
Creeping Wave Plot Properties 13-296
Modifying Creeping Wave Plots 13-297
Design Edits that Invalidate the Mesh Invalidate Creeping Ray VRT Plots 13-299
Visual Ray Trace CW Folder Properties 13-299
Rendering Properties Tab 13-299
Filter Properties Tab 13-301
Creeping Wave for Antenna Placement on Curved Surfaces in SBR+ Simulation 13-302
Prerequisites 13-303
Process Flow for Setting up and Simulating an SBR+ Design for Creeping Waves for Antenna Placement 13-304
Using Visual Ray Trace (VRT) with Creeping Wave for Antenna Placement 13-309
Creeping Wave Plot Properties 13-313
Modifying Creeping Wave Plots 13-314
Design Edits that Invalidate the Mesh Invalidate Creeping Ray VRT Plots 13-316
Visual Ray Trace CW Folder Properties 13-316
Rendering Properties Tab 13-316
Filter Properties Tab 13-318
Creeping Wave for RCS Analysis of Curved Surfaces in SBR+ Simulation 13-323
Prerequisites 13-325
Process Flow for Setting up and Simulating an SBR+ Design for Creeping Waves for RCS 13-325
Exporting ROM Radar to VRXPERIENCE (Beta) 13-328
Prerequisites 13-329
Synthesizing and Exporting an FF-RRP ROM File 13-330
Near Field Analysis using Antenna Ports in SBR+ Design 13-344
Prerequisites 13-345
Process Flow for Near Field Analysis in SBR+ Design 13-346

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

Process Flow for Near Field Analysis in HFSS Modal or Terminal Design with SBR+ Hybrid Regions 13-354
Near Field Analysis using Incident Plane Wave in SBR+ Design 13-361
Prerequisites 13-362
Technical Notes 13-363
Process Flow for Near Field Analysis in SBR+ Design 13-363
Process Flow for Near Field Analysis in HFSS Modal or Terminal Design with SBR+ Hybrid Regions 13-372
SBR+ Fast Frequency Looping for High Density Frequency Sweeps 13-380
Technical Background 13-382
Limitations on the Fast Frequency Looping Feature 13-384
Summary Workflow: 13-385
SBR+ Enhanced Radiated Power Accuracy 13-386
14 - HFSS Multipaction Analysis 14-1
Prerequisites 14-2
General Multipaction Analysis Process 14-2
Prepare a Driven Modal or Driven Terminal Design 14-2
Add Multipaction Charge Region Excitations 14-2
Adding Multipaction Secondary Electron Emission (SEE) Boundaries 14-4
Adding a Multipaction Setup 14-6
Optional Multipaction DC Bias 14-15
Running the Multipaction Simulation 14-17
Plotting the Charge Evolution 14-18
Displaying and Saving Charged Particle Animations 14-21
To Store and Obtain Power via a Script 14-27
15 - HFSS RF Discharge 15-1
Save Fields for a Driven Terminal or Driven-Terminal Project 15-1
Add an RF Discharge Analysis to a Driven Solution 15-2

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

Select the Source and Port Excitations 15-4
Gas Type for RF Discharge Analysis 15-6
Add an RF Discharge Analysis to an Eigenmode Solution 15-6
(Optional) Assign DC Magnetic Bias 15-8
Run RF Discharge Simulation 15-10
Create an RF Discharge Report 15-10
(Optional) Change the Pressure Unit 15-12
Create an Electron Density Plot or the RF Simulation 15-14
16 - Specifying Solution Settings 16-1
Adding a Solution Setup to an HFSS Design 16-2
Auto Solution Setup 16-3
Advanced Solution Setup 16-7
Add Dependent Solve Setup 16-14
Renaming a Solution Setup 16-14
Copying a Solution Setup 16-15
Deleting a Solution Setup 16-15
Disabling and Enabling an Analysis Setup 16-15
Setting the Solution Frequency 16-16
Solving for Ports Only 16-20
Setting the Minimum Frequency 16-20
Setting the Number of Modes 16-21
Adding a Solution Setup to an HFSS with Hybrid and Arrays Design 16-21
Auto Solution Setup 16-22
Advanced Solution Setup 16-27
Adding an Eigenmode Solution Setup 16-35
Adaptive Meshing Methods In HFSS 16-37
Single Solution Frequency 16-38
Broadband Frequency 16-39

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

Multi-Frequencies 16-42
Setting Adaptive Analysis Parameters for HFSS 16-44
Setting Hybrid Region Parameters for HFSS 16-47
Additional Options for SBR+ 16-48
MLFMM Usage Guidelines 16-57
The MLFMM Solver 16-57
Further Details of the MLFMM Algorithm 16-59
Guidelines for Choosing Between ACA and MLFMM IE Solver Options 16-60
Decision Flowchart for Deciding on the IE Solver Options 16-62
Setting the Maximum Number of Passes 16-62
Setting the Maximum Delta S Per Pass 16-63
Setting the Maximum Delta Energy Per Pass 16-65
Setting the Maximum Delta Frequency Per Pass 16-65
Specifying Convergence on Real Frequency Only 16-65
Specifying Expressions for Adaptive Convergence 16-66
Specifying a Source for the Initial Mesh 16-69
Clearing Linked Data 16-72
Mesh Linking 16-72
Mesh Link Requirements 16-72
Setting Lambda Refinement 16-72
Setting the Percent Maximum Refinement Per Pass 16-74
Setting the Maximum Refinement 16-74
Setting the Minimum Number of Passes 16-74
Setting the Minimum Number of Converged Passes 16-75
Setting Matrix Convergence Criteria 16-75
Setting the Order of Basis Functions 16-77
Enable Direct Solver 16-77
Enable the Direct Solver 16-78

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

Enable Iterative Solver ... 16-78
Enable the Iterative Solver ... 16-78
Enable Domain Decomposition ... 16-78
Beta Option for Domain Decomposition Allowing Distributed Direct Solver ... 16-81
Distributed Right-Hand-Side Solver in Domain Decomposition ... 16-82
Balancing Resources for Domains ... 16-82
Domains with FE-BI Radiation Boundaries or IE Regions ... 16-83
Use Radiation Boundary on Ports ... 16-83
Port Options ... 16-83
Beta Option: Enhanced Tau Port Meshing ... 16-85
Produce Derivatives for Selected Variables ... 16-85
Units for Calculated Derivatives ... 16-87
Adding a Frequency Sweep (HFSS) ... 16-87
Selecting the Sweep Type ... 16-90
Options for Discrete Sweeps ... 16-91
Options for Fast Sweeps ... 16-92
Options for Interpolating Sweeps ... 16-93
DC Extrapolation options ... 16-95
Set up Interpolating Sweep Advanced Options ... 16-96
Setting the Error Tolerance ... 16-99
Setting the Maximum Number of Solutions ... 16-100
Interpolation Basis Convergence ... 16-100
Specifying the Frequency Points to Solve ... 16-102
Specifying Frequency Points with a Linear Step Size ... 16-103
Specifying a Linear Count of Frequency Points ... 16-106
Specifying a Logarithmic Spaced Frequency Sweep ... 16-107
Specifying Single Frequency Points ... 16-107
Change the Value of an Existing Frequency Point ... 16-108

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

Deleting Frequency Points ... 16-109
Inserting Frequency Points ... 16-109
Choosing Frequencies for Full-Wave SPICE ... 16-110
Guidelines for Calculating Frequencies for Full-Wave SPICE ... 16-111
Requirements for Full-Wave SPICE ... 16-112
Disabling or Enabling a Frequency Sweep ... 16-112
Specifying the Number of Processors ... 16-113
Specifying the RAM Limit Percentage ... 16-113
Distributed Memory Solutions with HFSS ... 16-115
Installation Requirements for Distributed Memory Solutions with HFSS ... 16-116
Setting up HFSS and Running Distributed Memory Solutions ... 16-116
MPI Vendor and MPI Version ... 16-117
Setting the Intel MPI Interconnect ... 16-119
Running Distributed Memory Solutions from the Command line ... 16-120
Discussion of HFSS Distributed Memory Solutions ... 16-120
Interconnects for HFSS Distributed Memory Simulation ... 16-122
Authentication on Linux (RSH & SSH) for HFSS ... 16-122
Troubleshooting for Distributed Memory Simulations ... 16-122
17 - Assigning Boundaries in HFSS ... 17-1
Boundaries for Open or Closed Models ... 17-3
Boundaries for Simplifying Models ... 17-4
Zoom to Selected Boundary ... 17-5
Setting Default Boundary, Hybrid Region and Excitation Base Names ... 17-5
Excitation Name Ordering Conventions ... 17-6
Automation in Setting up an Open Problem ... 17-7
Recalculate the Padding for Regions ... 17-11
Assigning Perfect E Boundaries ... 17-11
Assigning Perfect H Boundaries ... 17-12

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

Assign Impedance Boundaries 17-13
Assigning Radiation Boundaries 17-14
Set as Frequency Selective Surface Reference 17-15
Assigning PML Boundaries 17-16
Creating PMLs Automatically 17-19
Creating PML Boundaries Manually 17-24
Guidelines for Assigning PML Boundaries 17-29
Modifying PML Boundaries 17-30
Assigning Finite Conductivity Boundaries 17-31
Spatially Dependent Boundaries 17-35
Assigning Symmetry Boundaries 17-38
Assigning Coupled Lattice Pair Boundaries 17-39
Assigning Primary Boundaries for a Coupled Lattice Pair 17-43
Assigning Secondary Boundaries for Lattice Pair 17-45
Auto Identify Lattice Pair for Coupled Boundaries 17-49
Assigning Lumped RLC Boundaries 17-51
Vector Line 17-52
Assigning Anisotropic Impedance Boundaries 17-52
Assigning Layered Impedance Boundaries 17-53
Assigning Linked Impedance Boundaries 17-58
Get Impedance from External Design 17-60
Assign Gradient Surface Roughness Boundaries 17-62
Assigning Gradient Surface Roughness Boundary 17-63
Assigning Half Space Boundaries in HFSS 17-63
Assigning (SBR+) Perfect Absorber Boundaries 17-65
Assigning (SBR+) Fresnel Table Boundaries 17-66
Designating Infinite Ground Planes 17-70
Modifying Boundaries 17-70

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

Deleting Boundaries ... 17-71
Reassigning Boundaries ... 17-71
Reprioritizing Boundaries ... 17-72
Editing The Global Material Environment ... 17-73
Duplicating Boundaries and Excitations with Geometry ... 17-74
Showing and Hiding Boundaries and Excitations ... 17-74
Showing and Hiding Boundaries and Excitations in the Active View Window ... 17-74
Showing and Hiding Boundaries and Excitations in Every View Window ... 17-76
Reviewing Boundaries and Excitations in the Solver View ... 17-76
Setting Default Values for Boundaries and Excitations ... 17-77
18 - Assigning Hybrid Regions in HFSS ... 18-1
Assigning FE-BI Hybrid Regions ... 18-4
Assigning IE Regions ... 18-5
To Assign an IE Region ... 18-6
Applications and Technical Considerations for Using IE Regions ... 18-8
IE Region Ports ... 18-10
Assigning PO Region Hybrid Region ... 18-10
To Assign a PO Region: ... 18-12
Examples of PO Hybrid Simulations ... 18-13
Horn Fed Reflector Antenna with PO ... 18-13
Antenna with Raydome using Dielectric PO Hybrid Region ... 18-15
Cassegrain Reflector: PO and IE Hybrid Regions in One Design ... 18-17
Technical Notes for Hybrid Regions and PO Solver ... 18-18
Assigning SBR+ Hybrid Regions ... 18-19
Configure Simulation with SBR+ Antenna Geometry Blockage (HFSS with Hybrid and Array Solution Type): ... 18-38
Assigning a Dielectric Cavity ... 18-41
Set Coupling for Hybrid Regions ... 18-42

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

Set SBR+ Source Regions 18-44
Set Current Source Option 18-50
Lightweight Geometry for HFSS Hybrid Regions Designs 18-52
Initial Mesh Operations for Lightweight Geometries 18-54
Lightweight Geometry Verification for the IE Solver 18-55
Permitted Modeler Operations on Faceted Objects 18-56
Other Characteristics of Faceted Objects 18-57
19 - Assigning Excitations for HFSS or HFSS-Transient 19-1
HFSS Excitations 19-1
HFSS Transient Excitations 19-3
Assigning Circuit Ports 19-4
To assign a Circuit Port: 19-4
Swap Assignment 19-7
Create a Circuit Element from a Circuit Port 19-7
Convert a Circuit Port to a Lumped Port 19-7
Convert a Lumped Port to a Circuit Port 19-8
Multimode Simulations in HFSS Terminal Solutions 19-9
Wave Ports 19-11
Wave Port Size 19-13
Wave Port Placement 19-15
Number of Modes 19-16
Automatic PEC Creation on Wave Ports 19-16
Example Using the Waveguide Combiner Example 19-17
Wave Port Dialog for Modal Solutions 19-19
General Tab: Wave Ports 19-20
Modes with Default Settings 19-21
Set Mode Polarity Using Integration Line 19-22
Align Modes Analytically Using Coordinate System 19-24

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

Align Modes Using Integration Lines ... 19-26
Alignment Groups ... 19-28
Characteristic Impedance Column ... 19-30
Port Post Processing Tab: Modal Solutions ... 19-32
Deembed Settings ... 19-33
Renormalization ... 19-34
Assign Wave Ports for Modal Solutions ... 19-35
Automatic PEC Creation on Modal Wave Ports ... 19-36
Define Mode Alignment and Polarity ... 19-37
Use Set mode polarity using integration lines ... 19-38
Use Align modes analytically using coordinate system ... 19-38
Use Align modes using integration lines ... 19-40
Post Processing: Wave Ports ... 19-40
Wave Ports for Terminal Solutions ... 19-43
Terminals and Modes ... 19-44
Example of Terminals ... 19-44
Reference Conductors ... 19-45
Scenario 1: Explicit Reference Conductor ... 19-46
Scenario 2: Implicit Reference Conductor ... 19-47
Assign Wave Ports for Terminal Solutions ... 19-48
Multi-Select Ports and Terminals ... 19-51
Automatic PEC Creation on Terminal Wave Ports ... 19-52
Associating Terminals and Ports ... 19-53
Auto Assign Terminals ... 19-54
Post Processing Operations ... 19-54
Set Renormalizing Impedance for Terminals ... 19-55
To Set the Reference for All terminals on a Port: ... 19-56
Renormalize Terminals from the Port Level ... 19-57

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

Set Differential Pairs 19-59
Lumped Ports 19-64
Examples of Lumped Port 19-65
Assign Lumped Ports for Modal Solutions 19-66
Selecting a Face or Sheet and Manually Drawing an Integration Line 19-66
Selecting Two Linear Edges that Do Not Intersect 19-69
Selecting a Planar Face and a Linear Edge 19-71
Lumped Port: Post Processing 19-74
Assign Lumped Ports for Terminal Solutions 19-77
Selecting a Face or Sheet 19-78
Selecting Two Linear Edges that Do Not Intersect 19-79
Selecting a Planar Face and a Linear Edge 19-81
Floquet Ports 19-83
Example of Floquet Ports 19-84
Assign Primary and Secondary Boundaries 19-85
Direction of the U-V Vectors 19-87
Assign Floquet Ports 19-87
Floquet Port Dialog Box 19-89
Floquet Port: Mode Setup 19-89
Floquet Port: Post Processing 19-91
Floquet Port: 3D Refinement 19-91
Floquet Ports: Lattice Coordinate System 19-92
Floquet Port: Modes Calculator 19-92
Incident Waves 19-94
Radiation Boundary Panel 19-94
Global Material Environment 19-95
Incident Plane Wave 19-96
Cartesian Vector Setup 19-98

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

Spherical Vector Input Format 19-101
Defining Multiple Plane Waves 19-105
Hertzian-Dipole Wave 19-108
Cartesian Coordinates 19-109
Incident Cylindrical Wave 19-110
Gaussian Beam Wave 19-111
Linear Antenna Wave 19-111
Far Field Wave 19-111
Far Field Wave Options 19-111
Legacy Translation of Source Origin Relative to This Design 19-112
External Data File for Far Field Wave 19-114
Frequency Independent Far Fields 19-114
Frequency Dependent Far Fields 19-115
Data File Example 19-116
Setup Link for Far Field or Near Field Wave 19-118
Example That Uses Far Field Links 19-120
Near Field Wave 19-123
Assign Near Field Wave 19-124
Preserve source design solution 19-130
Example that Uses Near Field Links 19-131
Example with Interior Source for Near Field Links 19-135
Access Near Field External Data File 19-139
Near/Far Field Polarization Definition Table for HFSS, SBR+ and Hybrid Designs 19-144
Ansys Near Field Data File 19-145
Sample *.and File Sphere Geometry 19-145
Sample *.and File, Cylinder Geometry 19-146
Sample NFD File in Cartesian Coordinate System 19-146
Sample NFD File in Spherical Coordinate System 19-147

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

Sample NFD File in Cylindrical Coordinate System 19-148
Generate Near Field Data File 19-149
Using Field Solutions from Other Simulations 19-150
Using a Dynamic Link Between HFSS and HFSS 3D Layout (Example) 19-151
Cable Network 19-161
Link from Driven Modal or Terminal Antenna to SBR+ Design 19-166
Creating a Link from Driven Modal or Driven Terminal Source to SBR+ Target 19-170
Simulation Setup for Driven Modal Antenna to SBR+ Design Link 19-187
Viewing Results in the SBR+ Simulation 19-189
Requirements for Creating a Gain Report for SBR+ Design with Driven Modal or Terminal Source 19-190
Select Tx/Rx Antennas for SBR+ Solutions 19-191
Prerequisites 19-192
To Assign Transmit and Receive Assignments for Antennas in SBR+ 19-192
Assigning Voltage Sources 19-199
Assigning Current Sources 19-200
Modifying Voltage or Current Sources 19-201
Assigning Magnetic Bias Sources 19-201
Applications for Deembedding 19-203
Extract Input Impedance: Use Deembedding 19-203
Setup Link Window 19-203
Modifying Excitations 19-204
Deleting Excitations 19-205
Reassigning Excitations 19-205
Setting the Impedance Multiplier 19-207
Show Nets for DC Continuity for 3D Conductors 19-208
Show Nets for Layout Components in HFSS 3D Geometries 19-213
Define an Integration Line 19-223

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

Duplicating Integration Lines 19-225
Modifying Integration Lines 19-227
20 - Circuit Elements in HFSS 20-1
Beta Feature: De-embedding for Circuit Elements 20-6
Creating a Single Port Circuit Element in HFSS 20-7
Import Multiple... Option 20-10
Creating a Multi-Terminal Circuit Element in HFSS 20-13
Import Multiple... Option 20-17
Import Touchstone Files for Circuit Elements in HFSS 20-21
Creating Circuit Models and Managing Libraries in HFSS 20-29
21 - Assigning Arrays in HFSS 21-1
Creating a Unit Cell for an Array 21-5
Create Array Command for Native Geometries 21-7
Create Array... Command Procedure 21-7
Multiple Finite Arrays (Beta) 21-12
Create Array Through CSV 21-15
Create Array Through CSV... Command Procedure 21-16
Exporting Array to CSV File 21-20
Example CSV File 21-20
Create Array Command for 3D Component 21-22
Multiple Finite Arrays (Beta) 21-25
Create Array Command for 3D Component 21-31
Create Array for 3D Component Process Flow 21-31
SBR+ and 3D Component Array 21-54
Sizing Variable for a New 3D Component Array 21-59
Array Visualization 21-64
Set up and Run an Array Simulation 21-66
Beta Feature: Parallel Component Mesh Adapt 21-69

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

Beta Feature: Composite Subarray in HFSS Finite Array Design 21-69
Composite Array Post-Processing 21-75
Multi-band Composite Subarray 21-77
General Setup for Virtual Array Simulation 21-80
Interpolating Sweep Advanced Options for Arrays 21-81
Post Processing for Array Models 21-82
Reports for Arrays 21-84
Field Plots for Arrays 21-84
Fields Calculator Applications for Arrays 21-85
Fields Post Processing on a Designated Array Cell 21-85
22 - Running Simulations 22-1
Local Analysis 22-2
Solving a Single Setup or Sweep 22-3
Running More than One Simulation 22-4
Submitting a Job 22-5
Monitoring Queued Simulations 22-10
Monitoring the Solution Process 22-11
Changing a Solution Priority for System Resources 22-12
Aborting an Analysis 22-13
Re-solving after Modifying a Design 22-14
Re-solving after Ansys Workbench Thermal Feedback 22-14
23 - Postprocessing and Generating Reports 23-1
Viewing Solution Data 23-2
Viewing Convergence Data 23-3
Viewing the Number of Completed Passes 23-6
Viewing the Max Magnitude of Delta S Between Passes 23-6
Viewing the Delta Magnitude Energy 23-7
Viewing the Magnitude Margin 23-7

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

Viewing the Phase Margin 23-7
Viewing the Max Delta (Mag S) 23-8
Viewing the Max Delta (Phase S) 23-8
Viewing the Maximum Delta Frequency 23-9
Plotting Convergence Data 23-9
Viewing a Solution Profile 23-9
Viewing Matrix Data 23-14
Selecting the Matrix Display Format 23-19
Exporting Matrix Data 23-21
Renaming Matrix Data 23-24
Reordering Matrix Data 23-25
Exporting Equivalent Circuit Data 23-27
Exporting W-element Data 23-30
Viewing Mesh Statistics 23-33
Viewing Eigenmode Solution Data 23-35
Deleting Solution Data 23-36
Creating Reports 23-37
Creating a Quick Report 23-38
Creating a New Report 23-39
Context Section for Reports 23-44
Using Families Tab for Reports 23-45
Filtering Quantity Selections for the Reporter 23-48
Deleting Reports 23-49
Modifying Reports 23-49
Modify Report: Selecting Use All Values or Making Selection 23-53
Modify Report: Using the Edit Sweep Dialog Box 23-54
Generating Reports For Characteristic Mode Analysis 23-54
Characteristic Mode Data Report 23-59

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

Modal Significance vs Frequency 23-62
Characteristic Value vs Frequency 23-63
Characteristic Angle vs Frequency 23-64
Modal Coefficient Plot 23-65
Generating Reports For Monostatic RCS 23-65
Monostatic RCS Reports 23-68
Creating a Report from an Ansoft Report Data File 23-70
Zooming and Fitting Reports 23-70
Modifying the Background Properties of a Report 23-70
Modifying the Legend in a Report 23-73
Exporting Ansoft Report Data Format Files 23-76
Exporting Reports as Graphics 23-78
Report File Formats 23-79
Creating Custom Report Templates and Defaults 23-81
Selecting the Report Type 23-82
Selecting the Display Type 23-85
Creating 2D Rectangular Plots 23-86
Creating 2D Rectangular Stacked Plots 23-88
Creating 3D Rectangular Plots 23-91
Creating 3D Rectangular Bar Plots 23-97
Creating Rectangular Contour Plots 23-110
Creating Sine Space Plots 23-113
Radiated Fields in HFSS 23-113
Sine Space Plots 23-114
Creating 2D Polar Plots 23-117
Reviewing 2D Polar Plots 23-118
Creating 3D Polar Plots 23-119
Creating 3D Spherical Plots 23-128

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

Creating Smith Charts 23-138
Creating Smith Contour Charts 23-139
Creating Data Tables 23-141
Creating Radiation Patterns 23-143
Delta Markers in 2D Reports 23-145
Plotting in the Time Domain 23-145
TDR Windowing Functions 23-150
Working with Traces 23-152
Editing Trace Properties 23-154
Editing the Display Properties of Traces 23-155
Adding Data Markers to Traces 23-158
Y Markers in Stacked XY Plots 23-160
Discarding Report Values Below a Specified Threshold 23-167
Adding Characteristics to a Trace 23-168
Adding a Recently Used Trace Characteristic 23-168
Adding a Trace Characteristic from Favorites 23-168
Adding Trace Characteristics to your Favorites 23-169
Adding Characteristics Using the Add Trace Characteristics Window 23-171
Removing All Trace Characteristics 23-173
Removing Traces 23-173
Copy and Paste of Report and Trace Definitions 23-173
Copy and Paste of Report and Trace Data 23-174
Limit Lines in Cartesian Plots 23-175
Sweeping a Variable in a Report 23-182
Sweeping Values Across a Distance 23-183
Sweeping Values Across a Sphere 23-183
Selecting a Function for a Plot 23-184
Selecting Solution Quantities to Plot 23-191

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

Selecting a Field Quantity to Plot 23-196
Selecting a Far-Field Quantity to Plot 23-196
Prerequisites 23-196
Plotting Vertical Cross-Sections of Far Fields 23-204
Plotting Horizontal Cross-Sections of Far Fields 23-204
Selecting a Near-Field Quantity to Plot 23-205
Prerequisites 23-205
To select a near-field quantity to plot: 23-206
Overlaying Near Field Reports 23-211
Selecting Antenna Parameters to Plot 23-213
Selecting an Emission Test Quantity to Plot 23-215
Plotting Imported Solution Data 23-216
Setting a Range Function 23-216
Range Functions 23-217
Perform FFT on a Report 23-222
FFT Window Functions 23-223
Apply FFT to Report Functions 23-225
Perform TDR on Report 23-227
Window Functions and Time Domain Plotting 23-227
Animated Reports 23-241
Overlaying Surface Currents on a 3D View 23-241
Overlaying Far Fields on a 3D View 23-244
Overlaying Near Fields on a 3D View 23-246
Frequency Animation 23-249
Phase Animation 23-252
Changing the Design Point 23-256
Specifying Output Variables 23-257
Function List for Output Variables 23-260

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

Scaling a Source's Magnitude and Phase 23-260
Scaling Sources for HFSS 23-262
Guidelines for Scaling a Source's Magnitude and Phase 23-266
Scaling Sources and Setting Delays for HFSS Transient Network 23-267
Editing Sources for Spectral Fields for Transient Network 23-268
Editing Sources for Transient Fields for Transient Network 23-269
Ports and Edit Sources Behavior for Transient Network 23-270
Specifying Source Contexts for Creating Radiated Field Reports 23-272
Edit Sources Panel for CMA 23-273
Derivative Tuning for Reports 23-275
Port Field Display 23-280
Plotting Field Overlays 23-281
Generating a Sampled Near Field Plot 23-288
Overlaying 3D Polar Plots of Near or Far Fields on Models 23-291
Overlaying 2D Radiation Field Plots on Models 23-294
Overlaying Monostatic and Bistatic RCS 2D Plots 23-297
Overlay of Contour Plot of Near Field on Rectangle 23-300
Overlaying DC Temperature Fields (General Mode Only) 23-306
Overlay Visibility 23-309
Plotting Derived Field Quantities 23-314
Creating 2D Reports from Named Expressions 23-315
Creating Scalar Field Plots 23-317
Working with Scalar Field Plot Markers 23-317
Adding a Field Plot Marker 23-318
Deleting a Field Plot Marker 23-319
Exporting a Field Plot Marker Table 23-320
Clearing All Field Plot Markers 23-320
Editing Field Plot Markers 23-320

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

Showing Field Plot Markers 23-321
Creating Vector Field Plots 23-322
Modifying Field Plots 23-325
Setting Field Overlay Attributes 23-329
Modifying Field Plot Colors 23-338
Color Key Settings 23-342
Modifying the Field Plot Scale 23-343
Modifying Vector Field Plot Arrows 23-346
Setting the Mesh Visibility on Field Plots 23-349
Modifying Scalar Field Plot Isovalues 23-349
Modifying Scalar Field Plot Shading 23-353
Mapping Scalar Field Plot Transparency to Field Values 23-355
Modifying Markers on Point Plots 23-356
Modifying Line Plots 23-356
Setting a Plot's Visibility 23-357
Saving a Field Overlay Plot 23-360
Opening a Field Overlay Plot 23-360
Deleting a Field Overlay Plot 23-360
Setting Field Overlay Plot Defaults 23-361
Exporting Field Plots 23-361
Field Plot Data Export Format (*.aedtplt) 23-363
File Format Description 23-365
Create Surface Mesh Summary 23-367
Creating Animations 23-370
Creating Phase Animations 23-372
Creating Frequency Animations 23-375
Creating Normalized Distance Animations 23-378
Creating Geometry Animations 23-381

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

Creating Parametric Animations 23-384
Creating Mesh Animations 23-388
Animating a Sampled Near Field Plot 23-391
Creating 2D Radiation Field Plot Overlay Animations 23-396
Animating VRT Plots 23-398
Controlling the Animation's Display 23-400
Modifying an Animation Setup 23-402
Exporting Animations 23-405
Exporting Ensight from the Select Animation Dialog 23-405
Exporting from the Animation Control Dialog 23-407
Modifying SAR Settings 23-409
Using the Fields Calculator 23-411
HFSS Support Expression Input in Fields Calculator (Beta) 23-413
Scripting Support 23-414
Preliminary Considerations for Using the Fields Calculator 23-414
Field Convergence and Accuracy 23-415
Fast Sweep and Dispersive Models 23-415
Inputs/Excitations 23-415
Units 23-416
Eigenmode Solutions 23-416
Opening the Fields Calculator 23-416
Context Area 23-417
The Calculator Stack 23-418
Stack Commands 23-419
Pop 23-420
RlUp 23-420
RlDn 23-420
Exch 23-420

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

Clear 23-420
Undo 23-420
Registers 23-420
Enlarging the Register Display Area 23-423
Units of Measure 23-423
Input Commands 23-423
Quantity Command 23-425
Geometry Command 23-427
Constant Command 23-431
Number Command 23-432
Function Command 23-433
Geom Settings Command 23-434
Read Command 23-435
Output Vars [Input for Eigenmode problems] 23-435
General Commands 23-435
Steps for Calculating the Complex Vector Electric Field 23-439
Scalar Commands 23-439
Vec? Command 23-441
1/x (Inverse) Command 23-441
Pow Command 23-441
(Square Root) Command 23-442
Trigonometry Commands 23-442
d/d? (Partial Derivative) Command 23-442
(Integral) Command 23-442
Min Command 23-443
Max Command 23-443
Ñ (Gradient) Command 23-444
Ln Command 23-444

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

Log Command 23-444
Std Command 23-444
Vector Commands 23-444
Scal? Command 23-446
Matl Command 23-446
Mag Command 23-447
Dot Command 23-447
Cross Command 23-447
Divg Command 23-447
Curl Command 23-447
Tangent Command 23-447
Normal Command 23-448
Unit Vec Command 23-449
X Form 23-449
Output Commands 23-450
Value Command 23-451
Eval Command 23-453
Write Command 23-454
Export Command 23-455
Calculating Derived Field Quantities Using Named Expressions 23-460
Named Expression Library 23-462
Exiting the Fields Calculator 23-464
Fields Calculator Recipes 23-464
Calculating Numerical Quantities 23-464
Calculating Quantities for 2D (Line) Plot Outputs 23-465
Calculating Quantities for 3D (Surface or Vector) Plot Outputs 23-465
Calculating Quantities for 3D (Volume) Plot Outputs 23-465
Calculating Quantities for Animated Outputs 23-465

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

Creating User Defined Named Expressions Library 23-465
Recipe Format 23-465
EXAMPLE: Title of Current Calculation 23-465
Calculating the Current along a Wire or Trace 23-466
Calculating the Voltage Drop along a Line 23-467
Calculating Net Power Flow through a Surface 23-468
Calculating the Average of a Field Quantity on a Surface 23-469
Calculating the Peak Electrical Energy in a Volume 23-471
Calculating the Q of a Resonant Cavity 23-472
Calculating the Max Value of a Field Quantity in a Volume 23-474
Calculating Dielectric (or Conduction) Loss in a Volume 23-475
Plotting Wave Impedance along a Line 23-476
Plotting the Phase of E Tangential to a Line/Curve 23-477
Plotting the Maximum Magnitude of E Tangential to a Line/Curve 23-478
Plotting the E-Field Magnitude Normal to a Surface 23-480
Generating an Iso-Surface Contour for a Given Field Value 23-481
Generating an Animation on Multiple Planes with a Positional Variable 23-482
Generating Cartesian/Cylindrical/Spherical-Coordinate Field-Components Library 23-483
Radiated Fields Observation Regions 23-484
Setting up a Near-Field Sphere 23-486
Setting up a Near-Field Rectangle 23-488
Setting up a Near-Field Box 23-491
Setting up a Near-Field Line 23-493
Setting up an Arbitrary Near Fields Point List 23-495
Generating a Report from Point List 23-498
Animating an Near Field Points List Plot 23-500
Computing Maximum Near-Field Parameters 23-506

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

Setting up a Far-Field Infinite Sphere 23-508
Setting up a Source Group for a Finite Array 23-516
Radiated Power Calculation Method 23-520
Defining Antenna Arrays 23-521
Defining a Parametric Antenna Array 23-523
Defining a Custom Antenna Array 23-527
Computing Antenna Parameters 23-530
Exporting Antenna Parameters and Maximum Field Data 23-536
Export Element Pattern from Antenna Parameters 23-544
Antenna Parameters: Save for Overlay 23-551
Creating and Displaying Antenna Parameter Overlays 23-553
Plotting the Mesh 23-556
Example of Mesh Plots for an HFSS Design with Layout Components 23-560
Setting Mesh Plot Attributes 23-565
User Defined Outputs (UDOs) 23-568
Named Probes and Properties in User Defined Outputs 23-569
Computation of Traces Based UDO Calculations 23-570
Dimensions Reduction by UDO Calculations 23-570
Dynamic Probes 23-571
User Defined Outputs: Python Script API 23-571
UDO Extension Implementation 23-571
Import Statements 23-572
UDOExtension Class 23-572
Validate 23-572
IUDOPluginExtension Abstract Class 23-573
GetUDSName 23-574
GetUDSDescription 23-574
GetUDSSweepNames 23-575

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

GetCategoryNames 23-575
GetQuantityNames 23-576
GetQuantityInfo 23-576
GetInputUDSParams 23-577
GetDynamicProbes 23-580
Compute 23-580
Data Types Used in UDO Python Scripts 23-583
Constants 23-583
Abstract Classes 23-583
IUDSInputData 23-583
GetDoubleProbeData 23-584
GetSweepsDataForProbe 23-584
GetComplexProbeData 23-585
GetSweepNamesForProbe 23-586
GetRequiredQuantities 23-586
GetVariableValues 23-587
GetInterpolationOrdersData 23-587
IUDSOutputData 23-588
SetSweepsData 23-588
SetDoubleQuantityData 23-589
SetComplexQuantityData 23-589
Working With Properties for UDO 23-590
IPropertyList Abstract Class 23-590
IProperty Abstract Class 23-591
INumberProperty Abstract Class 23-591
ITextProperty Abstract Class 23-591
IMenuProperty Abstract Class 23-592
Other Application-Specific Classes Used in Python Scripts 23-593

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

Constants Class 23-593
UDSProbeParams Class 23-593
UDSDynamicProbes Class 23-594
QuantityInfo Class 23-594
IProgressMonitor Abstract Class 23-595
SetTaskName 23-596
SetSubTaskName 23-596
BeginTask 23-597
SetTaskProgressPercentage 23-597
CheckForAbort 23-597
EndTask 23-598
Using .NET Collection Classes and Interfaces in Python Scripts 23-598
User Defined Outputs: Messaging Methods 23-600
AddErrorMessage 23-605
AddInfoMessage 23-606
AddWarningMessage 23-606
User Defined Outputs: Script Organization 23-606
Using Script Libraries 23-607
Using Additional .NET Assemblies 23-607
User-Defined Documents (UDDs) 23-608
Create User-Defined Document Inputs 23-609
UDD Document Creation and Display 23-611
Managing Documents Listed in the Project Window Under Results 23-612
Documents Folder's Context Menu 23-613
Document Folder's Property Window 23-613
Viewing UDDs with an HTML Web Browser 23-614
UDD Script Libraries 23-615
User-Defined Documents: Python Script API 23-616

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

Import Statements 23-616
Data Types Used in UDD Python Scripts 23-617
Constants 23-617
Abstract Classes 23-617
IProgressMonitor Abstract Class 23-617
SetTaskName 23-618
SetSubTaskName 23-618
BeginTask 23-619
SetTaskProgressPercentage 23-619
CheckForAbort 23-620
EndTask 23-620
IUDDPluginExtension Abstract Class 23-621
GetUDDName 23-621
GetUDDDescription 23-622
ShowDefaultSetupDialog 23-622
GetUDDInputParams 23-622
Generate 23-624
SetupUDDInputParams 23-626
HandleUDDEvents 23-627
GetUDDSchema 23-628
GetUDDStyleSheetForHtml 23-628
GetUDDStyleSheetForPdf 23-629
GetFopExecutable 23-629
GetUDDAppContext 23-630
GetUDDDesignContext 23-630
UDDExtension Class 23-631
UDDInputData Class 23-631
UDDInputBool 23-632

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

UDDInputDouble ... 23-632
UDDInputSolution ... 23-632
UDDInputText ... 23-633
UDDInputTrace ... 23-633
UDDInputParams Class ... 23-634
UDD Input Interfaces ... 23-634
Explication of a Sample UDD Script ... 23-636
Document Generator Interfaces ... 23-638
Toolkit ... 23-643
Chip Model Analyzer (CMA) ... 23-645
PinToPinUtility ... 23-645
24 - Network Data Explorer ... 24-1
Network Data Explorer Overview ... 24-2
NDE Ribbon ... 24-3
Network Data Selection Pane ... 24-4
Cell and Frequency Selection Pane ... 24-4
Data View Pane ... 24-5
Loading Data into Network Data Explorer ... 24-5
Exporting Data from Network Data Explorer ... 24-6
Exporting SYZ Data ... 24-6
Exporting Macro Model ... 24-9
Comparing Original S-Parameters with Exported S-Parameters ... 24-15
Creating an NPort Model ... 24-15
Scripting for Network Data Explorer ... 24-15
Network Data Explorer Ribbon ... 24-16
Network Data Explorer Data Sources ... 24-16
Network Data Explorer Display Format ... 24-17
Network Data Explorer Display Full Port Names ... 24-18

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

Network Data Explorer Save or Reset Default Settings 24-19
Smoothing 24-19
Cell Filtering 24-20
Changing Port Properties and Reducing Matrix Size 24-21
Displaying Mixed-Mode Parameters using Differential Pairs 24-22
Reset All Port Properties 24-24
Data View Pane Context Menus 24-24
Multiple Frequency Statistics 24-24
Highlight Min/Max 24-25
Select Transpose 24-25
Color Legend Attributes 24-26
Matrix Entries Plot Menu 24-27
Exploring Network Data and Modifying the Display 24-31
Viewing a Matrix Table 24-32
Viewing a Color-Coded Matrix Plot 24-33
Displaying a Cell Graph Across All Frequencies 24-34
Displaying Matrix Statistics by Frequency 24-35
Displaying Individual Statistics for All Frequencies 24-36
Creating a Statistics Plot 24-37
Comparing Network Data 24-38
Causality Checking and Plots 24-39
Multithreading 24-46
25 - Scripting 25-1
26 - Ansys Workbench Integration Overview 26-1
Integrating Ansys EM Suite with Ansys Workbench 26-1
Workbench Data Integration Overview 26-3
Adding New Analysis Systems 26-5
Importing Existing Ansys EM Projects into Workbench 26-6

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

Editing Ansys EM Models in Workbench 26-7
Analyzing Ansys EM Models in Workbench 26-8
Performing Parameter Studies in Workbench 26-10
Scripting in Workbench 26-14
Ansys EM - Ansys Multiphysics Coupling 26-15
Multiphysics Coupling on Workbench with Ansys Thermal 26-15
Multiphysics Coupling on Workbench with Ansys Structural 26-16
Multiphysics Coupling between Ansys EM Field Systems on Workbench 26-17
Ansys EM CAD Integration through Workbench 26-18
CAD Integration Functionality 26-21
CAD Integration and Geometry Sharing 26-23
Bi-Directional CAD Integration 26-26
CAD Integration Model Edits 26-27
Multiple Geometry Links for CAD Integration 26-28
Creating Dynamic Links to EDT Designs 26-29
Healing with CAD Integration 26-30
Important Geometry Options for CAD Integration 26-30
Ansys EM to Ansys Geometry Transfer 26-32
Material Assignment Transfer 26-33
Geometry Transfer through Ansys DesignModeler (DM) 26-34
Workbench Material Data Transfer 26-34
Detailed Behavior 26-34
Stress Feedback to HFSS using Workbench 26-36
Interface Changes for Stress Feedback from Ansys Mechanical to HFSS 26-36
Process Flow for HFSS Workbench for Stress Feedback 26-39
Only Stress Feedback 26-44
Feedback Iterator 26-44
The Feedback Iterator System 26-46

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

Feedback Iterator in Use ... 26-47
Feedback Iterator Component Properties ... 26-48
Feedback Iterator GUI Operations ... 26-50
Resetting the Feedback Iterator ... 26-50
Resetting via the Ansys EM User Interface ... 26-51
Resetting via a WB Journal Script ... 26-52
Callback Interface ... 26-53
Callback and State API ... 26-53
Utility Functions ... 26-54
Output/Debugging Functions ... 26-55
Example Scenarios for Feedback Iterator ... 26-55
Setting up Iteration with Feedback Iterator ... 26-55
Breaking Iteration Control ... 26-56
Starting an Iterative Update ... 26-57
Running a Single Iteration ... 26-58
Interrupting an Iterative Loop ... 26-58
Resuming an Interrupted Iterative Loop ... 26-58
Modifying any of the Systems Involved in Iteration (Coupled Clients) ... 26-59
Iterating to Convergence ... 26-59
Surface Force Density in HFSS ... 26-62
27 - Meshing In HFSS ... 27-1
Introduction to Meshing in HFSS ... 27-1
Mesh Generation ... 27-2
Adaptive Mesh Refinement ... 27-3
Adaptive Mesh Refinement Process Flow ... 27-7
Stages of Meshing ... 27-8
Assign Mesh Operations ... 27-12
Operations Affecting Geometry Representation ... 27-13

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

Surface Approximation 27-15
Name 27-17
Maximum Surface Deviation (Length) 27-18
Maximum Normal Deviation (Angle) 27-19
Maximum Aspect Ratio 27-20
Surface Representation Priority for Tau Mesh (effective only for Tau Tolerant Mesh) 27-22
Apply Curvilinear Elements Discussion 27-24
Meshing for Curved Surfaces 27-25
Effective Surface Approximation For a Pillbox Resonator 27-25
Model Resolution Length 27-27
Use Model Resolution Length 27-27
Auto Simplify Using Effective Thickness 27-29
An MRL Scenario 27-30
Operations Refining Initial Mesh 27-33
Mesh Refinement Settings (Optional) 27-33
Example 1: Dielectric Resonator Antenna 27-34
Example 2: Microstrip Filter on a Perforated Ground Plane 27-36
Mesh Seeding Controls 27-40
Length-Based Mesh Refinement 27-41
Seed Refinement for Mesh Growth 27-42
Mesh Refinement Scenarios 27-43
Scenario 1: Assign Mesh Operations after the First Pass 27-43
Scenario 2: Assign Mesh Operations Upfront 27-46
Restrict Length of Elements 27-47
Defining a Model-Level Mesh Operation 27-50
Normal Deviation Angle 27-51
Maximum Surface Deviation Length 27-52

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

Aspect Ratio 27-53
Initial Mesh Settings 27-54
Meshing Method 27-56
The Phi Mesher 27-58
Phi Mesher for Multilayer Complex Package Layout 27-59
Troubleshooting Mesh Failures 27-61
Scenario 1:Model Resolution has Caused Significant Changes in Object Dimensions 27-62
UI Controls for Mesh Operations 27-66
Mesh Operations 27-66
Assign Curvilinear Elements 27-67
Access Surface Approximation (Object Level) 27-67
Mesh Assembly 27-69
Mesh Method Override 27-71
Example: Horn Antenna and a Dish 27-73
Mesh Fusion 27-76
Rules for Ranked Priority Level 27-82
Rules for Default Priority Level 27-82
Rules for Resolving Intersection between Two Objects 27-82
Rules for Resolving Intersection between Two Components' Mesh Fusion Envelope 27-83
Rules for Resolving Intersection between an Object and Another Component's Mesh Fusion Envelope 27-84
Putting it all together: 27-85
Rules to Resolve Containments 27-87
Rules for Simple Priority 27-87
Mesh Envelope Padding for Components without a User-defined Mesh Envelope 27-92
27-93
Mesh Method Override 27-95

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

Example 1: USB Socket using Priority Level 27-96
Example 2: USB Socket using Simple Priority 27-100
Component Priority 27-102
28 - HFSS Technical Notes 28-1
The Finite Element Method 28-1
Representation of a Field Quantity 28-2
Rectilinear Elements and Curvilinear Elements 28-2
Basis Functions in HFSS 28-3
Size of Mesh Vs. Accuracy in HFSS 28-3
The HFSS Solution Process 28-4
The Mesh Generation Process 28-5
Seeding the Mesh 28-6
Guidelines for Seeding the Mesh 28-6
Length-Based Mesh Refinement Process 28-7
Skin Depth-Based Mesh Refinement 28-7
Surface Approximation Settings 28-8
Guidelines for Modifying Surface Approximation Settings 28-9
Meshing Region Vs. Problem Region 28-9
Model Resolution 28-10
Port Solutions 28-11
Excitation Fields 28-11
Wave Equation 28-12
Mesh Refinement on Ports 28-12
Modes 28-13
Degenerate Modes 28-14
Multiple Ports on the Same Face 28-14
Port Accuracy 28-14
Calculating Characteristic Impedance 28-15

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

Calculating the PI Impedance 28-16
Calculating the PV Impedance 28-16
Calculating the VI Impedance 28-17
Impedance Multipliers 28-17
Calculating Terminal Characteristic Impedance Matrix 28-18
Complex Propagation Constant 28-18
Calculating the Effective Wavelength (Lambda) 28-18
Calculating the Relative Permittivity (Epsilon) 28-19
Maximum Delta S 28-19
Maximum Delta E 28-20
Percent of Tetrahedra Refined Per Pass 28-20
Magnitude Margin 28-20
Phase Margin 28-21
Maximum Delta Frequency 28-21
Max Delta (Mag S) 28-21
Max Delta (Phase S) 28-22
Matrix Solvers 28-22
Direct Matrix Solver 28-22
Iterative Matrix Solver 28-23
Guidelines for Using the Iterative Solver 28-24
Multiprocessing and the Iterative Solver 28-24
Iterative Matrix Solver Technical Details 28-24
Single Frequency Solution 28-26
Frequency Sweeps in HFSS 28-27
Fast Frequency Sweeps 28-27
Discrete Frequency Sweeps in HFSS 28-29
Interpolating Frequency Sweeps in HFSS 28-30
Solution Types in HFSS 28-31

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

Eigenmode Solutions 28-34
Calculating the Resonant Frequency 28-34
Calculating the Quality Factor 28-35
Q Factor of Resonant Cavities 28-35
References for Eigenmode Solutions 28-37
Theory of Characteristic Modes 28-38
Characteristic Angle 28-39
Modal Significance 28-40
Loop Antenna of a Cell Phone Chassis 28-41
Field Solutions 28-46
Field Overlay Plots 28-47
Field Quantities 28-47
Specifying the Phase Angle 28-49
Peak Versus RMS Phasors 28-50
Calculating the SAR 28-51
Transient Solution Theory 28-54
Discontinuous Galerkin Finite Element Method 28-55
Local Time Stepping 28-56
Materials in Time Domain 28-57
Excitations in Time Domain 28-58
Analysis Setup in Time Domain 28-63
Target Residual for Transient Analysis Duration 28-64
Transient Implicit Solver 28-64
References for Time Domain 28-64
Domain Decomposition Method 28-66
Basic DDM Theory 28-66
Computational Memory 28-68
Computational Time 28-68

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

HFSS SBR+ Technical Overview 28-69
Shooting and Bouncing Rays (SBR) 28-69
SBR workflow 28-70
Additional capabilities in SBR+ 28-72
Excitations for SBR+ 28-73
Antenna and SBR+ Hybrid Region in Driven Modal Solution Type 28-73
Antenna introduced via Linked Source 28-73
Incident Plane Wave 28-74
Parametric Antennas 28-74
By File Antenna 28-75
Current Conformance 28-75
Current Reduction 28-76
Near Field vs Far Field Source Representation 28-76
Geometry, Boundaries, and Dielectric Region Support in SBR+ 28-78
Boundaries Concept in SBR+ 28-78
Perfect E Boundary 28-81
Perfect H Boundary 28-81
Finite Conductivity Boundary 28-81
Layered Impedance Boundary 28-82
Impedance Boundaries 28-82
Fresnel (SBR+) Boundaries 28-83
Perfect Absorber 28-83
Reflection-Transmission Table (.rttbl) 28-83
PEC-like State of a Boundary 28-85
Meshing Operations for SBR+ 28-86
Surface Curvature Information, Curvilinear Meshing, and Curvature Extraction 28-86
Ray-tube Divergence Correction 28-87
Dynamic meshing, Tolerant Meshing, Lightweight geometry 28-88

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

Lightweight geometry 28-88
Dynamic meshing 28-88
Tolerant Meshing 28-89
HFSS SBR+ Technical Overview 28-89
Physical Theory of Diffraction (PTD) and Uniform Theory of Diffraction (UTD) 28-89
SBR+ Wedges 28-90
Physical Theory of Diffraction (PTD) 28-90
Uniform Theory of Diffraction (UTD) 28-92
Creeping Waves 28-92
GO Blockage Technical Note 28-96
Visual Ray Tracing (VRT) 28-97
Current Conformance and Array Conformance in SBR+ 28-100
Example of Current Conformance 28-100
Conformance in SBR+ Parametric Array 28-101
Array conformance (Conform to Geometry) for Parametric Array 28-101
Current Conformance for Parametric Array 28-109
Ray Statistics for SBR+ Analysis 28-109
SBR Ray Stats 28-110
CW Ray Stats 28-111
Integral Equation Method Used in for IE Regions 28-113
S-Parameters 28-115
Post Processed S-Matrices 28-116
Renormalized S-Matrices in HFSS 28-116
De-embedded S-Matrices in HFSS 28-117
Alternative Matrix Characterizations 28-119
Calculating the Z-Matrix 28-119
Calculating the Y-Matrix 28-119
Passivity 28-119

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

Radiated Fields 28-120
Spherical Cross-Sections 28-121
Maximum Near-Field Data 28-124
Maximum Far-Field Data 28-125
Array Factor Calculation 28-126
Cautionary Note for Array Factor Use 28-126
Theory of the Array Factor Calculation 28-126
Regular Uniform Arrays 28-127
Scan Specification for Regular Uniform Arrays 28-128
Custom Arrays 28-129
Power Normalizations 28-130
Antenna Parameters 28-131
Polarization of the Electric Field 28-132
Spherical Polar 28-132
Ludwig-3 Polarization 28-133
Circular Polarization 28-133
Axial Ratio 28-133
Polarization Ratio 28-134
Front to Back Ratio 28-135
Max U 28-135
Peak Directivity 28-135
Peak Gain 28-136
Peak Realized Gain 28-136
Radiated Power 28-137
Accepted Power 28-138
Incident Power 28-139
Radiation Efficiency 28-140
Total Efficiency 28-141

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

System Efficiency 28-141
Decay Factor 28-141
Ansys Near Field Description File Format Specification 28-141
NearFieldData Block 28-145
Emission Test 28-147
Emission Test: Set Up and Results 28-149
Mathematical Background: Spectra of Digital Signals 28-154
Using the Power Spectrum for Radiated Field Computations 28-177
Conclusions and Discussion 28-180
Calculating Finite Thickness Impedance 28-184
Modes to Terminals Conversion 28-185
Background 28-185
Terminal Voltages and Currents 28-186
Calculating the W-Elements 28-187
Terminal-Based Models for Circuit Analysis 28-188
Terminal Characteristic Impedance Matrix 28-190
Overview of the Technical Approach for Derivatives in HFSS 28-191
Best Practices for Accurate and Causal Broad Band Frequency Sweeps 28-192
Introduction to Causality Issues for Simulations 28-193
Physical Parameters Compared to Simulation 28-193
Modeling Issue: Solve Inside and Surface Impedance Boundaries 28-193
Modeling Material Properties 28-194
Modeling Surface Roughness 28-194
Surface Impedance Boundary Condition (SIBC) for Metal Traces of Finite Thickness 28-194
Using Causal Material Models 28-196
Surface Roughness Modeling 28-197
Deembedding Parasitic Port Effects 28-199

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

Conclusions 28-202
References for Accurate and Causal Broad Band Frequency Sweeps 28-202
Determining PCB Material Properties: Q2D and Nexxim Measurements for HFSS Simulation 28-203
The Beatty Standard and its Use 28-204
How the Beatty Standard Performs: Lossless 28-205
How the Beatty Standard Performs, TDR 28-206
What about Changing εr? 28-206
Dielectric Constant, continued: Zoom in, the better to see Changing εr 28-207
Dielectric Constant, continued. Now Look at TDR to see effects of Changing εr 28-209
What about Metal Conductivity? 28-209
What does this tell us about metal conductivity? There is a change in plot shape due to loss. 28-210
What about Loss Tangent? 28-211
Now Look at TDR for Loss Tangent 28-212
What about Surface Roughness? 28-213
Now Look at TDR for Surface Roughness 28-214
Observations so Far 28-215
De-embedding of Design Under Test from Test Fixture 28-216
Illustration of Fixture and Device Under Test (DUT) 28-216
De-embedding: Custom Calibration 28-217
De-Embedding: YZ De-embedding Described in Reference [4] 28-218
De-embedding: Modeling and Simulation 28-218
Example Determining PCB Material Properties: Q2D and Nexxim to HFSS 28-221
De-embed in Nexxim 28-223
Q2D Model 28-224
Nexxim with Q2D Model 28-225
Nexxim Model with Changed εr 28-226

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

Nexxim Model, Changing Conductivity 28-227
Observations So Far 28-228
Change Loss Tangent 28-229
Final Values from Nexxim and Q2D 28-229
Compare HFSS Simulation with Extracted Parameters 28-230
Final Tuning Values 28-230
Summary of the Method for using Nexxim and Q2D to Determine PCB Parameters for HFSS 28-230
References for Method for Determining PCB Material Properties 28-231
Large Scale DSO Theory 28-231
Geometric Objects 28-234
Bondwires 28-235
Braided Shield Models in Cable Modeling 28-236
Definitions 28-236
Kley's Model 28-236
Geometric Relations 28-238
Detecting and Addressing Model Problems to Improve Meshing 28-240
One: Healing during geometry import 28-240
Two: Healing after geometry import 28-241
Three: Removing Object Intersections 28-244
Four: Removing Small Features 28-245
Five: Aligning Objects 28-246
Six: Troubleshooting if meshing still fails 28-247
Handling Complicated Models 28-248
Interface Options for Complicated Models 28-248
Geometry Imperfections and Complicated Models 28-249
Object Overlap Settings for Complicated Models 28-249
Post Processing for Complicated Models 28-250

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

Boundaries Technical Notes 28-250
Perfect E Boundaries 28-251
Impedance Boundaries 28-251
Units of Impedance Boundaries 28-253
Radiation Boundaries 28-254
ABC and PML Boundaries 28-254
Boundary Integral Equations 28-255
PML Boundaries 28-255
Material Tensors Applied at PML Boundaries 28-256
Tensor Entries 28-257
Boundaries at PML Surfaces 28-258
Finite Conductivity Boundaries 28-258
Huray Surface Roughness Model 28-259
Groiss Surface Roughness Model 28-262
Modified Hammerstad Model 28-263
Obtaining Broadband Matching between Huray and Hammerstad Models 28-264
Obtaining Better Low Frequency Matching 28-266
Symmetry Boundaries 28-268
Perfect E Vs. Perfect H Symmetry Boundaries 28-268
Symmetry and Port Impedance 28-269
Symmetry and Multiple Modes 28-269
Primary and Secondary Boundaries 28-270
Calculating the E-Field on the Secondary Boundary 28-271
Anisotropic Impedance Boundaries 28-272
Lumped RLC Boundaries 28-272
Layered Impedance Boundaries 28-272
Impedance Calculation for Layered Impedance Boundary 28-273
Surface Roughness Calculation for Impedance Boundary 28-274

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

Shell Elements Theory 28-275
Infinite Ground Planes 28-285
Frequency-Dependent Boundaries and Excitations 28-287
Implicit Boundary Assignments 28-287
Gradient Model for Surface Roughness 28-288
General Approach for Scattered and Total Field Decomposition 28-289
Scattered and total field formulation pros and cons 28-290
Where the Solvers Apply Scattered or Total Field Formulations 28-291
For Scattered Field Decomposition 28-291
Total Field Description: Usual Approach 28-292
Total Field Description: Unusual Approach 28-292
Total Field Description: Space Port Concept 28-292
Summary for Total and Scattered Fields 28-293
Excitations 28-293
Wave Ports Theory 28-294
Mode Polarity 28-294
Mode Alignment 28-295
Analytic Port Types 28-296
Alignment Groups for Wave Ports with Multiple Modes 28-297
Lumped Ports Theory 28-297
Terminals Theory 28-298
Differential Pairs 28-300
Computing Differential Pairs 28-300
Differential Admittance and Impedance Matrices 28-301
Differential S-Matrices 28-302
Floquet Ports Theory 28-302
Formula Summary for HFSS Floquet Modes 28-303
Incident Waves Theory 28-306

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

Plane Wave 28-306
Evanescent Plane Wave 28-307
Elliptically Polarized Plane Wave 28-308
Electric Spherical Wave (Electric Hertzian Dipole) 28-309
Magnetic Spherical Wave (Magnetic Hertzian Dipole) 28-310
Cylindrical Wave 28-310
Gaussian Beam 28-311
Linear Antenna 28-311
Simulating Circuits Containing N-Port Models from a Full-Wave Field Solver 28-312
A 2-Port Model 28-314
Voltage Sources 28-321
Current Sources 28-322
Magnetic Bias Sources 28-322
Uniform Applied Bias Fields 28-323
Non-uniform Applied Bias Fields 28-324
Deembedding 28-324
Deembedding Port Inductance Numerically 28-330
When HFSS Needs Port Calibration (Deembedding) 28-332
Deembedding Extension/Transition Z-matrix Method (Simplified Approximation) 28-332
Z Matrix Method 28-333
S matrix calculation from the series impedances to be deembedded 28-333
Materials Technical Notes 28-335
Relative Permeability 28-336
Relative Permittivity 28-336
Bulk Conductivity 28-337
Dielectric Loss Tangent 28-337
Magnetic Loss Tangent 28-338
Ferrite Materials 28-338

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

Magnetic Saturation 28-338
Lande G Factor 28-339
Delta H 28-339
Ferrite Permeability Tensor in HFSS 28-340
References 28-341
Anisotropic Materials 28-341
Anisotropic Relative Permeability Tensors 28-342
Anisotropic Relative Permittivity Tensors 28-342
Anisotropic Conductivity Tensors 28-343
Anisotropic Dielectric Loss Tangent Tensors 28-344
Anisotropic Magnetic Loss Tangent Tensors 28-345
Anisotropic Materials and Ports 28-346
Debye Materials 28-346
Figure 5: Calculated and measured frequency response of S12 28-352
Multipole Debye Model 28-354
Djordjevic-Sarkar Model Parameter Calculation 28-356
Expressions for Permittivity and Conductivity 28-357
Conductivity and Permittivity Limits 28-358
Force per Unit Area in HFSS 28-360
Energy Normalization of Eigen Solutions in HFSS 28-364
29 - Glossary 29-1
Glossary: A 29-1
Glossary: B 29-4
Glossary: C 29-6
Glossary: D 29-10
Glossary: E 29-14
Glossary: F 29-17
Glossary: G 29-20

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

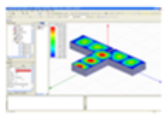
Glossary: H 29-22
Glossary: I 29-24
Glossary: J 29-25
Glossary: K 29-26
Glossary: L 29-27
Glossary: M 29-29
Glossary: N 29-32
Glossary: O 29-35
Glossary: P 29-37
Glossary: Q 29-41
Glossary: R 29-41
Glossary: S 29-43
Glossary: T 29-49
Glossary: U 29-52
Glossary: V 29-53
Glossary: W 29-54
Glossary: X 29-55
Glossary: Y 29-56
Glossary: Z 29-56
Index **Index-1**

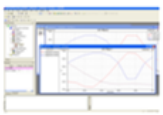
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

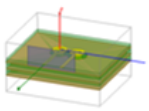
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

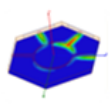
1 - Welcome to HFSS Help

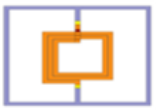
Click for help on these topics.

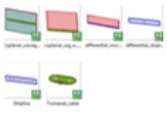

Getting Started HFSS

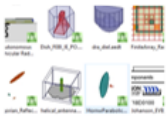

Getting Started Optimetrics

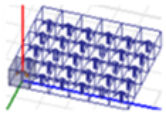

Signal Integrity


RF Microwave

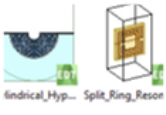

Integrated Circuits

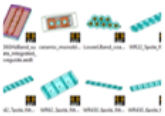

Transmission Lines


Antennas

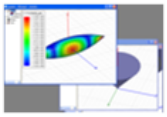

Arrays

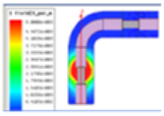
EMI EMC

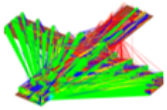

Metamaterial

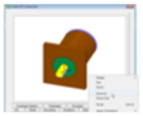

Filters

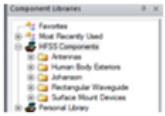
Finite Element Boundary Integral


IE Regions

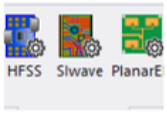

Introduction to HFSS Transient


SBR+

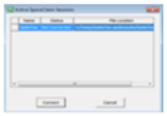

3D Components


3D Component Libraries

Antenna Design
Cable Modeling
Hearing Aid Compliance
MIMO
Wavelength Calculator
ACT Extensions/ Toolkits

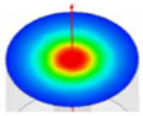


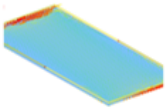
HFSS 3D Layout

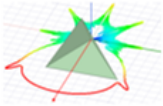

SpaceClaim Link

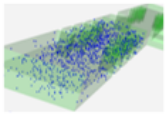
HFSS – EMA3D Datalink

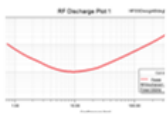
HFSS-EMIT


Eigenmode


Characteristic Mode

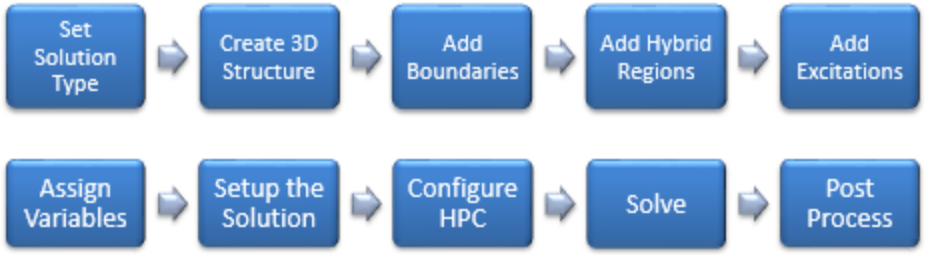

RCS


Multipaction


RF Discharge

Post Process

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



After you insert an HFSS design, you can access *An Introduction to HFSS* from **Help** > **Getting Started Guides**.

Click for Electronics Desktop and Working with Projects.

HFSS Getting Started Guides

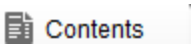
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modeling Quick Links

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

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

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

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

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

Materials Quick Links

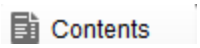
Use the following links for quick information on the following topics.

Assigning Materials
Searching for Materials
Assigning Material Property Types
Defining Frequency Dependent Material

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

Properties
Defining Functional Material Properties
Assigning DC Thickness
Solve Inside or On a Surface
Adding New Materials
Defining Variable Material Properties
Defining Material Properties as Expressions
Viewing and Modifying Material Attributes
Specifying Thermal Modifiers

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

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

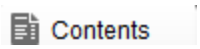
– Browse through the table of contents, expand entries, and close entries. Click an entry to see it in the content area.
- *A full text search* – To locate occurrences of a word or phrase that may be contained in the help, use the search function.

Meshing Quick Links

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

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

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

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

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

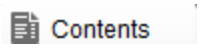
Analysis Quick Links

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

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

Specifying the HPC and Analysis Options
Configuring Distributed Analysis
Selecting Optimal Configurations for Distributed Analysis
Specifying Output Variable Convergence
Adding a Frequency Sweep
Running Simulations
Ansys Workbench Integration Overview
Remote Analysis
Specifying Solution Settings
Setting Adaptive Analysis Parameters
Setting the Order of Basis Functions
Options for Interpolating Sweeps
High Performance Computing Integration
Setup and Run an Array Simulation

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

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

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

Optimetrics Quick Links

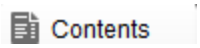
Use the following links for quick information on the following topics.

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

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

Adding a Cost Function
Running an Optimetrics Analysis
Getting Started Guide: Optimizing Waveguide T-Junction

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

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

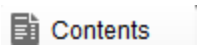
– Browse through the table of contents, expand entries, and close entries. Click an entry to see it in the content area.
- *A full text search* – To locate occurrences of a word or phrase that may be contained in the help, use the search function.

Results Quick Links

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

View solution data
Plot field overlay
Create 2D or 3D reports of S-parameters
Plot the finite element mesh
Create animations
Setting up a Near Field Sphere
Creating Reports
Scale an excitation's magnitude and modify its phase
Working with Traces
Using the Fields Calculator
Radiated Fields Post Processing
Setting up a Far Field Infinite Sphere

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

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

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

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

HFSS Getting Started Guides

General Guidance

In addition to the *Getting Started with HFSS* guides listed further down in this topic, the following general HFSS documentation is available:

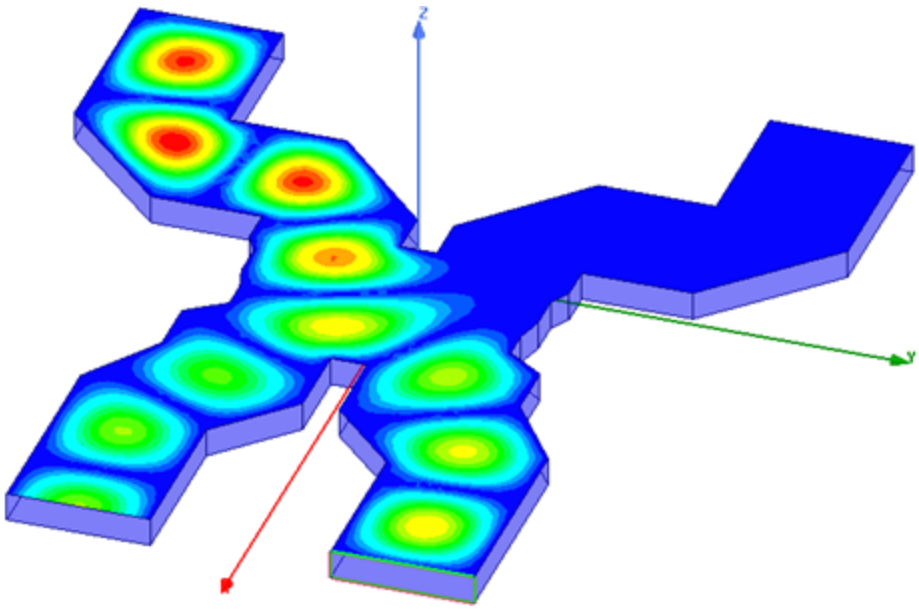
- *HFSS Scripting Guide*
- *An Introduction to HFSS*
- *HPC Administrator's Guide*
- *An Introduction to Multi-Frequency Adaptive Meshing in HFSS*

Getting Started with HFSS

The following guides are tutorials covering various types of HFSS designs. These *Getting Started* guides typically walk you through the steps to build the geometry, set up the solution, run the analysis, and evaluate the results (by generating plots, overlays, and animations):

- *20 GHz Waveguide Combiner*

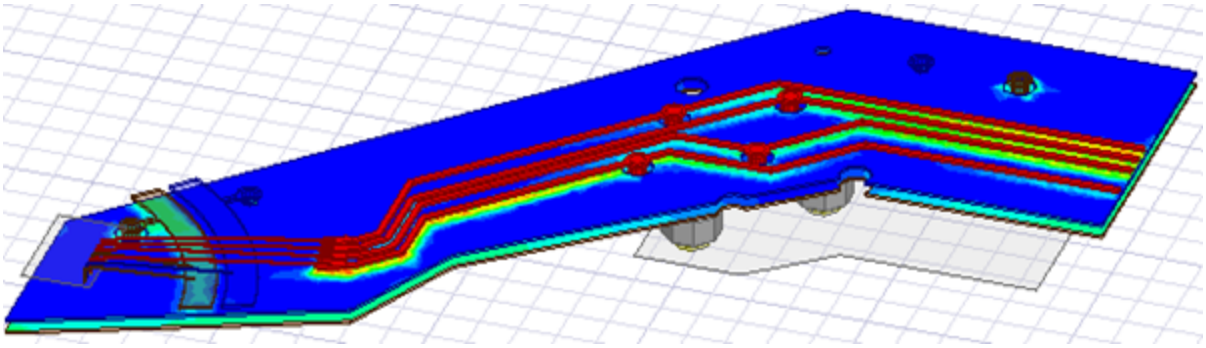
  This *Getting Started Guide* describes how to set up and solve a two-way, low-loss waveguide combiner.



- *Ball Grid Array IC Package*

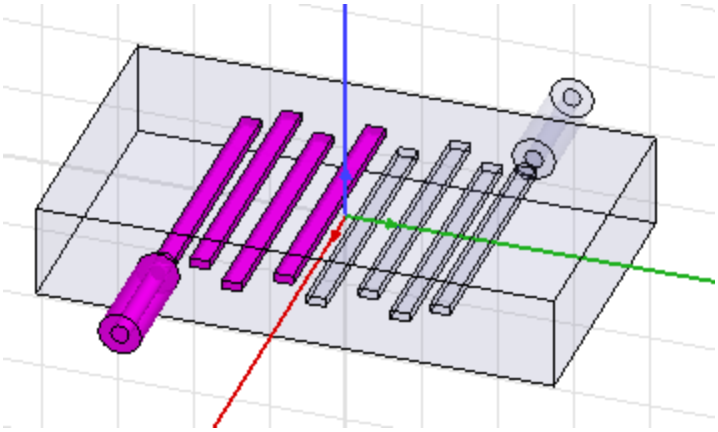
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This *Getting Started Guide* is written for HFSS Transient beginners as well as experienced users who are using HFSS for the first time. This manual guides you through the setup, solution, and analysis of a transient simulation.



- *Bandpass Filter*

  This *Getting Started Guide* assumes some familiarity with HFSS. It guides you through the process of creating a bandpass filter. It includes the use of Duplicate Around Axis commands.



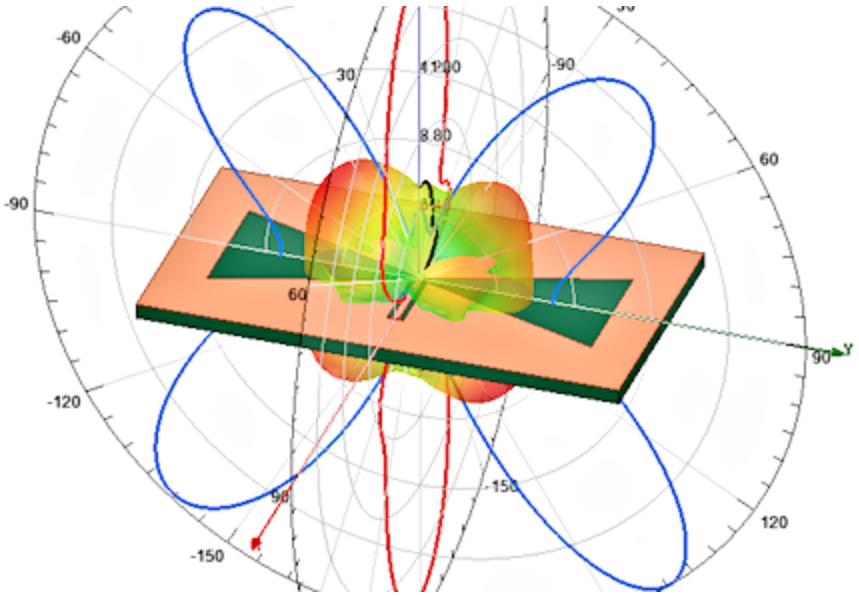
  This design has a frequency sweep. If you want to distribute the solutions of the frequencies for efficient simulation, you can use High Performance Computing (HPC). The HPC options are discussed in this guide. For more information about how to set up HPC, see the **Add HPC Analysis Setup** section in the **Getting Started Guide for Bandpass Filter**. You can access the design from the Examples folder, which also contains a short description on the HPC setup in the associated PDF.

- *Bow Tie Antenna*

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

This Getting Started Guide contains three separate tutorials that show you how to create a coplanar, waveguide-fed, bow tie antenna in Ansys HFSS. There are multiple ways to create and solve designs in HFSS. A bow tie antenna is a simple design that can be created using any of the following methods:

- **Parametric Method:** Makes use of advanced 3D Modeler commands
- **Linear Method:** The traditional approach of creating the antenna in a "linear" fashion
- **Antenna Design Toolkit:** Automatically generates the antenna in a ready-to-solve state

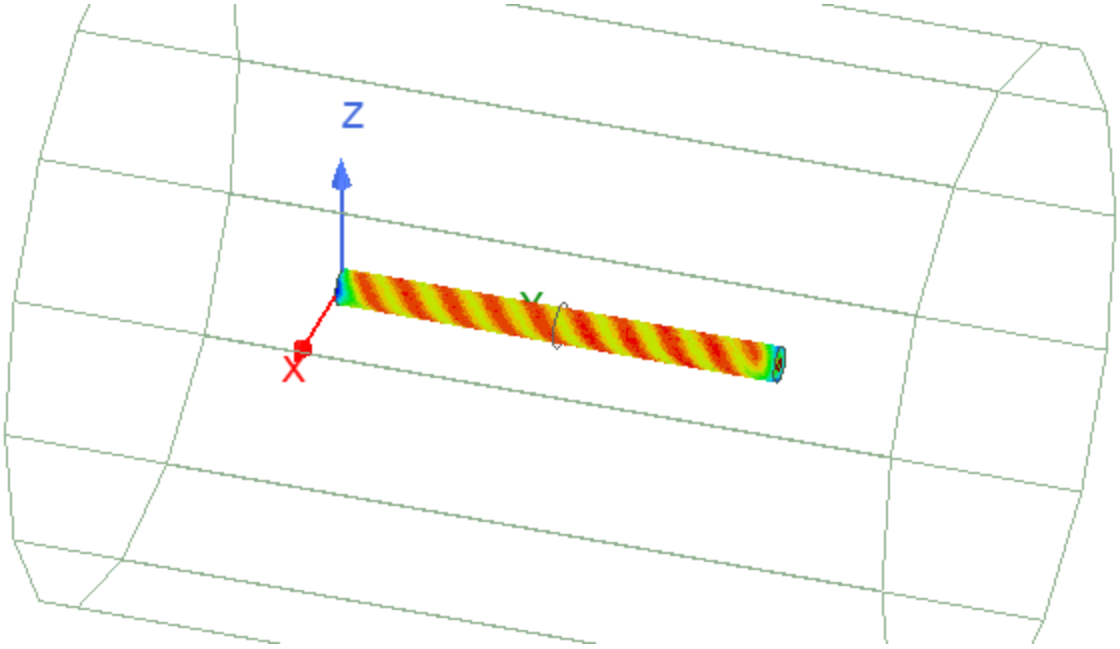


The first tutorial covers the parametric method, the second one covers the "linear" method, and the third one covers the automatic method of creating the antenna. You will create 2D plots of the radiation pattern and create a 3D gain plot (far field pattern). You will also overlay the 3D gain plot and the 2D radiation pattern plots on the model geometry.

- *Cable Modeling Solutions*

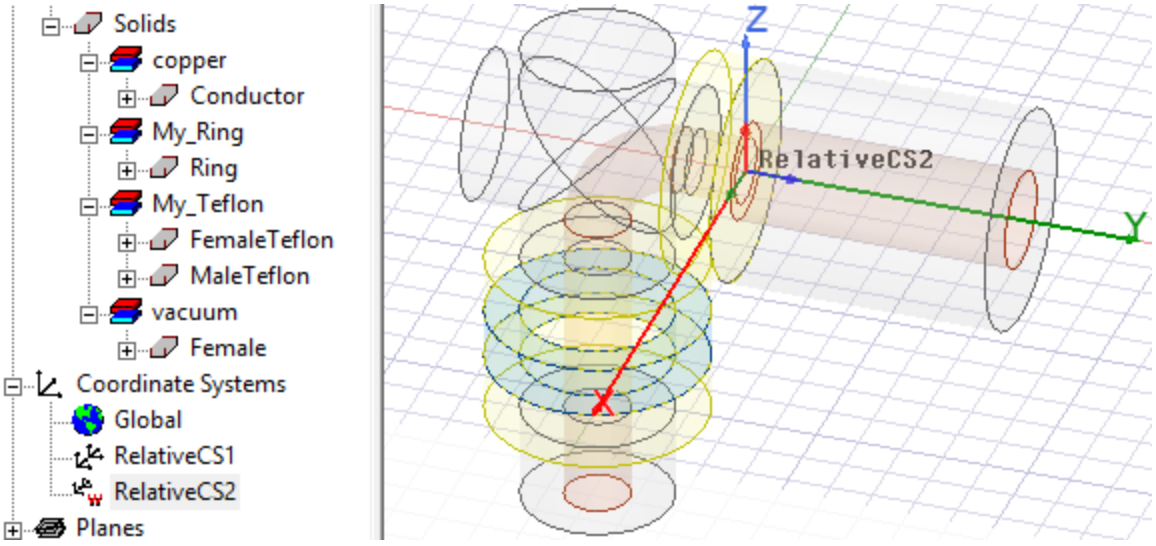
  This *Getting Started Guide* covers the cable modeling workflow. You can now complete the entire cable modeling workflow within a single HFSS design. Cables are defined as 3D components. Required 2D Extractor and Circuit simulations are set up and run automatically in the background as part of the HFSS solution process. No manual setup, linking, or transfer of data is required on the analyst's part.

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



- *Coax Connector*

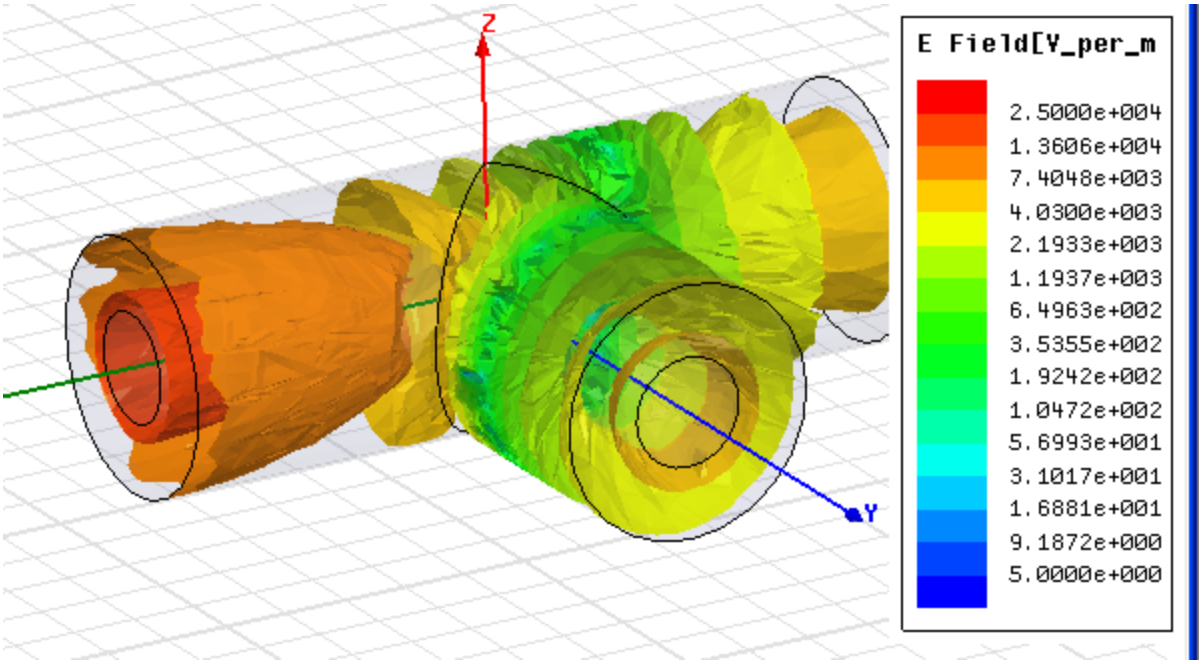
  This *Getting Started Guide* assumes some familiarity with HFSS. It guides you through the process of creating a coax connector. This includes the use of relative coordinate systems as an aid to building the model. It also uses Boolean and sweep operations.



- *Coax Tee*

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

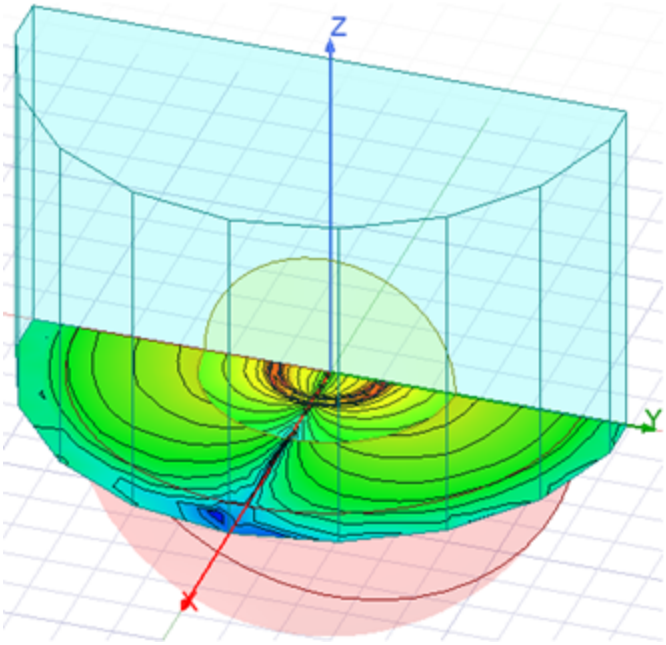
This *Getting Started Guide* is suitable for beginners and advanced users. It includes instructions to create and solve a Coax Tee and evaluate the results of the simulation (including creating an S-parameters vs. Frequency plot and an E-Field overlay). The E-Field is overlaid on the Coax Dielectric, which is modeled as a vacuum region surrounding the Coax Center Pin.



- *Dielectric Resonator Antenna*

  This *Getting Started Guide* is written for Ansys Electronics Desktop beginners as well as experienced users who are using HFSS for the first time. This guide leads you step-by-step through creating, solving, and analyzing the results of a dielectric resonator antenna.

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

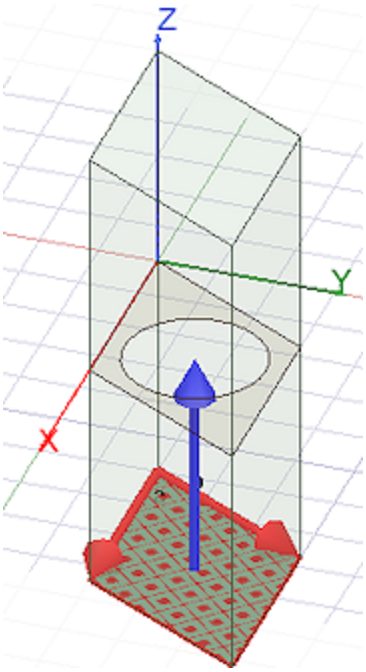


- *Floquet Ports*

  This *Getting Started Guide* is intended as supplementary material to HFSS users. We assume you have some experience designing projects using HFSS. The document includes instructions to create, solve, and analyze results of models that are excited with Floquet ports.

  By following the steps in this guide, you will learn how to set up Floquet ports in Ansys Electronics Desktop.

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



- *Keyboard Shortcuts*

  For those who prefer communicating instructions using a keyboard more often than a mouse, this *Getting Started* guide presents two different techniques for working with Ansys Electronics Desktop:

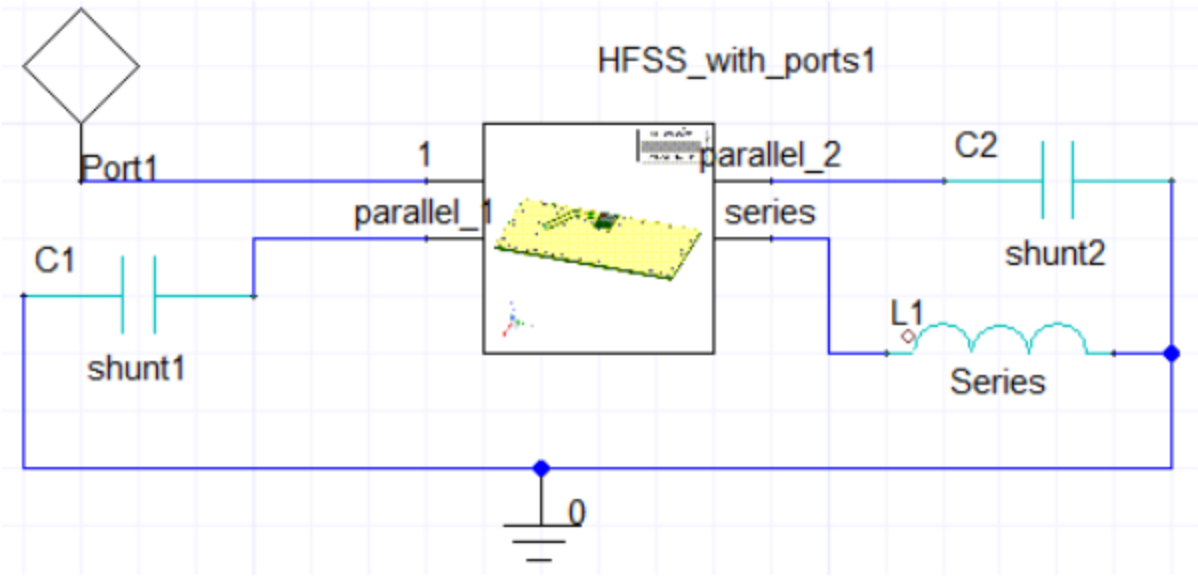
  - Shortcut keys (either predefined or user-defined)
  - Keystroke sequences for menu bar navigation to trigger desired actions

  This guide shows you how to create, simulate, and analyze a probe fed, slot coupled patch antenna using the HFSS design environment, with an emphasis on using keyboard shortcuts to complete the work.

- *Matching Network – Using Tuning in Circuits*

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

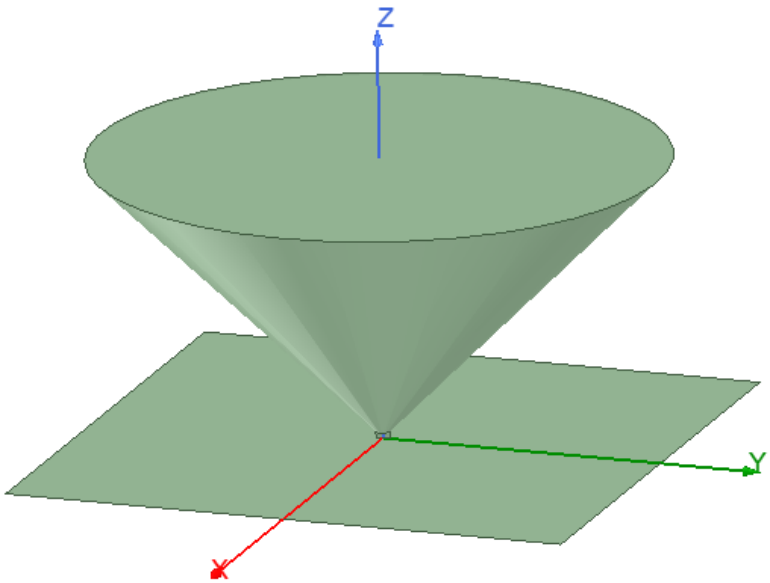
In this *Getting Started Guide*, you will learn about dynamic linking capabilities between HFSS and Circuit designs in the Ansys Electronics Desktop application. Specifically, the guide provides an example of a matching network for a Bluetooth chip antenna.



- *Monocone Antenna*

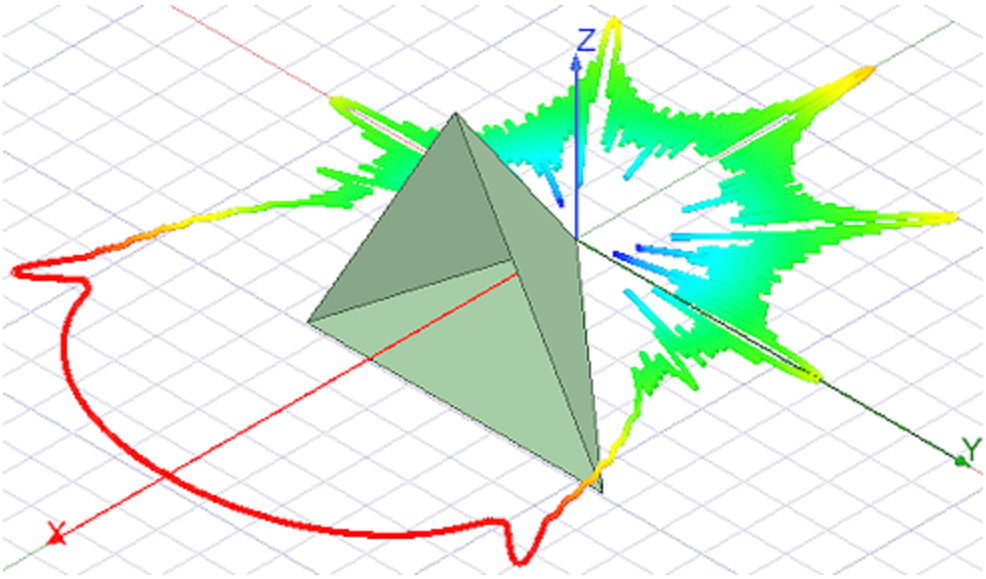
  This *Getting Started Guide* looks at the radiation pattern from a monocone antenna using the HFSS design type with an IE hybrid region applied to the model objects. An IE Region results in a fullwave Integral Equation solver being used to analyze the model. This solver is particularly well suited for analyzing large conducting objects, and it eliminates the necessity of creating an open region (that is, a vacuum or air box) around the model to determine far field effects.

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



- *Monostatic RCS with SBR+*

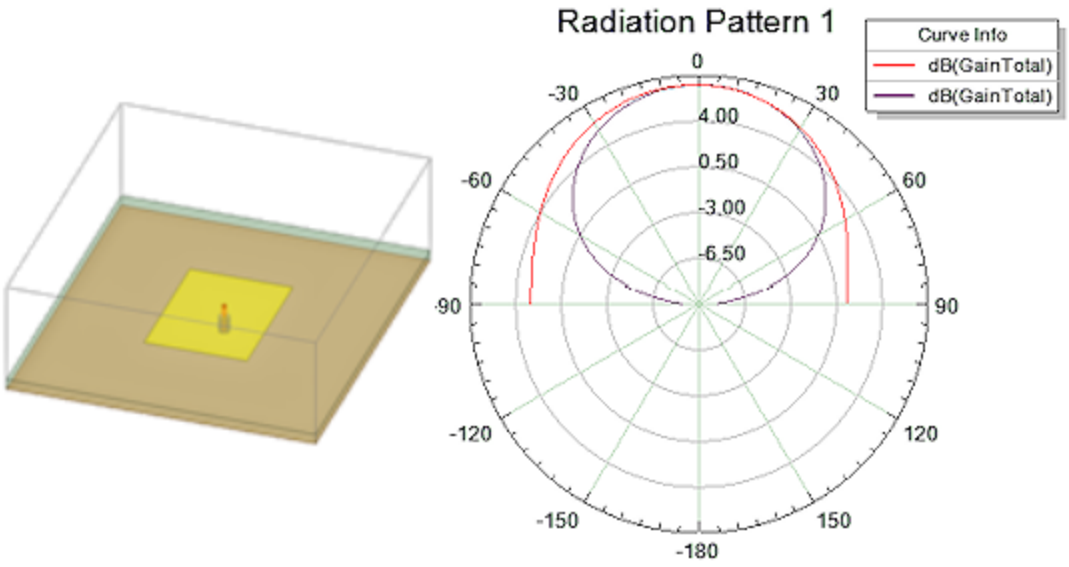
  This *Getting Started Guide* describes how to use HFSS to model a Monostatic Radar Cross Section (RCS) of a metal trihedral reflector using an SBR+ Hybrid Region.



- *Probe Feed Patch Antenna*

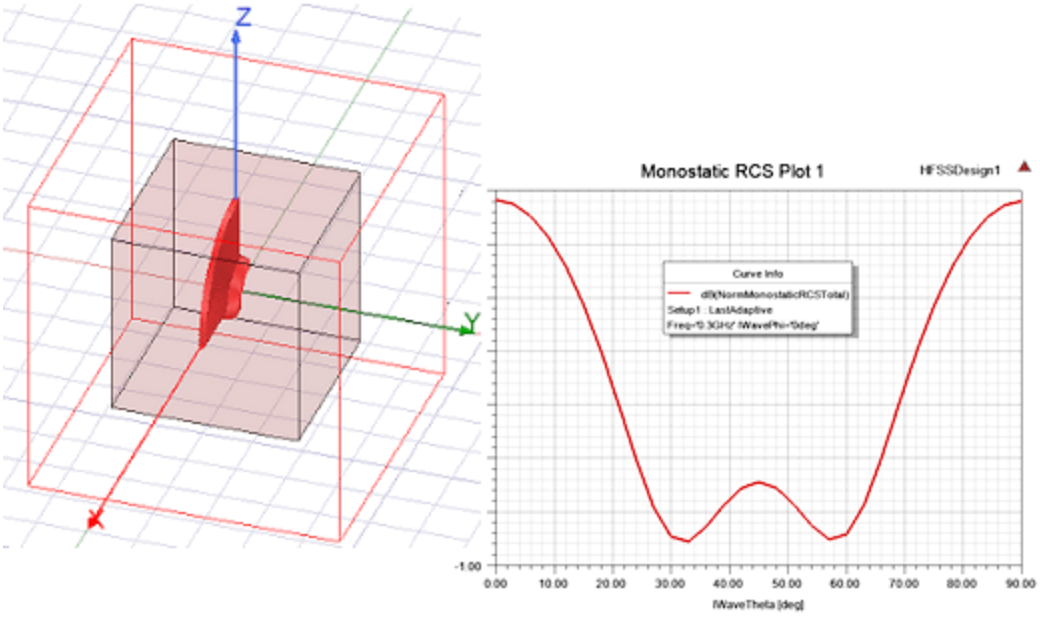
  This *Getting Started* guide assumes some familiarity with HFSS. It includes the use of Perfect E and Radiation boundaries and a Radiation Pattern plot.

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



- *Radar Cross Section*

  This guide shows how to create and solve a model for computing a radar cross section (RCS).

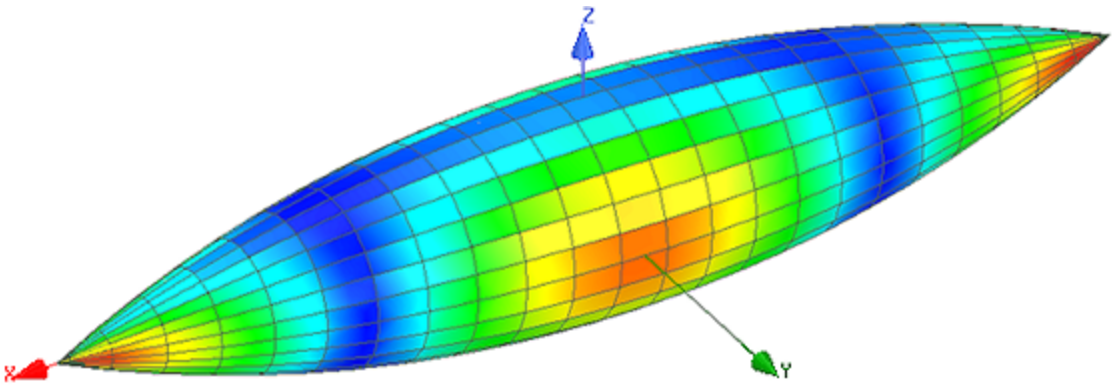


- *RCS Test Model (Ogive)*

  This *Getting Started Guide* looks at the scattering from a standard Radar Cross Section (RCS) test model (specifically, the ogive) using the HFSS design type with an IE region

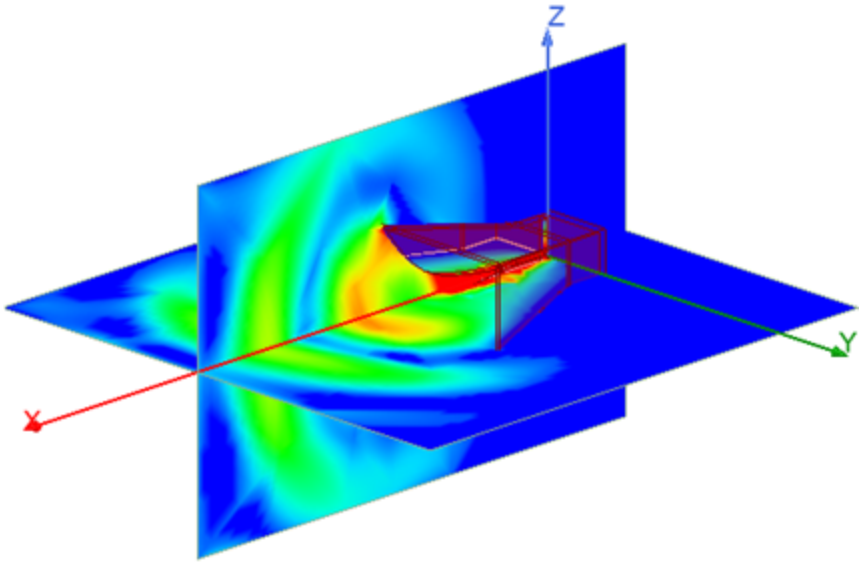
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

assigned to the ogive. An IE region results in a full-wave Integral Equation solver being used to analyze the model. This solver is particularly well suited for analyzing large conducting objects, and it eliminates the necessity of creating an open region (that is, a vacuum or air box) around the model to determine far field effects.



- *Ridged Horn Antenna*

  This *Getting Started Guide* is written for HFSS Transient beginners as well as experienced users who are using HFSS for the first time. This manual guides you through the setup, solution, and analysis of a transient horn antenna problem.

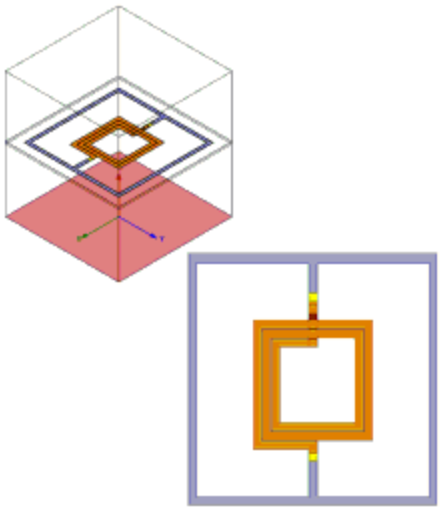


  By following this guide, you will learn how to set up HFSS transient antenna problems.

- *Silicon Spiral Inductor*

  This *Getting Started* guide assumes some familiarity with HFSS. It includes the use of Perfect E and Radiation boundaries and Output Variables in generating plots.

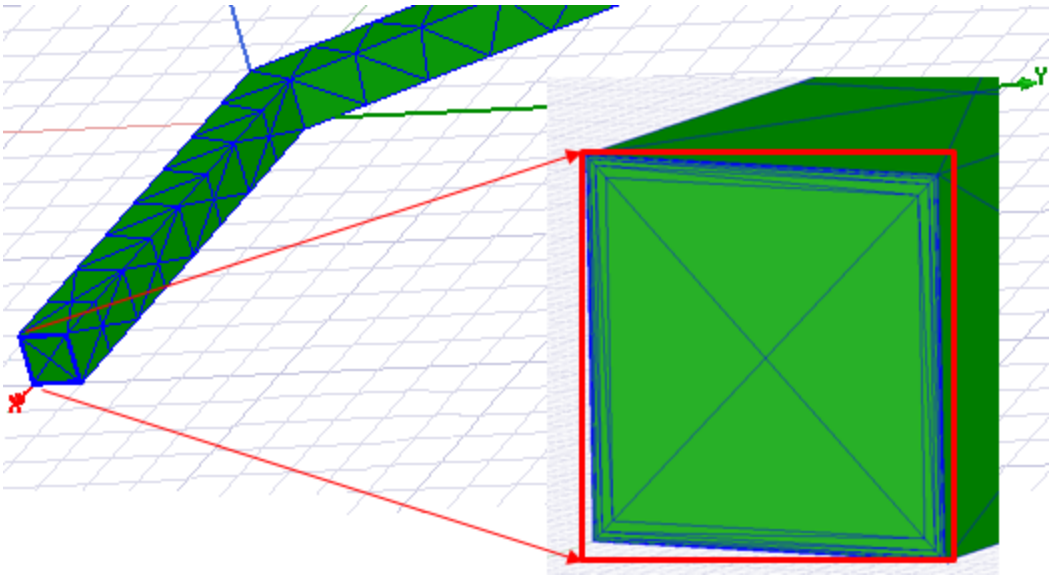
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Related Topic: Application Specific Modeling Guide: Spiral Inductors on Silicon Substrate

- *Skin Depth Seeding*

  By following the steps in this guide, you will learn to use Skin Depth-Based mesh refinement for HFSS or Maxwell Modelers to specify a number of layers and a skin depth as an alternative to modeling physical layers.

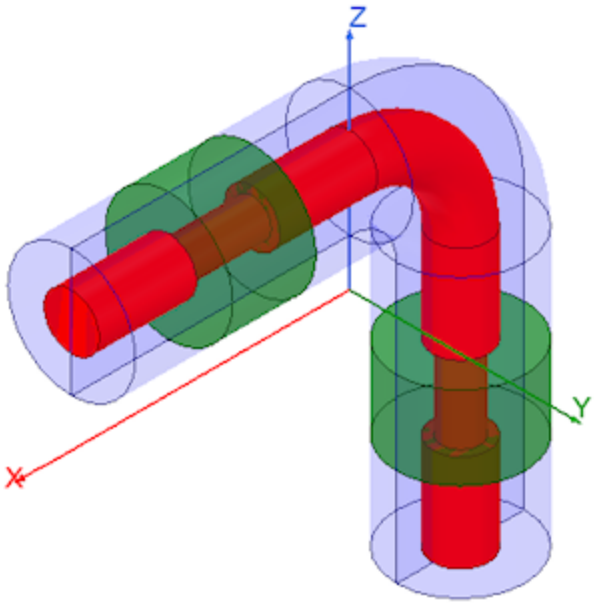


  The guide also shows how to use mesh plots with cut planes, as well as a repurposed **Model Analysis** dialog to view and obtain statistics on layering success.

- *TDR for Coax Bend*

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

This *Getting Started Guide* is written for HFSS Transient beginners as well as experienced users who are using HFSS for the first time. This manual guides you through the setup, solution, and analysis of a time delayed response problem for a coax bend.

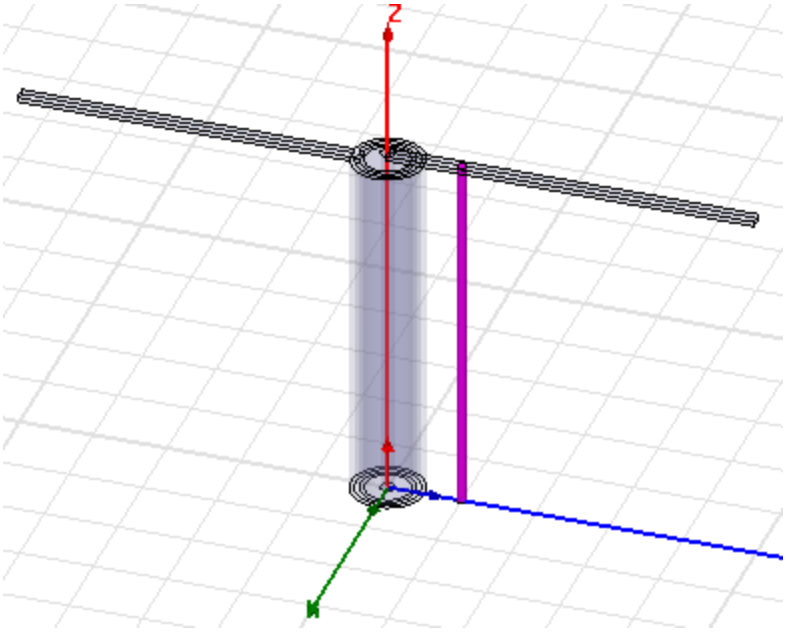


By following this guide, you will learn how to set up HFSS transient problems.

- *Ultra High Frequency Probe*

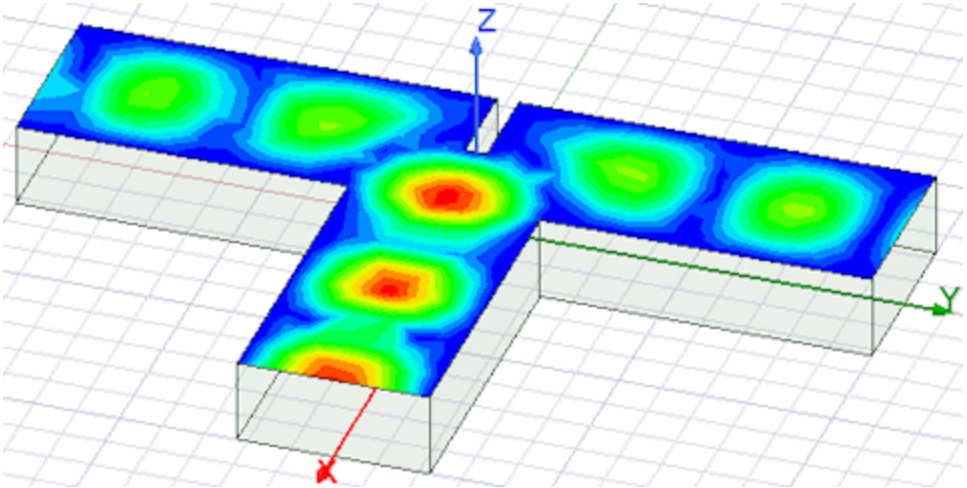
  This *Getting Started Guide* assumes some familiarity with HFSS. It includes the use of boolean operations and the use of a ground plane and radiation boundaries.

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



- *Waveguide T-Junction*

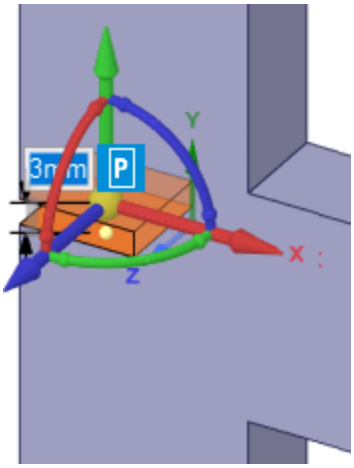
  This *Getting Started Guide* is written for Ansys Electronics Desktop beginners as well as experienced users who are using the HFSS design type for the first time. This guide will lead you step-by-step through creating, solving, and analyzing the results of a T-shaped waveguide with an inductive septum. This type of structure is used to split an incoming microwave signal into two outgoing signals. The waveguide's transmission and reflection of the signal will depend on the position of the septum.



- *Waveguide T-Junction Optimization*

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

This *Getting Started Guide* is written for Optimetrics beginners as well as experienced users who are using Optimetrics for the first time. You must have completed *Getting Started with HFSS: A Waveguide T-Junction* before you begin this guide.

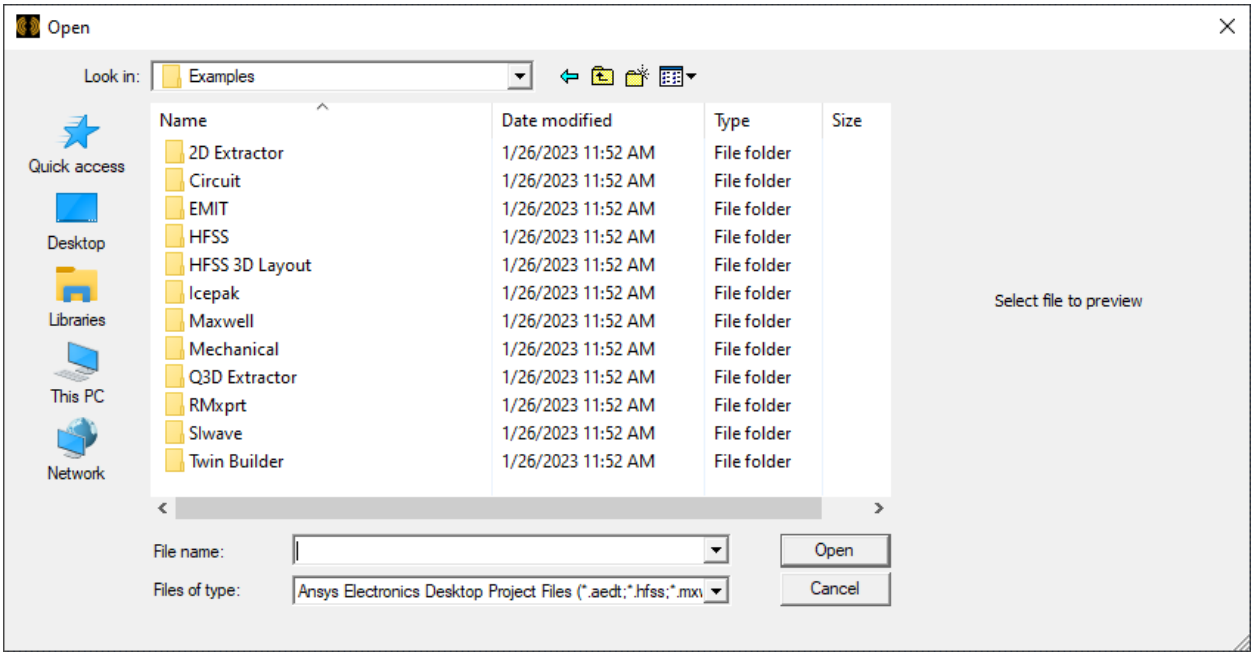


You will use Ansys Electromagnetics Optimetrics software to find an optimal position for the septum. Prior to performing the optimization, you will set up and solve a parametric analysis.

Example Projects

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

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



Example projects are organized by the design type. The examples under Circuit, HFSS, and HFSS 3D Layout include subfolders, organizing the projects according to different applications:

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

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

- HFSS Examples Associated with a Getting Started Guide

Many other projects include descriptions in the help.

HFSS Examples

The example models are provided as an additional applications resource and are included with the software installation.

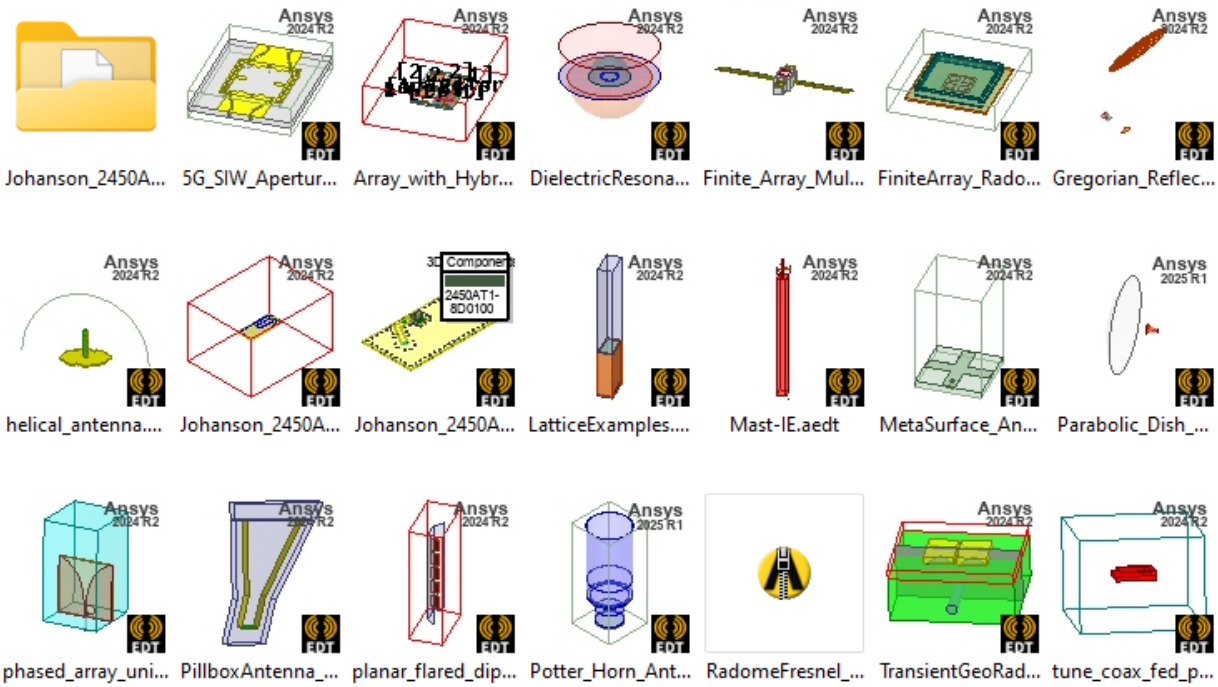
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Many, but not all, of the following HFSS example projects have descriptions included in the help. They are organized by category—ADAS, Antennas, EMI EMC, RCS, Signal Integrity, Metamaterial, RF Microwave, and Transmission Lines. For models with included descriptions in the HFSS help, the icons in the figures are linked to the associated topic.

ADAS Examples (Advanced Driver Assistance Systems)

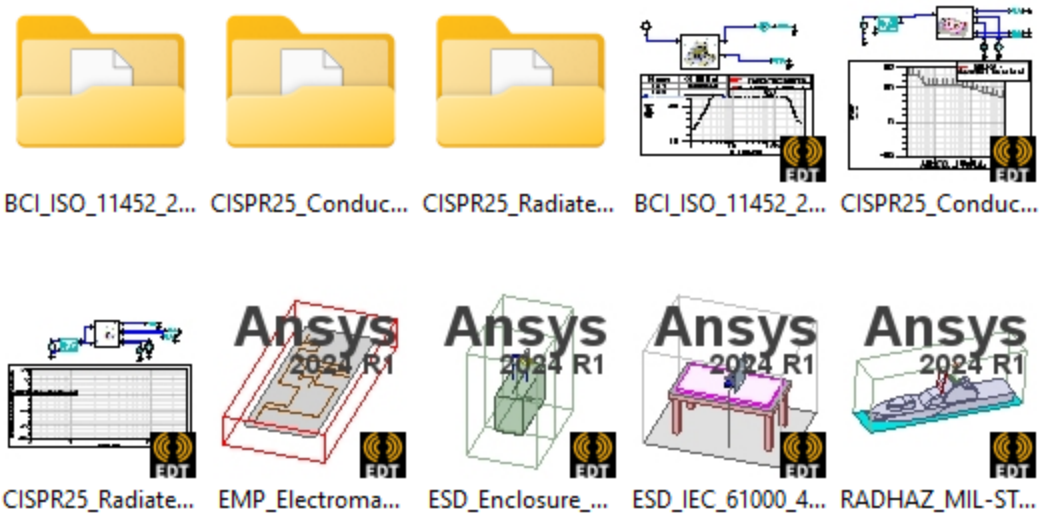
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Antenna Examples

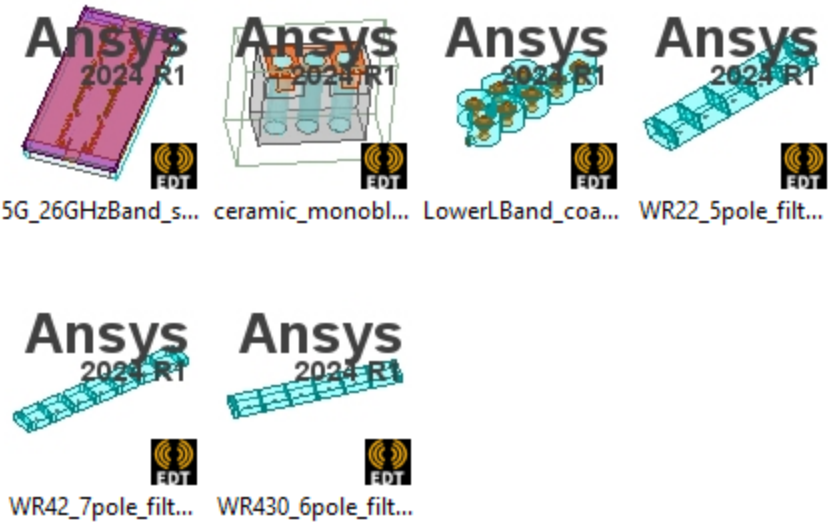


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EMI EMC Examples (Electromagnetic Interference, Electromagnetic Compatibility)



Filter Examples



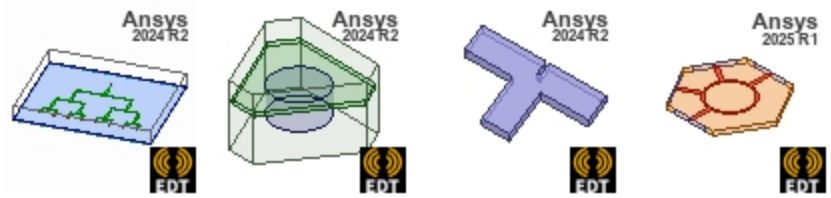
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

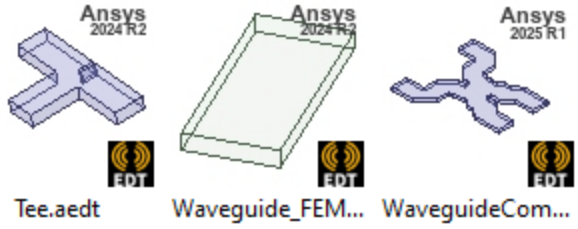
Metamaterial Examples

RCS Examples (Radar Cross Section)

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

RF Microwave Examples

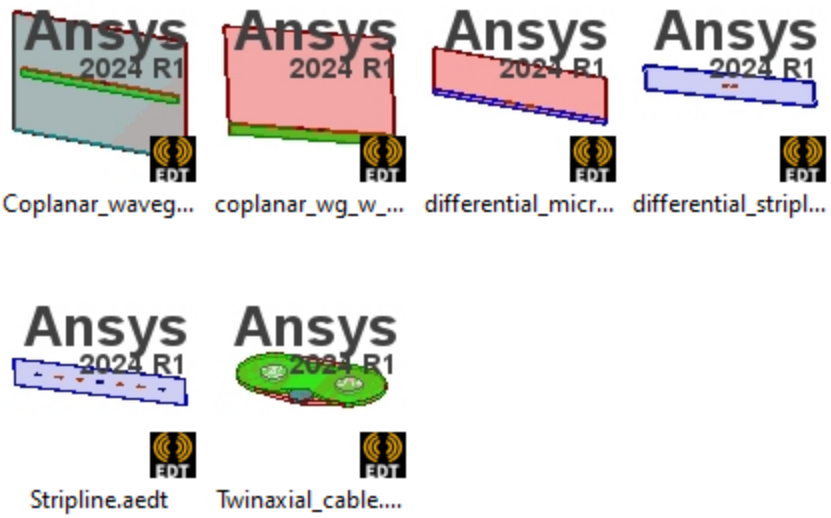




Signal Integrity Examples

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

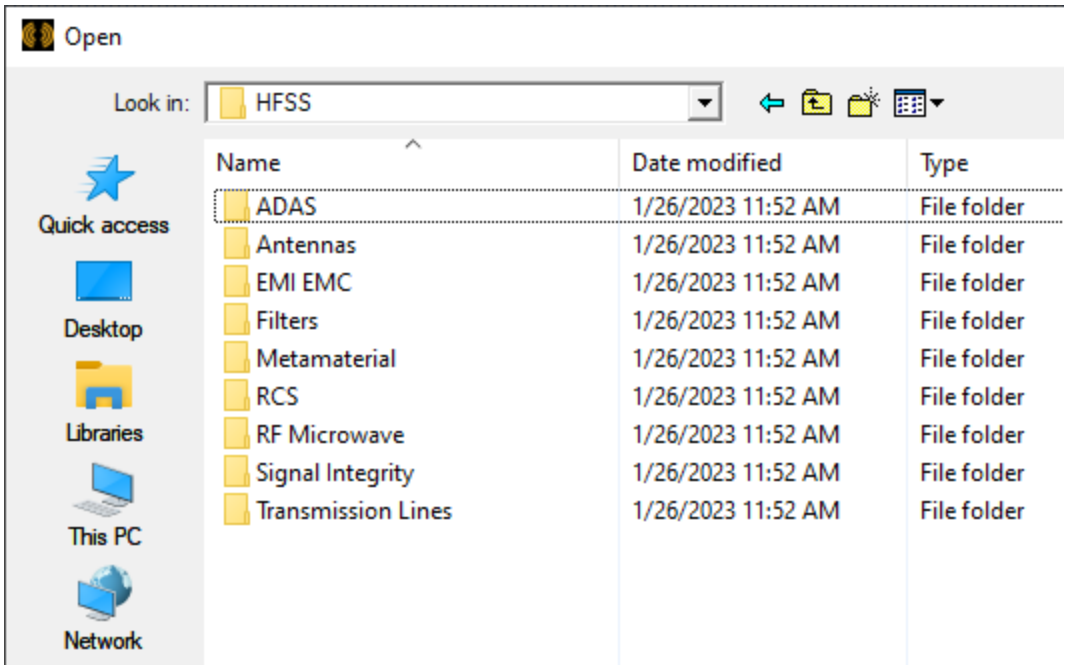
Transmission Line Examples



For further examples, see the HFSS Getting Started Guides (link below).

HFSS Examples Associated with Getting Started Guides

The HFSS examples folder has subfolders for ADAS, Antennas, EMI EMC, Metamaterial, RCS, RF Microwave, Signal Integrity and Transmission Lines:



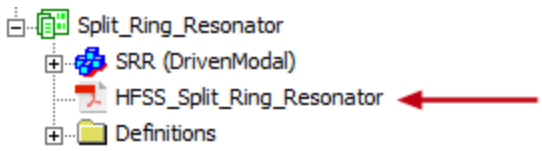
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following HFSS example projects are associated with detailed Getting Started Guides:

- Bandpass Filter
- Dielectric Resonator Antenna
- Johanson_2450AT18D0100-EB1SMA (Bluetooth chip antenna evaluation board, with matching network tuning in a linked Circuit design)
- Monostatic RCS with SBR+
- RCS Test Model (Ogive)
- Tee and OptimTee Waveguide Projects
- Waveguide Combiner Example Project

Also see Example Projects for an index of all HFSS examples grouped by category (ADAS, Antennas, EMI EMC, Filter, RCS, Signal Integrity, Metamaterials, RF Microwave, and Transmission Lines). For projects that have descriptions included in the help, the model thumbnail images in the figures are linked to the description topics.

Example models typically reference a PDF document, which is located in the same folder and explains the project.



Additionally, the following topics are each specific to a single example category:

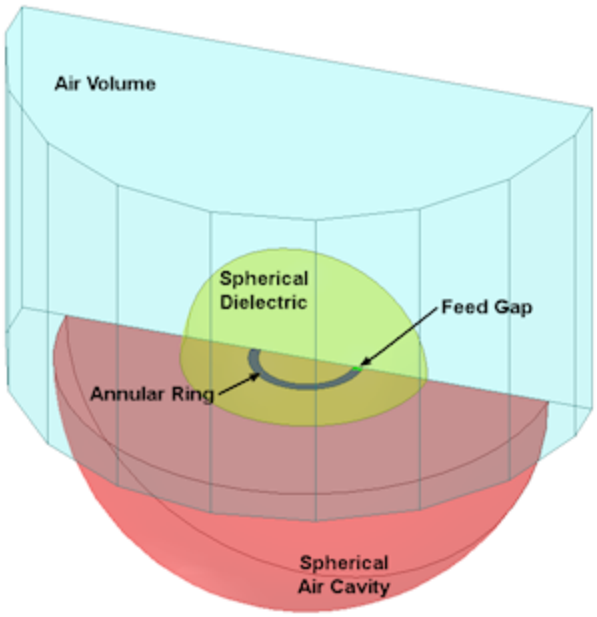
- ADAS Examples (Advanced Driver Assistance Systems)
- Antenna Examples
- EMI EMC Examples (Electromagnetic Interference, Electromagnetic Compatibility)
- Metamaterial Examples
- RCS Examples (Radar Cross Section)
- RF Microwave Examples
- Signal Integrity Examples
- Transmission Line Examples

Dielectric Resonator Antenna

The dra_diel model in the **...Examples/Electronics Desktop/HFSS/Antennas** subfolder contains the modal solution project described in the following getting started guide:

- Getting Started with HFSS: Dielectric Resonator Antenna (DRA)

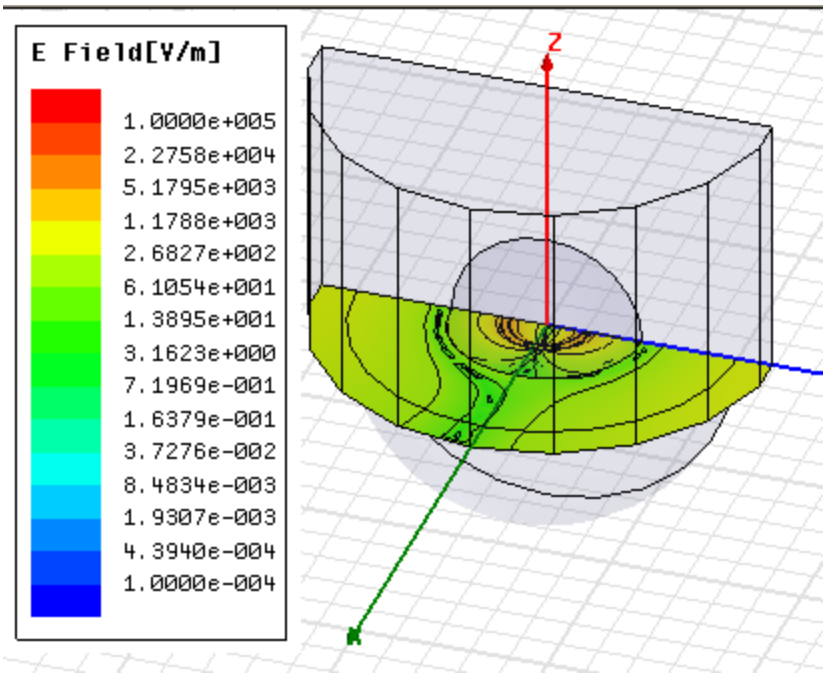
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This design demonstrates the use of:

- Boolean operations on geometries
- the use of symmetry and radiation boundaries
- meshing
- lumped ports
- modifying the impedance multiplier because of symmetry
- animation of a field plot
- setting up an infinite sphere and computing antenna parameters

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



Johanson 2450AT18D0100-EB1SMA

Abstract

This example is one of two Johanson Technology chip antenna and evaluation board (EVB) models that you can find in the **Examples\HFSS\Antennas** subfolder included in the Ansys Electromagnetics Suite installation. This model is associated with the following HFSS *Getting Started* guide:

Getting Started with HFSS: *Matching Network – Using Tuning in Circuits*

The example demonstrates the following:

- Using Johanson 3D Components
- Dynamically linking an HFSS design to a Circuit simulation
- Using lumped components and the tuning feature in the Circuit design to match the antenna (optimizing its performance)
- Generating several types of plots and overlays
- Pushing excitations from a Circuit design to an HFSS design

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

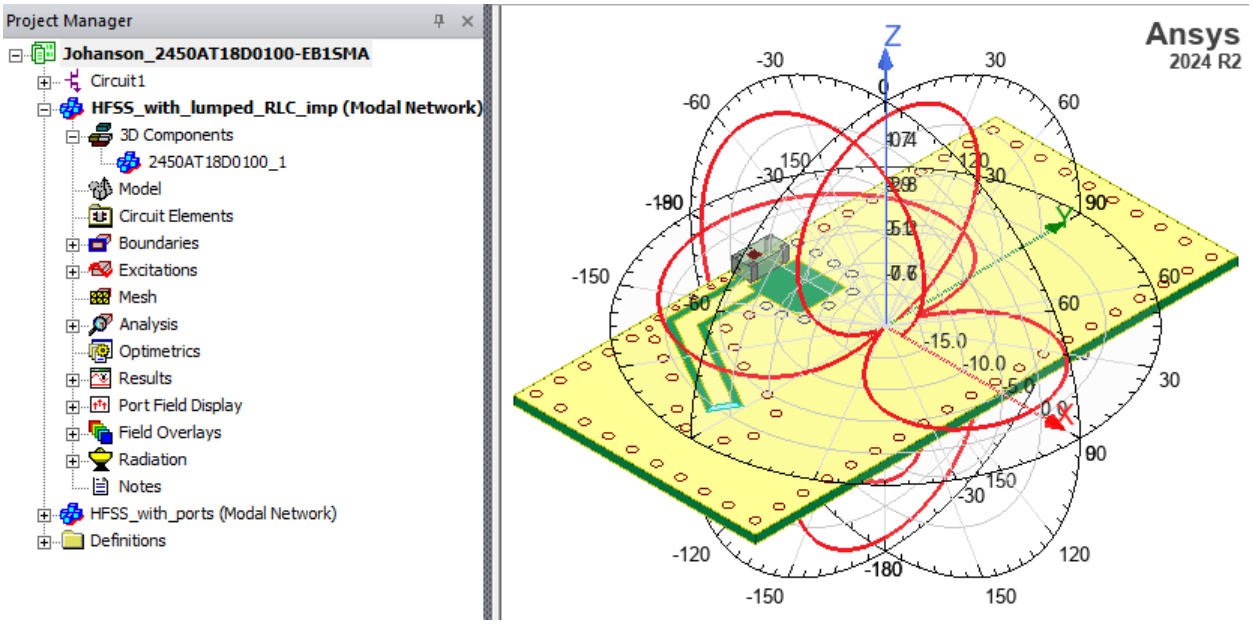


Figure 1: Johanson Chip Antenna and EVB with Overlaid 2D Gain Plots

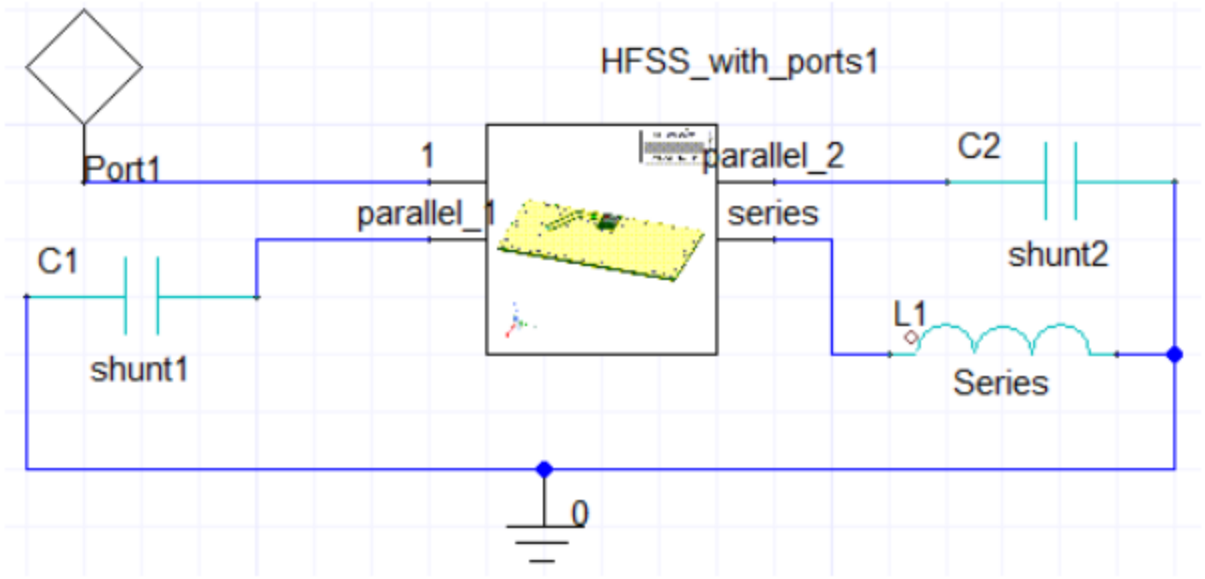


Figure 2: Matching Circuit Schematic

Version: 2024 R2

Simulation Time: 2 minutes: 20 seconds (using 8 cores), less than 1.4 GB RAM used

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

Model and Setup Details

The model consists of a single chip antenna mounted to an EVB – Johanson Technology, legacy part number: 250AT18D0100-EB1SMA. The antenna is a vendor model included as an encrypted 3D component. It is excited by a lumped port and contained within an automatically defined open region with a radiation boundary, as detailed below.

The example project contains three designs, summarized as follows:

- **Circuit1:** This design is dynamically linked to the third design (*HFSS_with_ports*). You can vary the discrete component values and immediately see the effect on the results. This is a fast and easy way of finding the ideal component values to optimize the design performance. You can then push the optimized parameters to the third design (*HFSS_with_ports*), where they are represented as lumped port excitations. For details on this process, refer to the *Matching Network* getting started guide referenced in the previous *Abstract* section.
- **HFSS_with_lumped_RLC_imp (Modal Network):** This design is the fastest and most straightforward way of determining the antenna and matching network performance when you do not intend to modify the RLC values. The tuning components are represented as lumped RLC boundaries. If you choose to modify any component value, you must run a new solution to see the updated behavior. Therefore, this method is not as desirable as the linked Circuit approach when the point of the analysis is to tune the component parameters for optimal results.
- **HFSS_with_ports (Modal Network):** In this design, tuning components are represented as lumped port excitations with renormalized impedances corresponding to the tuned component values pushed from the first design (*Circuit1*). When solved, this setup produces the same results as determined for the final tuned RLC component values in the Circuit design.

> **Note:**
>
> As an alternative approach to circuit tuning, you can substitute postprocessing variables for the renormalized impedance values. By varying the postprocessing variables, you can immediately see the effect of the design behavior (without having to rerun the solution). Similarly, you could scale the field amplitudes in the *Edit Sources* dialog box to see immediate tuning results.

This topic describes the setup of the HFSS antenna evaluation board solution and its results only. For details concerning the dynamic circuit link, adding lumped components for tuning, the tuning process, and pushing the excitations from the tuned circuit design to HFSS, see the above-referenced *getting started guide*.

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

> **Note:**
>
> The project filename and the design names differ between the example model and the getting started guide, but the setup procedure and results are otherwise the same.

Boundaries:

- Open Region with Radiation boundary (*AutoOpen1*), based on 2440 MHz adaptive solution frequency (**HFSS** > **Model** > **Create Open Region**)
- Finite Conductivity (*FiniteCond1*), 58 x 10^6 Siemens/m, classic infinite thickness model, at bottom face of PCB (**HFSS** > **Boundaries** > **Assign** > **Finite Conductivity**)

Excitations:

- Lumped Port (*P1*), 50 Ω impedance, single mode, at the beginning of the antenna feed

Mesh Setting:

- No mesh operations
- Initial Mesh Settings: *Curved Surface Meshing* slider three ticks coarser than default (to reduce the element count at the cylindrical grounding vias):

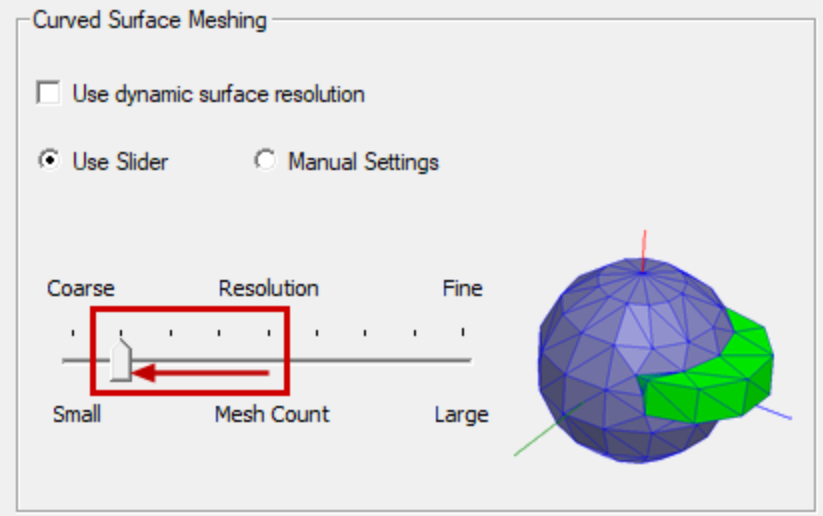


Figure 3: Initial Mesh Settings Dialog Box, Curved Surface Meshing Section

Setup:

- **Bluetooth:** 2440 MHz single solution frequency, *Do Lamda Refinement* enabled, 30% max refinement per pass, *Auto Select Direct/Iterative* selected, *Save Fields* enabled
- 1 – 4 GHz sweep in 0.001 GHz steps, 250 maximum solutions, 0.5% error tolerance

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

Postprocessing

After solving (**Simulation** > **Analyze all**), you can view different post-processing results. Look in the Project Manager under *Results* and double-click on the different predefined reports (**Figure 4**):

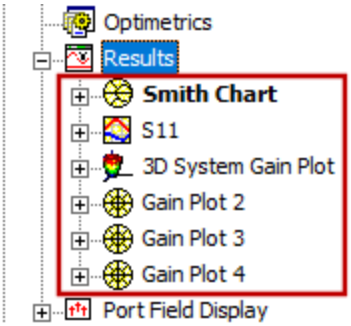


Figure 4: Predefined Reports Listed in the Project Manager

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

Predefined Reports:

Figures 5 through 10, which follow, show each of the six predefined reports in the HFSS designs of the example project. These results are based on the optimized matching network as solved in the third design (*HFSS_with_ports*). That is, the excitations have been pushed from the Circuit design to the linked HFSS design after tuning had been completed):

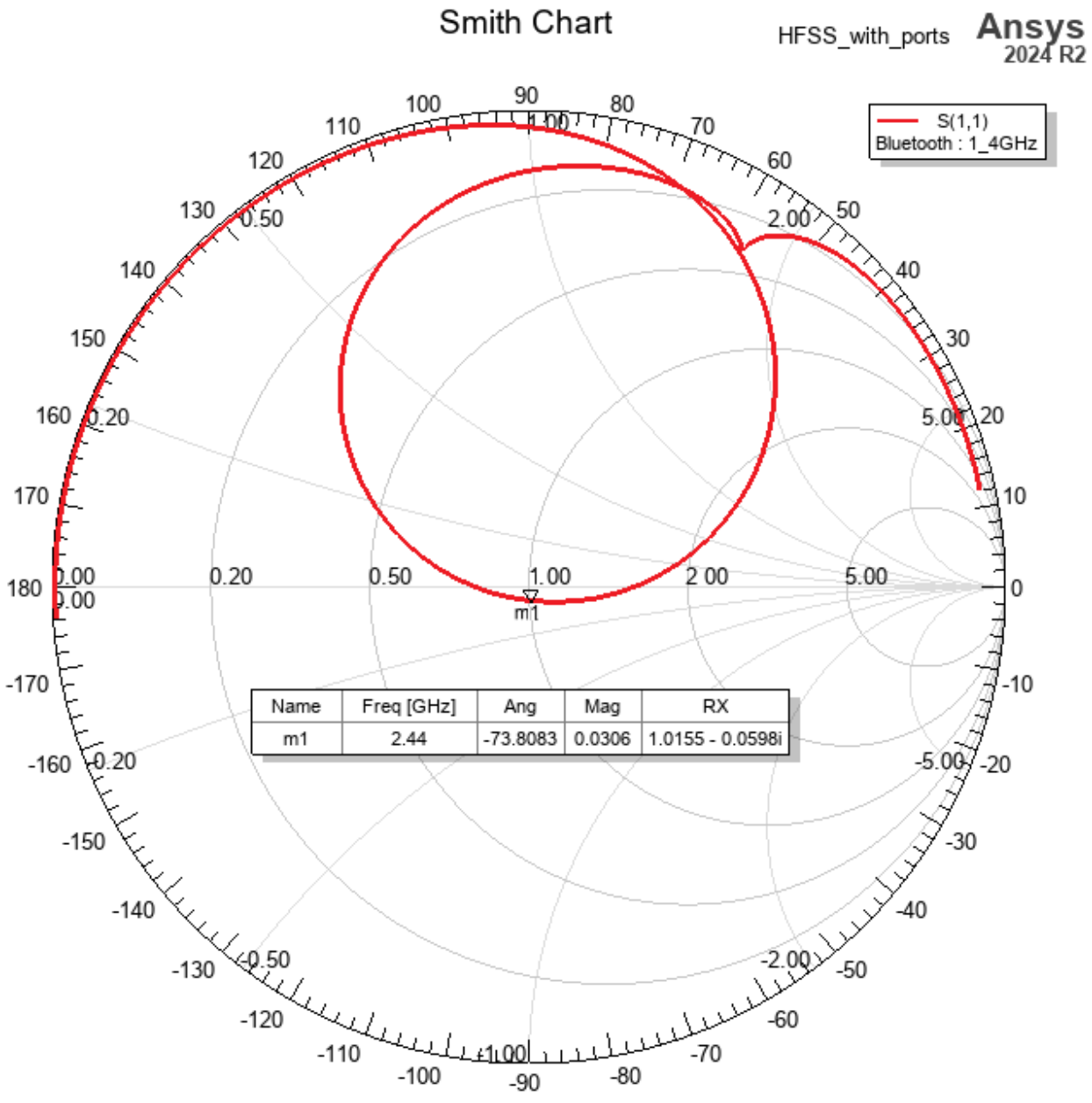


Figure 5: Smith Chart

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

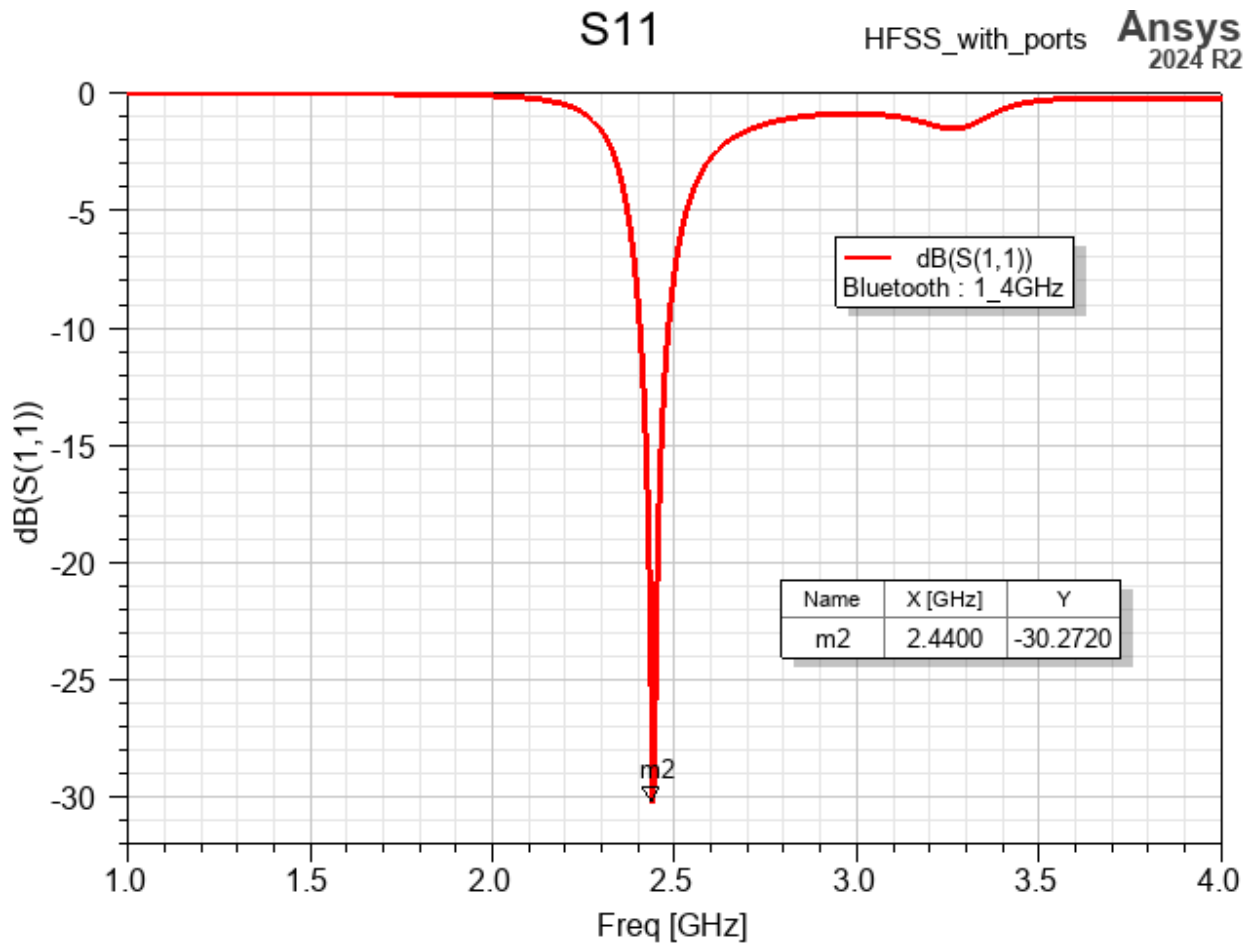


Figure 6: S11 Plot (dB)

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

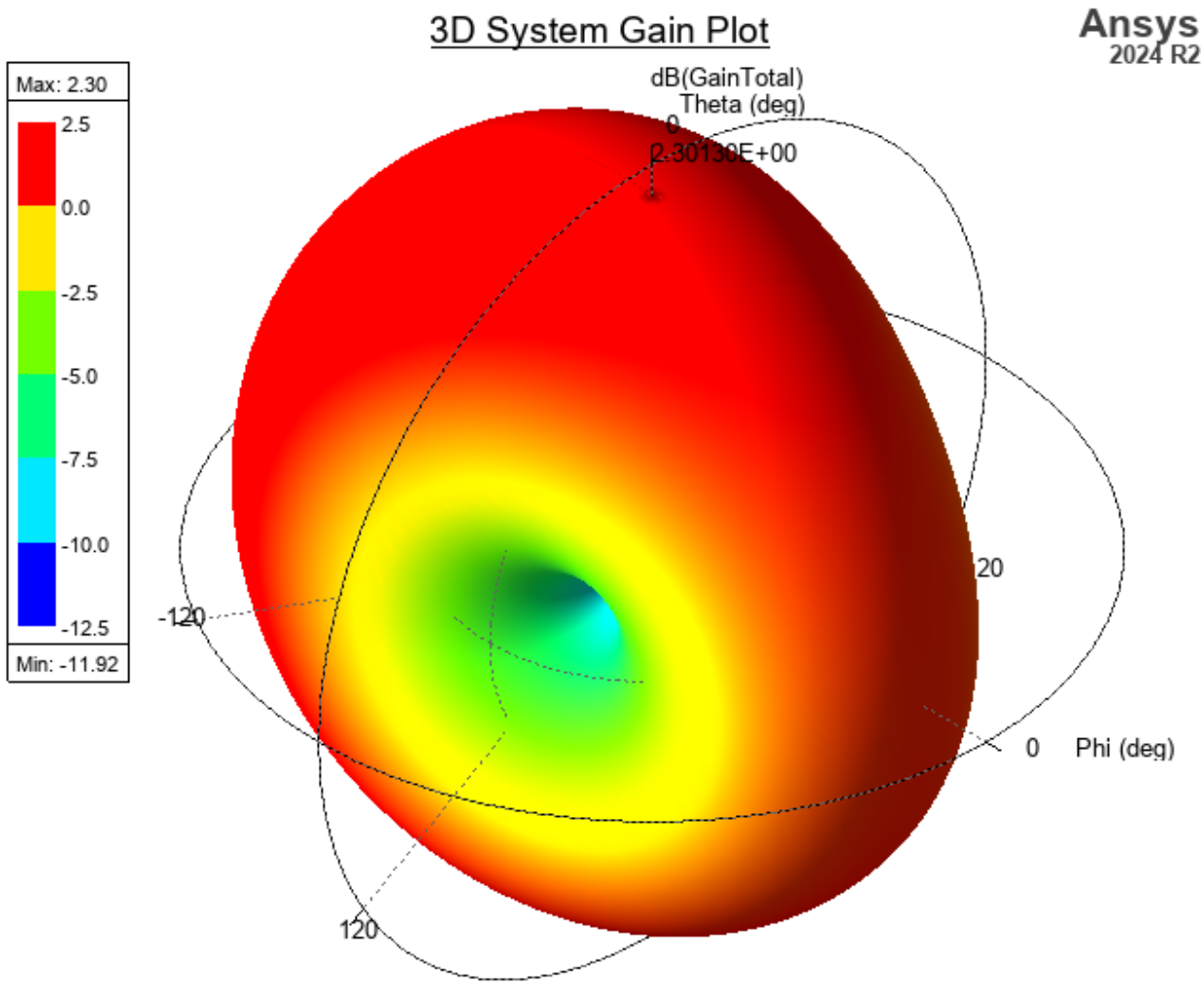


Figure 7: 3D System Gain Plot – Total Gain (dB) vs. Phi and Theta

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

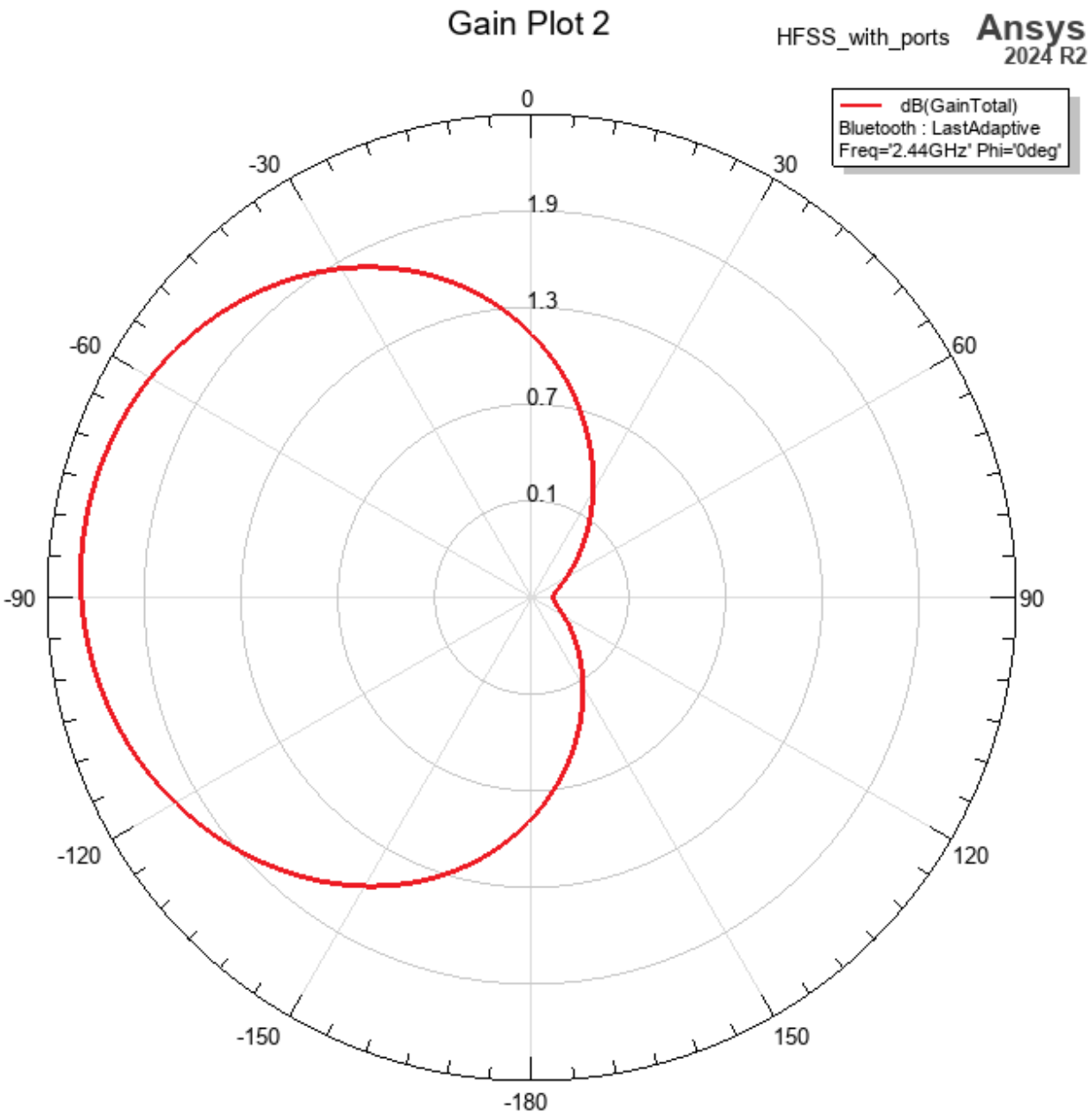


Figure 8: Gain Plot 2 – Total Gain (dB) vs. Theta, Phi = 0°

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

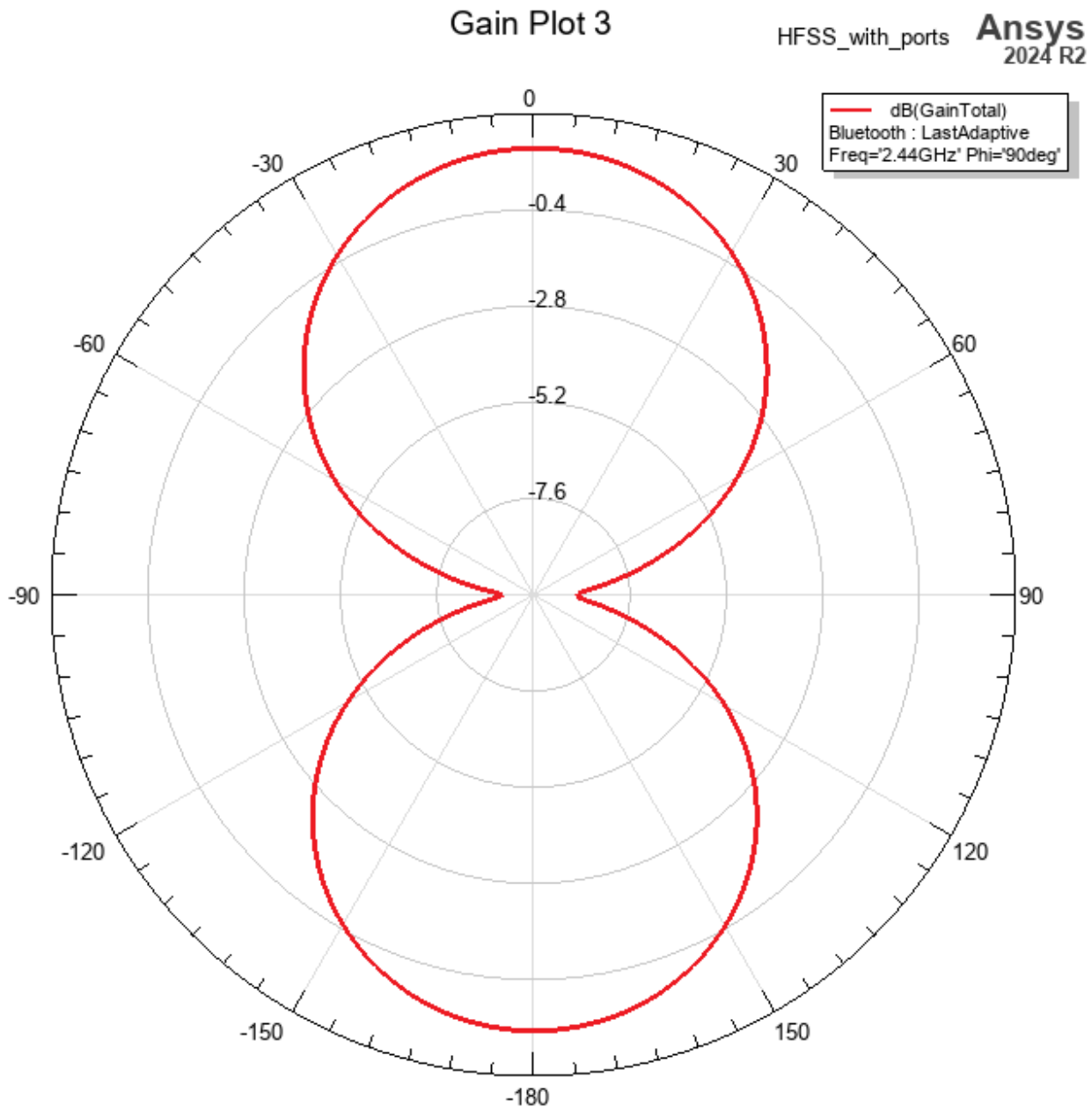


Figure 9: Gain Plot 3 – Total Gain (dB) vs. Theta, Phi = 90°

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

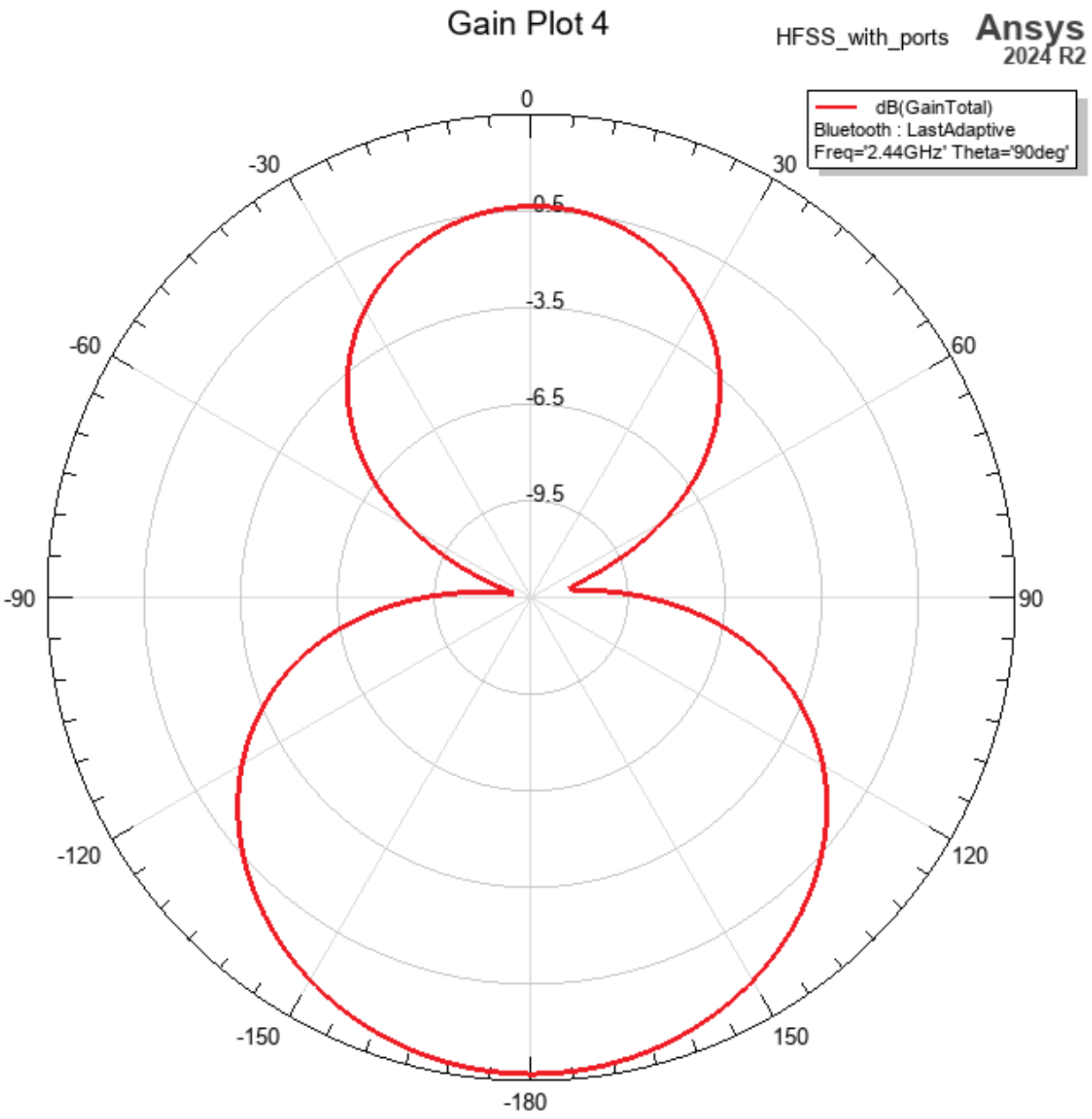


Figure 10: Gain Plot 4 – Total Gain (dB) vs. Phi, Theta = 90°

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

Overlaying Reports on the Model Geometry:

To overlay any of the gain plots on the model geometry, right-click in the **Modeler** window and choose **Plot Fields** > **Radiation Field** from the shortcut menu. In the *Overlay Radiation Field* dialog box that appears, select the checkbox in the **Visible** column for one or more of the available gain plots and click **Apply**. Adjust the **Transparency** or **Scale** as desired and click **Apply** again. Click **Close** when finished. Figure 11 below shows the first (3D) gain plot overlay:

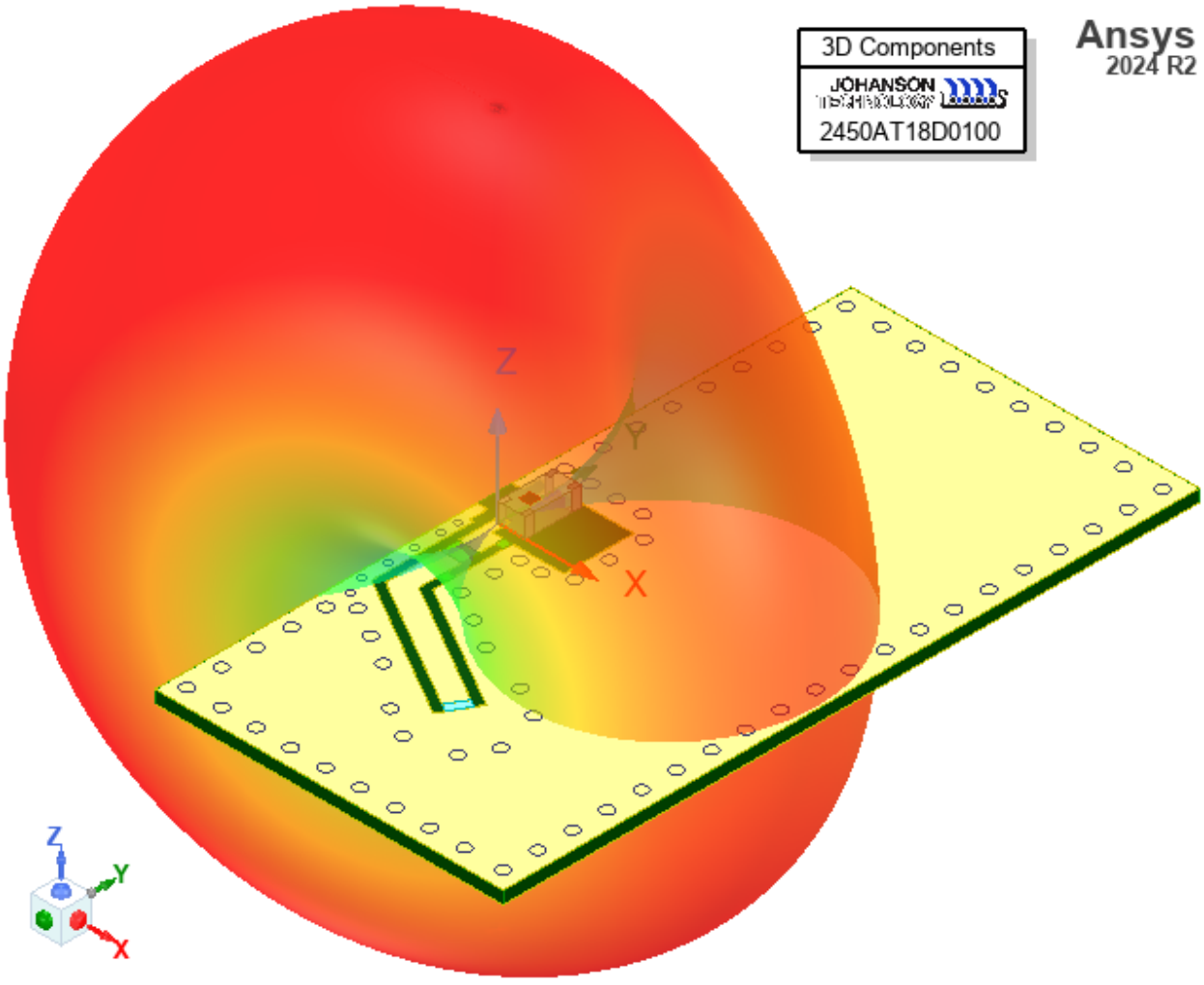
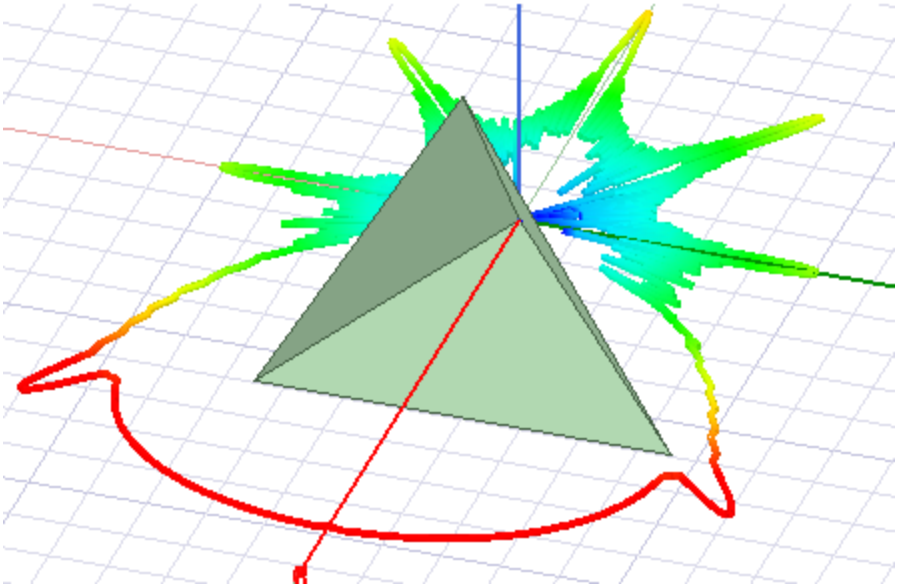


Figure 11: 3D System Gain Plot Overlaid on the Model Geometry

Monostatic Trihedral RCS Example Project

The tridehdral_rcs project is located in the Examples/HFSS/RCS folder. This project has an associated Getting Started Guide (see Monostatic RCS Using SBR+).

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



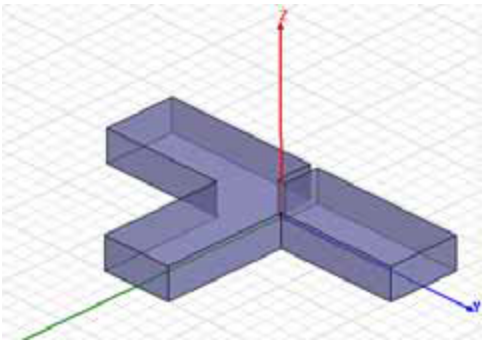
The project demonstrates:

- SBR+ Boundary
- SBR+ Wedge Settings
- Create 2D x-y plots of Monostatic RCS
- Create 3D Polar Plot and overlay it on the model

Tee and OptimTee Waveguide Projects

In the installation directory, the Examples/Electronics Desktop/HFSS/RF Microwave folder contains the versions of the waveguide t-junction modal solution project described in *Getting Started with HFSS: A Waveguide T-Junction*, and *Getting Started with HFSS: Optimizing a Waveguide T-Junction Using HFSS with Optimetrics*.

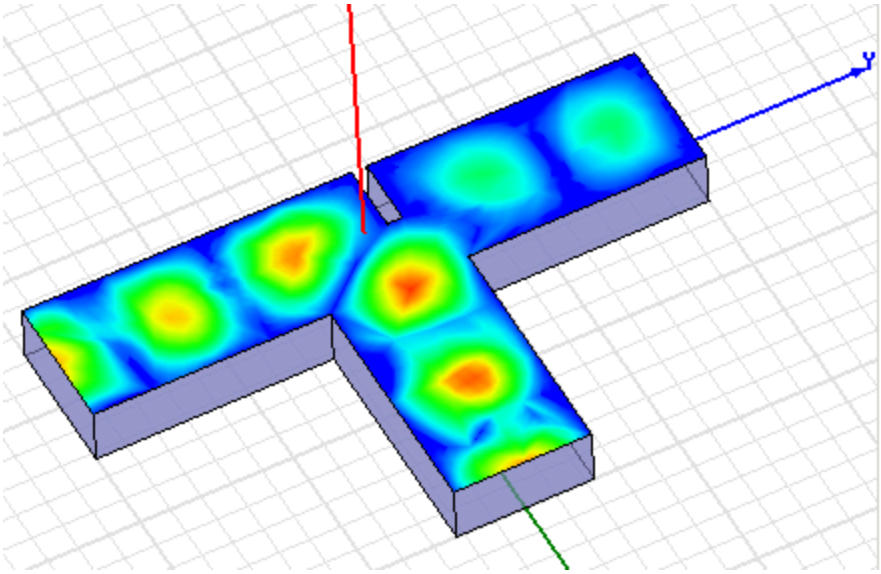
After you insert an HFSS design, you can access HFSS Getting Started Guides from **Help** > **Getting Started Guides**.



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

The waveguide T-junction illustrates the basic HFSS features, including:

- the Modeler
- parameterization of a design feature
- setup and analysis
- the use of the Reporter and field animation



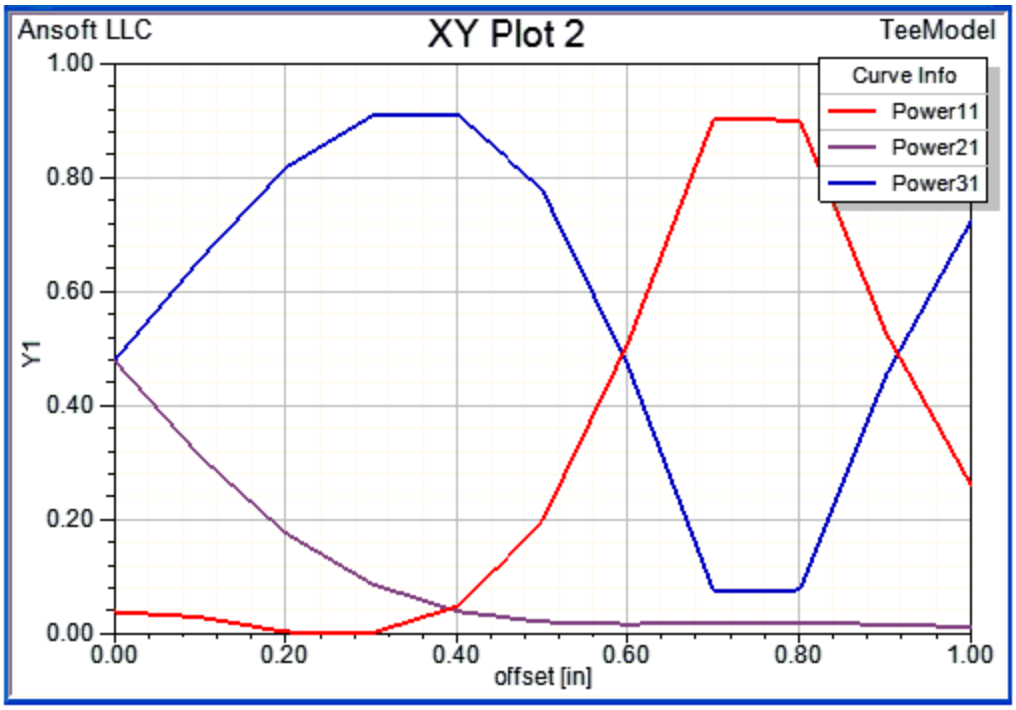
The animated *Mag_E1* plot of the E-field when the septum is located 0.2 inches closer to Port 2.

The second version of the wave guide t-junction demonstrates the use of the Optimetrics and provides an example of using the SpaceClaim link and parameterization.

- Parametric analysis
- Variable for optimization
- An optimization setup
- A cost function
- Optimization analysis
- plot of cost values versus solved iterations

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

- Use of output variables



See the following getting started guides in the help:

- Getting Started with HFSS: Waveguide T-Junction
- Getting Started with HFSS: Waveguide T-Junction Optimization

Waveguide Combiner

Abstract

This HFSS example model is a rectangular waveguide combiner operating at 20 GHz.

This example project has an associated getting started guide:

- *Getting Started with HFSS: 20 GHz Waveguide Combiner*

  Whereas the example model is fully set up, complete with predefined plots and overlays, and is ready to solve, the getting started guide offers an alternative learning path. Follow it to walk through the process of drawing the waveguide geometry, assigning the boundaries and excitations, setting up the analysis and frequency sweep, and postprocessing the results.

Solution Time: 4 cores in version 2025 R1 (meshing, adaptive solution, and frequency sweep)

Approximately 2 minutes, 255 MB max. RAM used

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

> **Note:**
>
> Simulation time and memory will vary depending on your available resources. In most cases, HPC (parallel resources) can be leveraged to significantly reduce simulation time and the memory footprint required by each CPU.

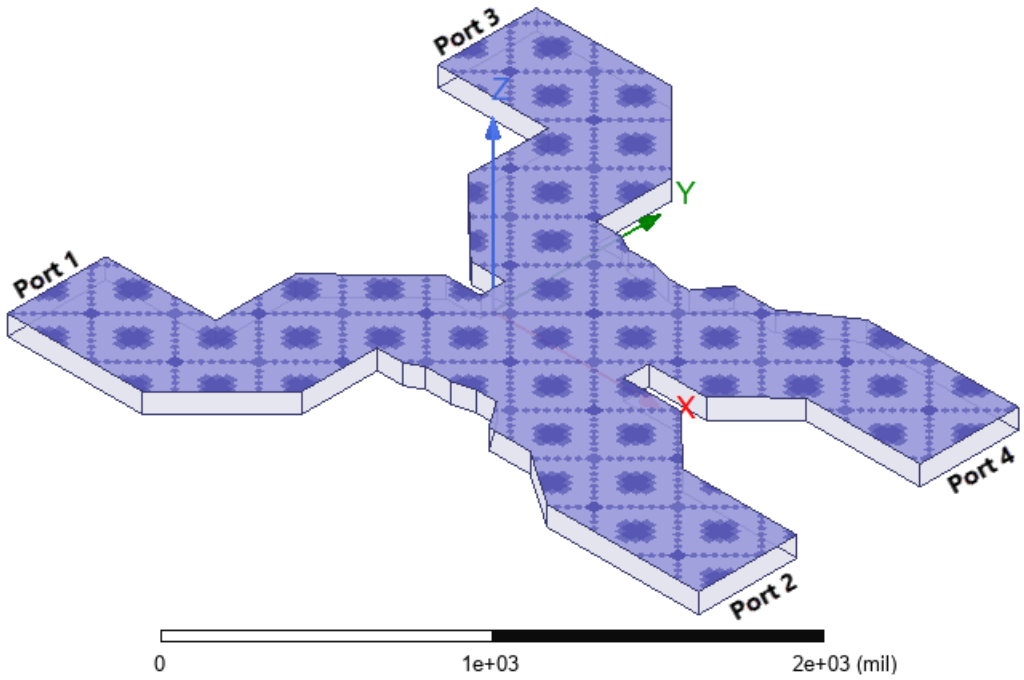
The WaveguideCombiner.aedt project file is located in the **...Examples\HFSS\RF Microwave** subfolder of the program installation path.

Setup Description:

The waveguide combiner example has the following characteristics:

- E-plane symmetry boundary in the XY plane, so the model is half the height of the waveguide combiner it represents
- Impedance multiplier set to a value of 2 to account for the symmetry condition
- Wave ports assigned at each of the four inputs/outputs
- Finite conductivity boundary based on the properties of the material, aluminum, assigned to the bottom and the remaining side surfaces
- Analysis setup with an adaptive frequency of 20GHz, delta S = 0.005, and using second order basis (suitable for waveguide structures)

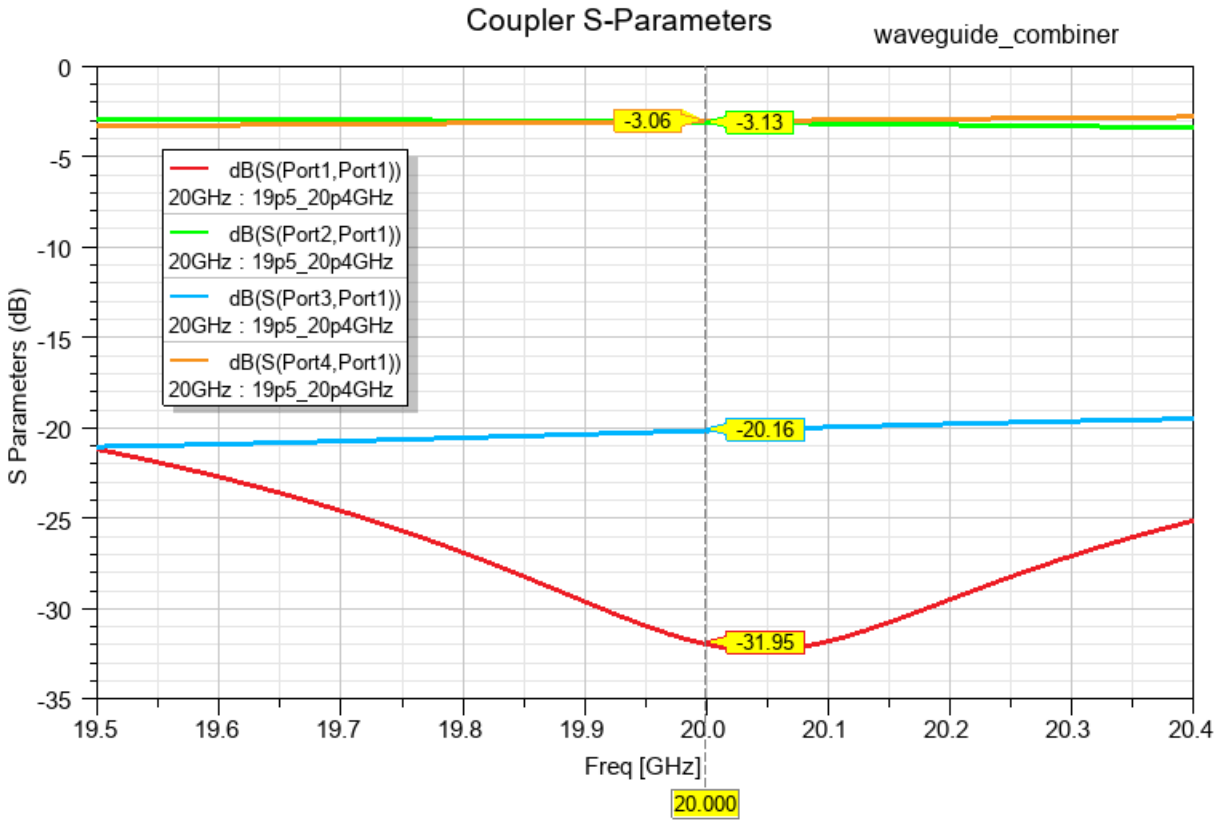
The following figure shows the waveguide combiner geometry with the symmetry boundary visualized:



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

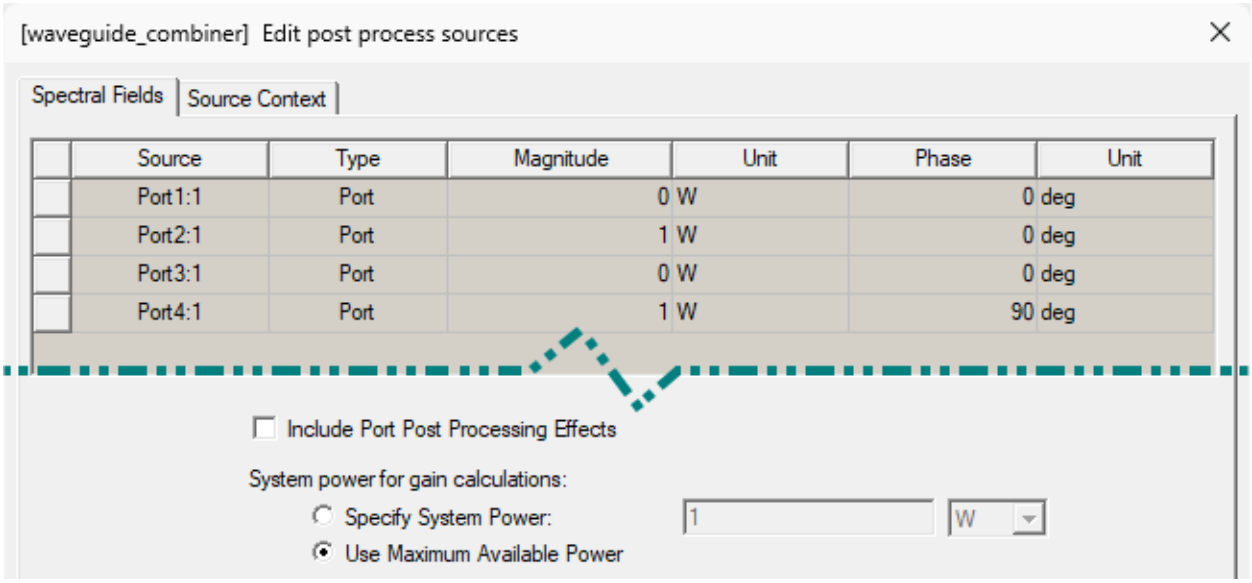
Postprocessing

After analysis, the S-parameters can be seen in the predefined report, **Coupler S-Parameters**. X markers have been added to point out values at the 20 GHz adaptive solution frequency:

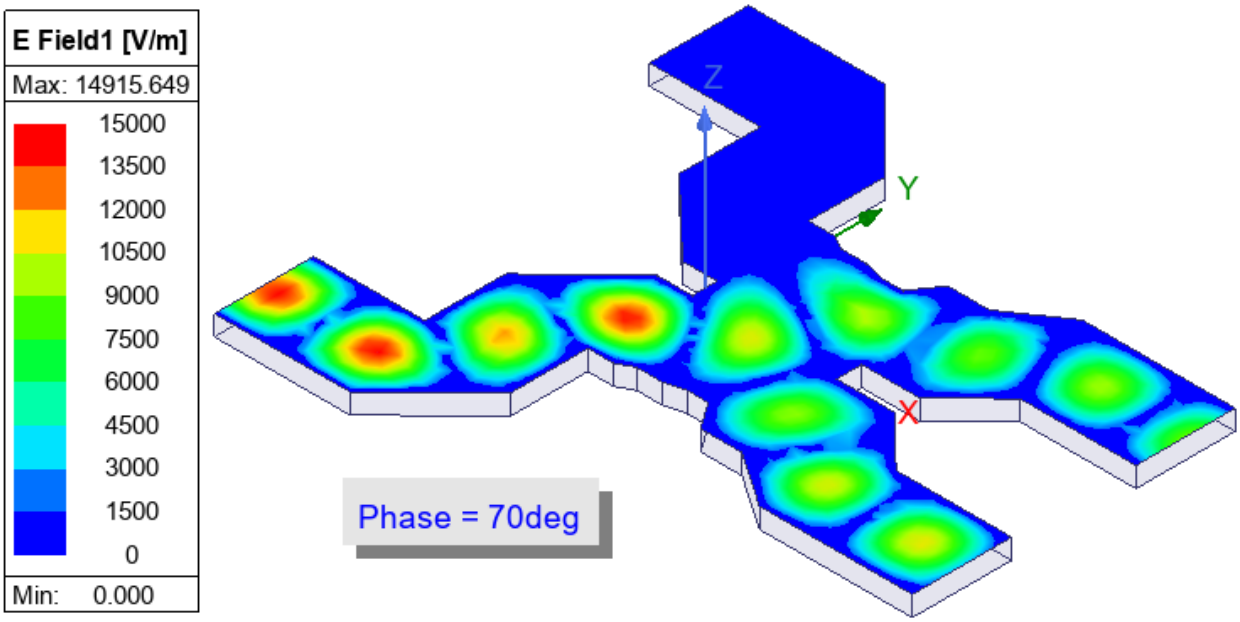


An E-Field overlay plot is also predefined to visualize the electric field distribution within the combiner. The field is controlled using the *Edit Sources* dialog box (to define the excitation magnitude and phase at each port). In this example, **Port 2** is excited at 1 W of power and 0° phase. **Port 4** is also excited at 1 W, but the phase angle is 90° (relative to the Port 2 phase). The remaining two ports are not excited (0 W):

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



With the sources defined as shown above, the power from the two 1 W excitations will be combined, and essentially all of the output will be directed to Port 1:



The *Phase* annotation that appears in the Modeler window indicates the instantaneous phase of the excitation at Port 1 for each frame of the animation.

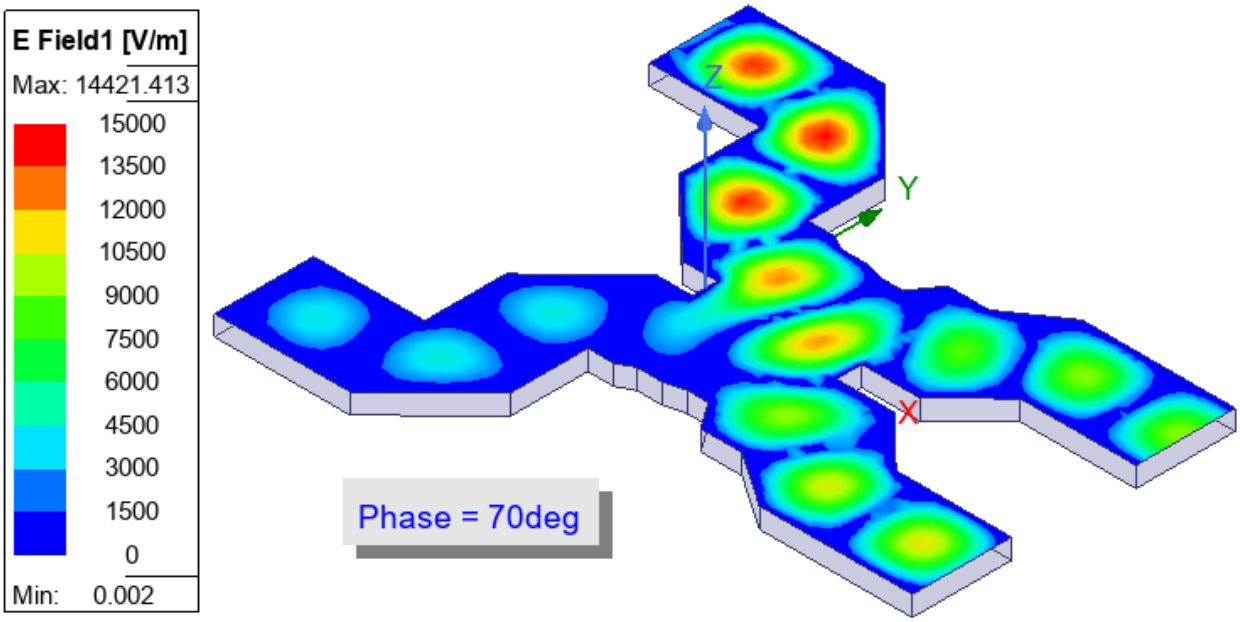
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optionally, change the Port 4 Phase value to various angles between -90° and 90°. You can **Apply** the port sources changes while the animation is running to immediately see the effect on the E-Field distribution (without closing the *Edit Port Sources* dialog box).

Observations:

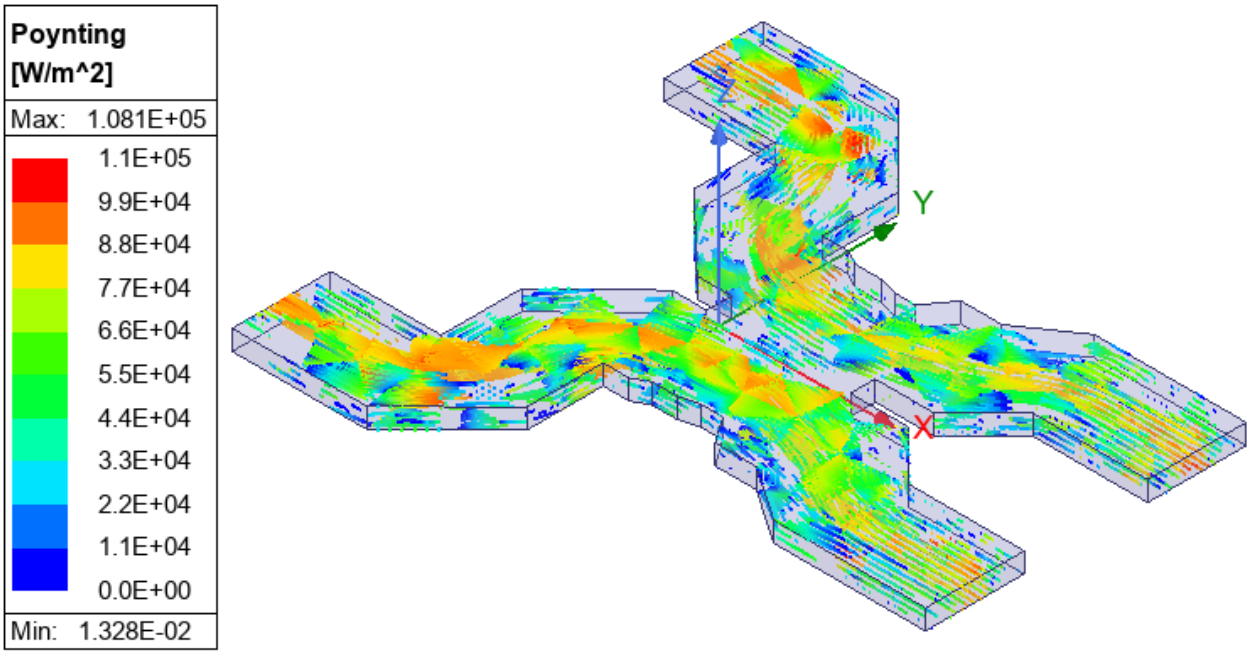
- With the Port 4 excitation at -90° phase, the input power is combined and essentially all of the output is directed to Port 3.
- At 0° phase, the outputs at Ports 1 and 3 are balanced.
- For Port 4 phase angles of -75° to -90° or 75° to 90°, the field strength will be very low at one of the output ports, and you can use a logarithmic scale to better see the weaker field color contours.

The following overlay animation shows the results with 1 W applied to both input ports and with a -60° phase angle at Port 4. In this case, the majority of the power is output at Port 3, with a lesser portion directed to Port 1:



Streamline plots for vector quantities, such as Poynting Vector, can also be displayed. An example of the Poynting Vector is predefined and shown below. Select **Poynting** > **RealPoynting1** under *Field Overlays* to visualize this result. Adjust the streamlines as desired by double-clicking the plot legend and adjusting the **Seeds density** in the *Plots* tab. This overlay shows the vectors with 1 W power and 0° phase at both input ports (2 and 4), in which case the output power is balanced between the two ports:

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



ADAS Examples (Advanced Driver Assistance Systems)

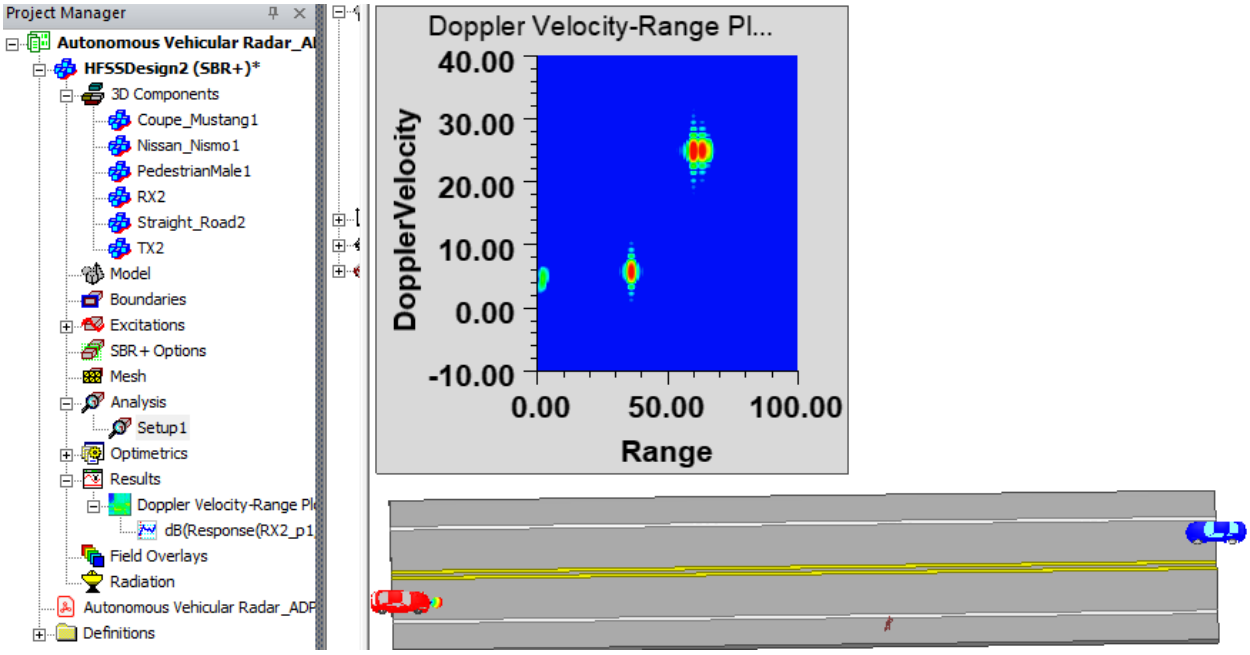
Currently, one of the ADAS examples is described in the help. Click the *Autonomous Vehicular Radar ADP (SBR+)* thumbnail (upper-left) for a description of this example:

For the remaining ADAS examples, see the PDF files that accompany the projects (located in the same folder).

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

Autonomous Vehicular Radar ADP (SBR+)

The example demonstrates a hybrid hierarchical workflow for simulating a full-scale traffic scene at 76.5 GHz by leveraging Ansys SBR+ (shooting and bouncing rays) solver. The project contains only one design “Full-scale traffic scene”.



This design “Full-scale traffic scene” consists of a complex real-world scenario and is solved using SBR+ in Ansys HFSS. This asymptotic technique leverages the equivalence theorem by approximating the equivalent electric and magnetic sources (Js and Ms, respectively) on the surface, instead of performing full wave simulation. It enables precise and efficient simulation for electrically large problems, with dimensions. The Traffic scenario has electrical dimension as large as 16,586λ at 76.5 GHz. The scene is populated with the ego vehicle, one car and a pedestrian. Vehicle body is modeled as a PEC (perfect electric conductor), while the road is modeled as an asphalt surface. The pedestrian is simulated using the average dry skin human model as reported in [1] (relative dielectric permittivity (εr) of 6.4 and conductivity of 38.38 S/m).

The design demonstrates the use of:

- HFSS SBR+
- Parametric Optimetrics

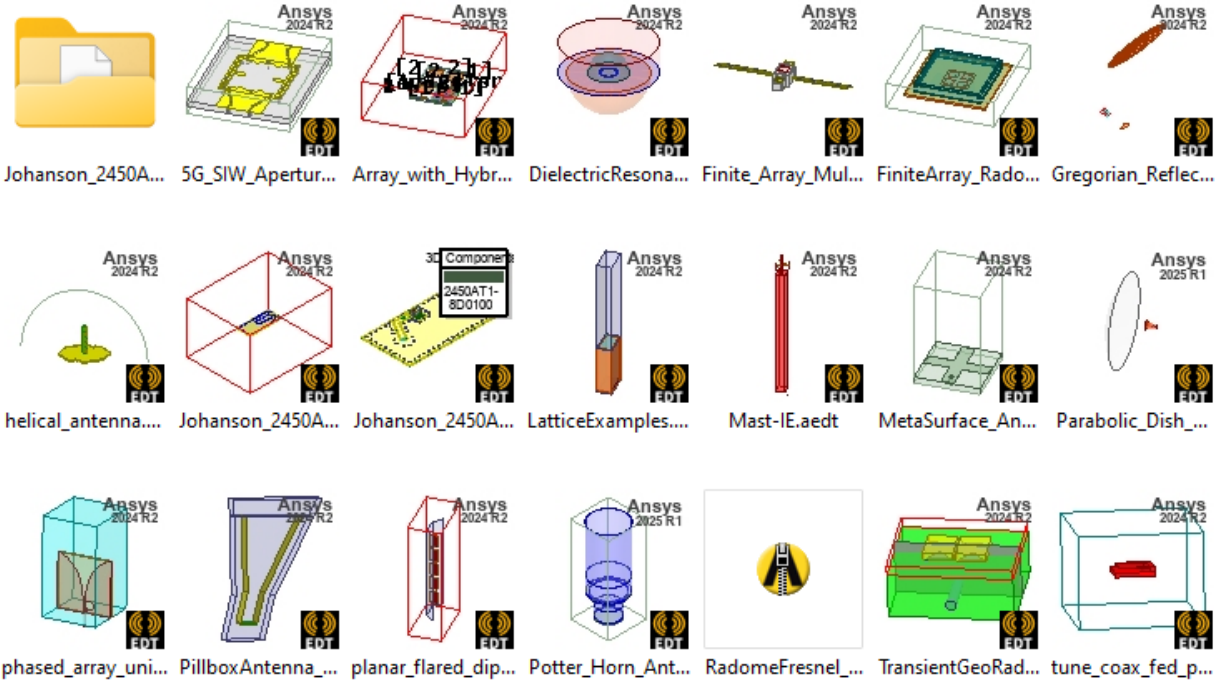
References:

[1] Castro, Juan D, et al. “Enabling Safe Autonomous Vehicles by Advanced Mm-Wave Radar Simulations.” Institute of Electrical and Electronics Engineers, IMS: 2019 IEEE MTT-S International Microwave Symposium: 2-7 June 2019, Boston, MA, USA, 2019.

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

Antenna Examples

Click graphics for descriptions of the following HFSS antenna examples and documentation of antenna creation tools:



Click for HFSS Application Modeling Guide: Antennas.

5G Antenna Using HFSS 3D Components

Introduction

Fifth-generation (5G) technology is the new standard for wireless communication, offering a host of advantages over 4G/4.5G. By shifting to millimeter-waves, 5G offers unprecedented data rates and latency but also presents engineers with significant challenges. Due to high propagation loss at these frequencies and requirements for integration with technology such as massive MIMO, careful design of the antennas in their target environment is required to meet all the 5G specifications. Simulation played a crucial role in achieving the 2020 deadline for 5G rollout. Full-wave electromagnetic simulators such as ANSYS HFSS enable 5G researchers and engineers to design and analyze millimeter-wave antennas with little cost and effort.

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

The obstacles to 5G implementation require interdisciplinary collaboration between each level of the supply chain. HFSS 3D Components offer a way to easily manage and share HFSS models encompassing all the geometries, materials, excitations, boundaries, and mesh operations. In turn, it is easy to see that 5G simulations are a natural application for HFSS 3D Components.

Description

Differentially Fed Circularly Polarized Planar Aperture Antenna

In this project, a 3D component of a planar antenna element [1] was created and inserted into a target design, in this case a differential feed structure. Then, the component was copied to make a total of four instances of it. This antenna produces circularly polarized light with wide bandwidth and has a peak gain of around 13 dB at 60 GHz, making it potentially suitable for 5G.

The goal of this example is to highlight a few best practices for inserting 3D components into target designs in the context of 5G antenna simulation. The project contains three designs, as detailed below in the Model Construction section.

Model construction:

Open the project 5G_SIW_Aperture_Antenna. You will see three HFSS designs, as follows:

- **0_5G Aperture Element:** This design contains the geometry used to create a 3D component of a single antenna aperture element.
- **1_SIW Feed Structure:** This design is a differential feed based on the substrate integrated waveguide (SIW). It is the structure into which four instances of the aperture element must be inserted to complete the 5G antenna model, which has been done in the third design.
- **2_SIW Feed Structure + 3Dcomp antenna:** This design contains the differential feed structure plus four instances of the 5G aperture component. To create it, the feed structure from the second design was copied into this design. Next, a single 3D component of the 5G aperture was inserted. Finally, the three additional apertures were created by copying the first one.

If all four antennae are not visible in the third design, click **View** > **Visibility** > **Show All** to make all objects visible.

The detailed construction sequence was as follows:

A new HFSS design was inserted into the project and named **5G Aperture Element**. Then, a box was drawn using the following command properties:

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

Properties: 5G_Planar_Aperture_Antenna - 5G Aperture Element - Modeler

Command

	Name	Value	Unit	Evaluated Value
	Command	CreateBox		
	Coordinate Sys...	Global		
	Position	0 ,0 ,0	mm	0mm , 0mm , 0mm
	XSize	Tx		2.9mm
	YSize	Ty		2.5mm
	ZSize	hsub		0.787mm

The values of **Tx**, **Ty**, and **hsub** were defined as **2.9 mm**, **2.5 mm**, and **0.787 mm**, respectively, and the name was changed from *Box1* to **Substrate**.

Next, an offset relative coordinate system was created and named **FeedGroundCS**, as shown below:

Properties: 5G_Planar_Aperture_Antenna - 5G Aperture Element - Modeler

Coord System

	Name	Value	Unit	Evaluated Value	Description
	Type	Relative			
	Name	FeedGroundCS			
	Reference CS	Global			
	Mode	Axis/Position			
	Origin	0mm ,Ty/2 ,0mm		0mm , 1.25mm ,...	
	X Axis	1 ,0 ,0	mm	1mm , 0mm , 0mm	
	Y Point	0 ,1 ,0	mm	0mm , 1mm , 0mm	

With **FeedGroundCS** set as the working coordinate system, another offset relative coordinate system was created and named **FeedCS**, as shown below:

Properties: 5G_Planar_Aperture_Antenna - 5G Aperture Element - Modeler

Coord System

	Name	Value	Unit	Evaluated Value	Description
	Type	Relative			
	Name	FeedCS			
	Reference CS	FeedGroundCS			
	Mode	Axis/Position			
	Origin	0mm ,0mm ,hsub		0mm , 0mm , 0....	
	X Axis	1 ,0 ,0	mm	1mm , 0mm , 0mm	
	Y Point	0 ,1 ,0	mm	0mm , 1mm , 0mm	

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

FeedCS was kept as the working coordinate system. From here, the metallic traces defining the antenna aperture were created. Eight rectangles with the name **Trace*n*** (n = 1, 2, … 8) were drawn using the command properties shown below. The parameters **tw1**, **tw2**, ... **tw8** were all set to equal **0.32 mm**, **gap** equal to **0.18 mm**, **ax** to **1.8 mm**, and **ay** to **1.7 mm**. Each rectangle was made yellow with a transparency value of **0**.

Properties: 5G_Planar_Aperture_Antenna - 5G Aperture Element - Modeler

Command

	Name	Value	Unit	Evaluated Value	D
	Command	CreateRectangle			
	Coordinate Sys...	FeedCS			
	Position	0mm ,-tw1/2 ,0mm		0mm , -0.16mm , 0mm	
	Axis	Z			
	XSize	(Tx-ax)/2		0.55mm	
	YSize	tw1		0.32mm	

Properties: 5G_SIW_Aperture_Antenna - 0_5G Aperture Element - Modeler

Command

	Name	Value	Unit	Evaluated Value	Description
	Command	CreateRectangle			
	Coordinate Sys...	FeedCS			
	Position	(Tx-ax)/2 ,tw1/2 ,0		0.55mm , 0.16m...	
	Axis	Z			
	XSize	tw2		0.32mm	
	YSize	-(ay+tw1)/2		-1.01mm	

Properties: 5G_SIW_Aperture_Antenna - 0_5G Aperture Element - Modeler

Command

	Name	Value	Unit	Evaluated Value	Description
	Command	CreateRectangle			
	Coordinate Sys...	FeedCS			
	Position	(Tx-ax)/2+tw2 ,-ay/2 ,0		0.87mm , -0.85...	
	Axis	Z			
	XSize	ax-tw2		1.48mm	
	YSize	tw3		0.32mm	

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

Properties: 5G_SIW_Aperture_Antenna - 0_5G Aperture Element - Modeler

Command

	Name	Value	Unit	Evaluated Value	Description
	Command	CreateRectangle			
	Coordinate Sys...	FeedCS			
	Position	(Tx-ax)/2+tw2+ax-tw2-tw4 ,-ay/2+tw3 ,0		2.03mm , -0.53...	
	Axis	Z			
	XSize	tw4		0.32mm	
	YSize	ay/2-tw3-gap-tw5/2		0.19mm	

Properties: 5G_SIW_Aperture_Antenna - 0_5G Aperture Element - Modeler

Command

	Name	Value	Unit	Evaluated Value	Description
	Command	CreateRectangle			
	Coordinate Sys...	FeedCS			
	Position	Tx ,-tw5/2 ,0mm		2.9mm , -0.16m...	
	Axis	Z			
	XSize	-(Tx-ax)/2		-0.55mm	
	YSize	tw5		0.32mm	

Properties: 5G_SIW_Aperture_Antenna - 0_5G Aperture Element - Modeler

Command

	Name	Value	Unit	Evaluated Value	Description
	Command	CreateRectangle			
	Coordinate Sys...	FeedCS			
	Position	Tx-(Tx-ax)/2 ,-tw5/2 ,0mm		2.35mm , -0.16...	
	Axis	Z			
	XSize	-tw6		-0.32mm	
	YSize	(ay+tw5)/2		1.01mm	

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

Properties: 5G_SIW_Aperture_Antenna - 0_5G Aperture Element - Modeler

Command

	Name	Value	Unit	Evaluated Value	Description
	Command	CreateRectangle			
	Coordinate Sys...	FeedCS			
	Position	(Tx+ax)/2+tw6 ,(ay)/2 ,0mm		2.03mm , 0.85m...	
	Axis	Z			
	XSize	-ax+tw6		-1.48mm	
	YSize	-tw7		-0.32mm	

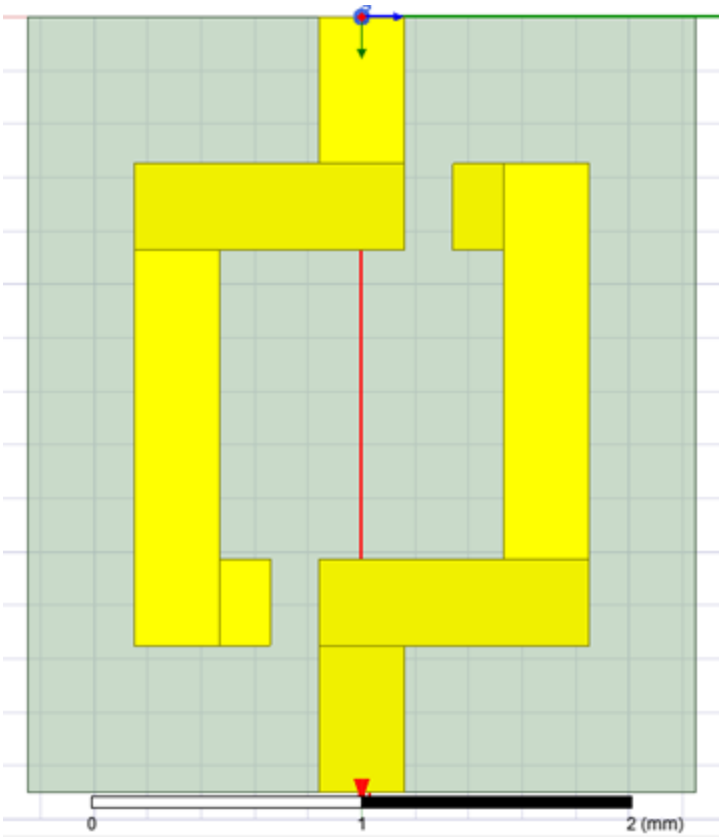
Properties: 5G_SIW_Aperture_Antenna - 0_5G Aperture Element - Modeler

Command

	Name	Value	Unit	Evaluated Value	Description
	Command	CreateRectangle			
	Coordinate Sys...	FeedCS			
	Position	(Tx-ax)/2 ,ay/2+tw7 ,0		0.55mm , 0.53m...	
	Axis	Z			
	XSize	tw8		0.32mm	
	YSize	-(ay/2+tw7+tw1/2-gap)		-0.19mm	

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

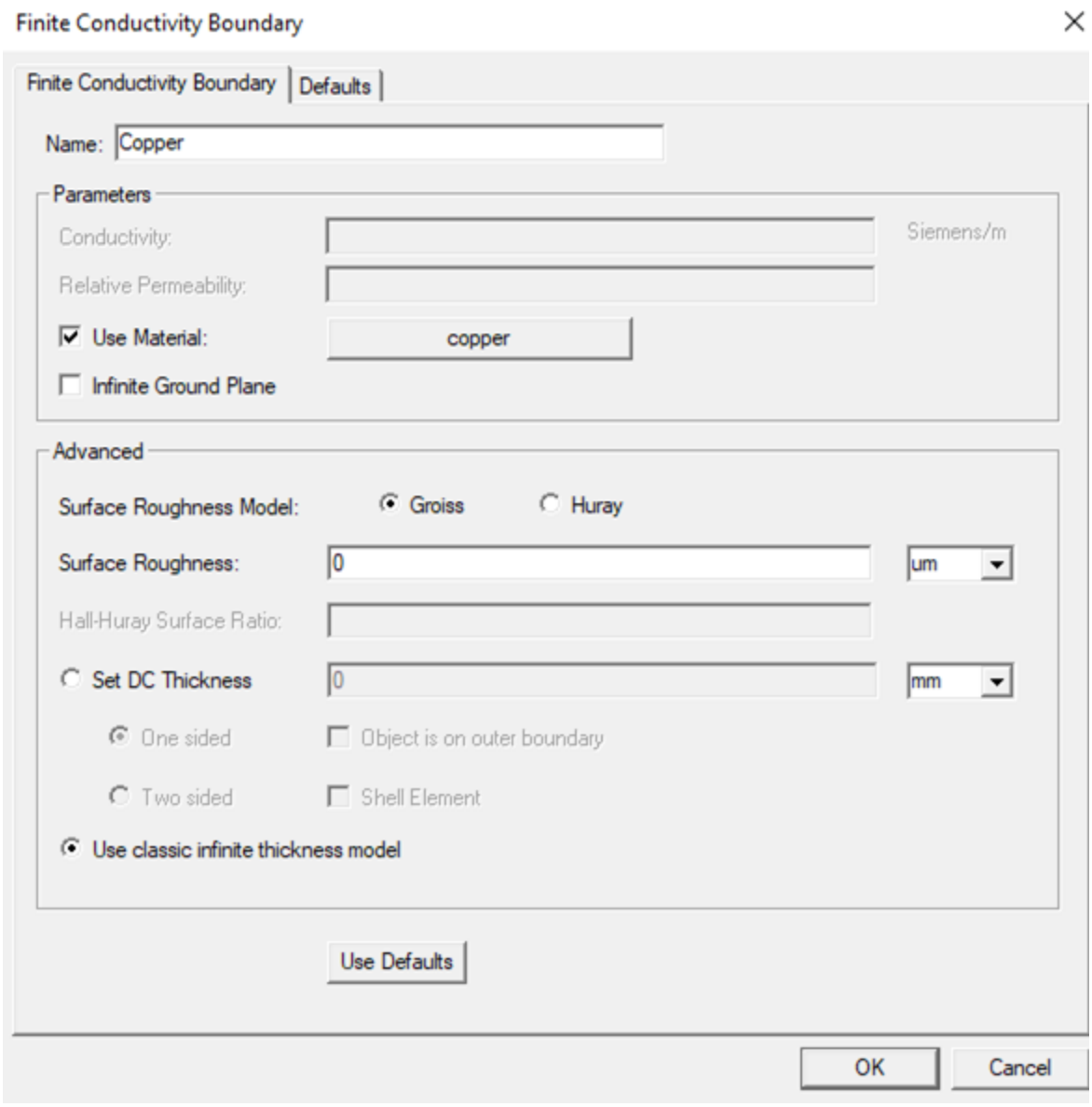
The resulting aperture element geometry should resemble the following figure:



Note that defining the geometry in this way allows us to individually modify each straight portion of the traces by simply changing the corresponding parameter values. Try modifying these parameters slightly and observe the changes in the geometry.

Next, **all** the rectangles just defined, starting with **Trace1**, were selected and united using the **Unite** command. With the traces still selected, a **Finite Conductivity** boundary was assigned to **Trace1** using the material **copper**, as shown below:

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



Next, the object, **Substrate**, was selected and assigned the material, **Rogers RT/duroid 5880 (tm)**. The geometry, material, and boundary settings are now complete. In this case, the created 3D component does not have its own excitations or mesh operations, as those are assigned in the target design.

At completion of the aperture geometry, the project was saved.

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

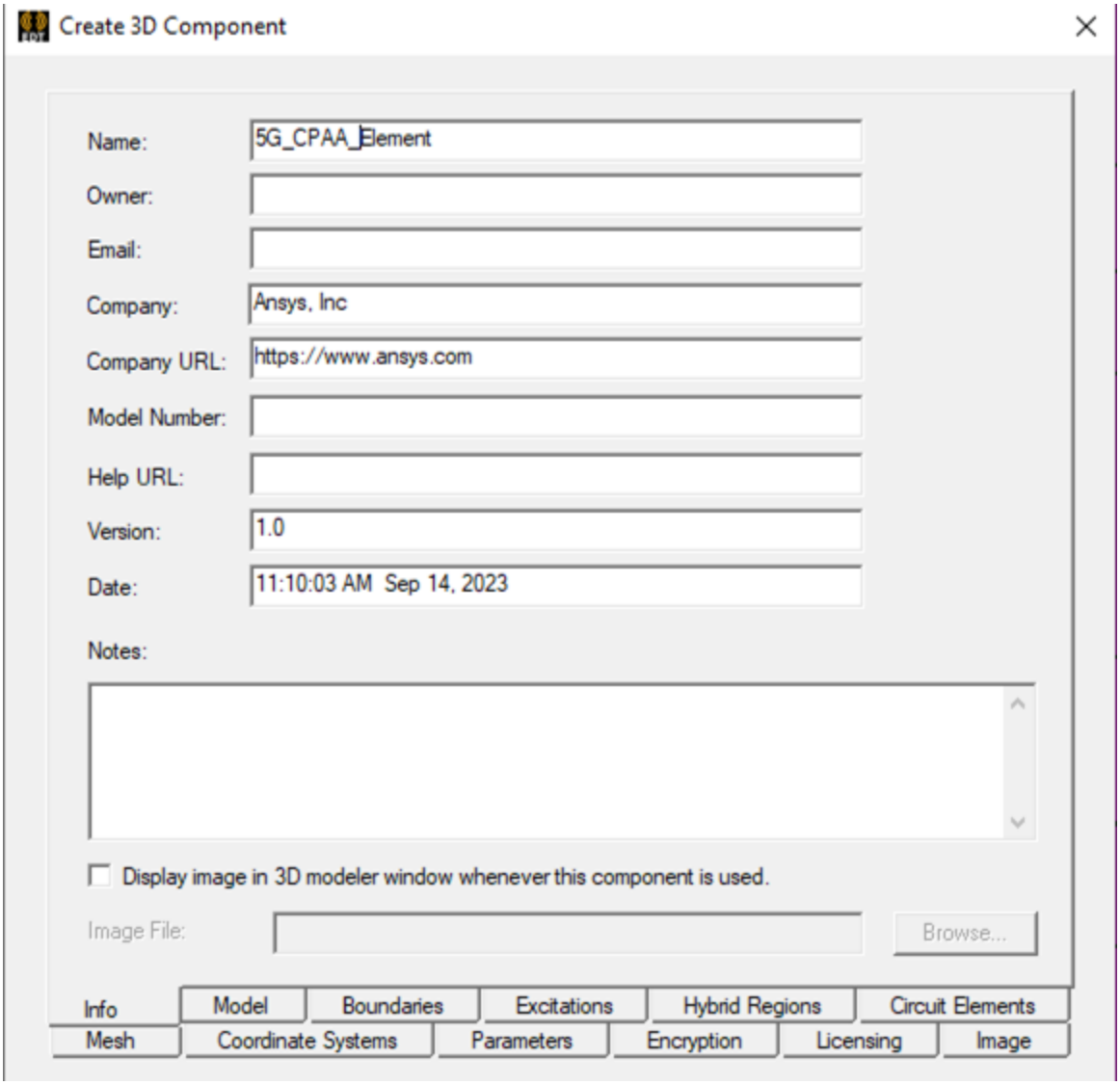
Creation of the 3D Component:

An important part of this procedure is to carefully consider the reference coordinate system to choose for the 3D component. A good choice can simplify the process of inserting the component into the target design, while a poor choice can complicate it. In the geometry that was just created, any of the coordinate systems that were defined would likely work. However, if you look in the target design you can see that there is already a coordinate system (named *FeedGroundCS*), which corresponds exactly to where we wish to place the ground of our first antenna element. Therefore, making *FeedGroundCS* the reference coordinate system for the 3D component becomes a convenient choice. Note that the chosen coordinate systems in the 3D component and the target design do not need to have the same name, but they can, as in this example.

To begin creating the 3D component, **FeedGroundCS** was made the working coordinate system in the *5G Aperture Element* design. Then, all model objects were selected (Ctrl + A).

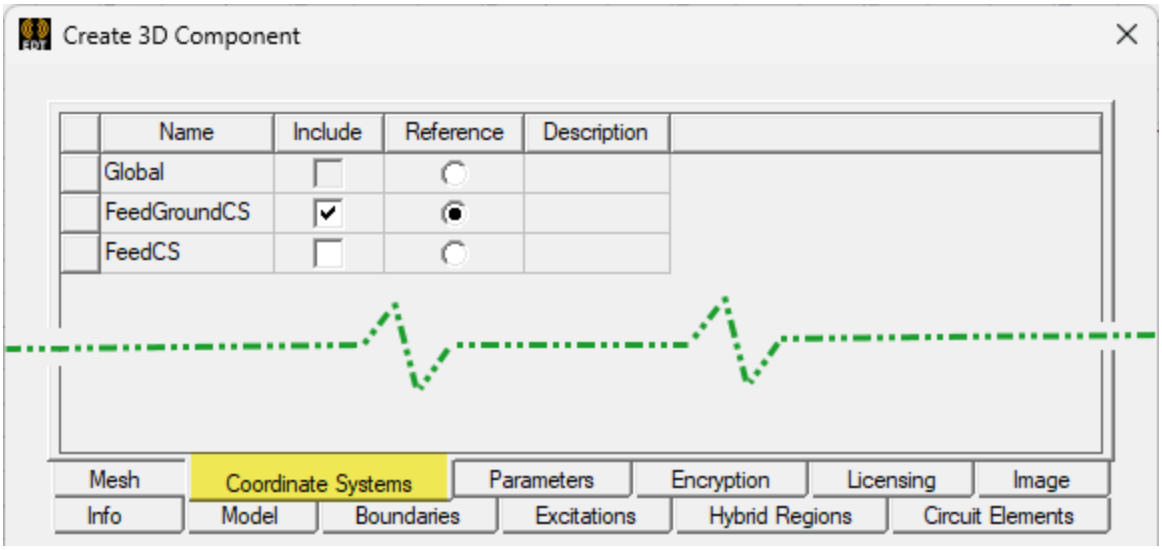
In the menu bar, **Draw** > **3D Component Library** > **Create 3D Component**… was used to create the component. This command opens a dialog box in which you can edit the attributes of the 3D component. Details in the **Info** tab can be specified as desired, and the name was set as **5G_CPAA_Element**, as shown below:

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



Next, in the **Coordinate Systems** tab, verify that the desired coordinate system is **Included** in the component and is selected as the **Reference** coordinate system. This has been done for **FeedGroundCS**, as shown in the following image:

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



Finally, in the **Parameters** tab, all listed parameters were selected. For your designs, you may wish to only include a few parameters, or none, depending upon what you wish to do with the component once it is inserted into a target design.

At this point the definition of the aperture component was complete. **OK** was clicked to create it, and to save it to a specified folder. The created component exists as an embedded item in the *5G_SIW_Aperture_Antenna* project. However, the 3D component file, *5G_CPAA_Element.a3dcomp*, is not included with the Ansys Electromagnetics Suite installation. If you wish to create a 3D component of the geometry in the first design (*0_5G Aperture Element*) and save it to your Ansoft\PersonalLib folder, duplicate the procedure described above.

> **Note:**
>
> This example model was created prior to more restrictive component naming rules being put into effect. The name may now include only letters, numeric digits, and the underscore character. Additionally, it can no longer begin with a number; the first character must be a letter or an underscore. Therefore, you must specify an alternative, compliant name, such as *CPAA_Element_5G*.

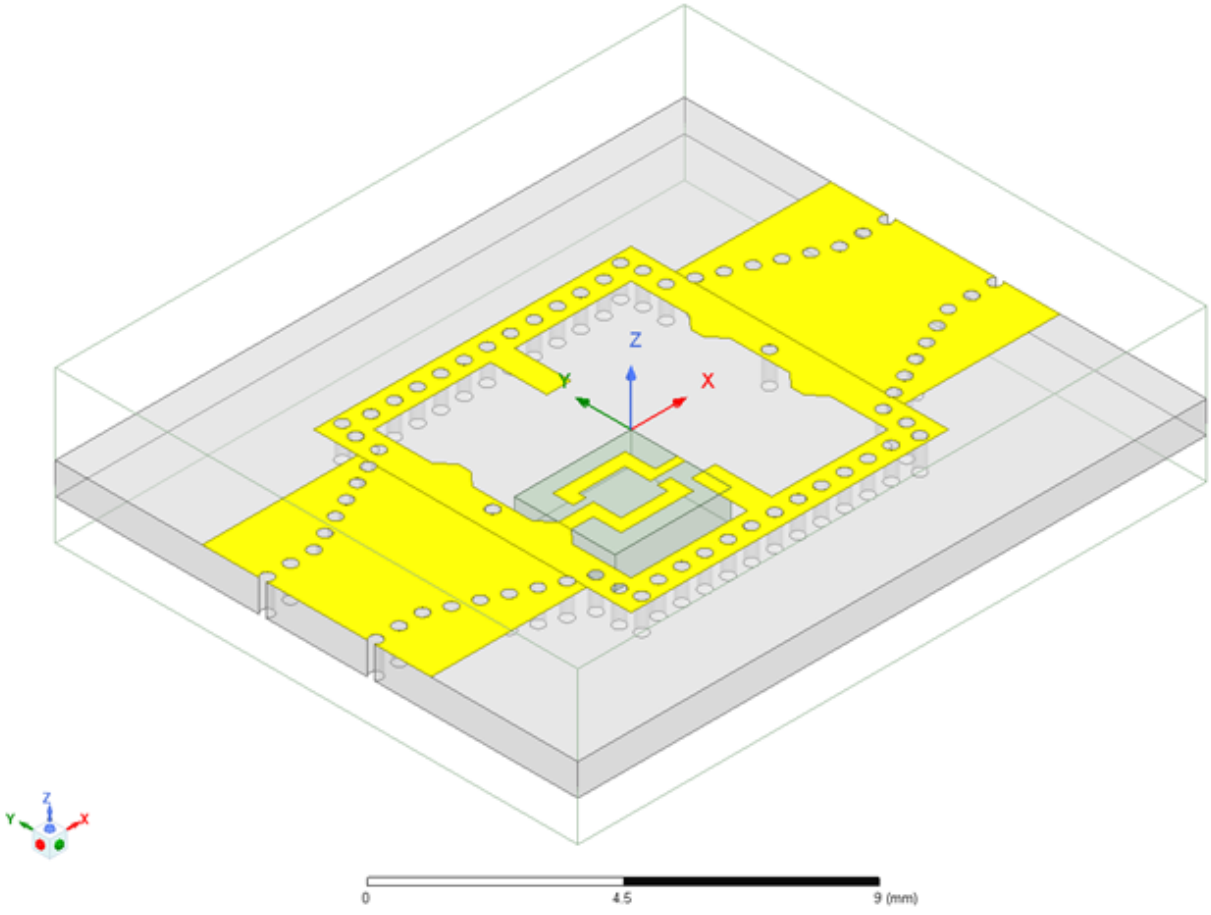
The target design for the 3D component is **2_SIW Feed Structure + 3Dcomp antenna**. This design was double-clicked in the Project Manager to make it the active design. Next, **FeedGroundCS** was made the working coordinate system in the target design. Then, the first instance of the component was inserted.

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

Tip:

One way to insert a 3D component is to right-click **3D Components** in the Project Manager and choose **Browse 3D Component** from the shortcut menu. In the resulting dialog box, navigate to the appropriate folder, select the desired component, and click **Open**. You can also drag and drop system components from the *Components Library* window into the **Modeler** window.

The appearance of the model after inserting the first instance of the 3D component is as shown in the following figure, with the antenna element automatically positioned exactly as desired:

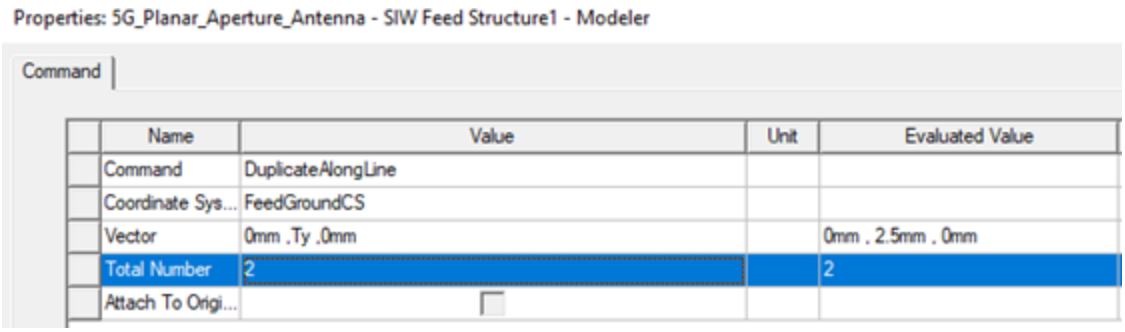


Also note that the 3D component can be seen in the History Tree. Double click '**5G_CPAA_Element1**' in the History Tree and go to the **Parameters** tab. You will see that the value of '**tw5**' has been overridden to '**t2**' (a separately defined variable with a value of 0.2 mm, which you can see listed in the design properties). You can temporarily change the dimension back to the

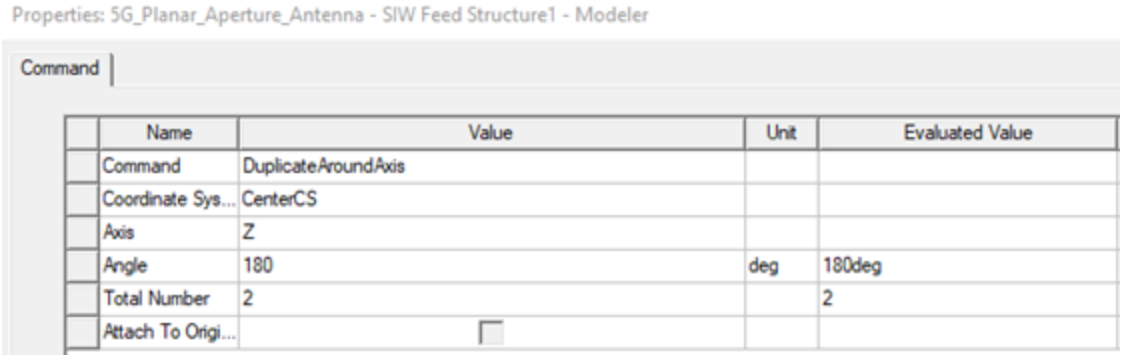
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

original value of **0.32 mm** and see that the corresponding trace width changes. Restore '**t2**' as the '**tw5**' value, since the solution will be based on this overridden parameter value.

Next, the three remaining antenna elements had to be added to the feed structure design. Instead of browsing to each component individually and inserting them into the design one at a time, it was easier to apply duplication commands to the component already in the model. "**5G_CPAA_Element1**" was selected in the History Tree and the **Duplicate Along Line** command used with the following parameters:

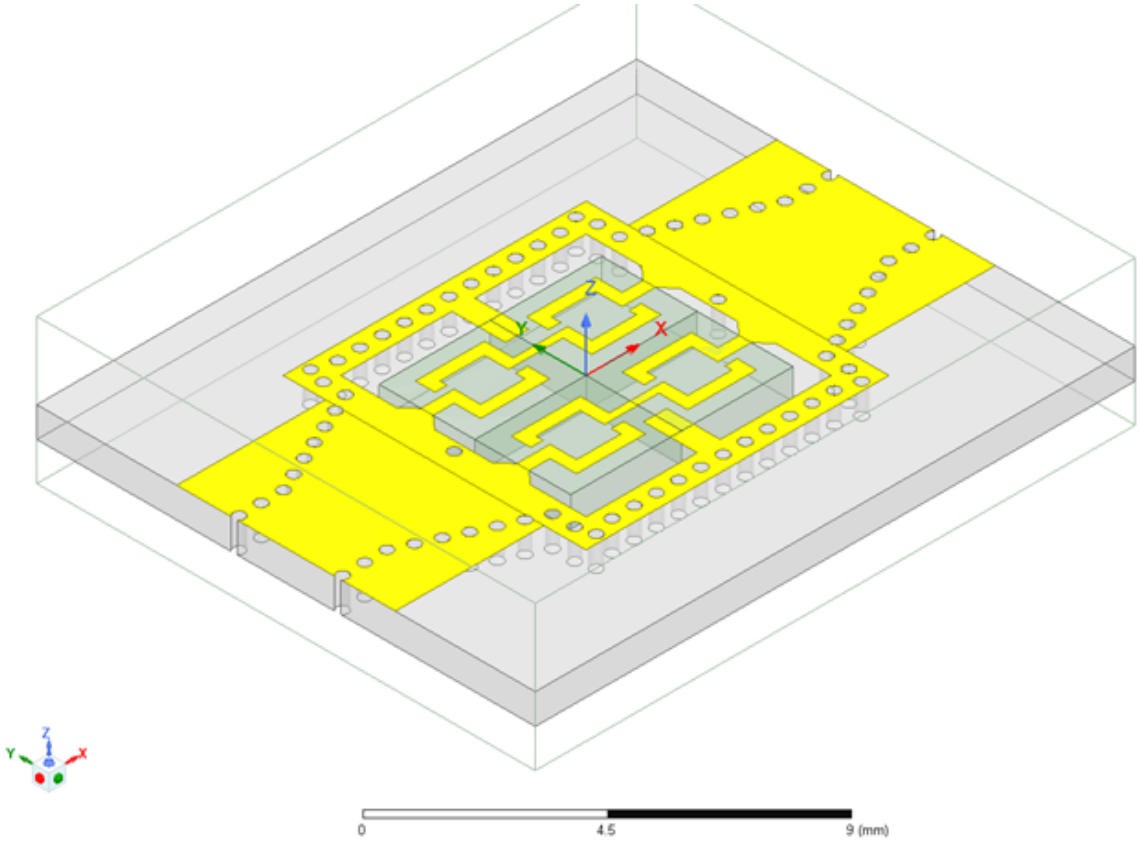


The active coordinate system was then changed to **CenterCS**, another previously defined coordinate system. **5G_CPAA_Element1** and **5G_CPAA_Element2** were selected and the **Duplicate Around Axis** command executed using the parameters shown below:



You can see that the origin and Z axis of *CenterCS* is perfectly defined to copy the first two elements to a position 180 degrees about the Z axis to create the third and fourth elements. This command completes the geometry of the antenna, and the appearance of the model is as shown below:

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

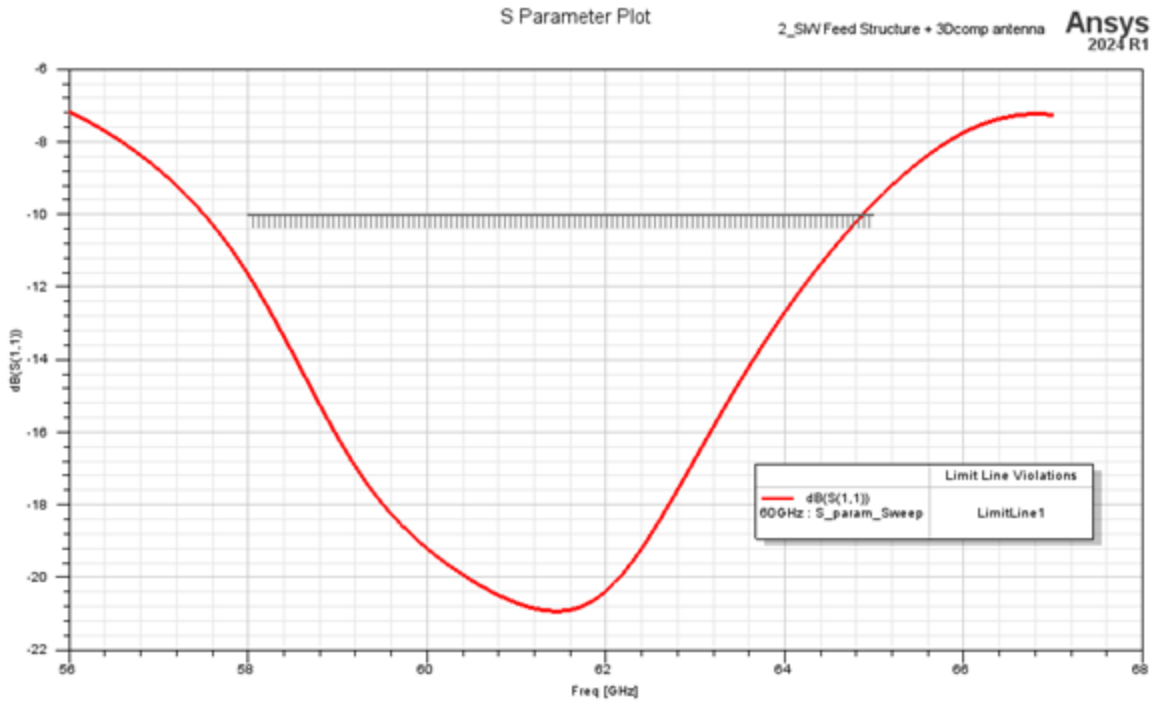


Since all the necessary boundaries and excitations are already defined in the target design, the model is ready to analyze. Click **HFSS** > **Analyze All** to run the adaptive solution and frequency sweep.

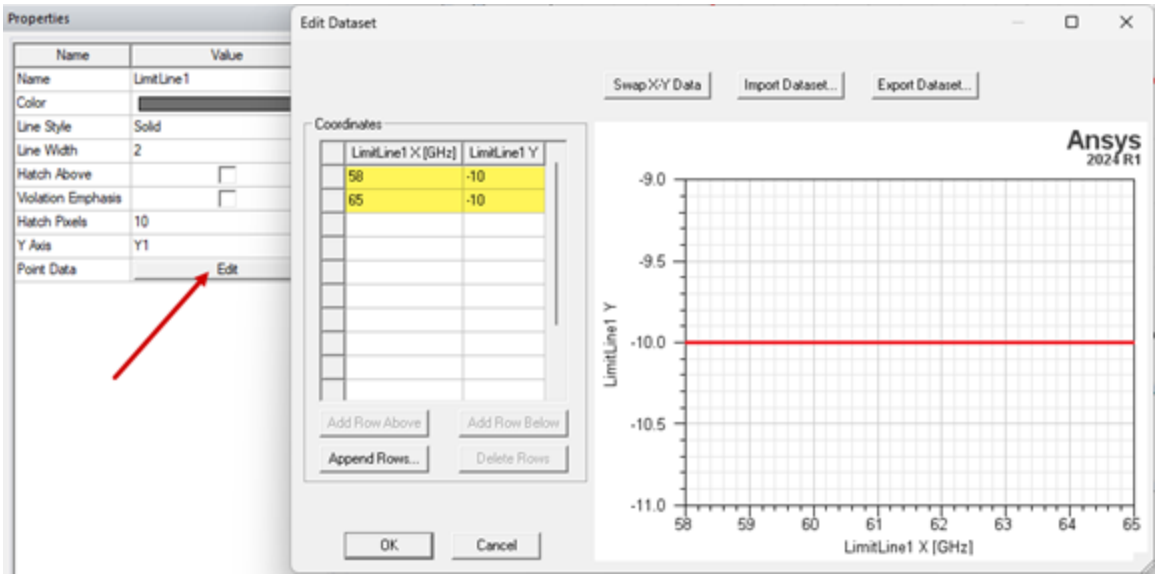
Postprocessing

After analyzing, verify that the results match the following plots.

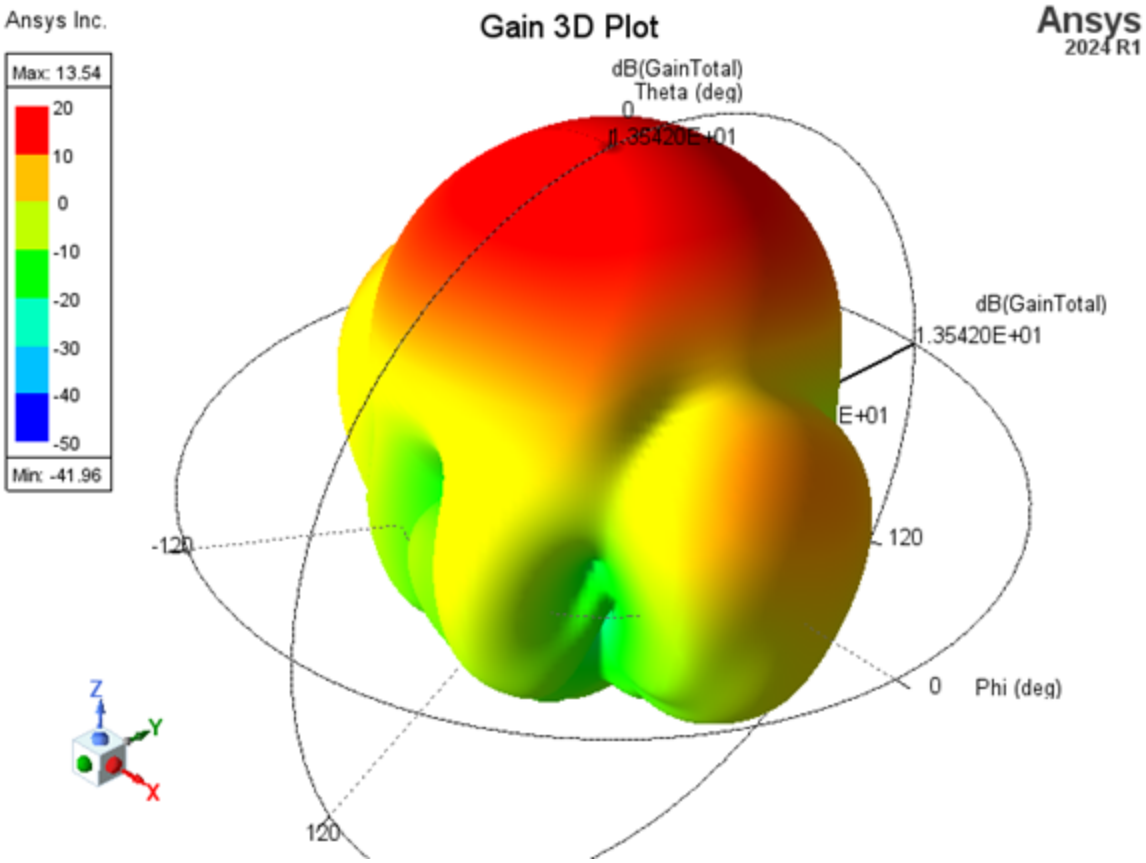
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



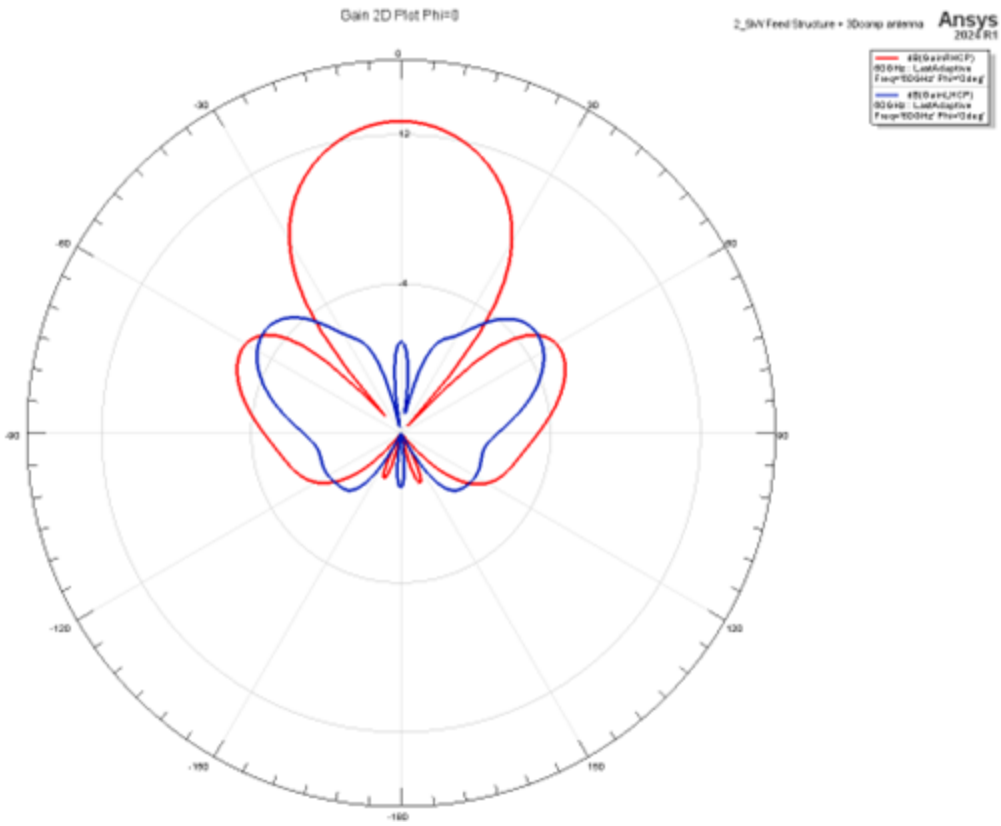
For the preceding plot, a limit line has been defined using the following settings. With the limit line in the plot window selected, click the Edit button in the docked Properties window to see the definition:



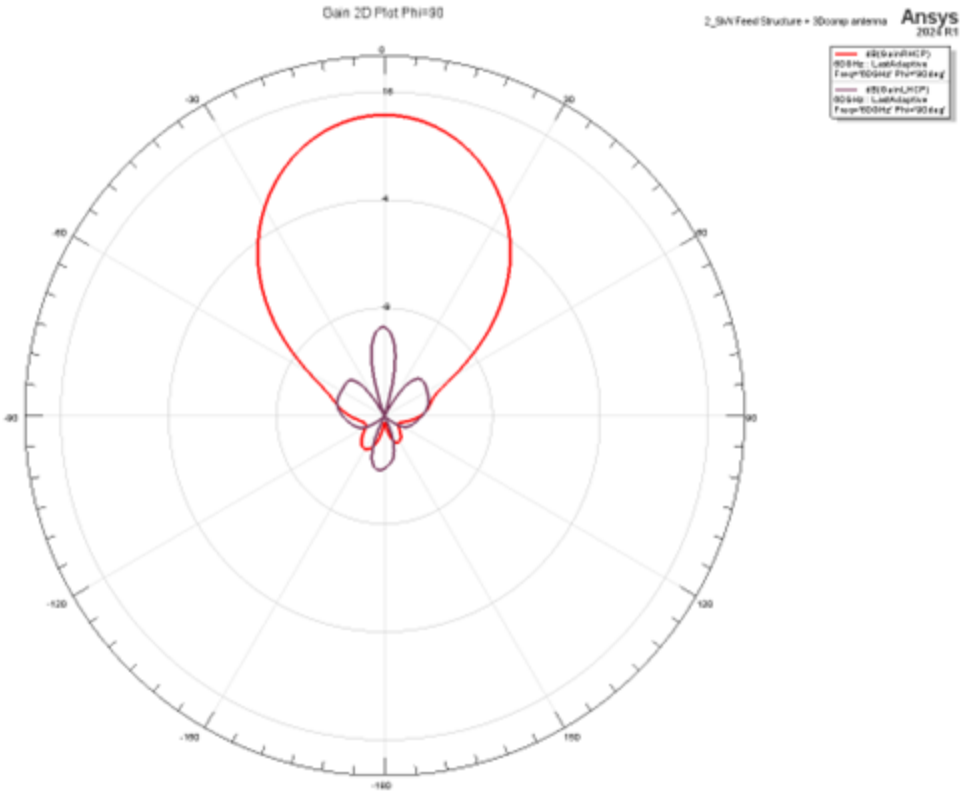
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

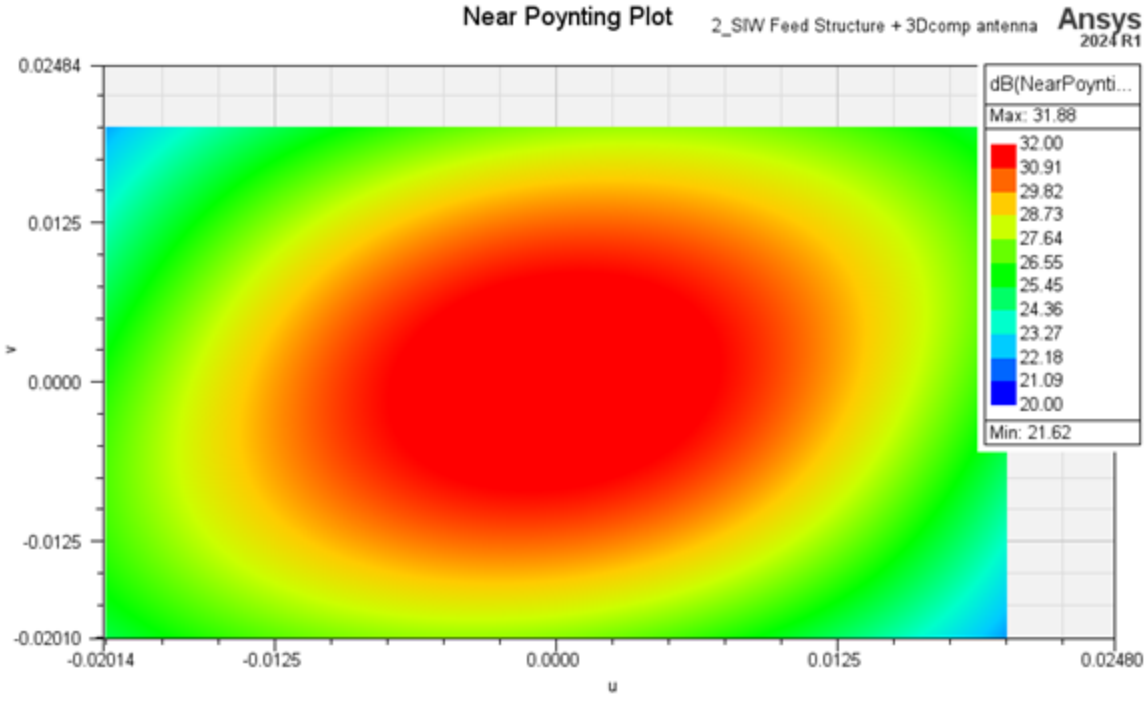


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



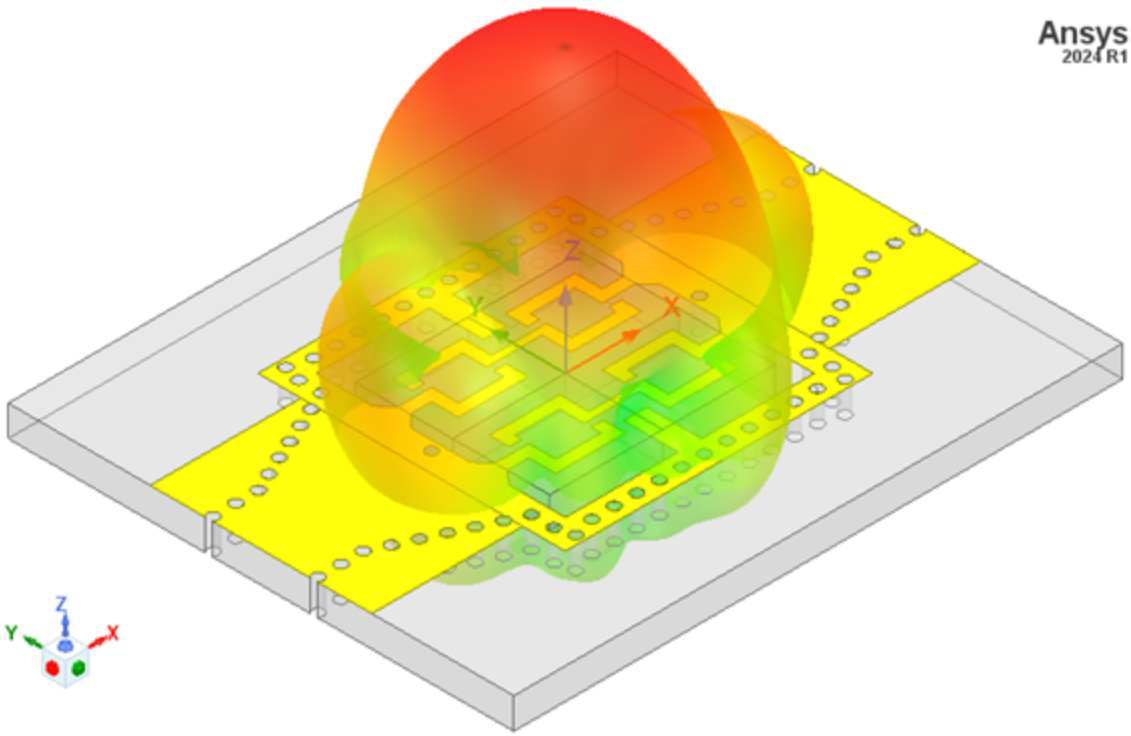
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





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

The following final image is an overlay of the Gain 3D plot on the model geometry:



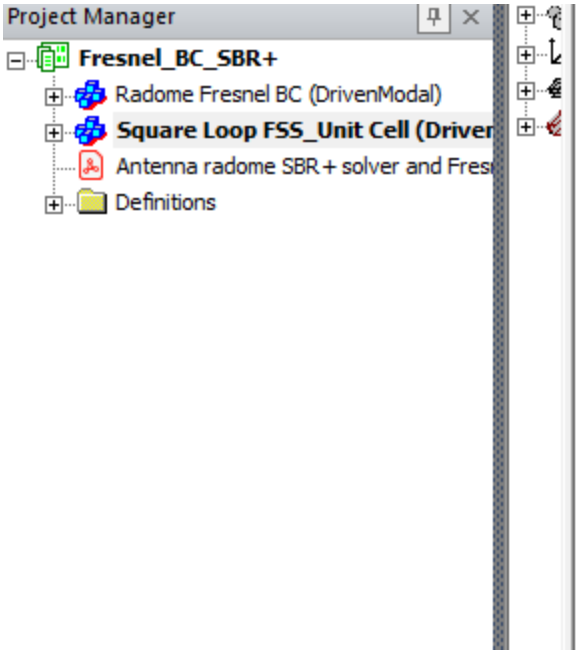
References

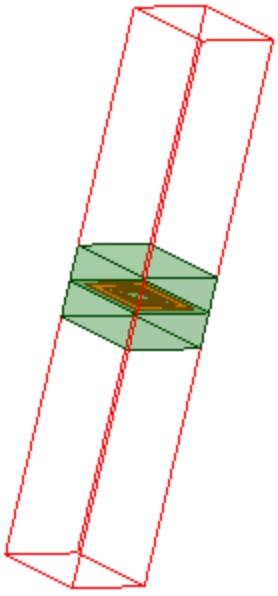
[1] Dia'aaldin, J. B., Liao, S., & Xue, Q., "High gain and low cost differentially fed circularly polarized planar aperture antenna for broadband millimeter-wave applications," IEEE Trans. Antennas Propag. 64(1), 33-42 (2016).

Antenna Radome SBR+ Using Fresnel Boundary

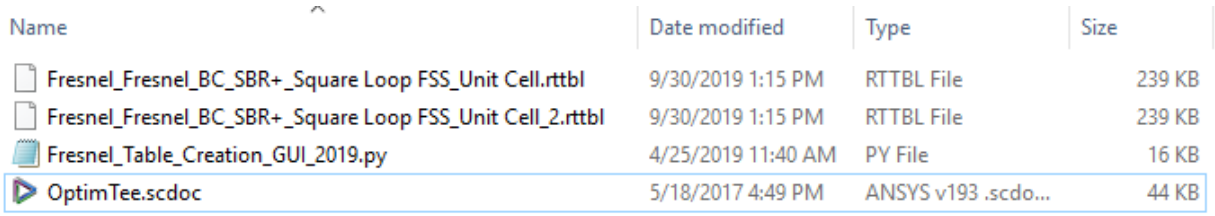
This example demonstrates how to apply Fresnel boundary condition in SBR+ to model an antenna radome. Frequency selective surface (FSS) was used for the antenna radome. The project Fresnel_BC_SBR+ consists of two designs: 1_Square Loop FSS_Unit Cell, and 2_ Radome Fresnel BC. The example starts with HFSS Unit Cell design that contains a dual square loop FSS. The pdf attached to the project describes how to simulate the models.

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





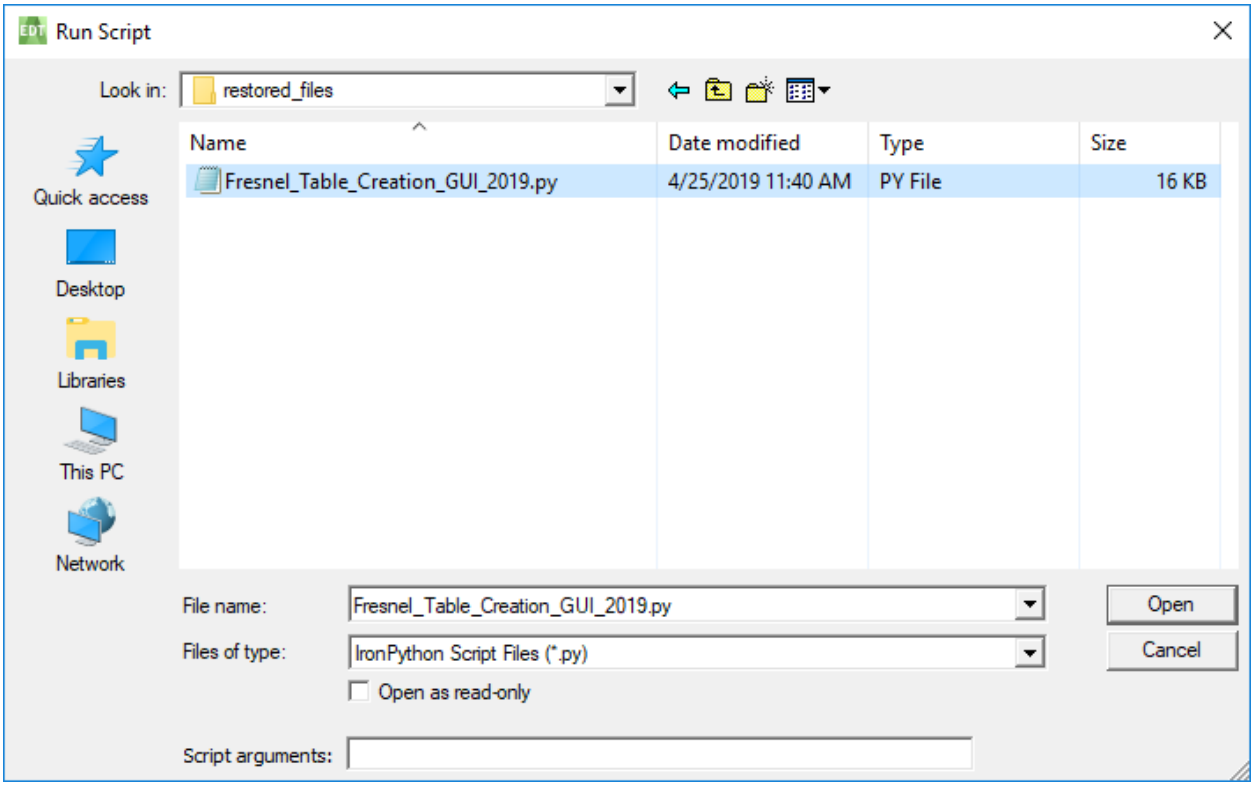
Once HFSS Unit Cell design is solved, script Fresnel_Table_Creation_GUI_2019.py should be run in order to obtain .rttbl table that is used to assign SBR+ Fresnel boundary condition for Radome design. The project is stored as an .aedtz archive in order to provide the scripts. Opening RadomeFresnel_BCPSBR+.aedtz from the **Examples** > **HFSS** > **Antennas** directory automatically asks you to specify a Restore location. In the save location you specify, you will see a **restored_files** folder. This folder contains the python script you need. Open the Antenna radome SBR+ solver and Fresnel Boundary Condition pdf for detailed instructions on how to work with the project and use the script.



Name	Date modified	Type	Size
Fresnel_Fresnel_BC_SBR+_Square Loop FSS_Unit Cell.rttbl	9/30/2019 1:15 PM	RTTBL File	239 KB
Fresnel_Fresnel_BC_SBR+_Square Loop FSS_Unit Cell_2.rttbl	9/30/2019 1:15 PM	RTTBL File	239 KB
Fresnel_Table_Creation_GUI_2019.py	4/25/2019 11:40 AM	PY File	16 KB
OptimTee.scdoc	5/18/2017 4:49 PM	ANSYS v193 .scdo...	44 KB

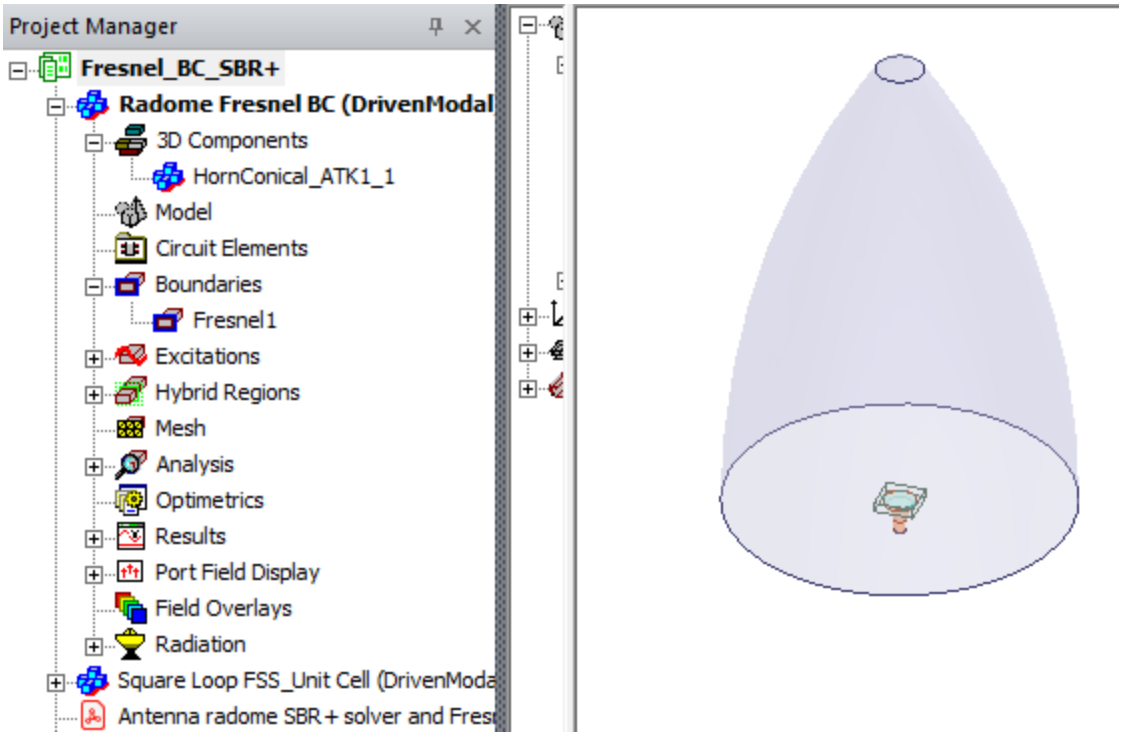
At one point you will use **Tools** > **Run Script...** to look in the restored_files folder. Set the Files of type line to Python to see and select the script.

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



The Radome is an SBR+ Region and all rays incident upon radome will be correctly captured even at obliques angles of incidence. With assigned Fresnel Table Boundary Condition to the radome, analyze the design. Make sure to use GPU for SBR+ solver if system supports it.

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



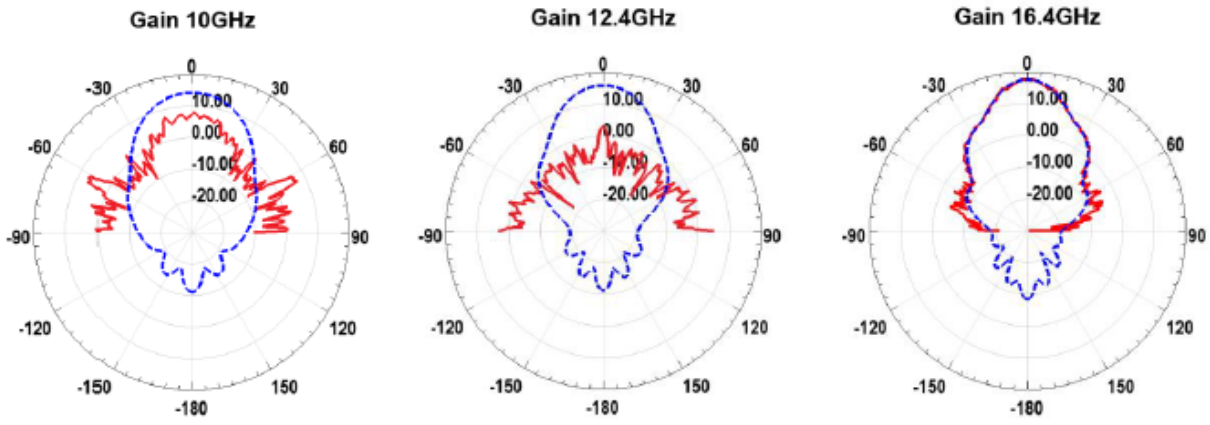


Figure 3: Gain patterns for isolated circular horn antenna (dashed blue line) and horn and radome system (solid red line) at 10 GHz, 12.4 GHz, and 16.4 GHz.

Antennas on Mast

Description - Three dipole antennas mounted on an aluminum mast, simulated in HFSS-IE. The mast is mounted on an infinite ground plane.

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

Model - The mast is 3.1 m high. The dipoles are modeled using 2D objects with a PerfE boundary. The excitations are lumped ports and the mounting structures are modeled by the polystyrene rods. Note under Boundaries in the Project tree, InfGndPlane1.

Setup - Adapt at 0.9 GHz.

> **Note:**
>
> To view a port or boundary, select the desired item in the Project Tree. It is then highlighted in the Model window and the properties will be displayed in the Properties window. Selecting an object in the History tree will also display its properties.

Post Processing Antennas Mounted on a Mast

After solving, you can view solution data by right-clicking on Setup1 and selecting **Profile** to display the **Solution** dialog. You also view the **Solution** tabs for **Convergence**, **Matrix Data**, and **Mesh Statistics**.

Before viewing the fields make sure all three antennas are excited. Go to **HFSS>Fields>Edit Sources** and uncheck all Terminated options. Set Scaling Factor to 1 for all three ports.

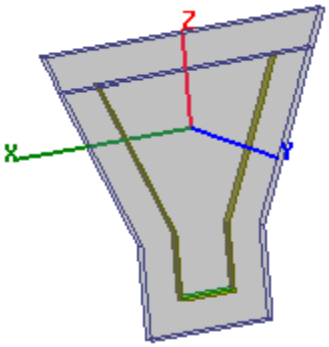
To view the radiation pattern shown, double-click Radiation Pattern 1 under Results in the Project tree. This is the $\theta = 90^{o}$ pattern cut.

To view the induced currents on the mast, double-click on Mag_J1 under **Field Overlays>J Fields**.

Determining Phase Center Using Optimetrics

The phase center is the point from which the electromagnetic radiation spreads spherically outward, with the phase of the signal being equal at any point on the sphere. This example illustrates a technique for using an optimization setup to search for the phase center.

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



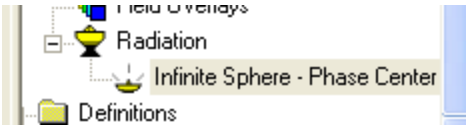
The technique involves the following tasks.

1. Define a relative coordinate system that uses a post processing variable.

   This allows the variable value to change the definition of the coordinate system and thereby change a solved solution without invalidating it.

2. Use a relative coordinate system in a Far field setup calculation.

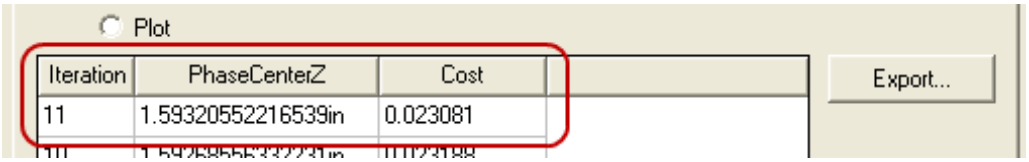
   This allows far field quantities to be re-calculated when the relative coordinate system is repositioned (demonstrated in the example by changing the value of the PhaseCenterZ variable), without re-simulation.



   The far field setup will also limit the angular range of the sweeping intrinsic (phi or theta).

3. Create an optimization setup to search for the phase center. The search will be efficient if you have a rough idea on the location of the phase center. The expression to be optimized is the peak to peak continuous angle of the quantity rEPhi. (The electric field of this antenna is Phi polarized.)
4. Analyze the Optimization setup and view results via the **View Analysis Result** dialog.

   The Z coordinate of the phase center will be the value of the post processing variable when Cost is at minimum.



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

The following sections detail these steps for setting up an Optimetrics project to calculate the Phase Center.

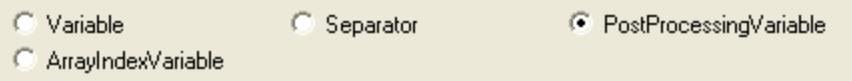
- Define the Relative Coordinate System using a Post Processing Variable
- Use the Relative Coordinate System in a Far Field Setup Calculation
- Create an Optimization Setup to Search for the Phase Center
- Analyze and View the Optimization Results

Define the Relative Coordinate System using a Post Processing Variable

First create a post processing variable that you can use to define a relative coordinate system.

1. Use **HFSS>Design Properties** to display the **Design Properties** dialog box.
2. Click **Add Variable**, to display the **Properties** dialog for defining a Design Variable. The example project uses the name “PhaseCenterZ”.

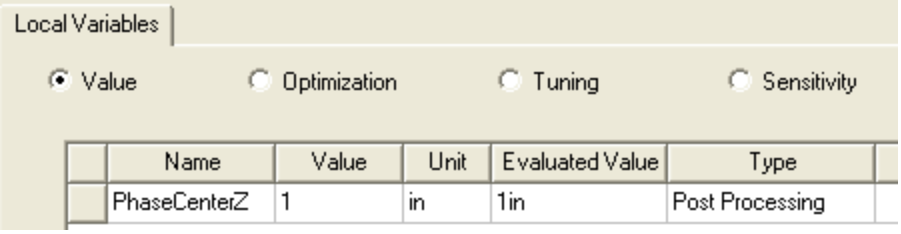
Designate the type as Post Processing variable:



Set the Unit Type as Length, the Units. This example uses “in” for the units and the value as 1.

Click **OK**.

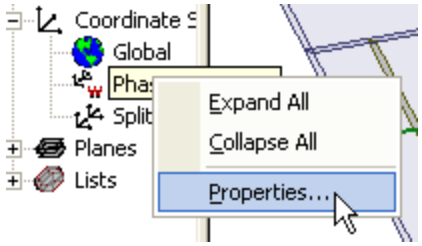
The PhaseCenterZ variable is listed as a Local variable with the Post Processing Type.



3. You can use the variable to define an offset relative coordinate system.

Use **Modeler>Coordinate System>Create>Relative CS>Offset** to create a coordinate system, using the variable as the z coordinate. You can verify the Properties.

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



In the example, the PhaseCenterZ variable defines the z coordinate of the origin for Phase Center CS.

Coord System

	Name	Value	Unit	Evaluated Value
	Type	Relative		
	Name	Phase Center CS		
	Reference CS	Global		
	Origin	0in ,0in ,PhaseCenterZ		0in , 0in , 1in
	X Axis	1 ,0 ,0	in	1in , 0in , 0in
	Y Point	0 ,1 ,0	in	0in , 1in , 0in

This variable allows you to change definition of the coordinate system during the optimization process without invalidating solved solution.

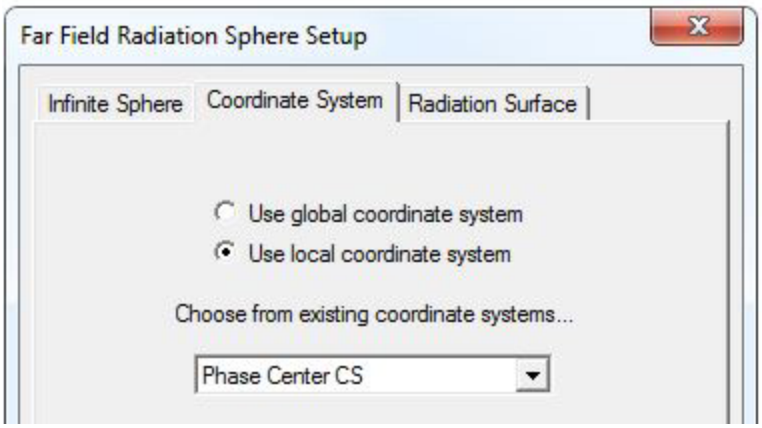
Next, Use the Relative Coordinate System in a Far Field Setup Calculation.

Use the Relative Coordinate System in a Far Field Setup Calculation

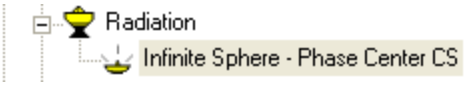
1. Click **HFSS>Radiation>Insert Far Field Setup>Infinite Sphere**, or right-click **Radiation** in the Project tree and on the shortcut menu, click **Insert Far Field Setup>Infinite Sphere**. This displays the dialog box.
2. Set the Infinite Sphere values appropriately, and on the Coordinates tab select the

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

Relative coordinate system that uses the Post Processing variable.



In the example project, the radiation sphere setup is named Infinite Sphere - Phase Center CS.



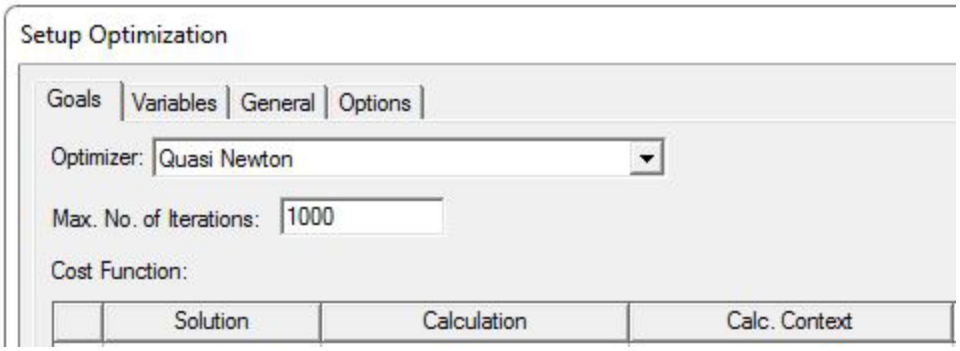
This allows far field quantities to be re-calculated when the relative coordinate system is repositioned (by changing the value of the PhaseCenterZ variable), without re-simulation.

The far field setup will also limit the angular range of the sweeping intrinsic (phi or theta). rEPhi and rETheta are the quantities of interest, depending on the radiation direction of the antenna.

Next, Create an Optimization Setup to Search for the Phase Center

Create an Optimization Setup to Search for the Phase Center

1. Add an Optimization setup.

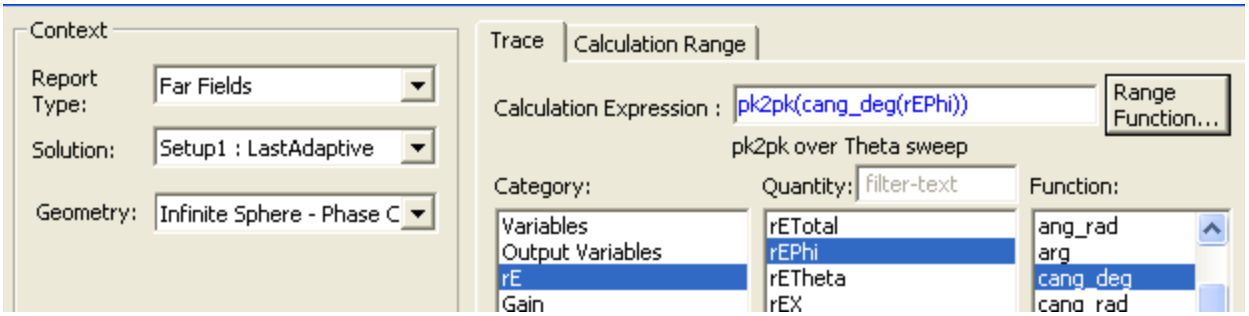


2. Click the Setup Calculations button to open the dialog for defining the Calculation Expression.

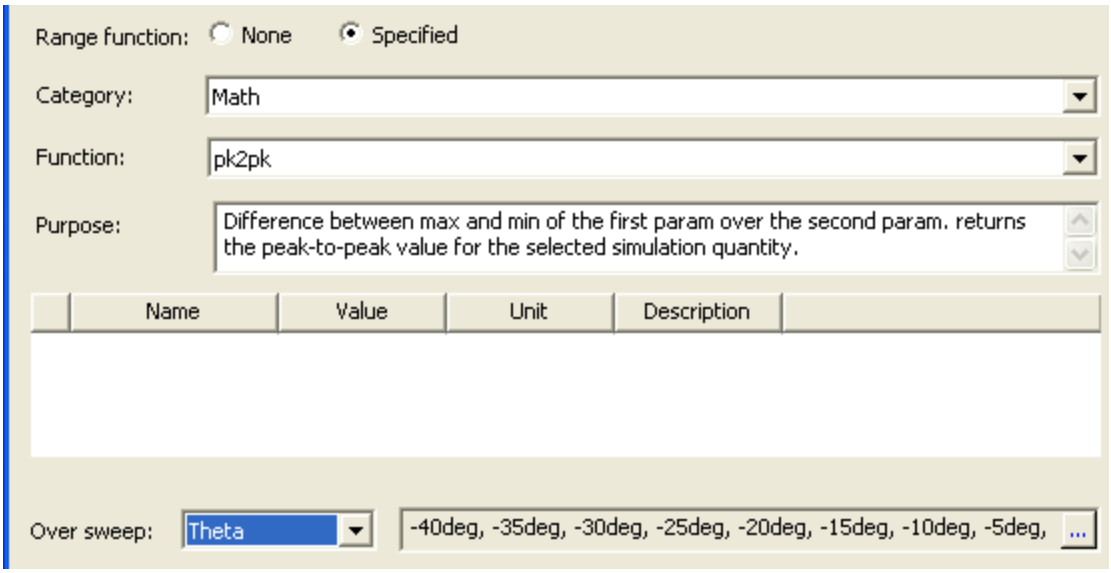
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You will define Optimization Goals designed to search for the phase center.

The expression to be optimized is the peak to peak continuous angle of the quantity rEPhi.

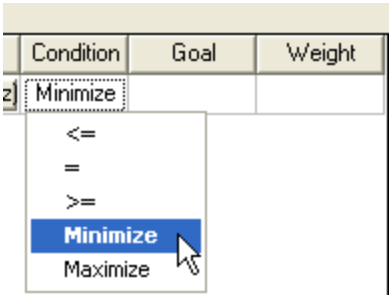


- The Report Type is Far Fields.
- The Solution is LastAdaptive.
- The Geometry is the Infinite Sphere defined using the PhaseCenterZ variable.
- The Calculation Expression includes the rEPhi Quantity with the cang_deg function. The antenna is Phi polarized for the Phi=0 plane (E-field along the y-axis).
- You then click Range Function button to select Math and pk2pk to apply over the Theta sweep.



- The Over sweep field defines the range of the sweeping intrinsic, with Theta from -40 deg to +40 deg. 0 deg is selected for the value of Phi.
- Return to the Setup Optimization dialog box. Here you may need to scroll to the right to see and set the Goal weight Condition to Minimize.

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



The “Minimize” goal condition doesn't require user to set up the goal value/weight and can be more natural for antenna related application. However, users need to be aware of below restrictions.

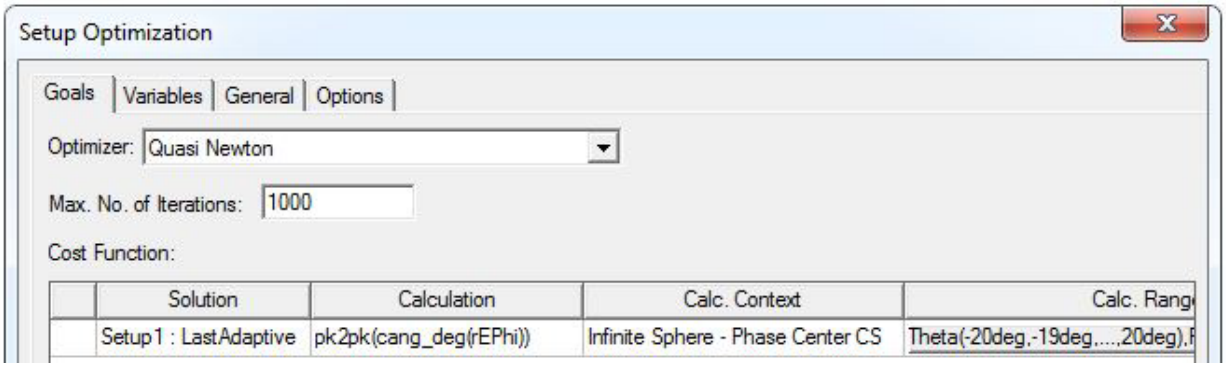
- “Minimize/Maximize” can only be used when a single calculation quantity is being optimized.
- Optimetrics only supports one dimension of range reduction, that is, no nested range function

The above restrictions implies “Minimize/Maximize” choices can only be used if

- Only one sweep in the Calculation Range has multiple values, and
- A Range reduction function is used in the Calculation Expression

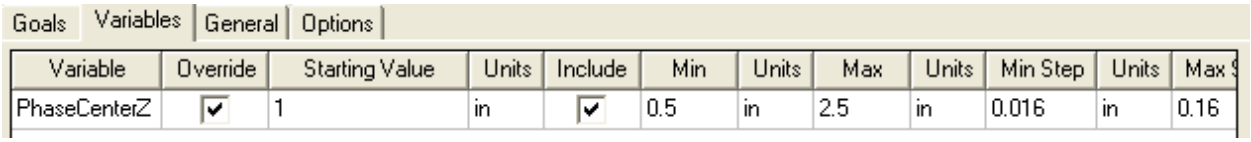
In the example project, OptimizationSetup2 uses the Minimize selection. For comparison, The OptimizationSetup1 uses the <= Condition with a Goal of 0 and a weight of 1, subject to the Noise setting of 0.0001. The final values for PhaseCenterZ in the two setups are very close.

The completed Goals for the Optimization setup2 for the example display as shown:



For the **Variables** tab, you should check to include the variable used to define the relative coordinate system. The search is restricted to a minimum of 0.5 in and a maximum of 2.5 in along the Z coordinates of the relative coordinate system.

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



For the **General** tab, uncheck Update design parameters value after optimization. This permits you to view a plot of the progress of the simulation.

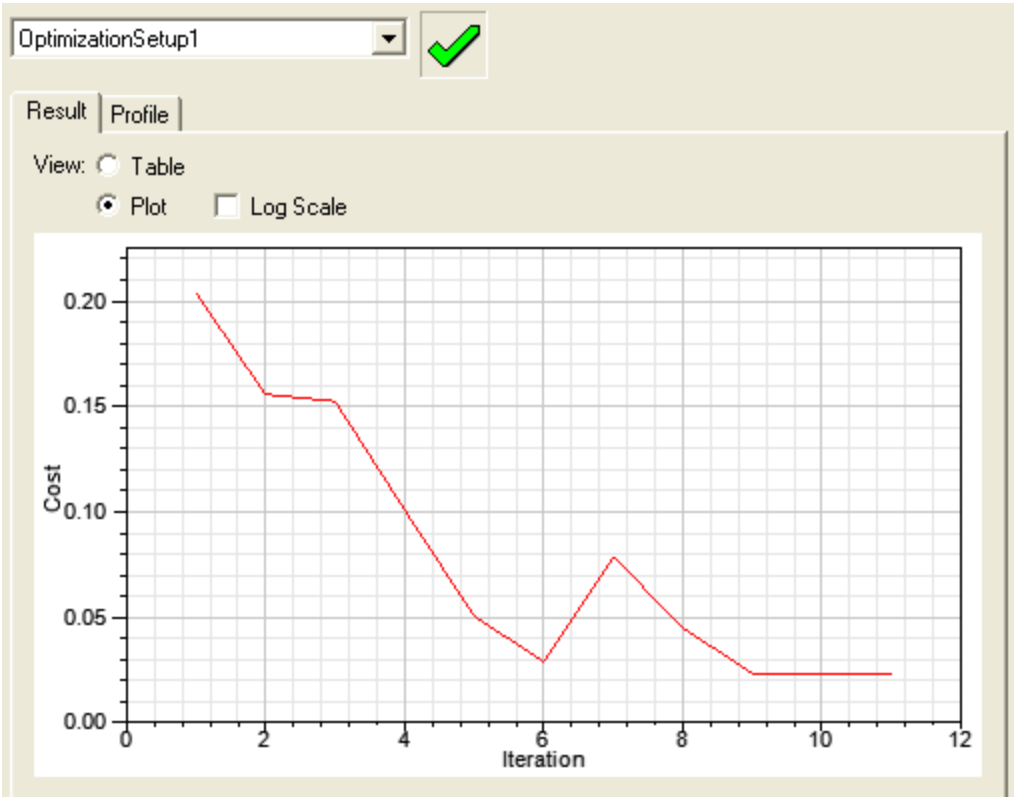
OK the completed Optimization Setup.

Next, you Analyze and View the Optimization Results.

Analyze and View the Optimization Results

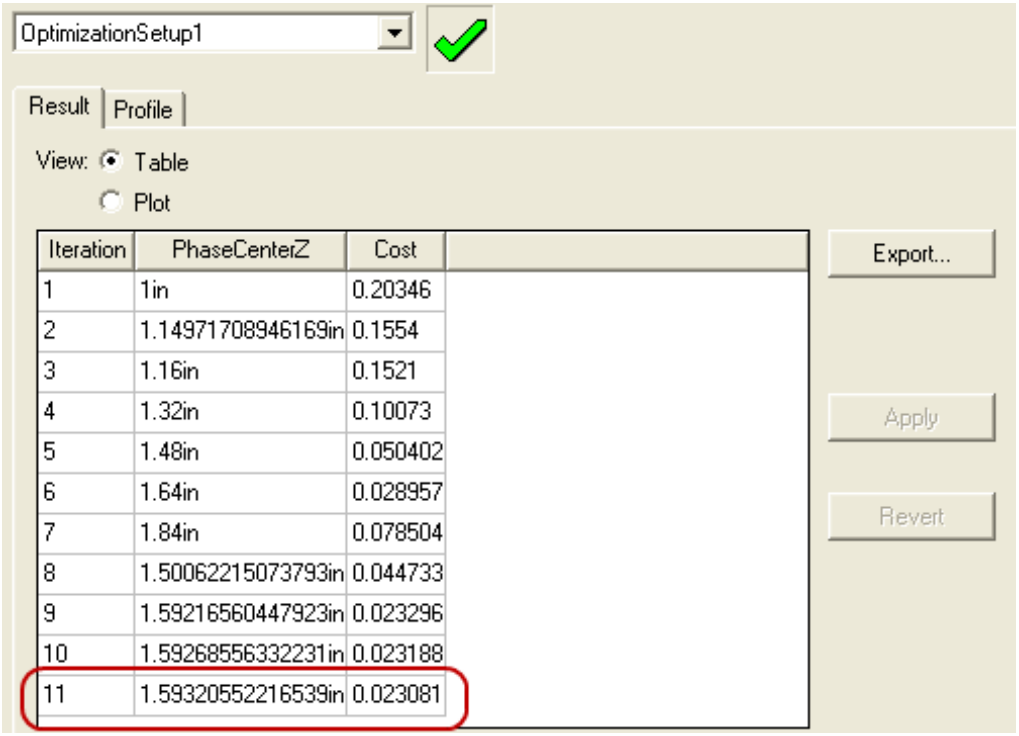
Right-click the Optimization Setup to view the short-cut menu and click **Analyze** to start the simulation. Then right-click again, and select View Analysis Result to view the Post Analysis dialog box. Select the Plot radio button to watch the progress of the simulation.

The plot form of the results shows how the Optimization search progresses toward the lowest cost.



The table form of the Results shows that the Z coordinate of the phase center is the value of the post processing variable when Cost is at minimum.

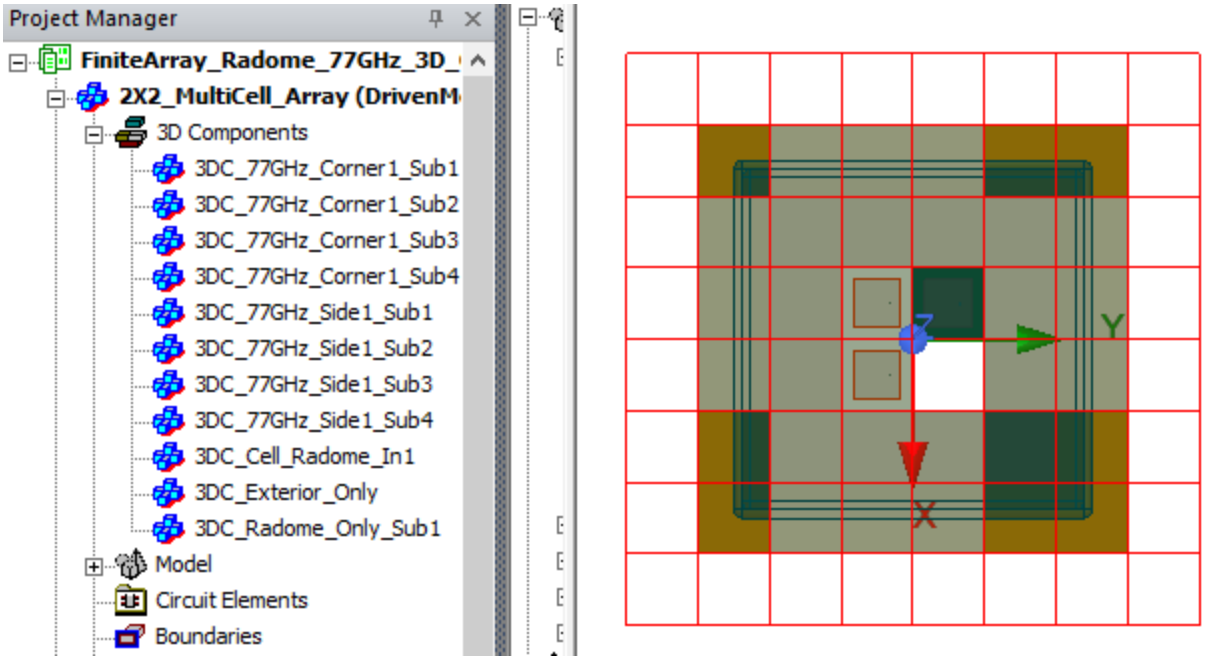
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Finite Array Radome - Multi Cell Array

The Finite Array Radome model in the Examples/Electronics Desktop/HFSS/Antennas folder contains a 77 GHz 2x2 phased array antenna and radome integration using the 3D component finite array workflow, also known as Component Array Domain Decomposition Method (CA-DDM).

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



This feature utilizes efficient iterative domain-decomposition-based finite element technique for modeling finite (semi-)periodic structures that contain non-identical unit cells. This simulation technique enables faster simulation times with memory usage reduction compared to simulating the finite array Domain Decomposition Method (FA-DDM) or explicit array, and it can leverage distributed computing resources with shared memory.

The included PDF 3D_Component_Finite_Array_&_Radome describes the model and techniques in detail, and describes how to run the simulation and generate a variety of reports.

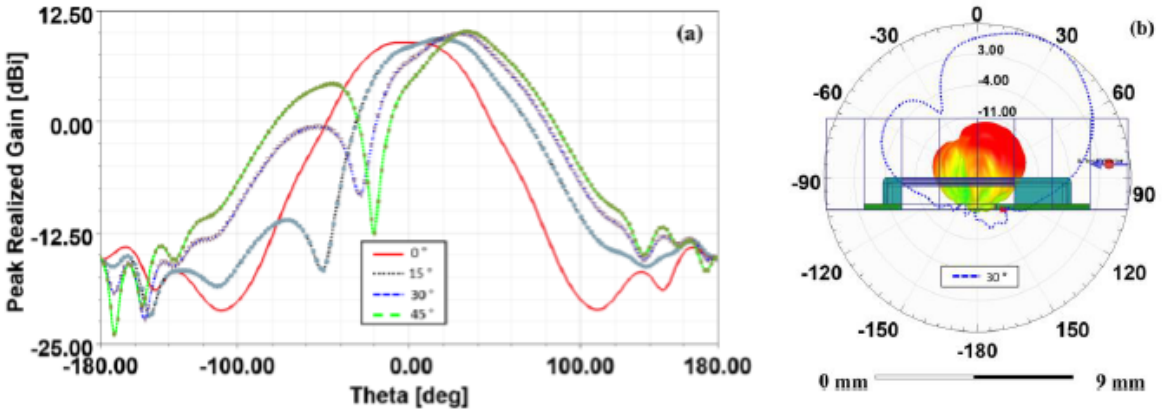


Figure 6: (a) Peak realized gain rectangular plot at different Scan Angle Theta (0°,15°, 30° and 45°). (b) Overlay representation of the peak realized gain 3D polar plot and radiation pattern at phi=0° cut with Scan Angle Theta in 30°, note that the beam has been steered at 30°.

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

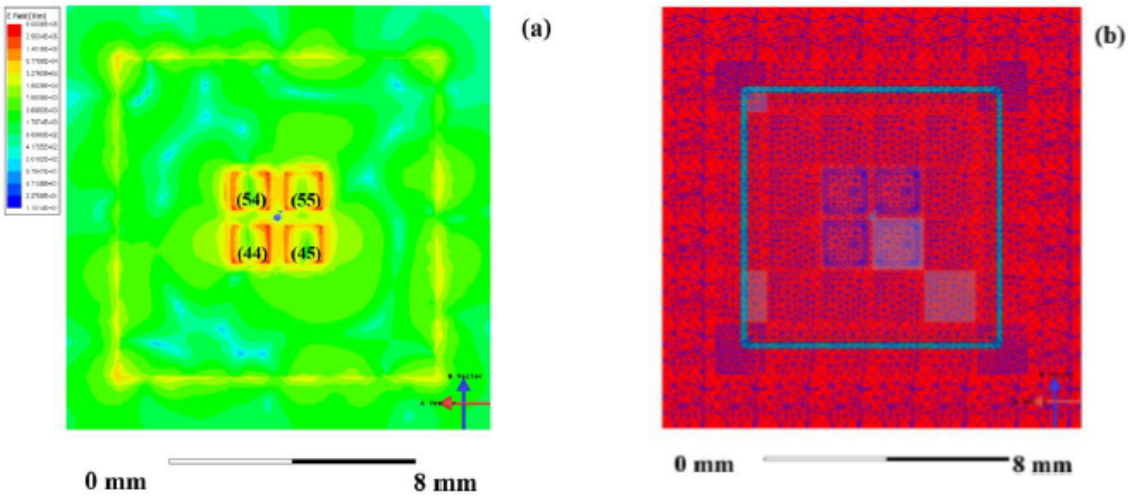


Figure 8: (a) Complex magnitude of the electric field (Complex Mag E) on substrate's top surface XY plane, (b) Adapted mesh on substrate's top surface XY plane.

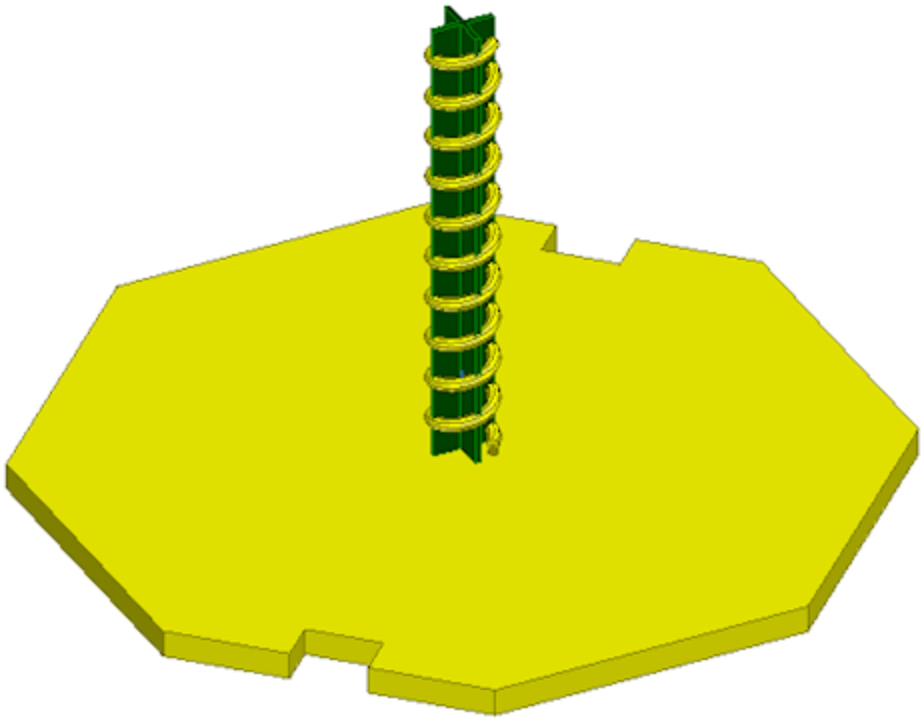
Helical Antenna

Abstract

A coax-fed helical antenna with a dielectric support on a finite ground plane makes up the design.

The antenna is set up to run at 3.5 GHz. The design illustrates how to build a parametric helical antenna. Using Material override, geometries can overlap without error.

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



Version: 2024 R1

Description

Model Construction:

- HFSS solution type: HFSS with Auto-Open Region Radiation selected
- The support is made of Teflon and the ground has a thickness of 0.5 in. The coax port is internal and is capped by a conducting object. You can create a helix similar to this model by drawing a cross-sectional circle or polygon for the starting face and then clicking **Draw** > **Helix** while it is still selected. You must define two points for the axis direction of the helix as well as the pitch, number of turns, right/left hand, and optional radius change per turn.

> **Note:**
>
> To view a port or boundary, select the desired item in the Project Manager. It is then highlighted in the *Modeler* window, and the properties are displayed in the docked *Properties* window. Selecting an object in the History Tree will also display its properties.

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

Variables are used for the helix cross section, radius, number of turns, and more for future variation.

Under the radiation tab, multiple Far Field or Near Field setups are defined.

One post processing variable “y_dist” is defined to illustrate the variation of the radiated field from the Near Field to Far Field zone.

Excitations:

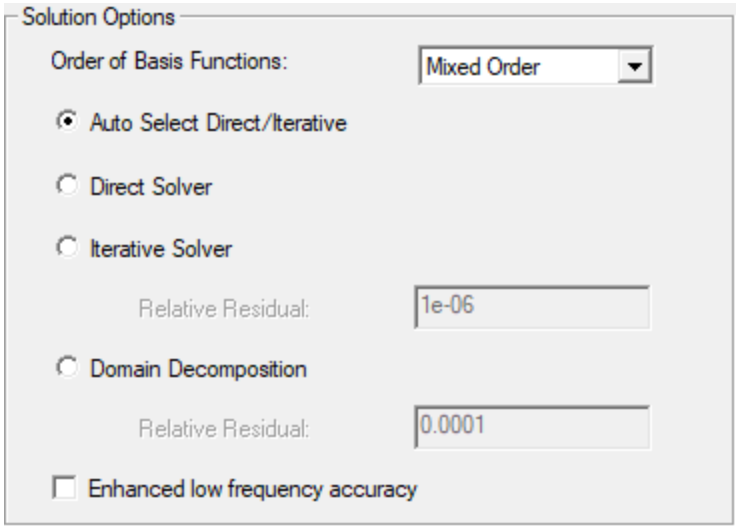
A wave port is defined internally so it will compute both the feed characteristic impedance and S parameters.

Setup:

Adaptive meshing is done at 3.5 Ghz - Convergence is set up to the default value (0.02).

Solution Options are as follows:

- **Order of Basis Functions** set to **Mixed order**, as the design contains a lot of small geometric details compared to wavelength. This setting can reduce the solution time.
- **Auto Select Direct/Iterative**



An interpolating sweep from 2.5 GHz to 4 GHz with 1001 points provides a smooth S parameters curve. Far fields, such as directivity or gain, are also computed for the frequencies directly included in the interpolating sweep. Specific frequencies can also be computed using a discrete sweep.

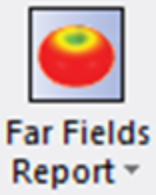
Simulation Time: 4 minutes (on a 6-core machine) – max RAM used 2GB

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

Postprocessing

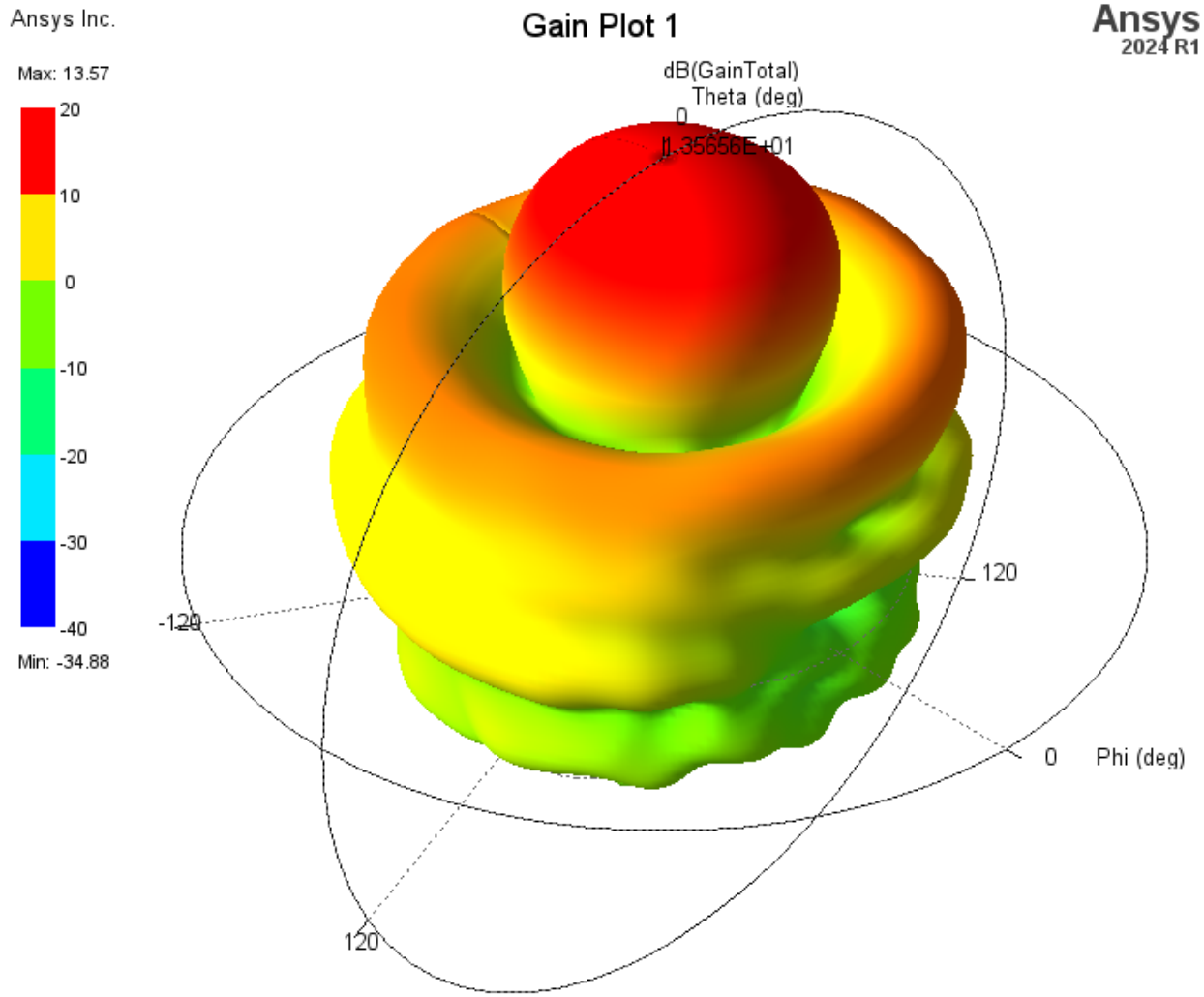
Several plots have been defined and are listed in the Project Manager under *Results*. Double-click any of them to see the plot. The plots are shown below, organized according to the type of report.

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



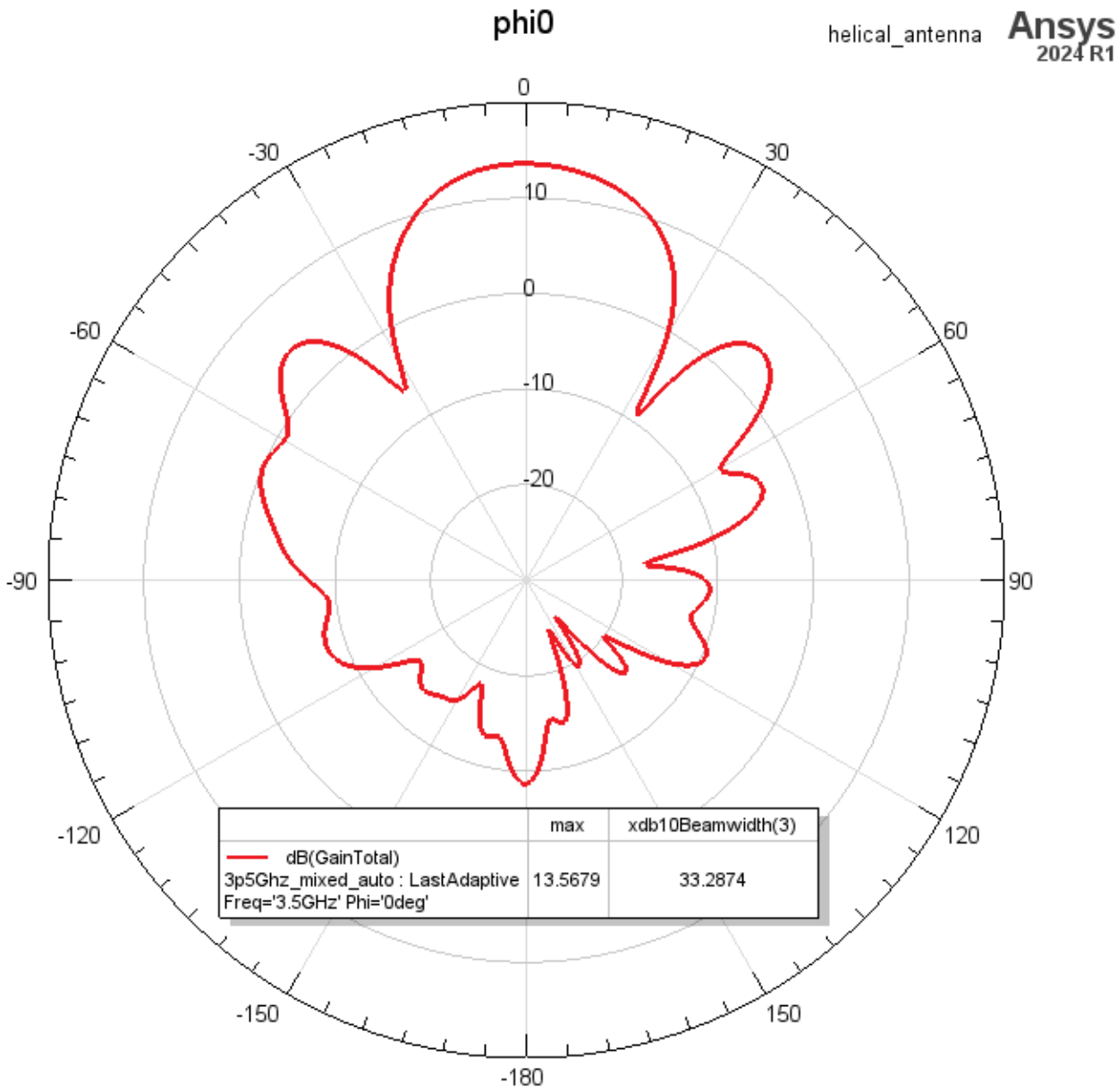
The following far fields reports have been predefined:

3D Polar Plot of Total Gain in Decibels:



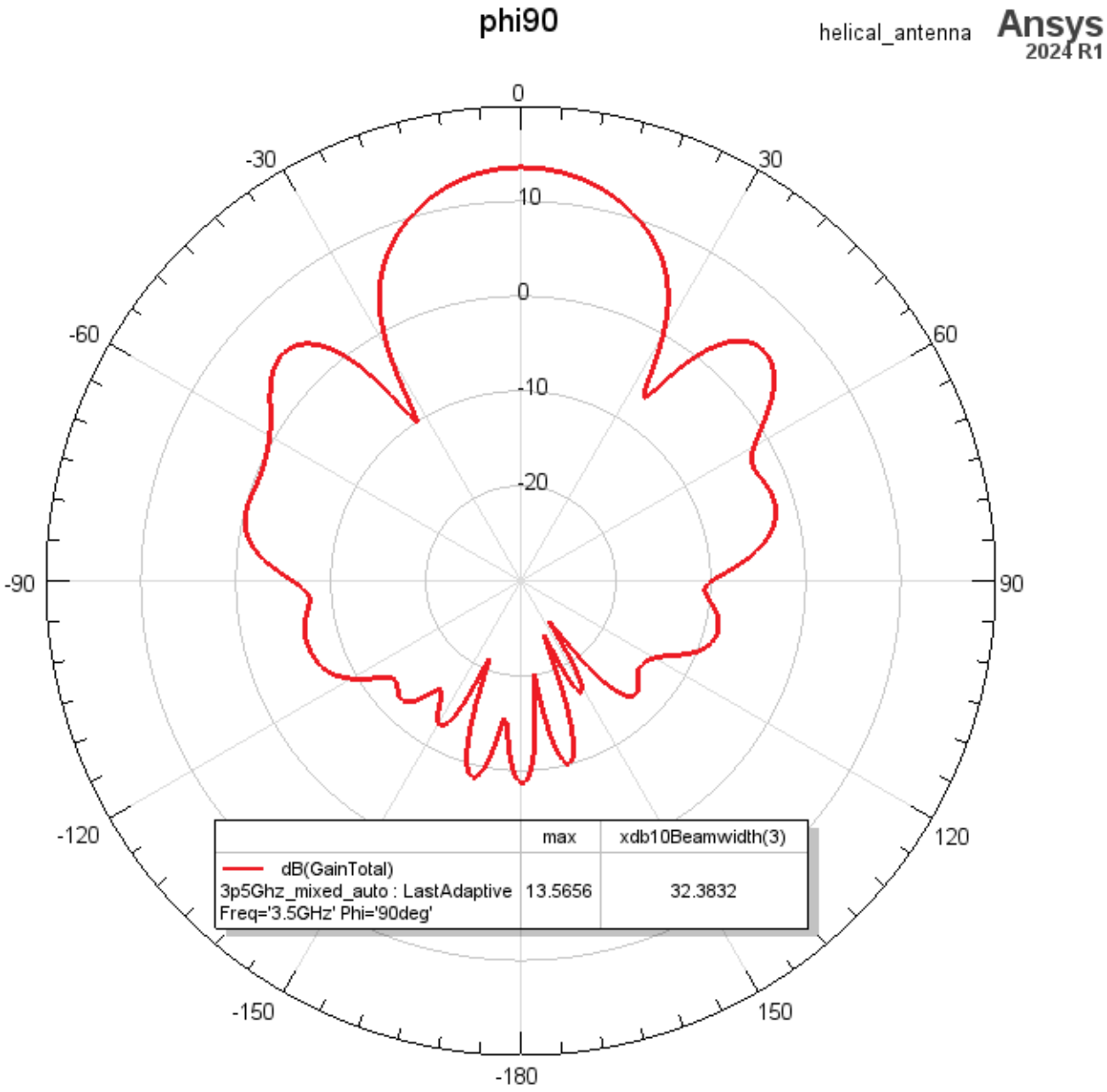
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2D Polar Plot of Total Gain in Decibels for phi = 0°:



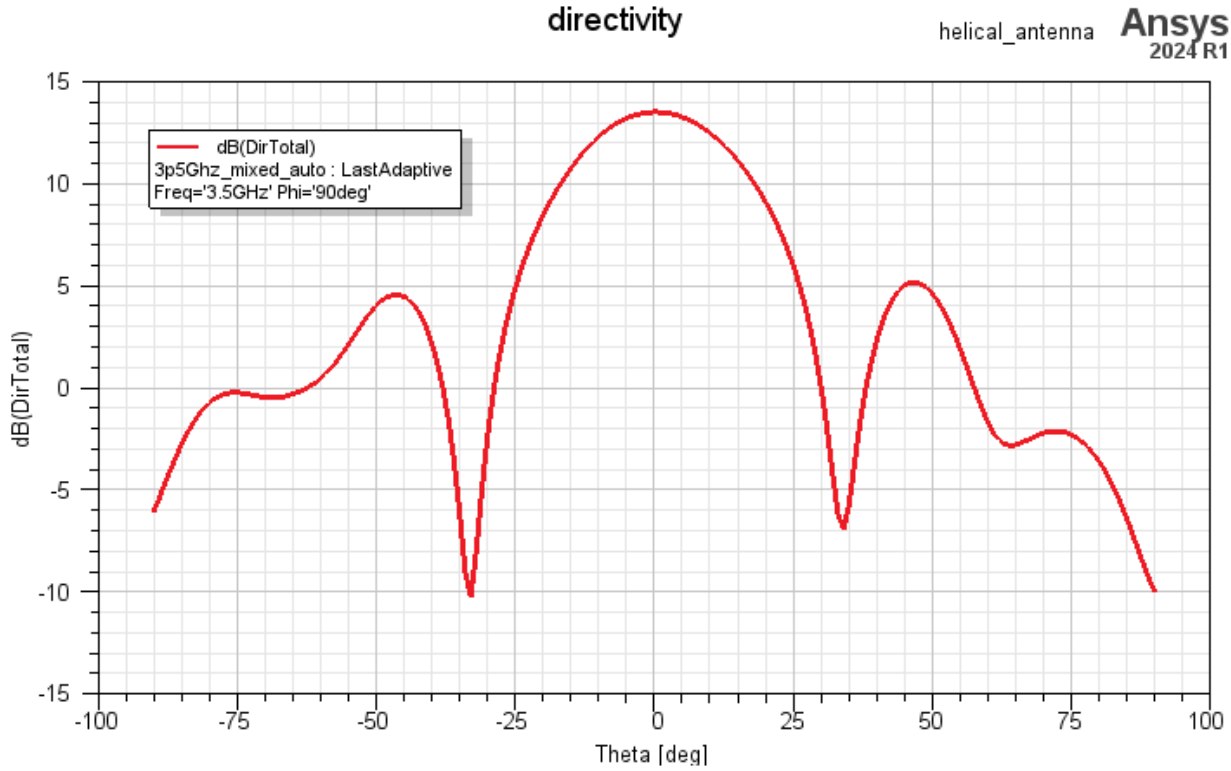
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2D Polar Plot of Total Gain in Decibels for phi = 90°:

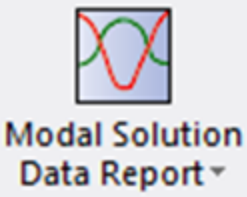


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Total Directivity in Decibels vs. Theta:

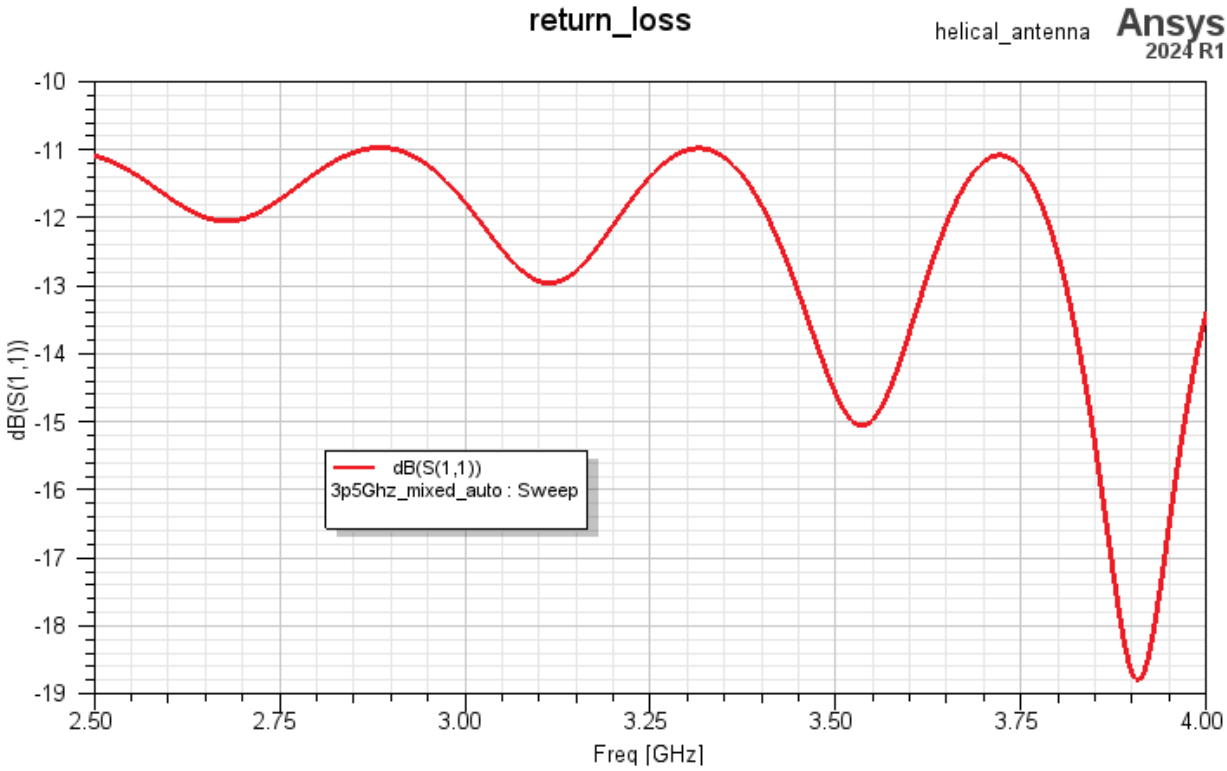


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

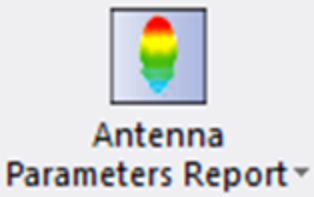


The following modal solution data report has been predefined:

S Parameter$_{1,1}$ – Return Loss vs. Frequency:

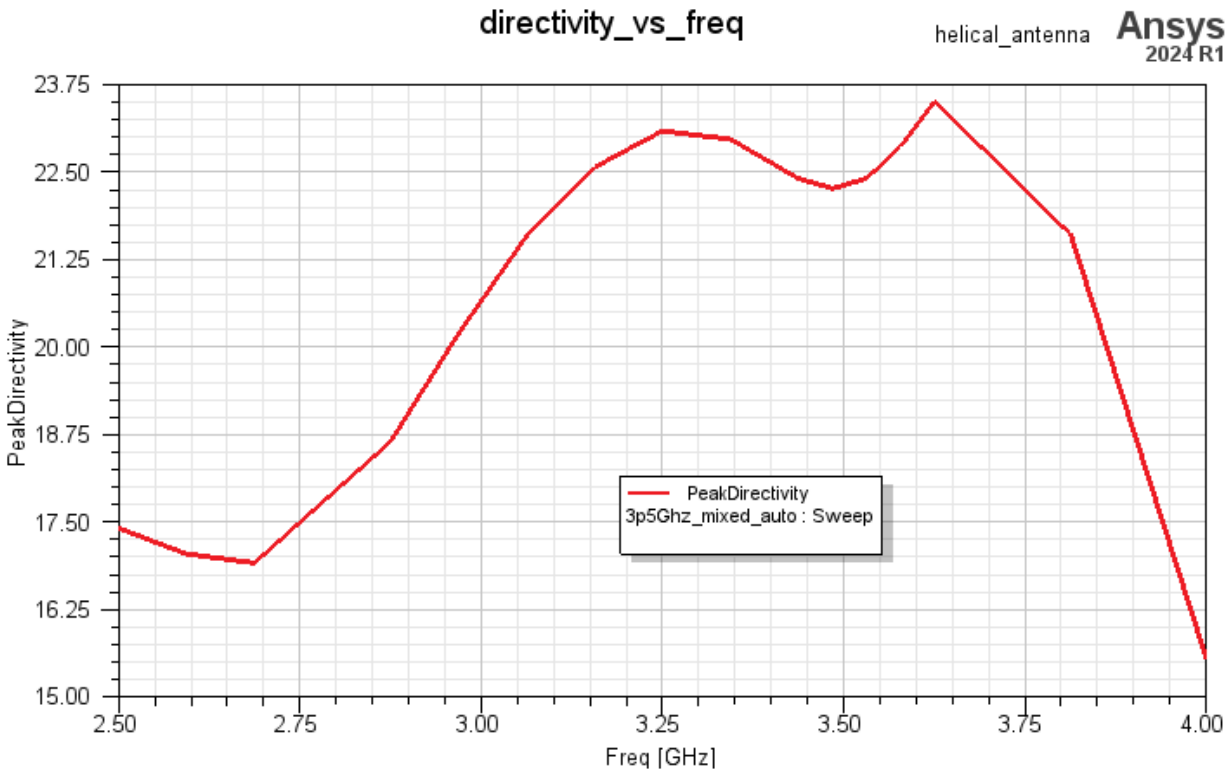


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

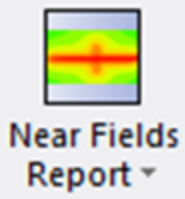


The following antenna parameters report has been predefined:

Peak Directivity vs. Frequency:

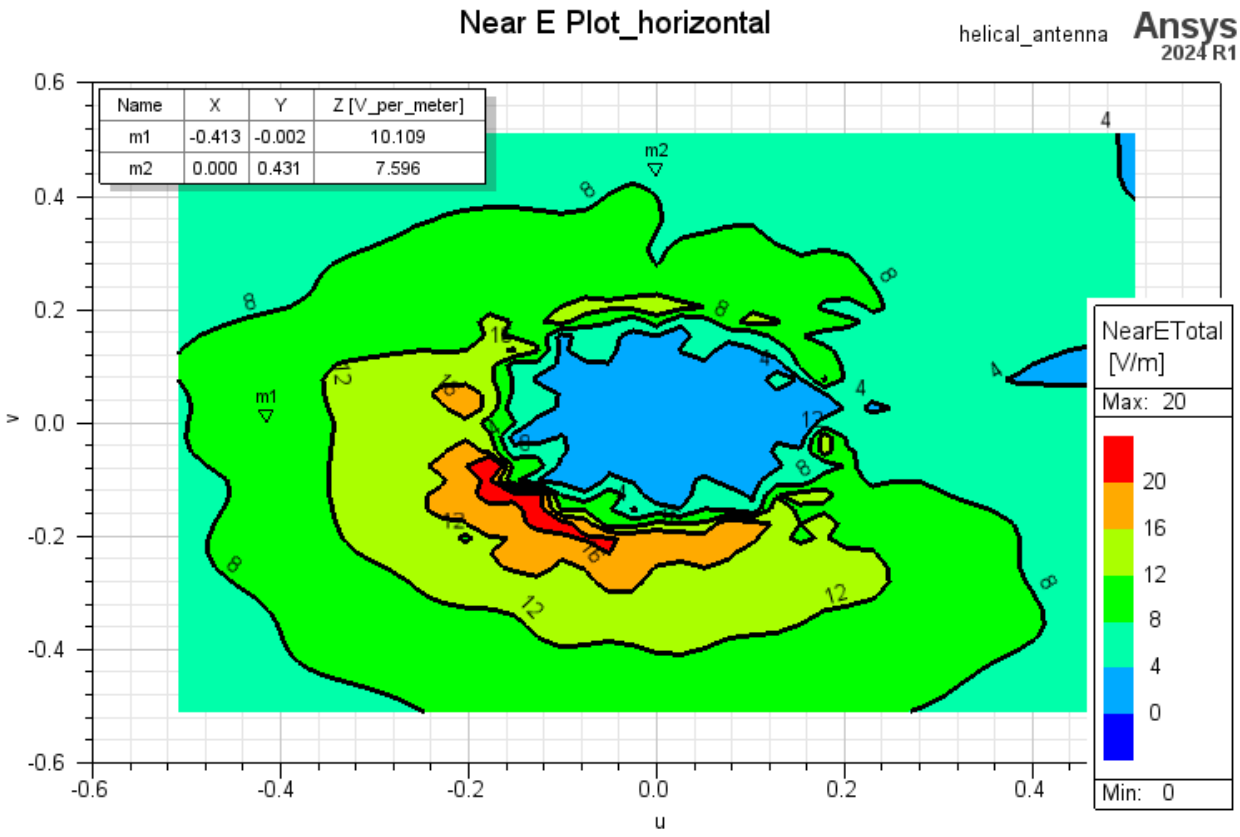


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



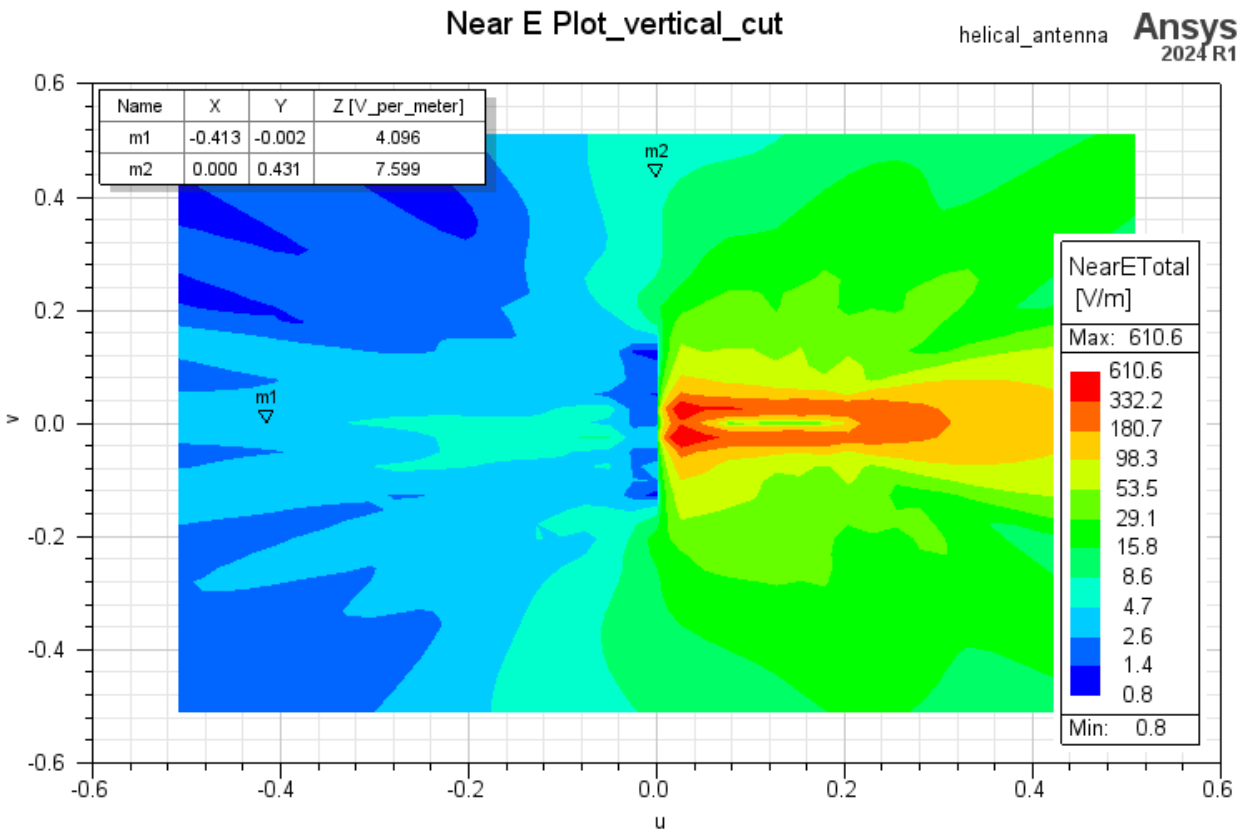
The following near fields reports have been predefined:

Near E_{Total}, 2D Contour Plot, Horizontal Cut (based on the *horizontal_near_field* radiation setup)**:**



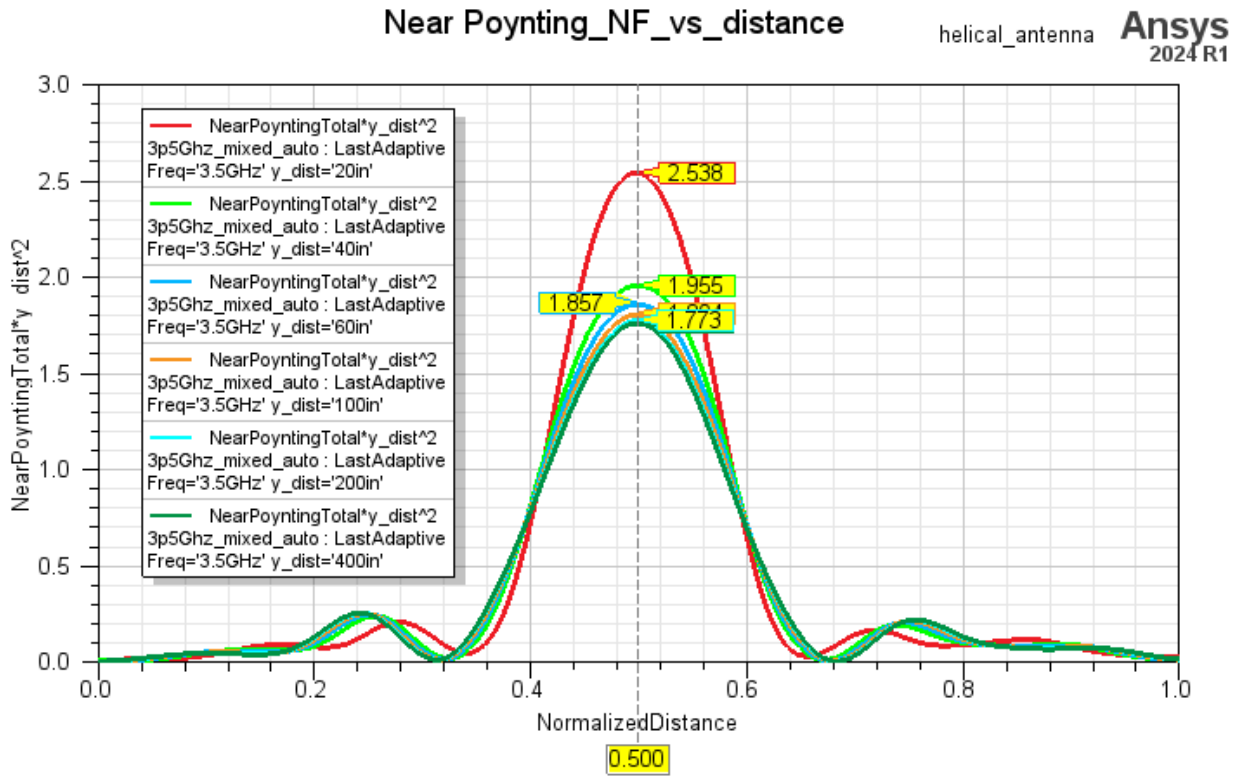
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Near E_{Total}, 2D Contour Plot, Vertical Cut (based on the *vertical_near_field* radiation setup)**:**



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

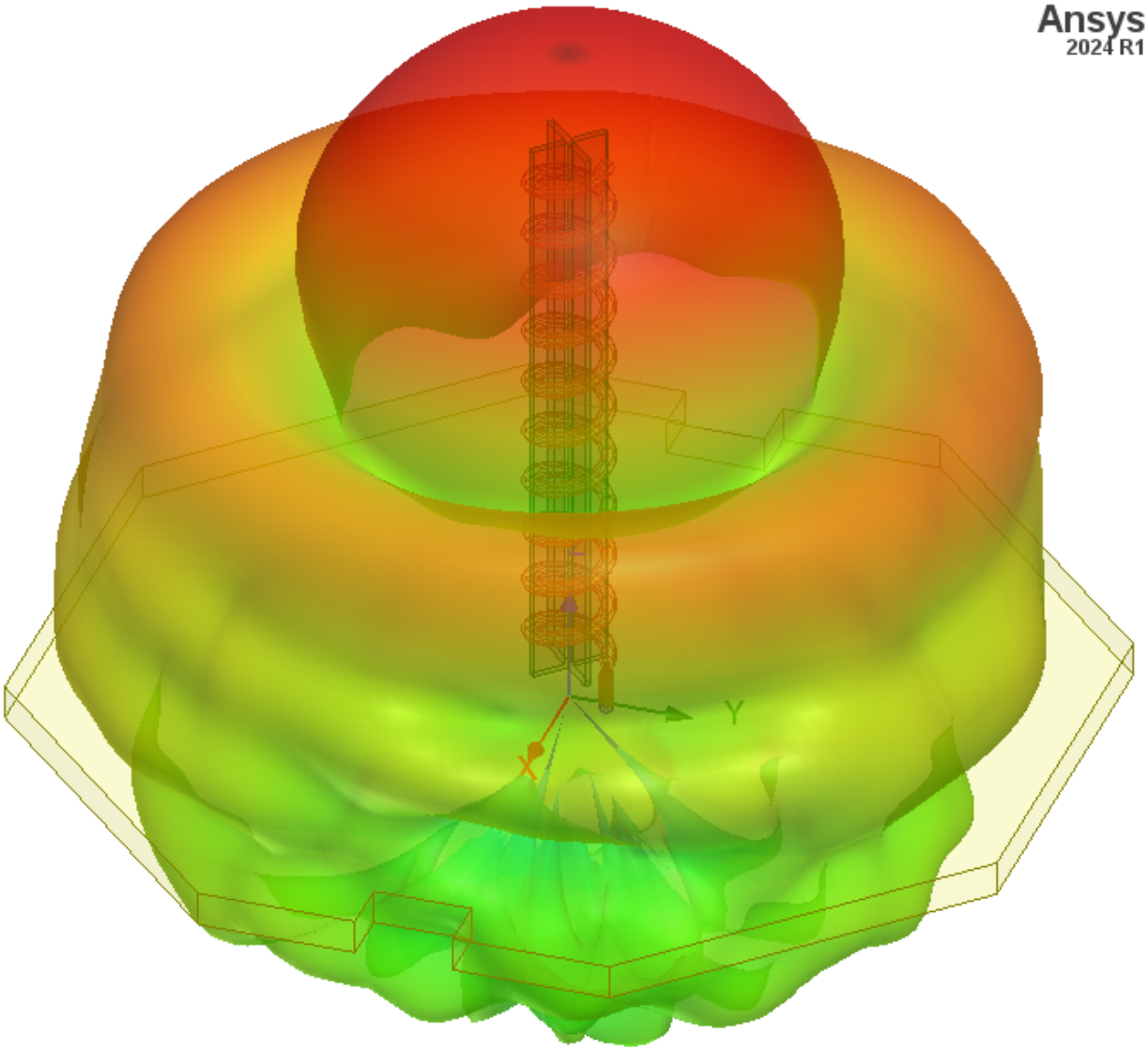
Magnitude of Poynting vs. Distance (“y_dist” variable) – illustrating Near Field / Far Field Zone**:**



You can add markers to any of the plots by right-clicking on the plot window and choosing **Marker** > **Add Marker**, **Add X Marker**, or **Add Y Marker**.

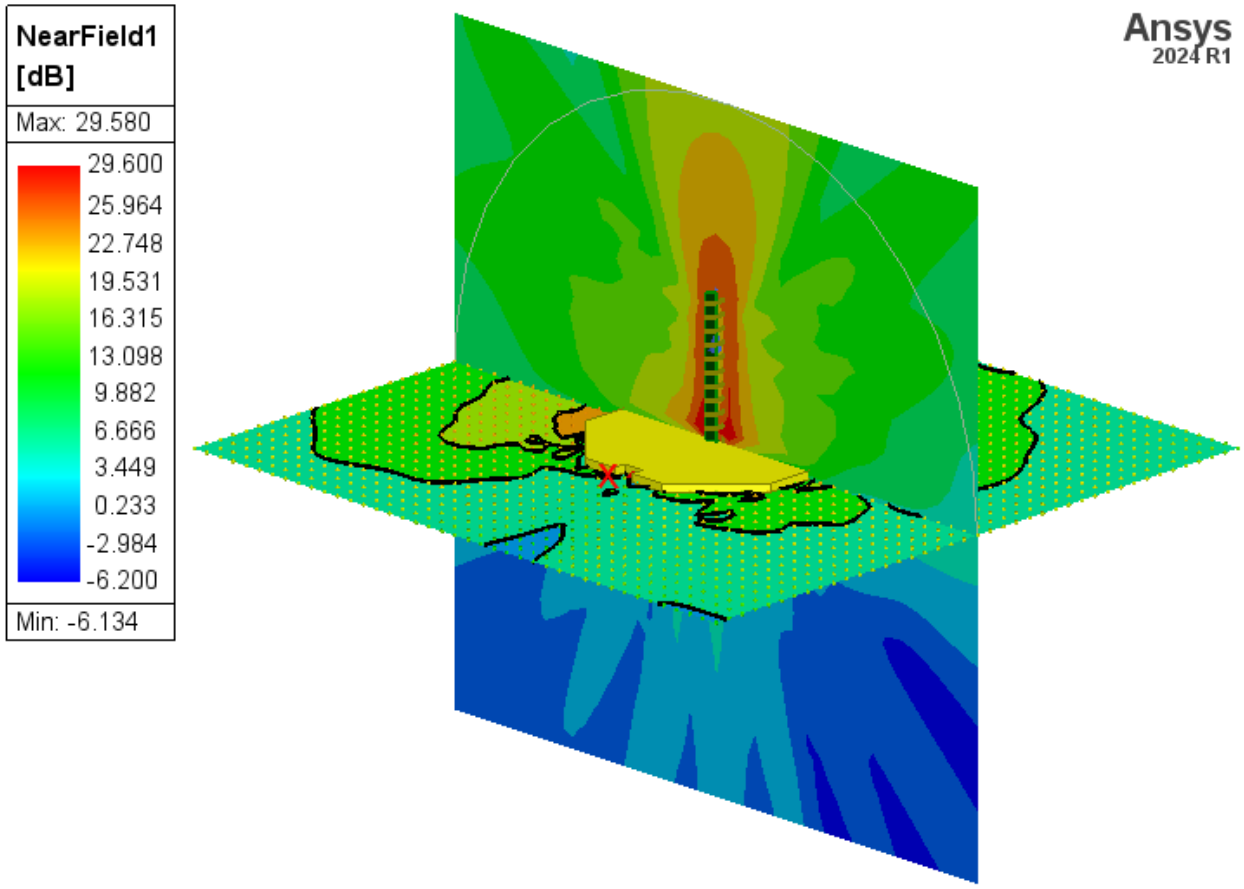
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To overlay the 3D gain plot on the model, click **HFSS** > **Fields** > **Plot Fields** > **Radiation Field** to display the *Overlay radiation field* dialog box. Check **Visible** for 3D Polar Plot 1 and set the transparency and scale as desired:



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

Similarly, here is the appearance when the **Near E-Plot_vertical_cut** and **Near E-Plot_ horizontal** fields are overlaid on the model:

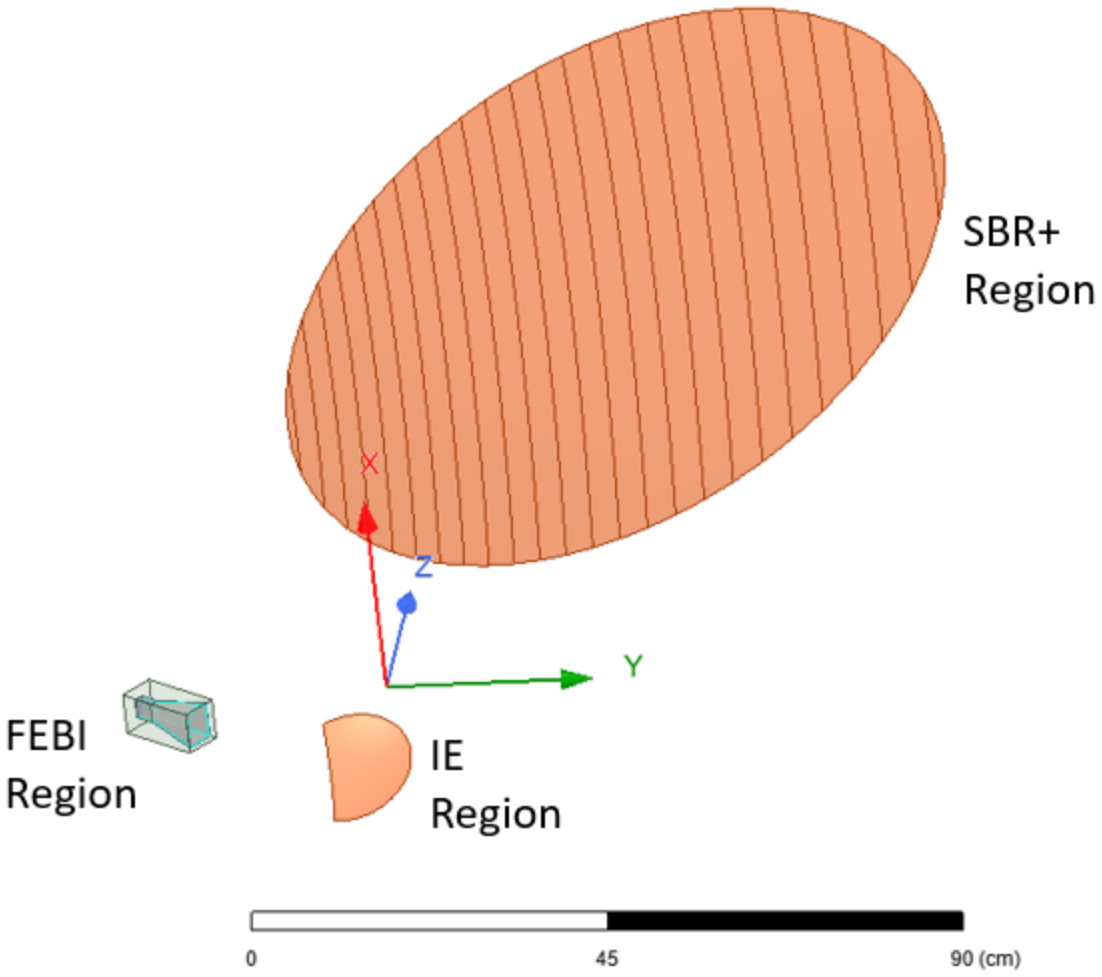


Add the legend by right-clicking **Field Overlays** in the Project Manager and choosing **Plot Sampled Near Field** from the shortcut menu. Finally, choose the **Near ETotal** and **dB** options.

HFSS SBR+ Hybrid Reflector Antenna Simulation Example

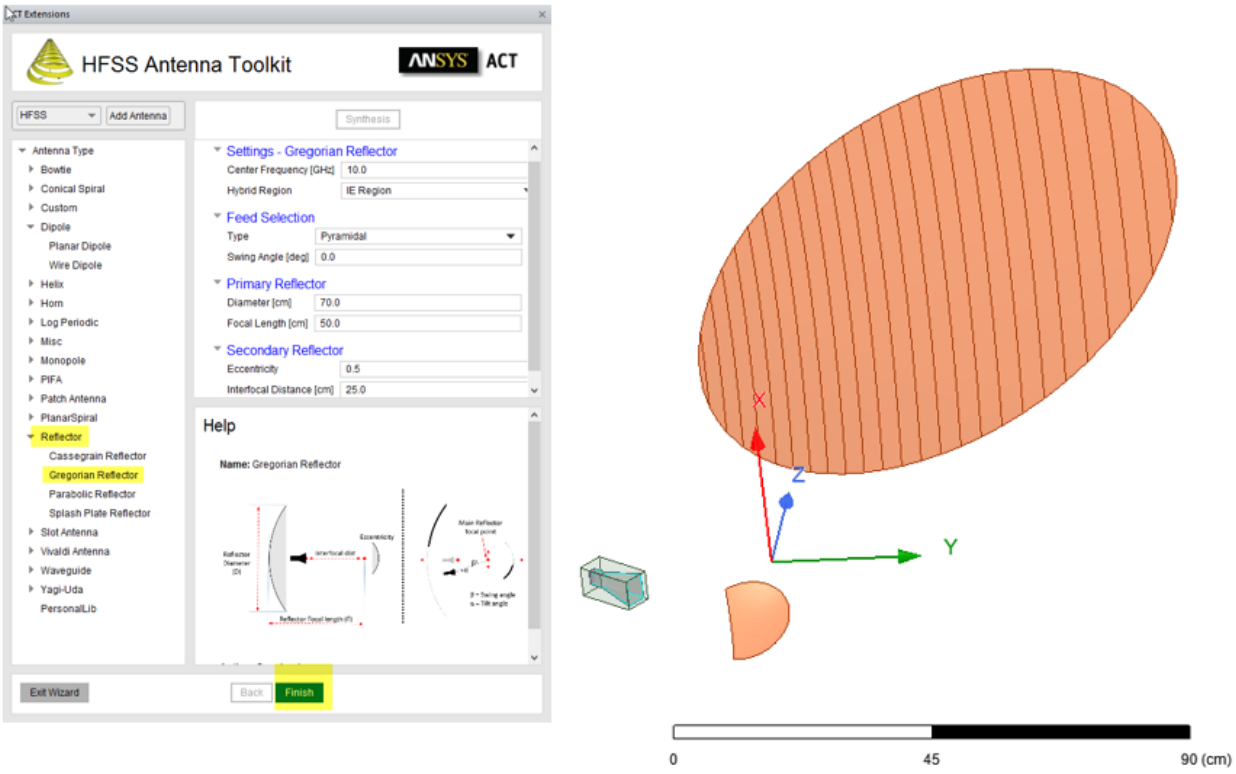
Description - HFSS SBR+ can provide significant time and memory savings for computing large reflector interaction. Most asymptotic solvers hybridize only to MoM codes. HFSS SBR+ hybridizes to both FEM and IE methods simultaneously. HFSS SBR+ can be hybridized to FEBI and IE Regions. FEBI or IE Regions may have sources, or be source-free regions. In this example, all full wave regions are first pre-computed, then applied to SBR+ Regions. All full wave regions (FEBI, IE) are considered independently. In practice, closely-spaced (near field) full wave regions should be grouped so that their 2-way couplings are considered prior to hybridization to SBR+ Region. Results exhibit comparable accuracy to IE simulations.

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



Model - The model is a 10 GHz Gregorian reflector system created with the Antenna Design Toolkit that is embedded with HFSS in the Ansys Electronics Desktop. The horn is set to FEBI, and the reflectors are set to IE Hybrid Regions. The largest reflector is changed to SBR+ Hybrid Region.

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



Full-wave meshing should be used for reflectors close to feed apertures.

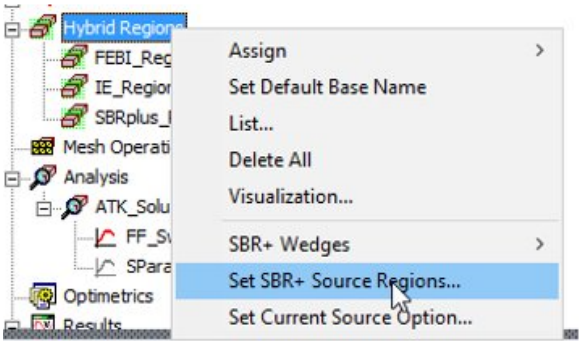
- Horn: HFSS FEM (FEBI boundary)
- Subreflector: IE Region

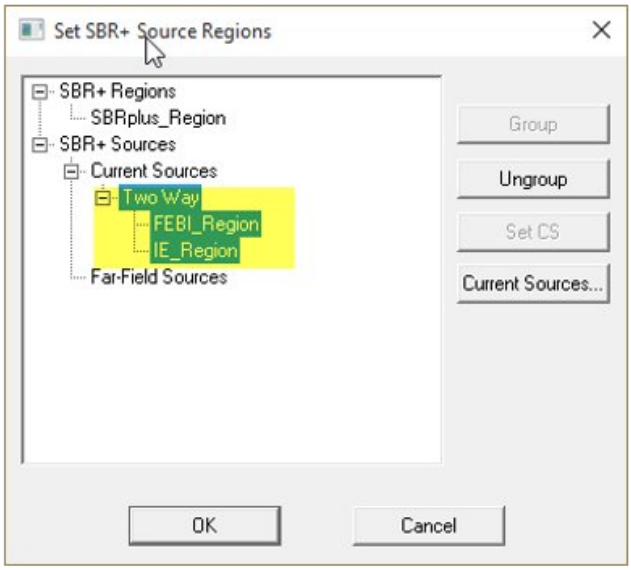
Large reflector can use HFSS SBR+ to speed up simulation dramatically

Set Two-way Interactions for full wave hybrid regions- •By default, full-wave regions will operate on the SBR+ Region independently. We want the FEBI and IE Regions to be simulated as coupled *before* applying them together to the SBR+ Region.

- Right-click on “Hybrid Regions” to select the **Set SBR+ Source Region...**
- Make sure the FEBI and IE Regions are grouped under “Two Way” sources

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

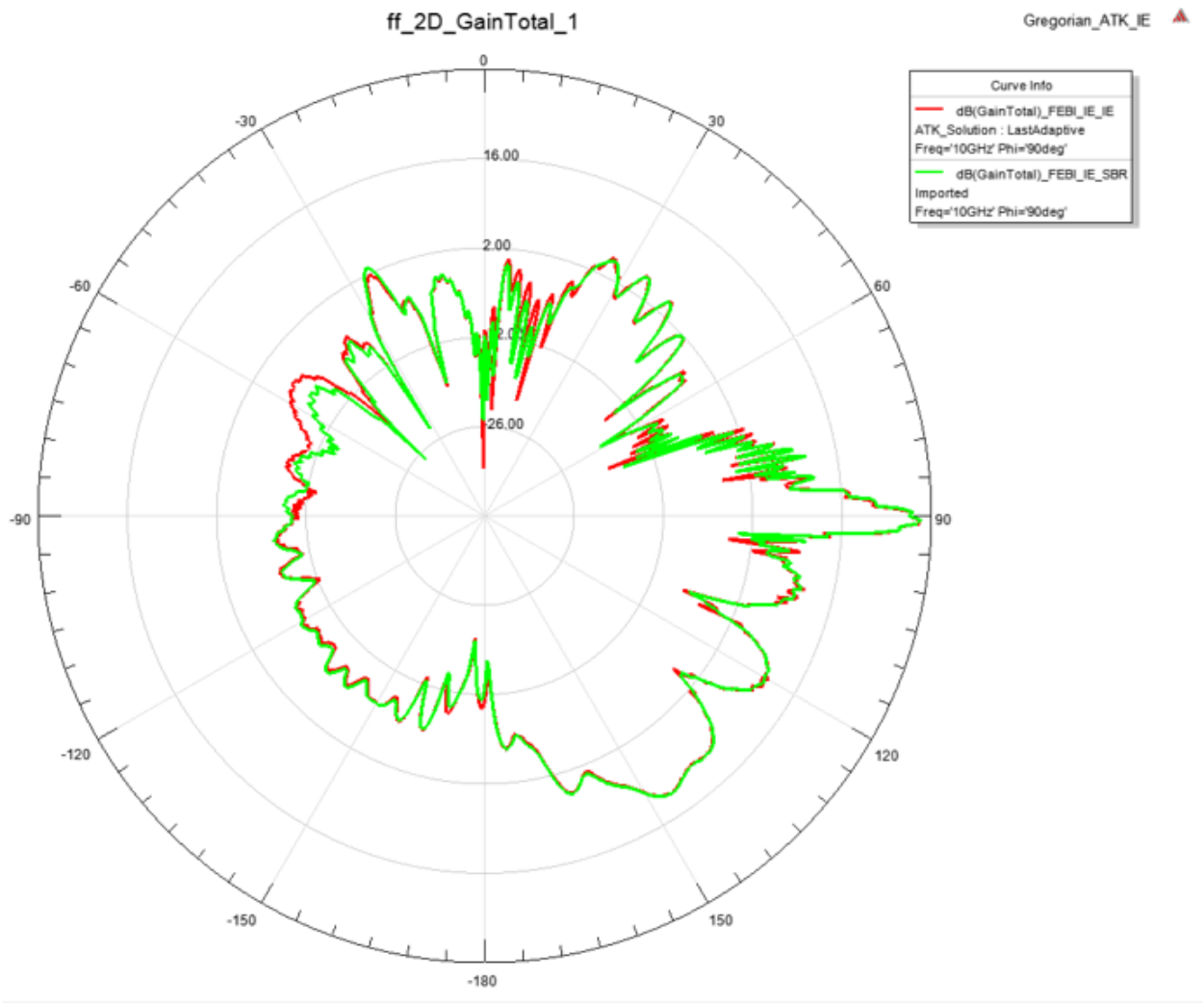




Results: If you compare a FEBI + IE Regions simulation with a Hybrid FEBI + IE + SBR+, there is excellent agreement. Using SBR+ reduces the overall simulation time (including FEB/FEBI and IE meshing by 63%.

- Reduction grows exponentially with electrical size of the problem.
- SBR+ makes it practical to simulate installed sidelobes for antenna system and its host platform (satellite payload, etc.)

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



Parabolic Dish with Horn

Abstract

A horn antenna with a parabolic dish for studying parametric placement of the horn along the dish's directional axis. This example also demonstrates usage of the mesh assembly option.

Solution Time: 14 cores in version 2025 R1 (meshing, adaptive solution, and parametric study)

- **System_IE:** 40 minutes: 6 seconds, 4.31 GB max. memory per process
- **System_PO:** 11 minutes: 50 seconds, 1.44 GB max. memory per process

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

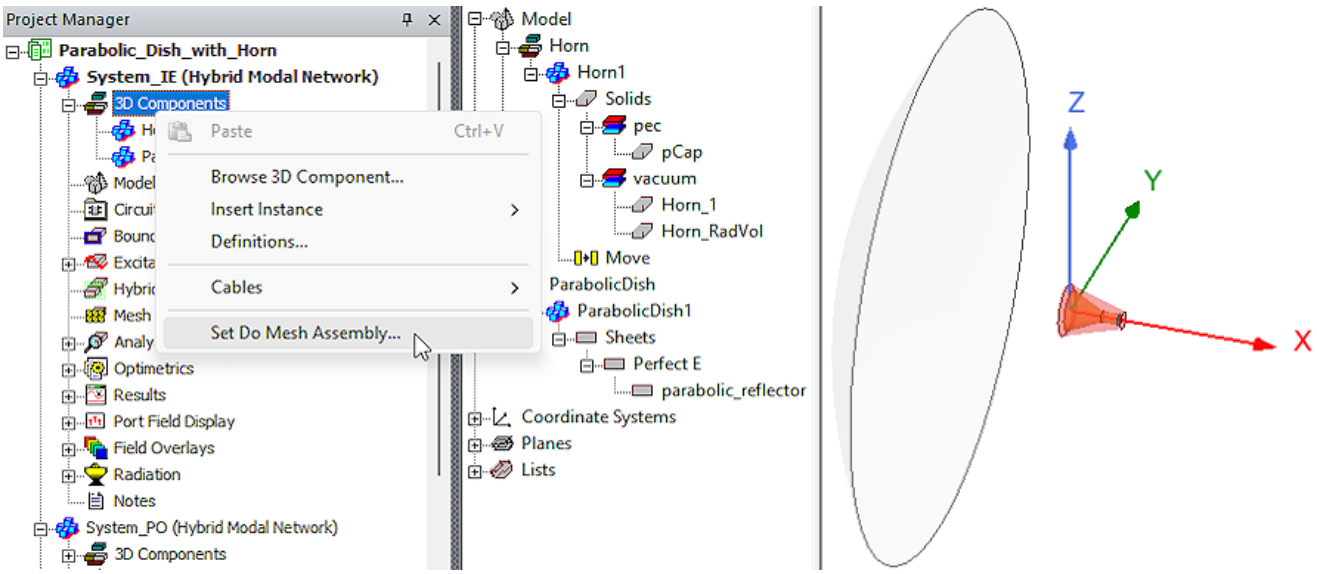
> **Note:**
>
> Simulation time and memory will vary depending on your available resources. In most cases, HPC (parallel resources) can be leveraged to significantly reduce simulation time and the memory footprint required by each CPU.

Description:

This example shows an electrically large problem for the placement study of a horn antenna illuminating a dish when the antenna feed is moved along the X-axis. The horn and dish are defined as 3D Components to enable Mesh Assembly. Two projects are included for comparison, **System_IE** and **System_PO**, utilizing either an IE or PO region for the reflector. For such designs, meshing the horn and the dish separately makes the simulation faster since only the currents on the dish and on the radiation boundary of the horn need to be modeled. Meshing the volume of the region between the horn and the dish, to model coupling occurring due to the currents on the dish and the radiation volume bounding the horn, is not required. Additionally, with mesh assembly, you can reuse the meshes of the individual components for performing placement studies.

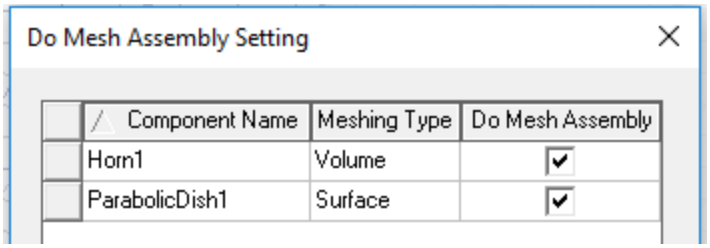
Simulation Summary:

For the best initial view of the model, choose the **Trimetric** view orientation rather than the default Isometric view. The curvature of the dish is more apparent from this viewpoint.



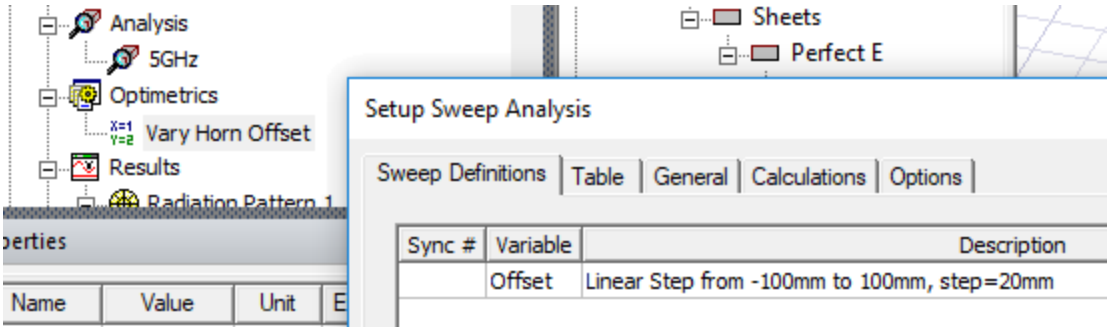
The Parabolic Dish is 2 meters in diameter and is defined as a 3D Component with an IE or PO Hybrid Region assigned. The Horn is defined as a 3D Component with an FE-BI Region assigned. Both designs have the component option to **Do Mesh Assembly** enabled:

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



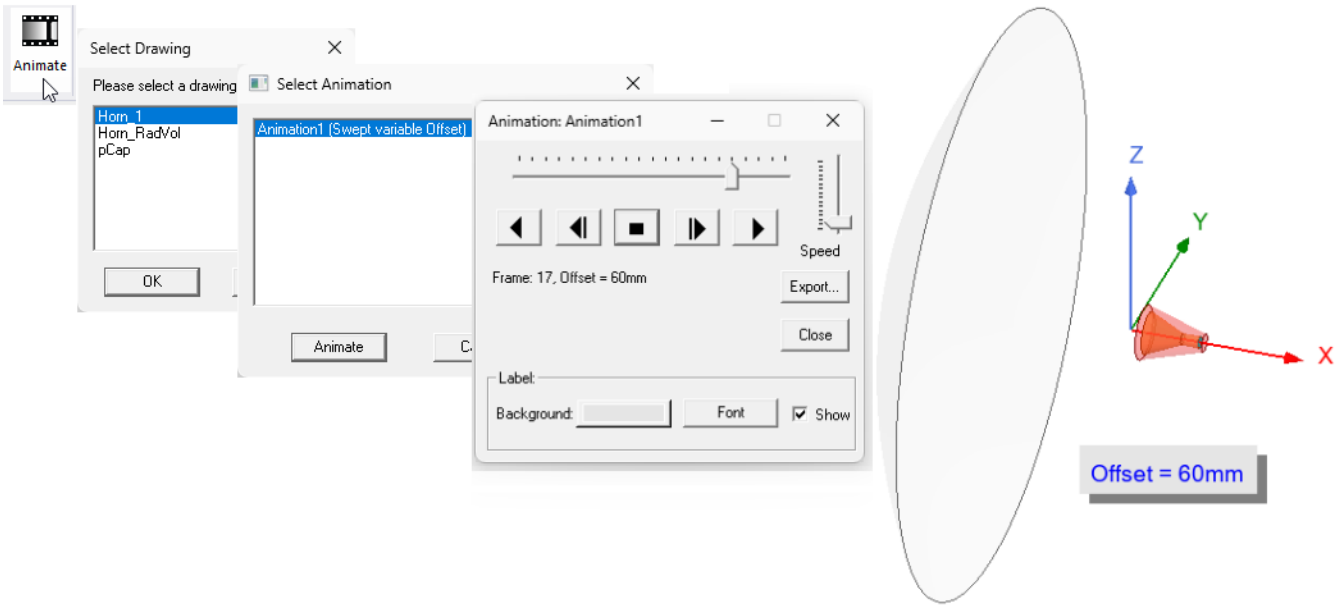
Solution Setup:

The adaptive solution frequency is 5 GHz. A design variable, *Offset*, is defined to vary the placement of the horn along the X axis for the Parametric Sweep setup ("*Vary Horn Offset*"):



Before solving a Parametric Sweep, you can use the Parametric Animation feature to see an animation of all the geometric variations that will be solved. Select the **View** tab of the Ribbon and click **Animate**. Then, in the *Select Drawing* dialog box, select **Horn_1** and click **OK**. An animation is predefined with *Offset* defined as the swept variable. Click **Animate** to generate the frames and see the range of motion of the horn.

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



Solve both designs, one at a time, before proceeding to the postprocessing section. In the Project Manager, right-click on the design name and choose **Analyze All** from the shortcut menu to mesh the model, run the adaptive solution, the frequency sweep, and the parametric sweep.

Postprocessing

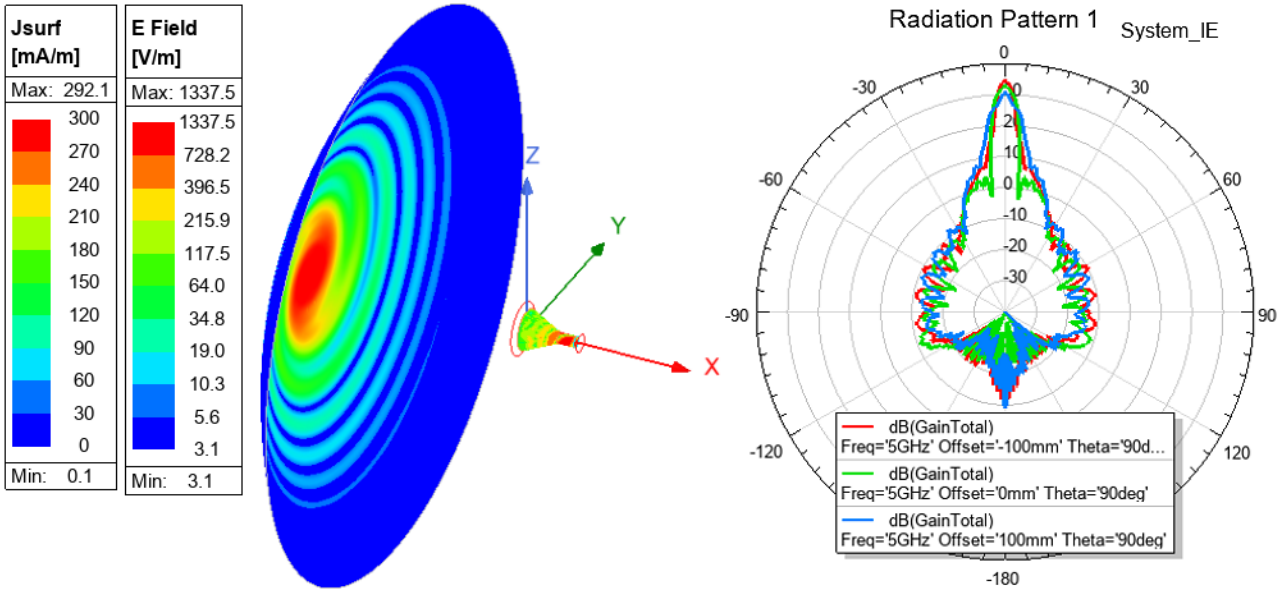
The example model includes two predefined plots and three field overlays per design, as listed below:

- **Radiation Pattern 1:** Total Gain in dB for Offsets of -100, 0, and 100 mm
- **Realized Gain Plot 1:** Realized Total Gain in dB at Nominal Offset (0 mm)
- **E-Field Magnitude Overlay** (Horn)
- **Jsurf Magnitude Overlay** (Dish)
- **Mesh Overlay** (Dish)

You might want to switch to the default Isometric view for reviewing the fields overlays. Alternatively, rotate the Trimetric view slightly about the Z axis for a better view of the overlay on the concave dish surface. You can constrain rotation to be about the Z axis by pressing and holding the Z key while clicking and dragging using the middle mouse button. Move the mouse in a small arc around the center of the Modeler window. The model viewpoint will rotate about the selected axis the same number of degrees as the arc along which you drag the mouse.

The following image shows the Jsurf overlay on the Dish, the E-Field Overlay on Horn, and the Radiation Pattern plot for the **System_IE** design:

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



Double-click the predefined plots in the Program Manager's *Results* branches and the overlays in the *Field Overlays* branches to see them and to compare results between the **System_IE** and **System_PO** designs. For the field overlays, click and drag the E-Field legend and place it to the right of the Jsurf legend so that both overlays can be viewed simultaneously.

Johanson 2450AT18B100-EB1SMA

Abstract

This example is one of two Johanson Technology chip antenna and evaluation board (EVB) models that you can find in the **Examples\HFSS\Antennas** subfolder included in the Ansys Electromagnetics Suite installation. This model demonstrates the following:

- The usage of Johanson 3D Components
- The usage of lumped RLC boundaries to represent discrete tuning components
- Several types of plots and overlays

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

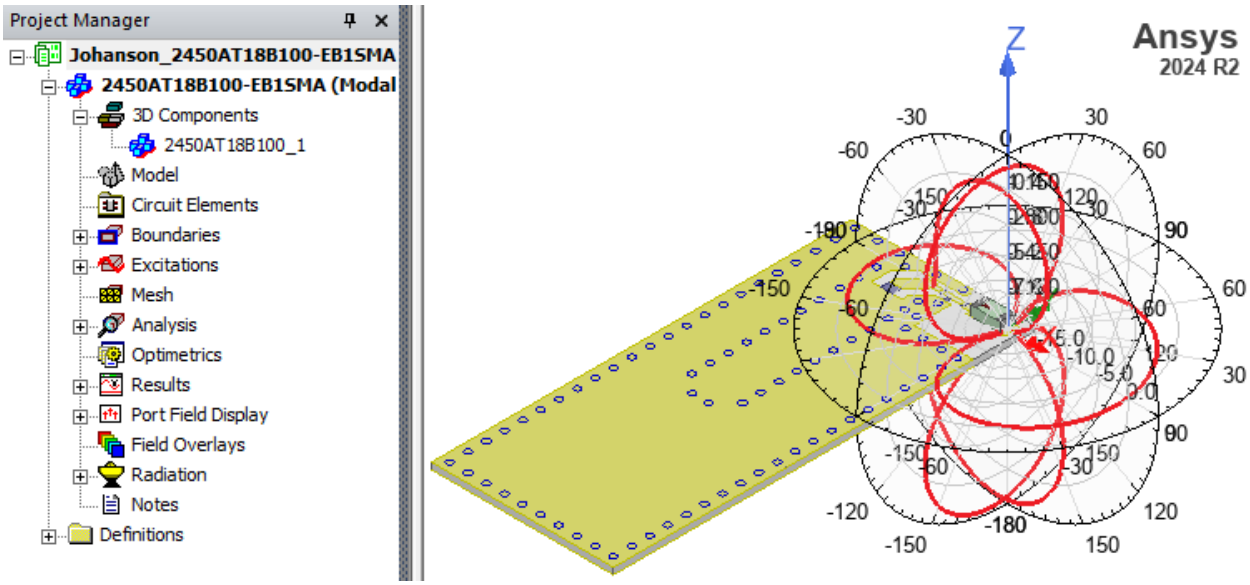


Figure 1: Johanson Chip Antenna and EVB with Overlaid Gain Plots

Version: 2024 R2

Simulation Time: 2 minutes: 20 seconds (using 8 cores), less than 1.4 GB RAM used

Model and Setup Details

The model consists of a single chip antenna mounted to an EVB – Johanson Technology, legacy part number: 250AT18B100-EB1SMA. The antenna is a vendor model included as an encrypted 3D component. It is excited by a lumped port, includes a matching circuit, and is contained within an open region with a radiation boundary, as detailed below. The matching circuit consists of three components represented using lumped RLC boundaries—two inductive and one capacitive. The capacitor and one inductor are in series with each other and with the chip antenna. The other inductor is a shunt to ground from the series inductor-to-series capacitor junction. Their values are specified to tune the antenna for the desired performance characteristics and operating frequency.

Boundaries:

- Finite Conductivity boundaries (*FiniteCond1* and *FiniteCond2*), 58 x 10^6 Siemens/m, classic infinite thickness model, at the bottom and top faces of PCB, respectively (**HFSS** > **Boundaries** > **Assign** > **Finite Conductivity**)
- Lumped RLC boundaries (**HFSS** > **Boundaries** > **Assign** > **Lumped RLC**):
  - *LumpRLC1*: Inductor, 3.3 nH (series, adjacent to antenna)
  - *LumpRLC2*: Inductor, 2.7 nH (shunt)
  - *LumpRLC3*: Capacitor, 1.2 pF (series, adjacent to feed port)

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

- Radiation boundary (*Open1*) assigned to the vacuum region faces – the region's padding is 1.40707 inches in all directions, which is one-third of the wavelength at 2800 MHz (**HFSS** > **Model** > **Create Open Region**)

Excitations:

- Lumped Port (*1*), 50 Ω *Full Port Impedance*, single mode, *Renormalize all modes* selected, located at beginning of antenna feed (**HFSS** > **Excitations** > **Assign** > **Port** > **Lumped Port**)

Mesh Setting:

- No mesh operations
- Initial Mesh Settings: Mesh slider one tick coarser than default (to reduce the element count at the grounding vias):

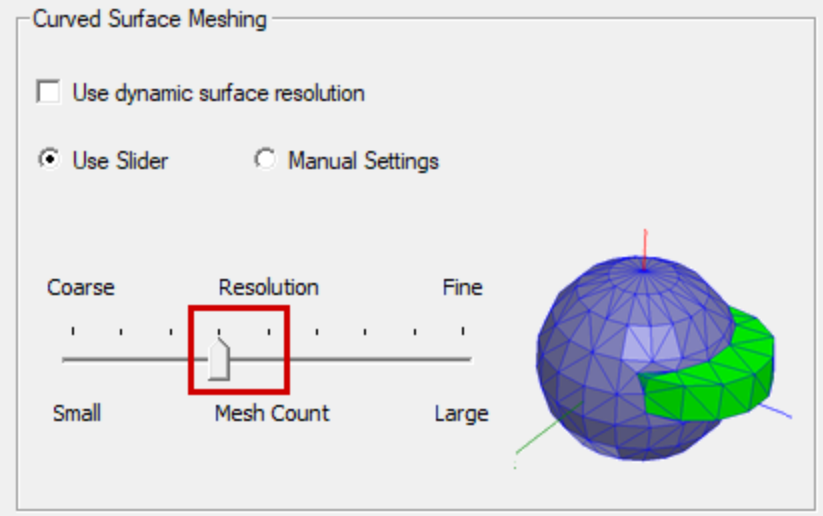


Figure 2: Initial Mesh Settings Dialog Box, Curved Surface Meshing Section

Setup:

- 2800 MHz single solution frequency, *Do Lamda Refinement* enabled, 30% max refinement per pass, *Save Fields* enabled
- 1 – 4 GHz sweep in 0.001 GHz steps, 250 maximum solutions, 0.5% error tolerance

Postprocessing

After solving (**Simulation** > **Analyze all**), you can view different postprocessing results. Look in the Project Manager under *Results* and double-click on the different predefined reports (**Figure 3**):

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

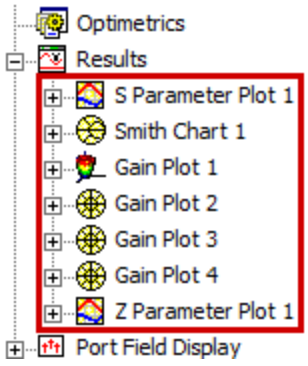


Figure 3: Predefined Reports Listed in the Project Manager

Predefined Reports:

Figures 4 through 10, which follow, show each of the seven predefined reports in the example project:

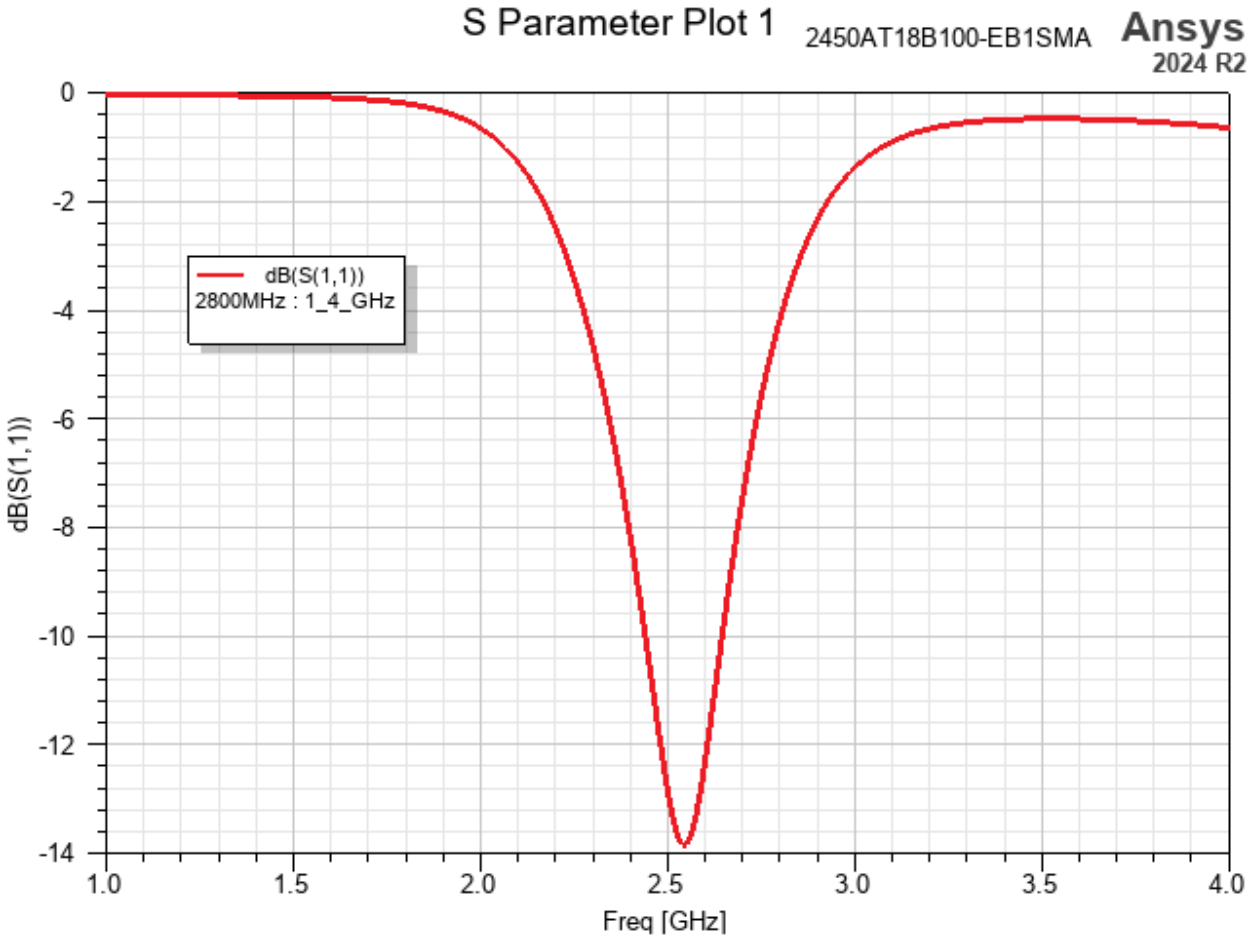


Figure 4: S Parameter Plot 1 (dB)

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

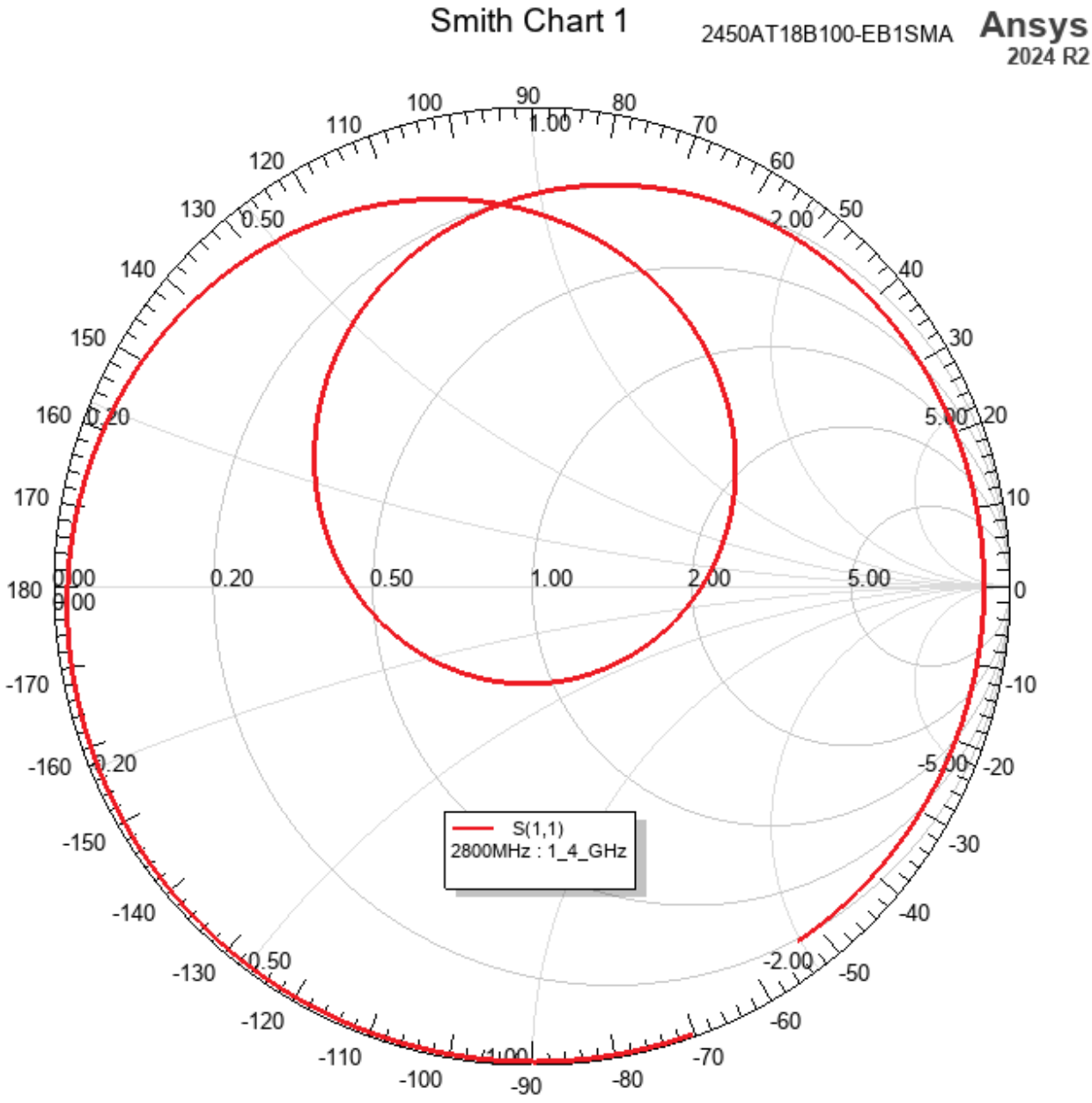


Figure 5: Smith Chart 1

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

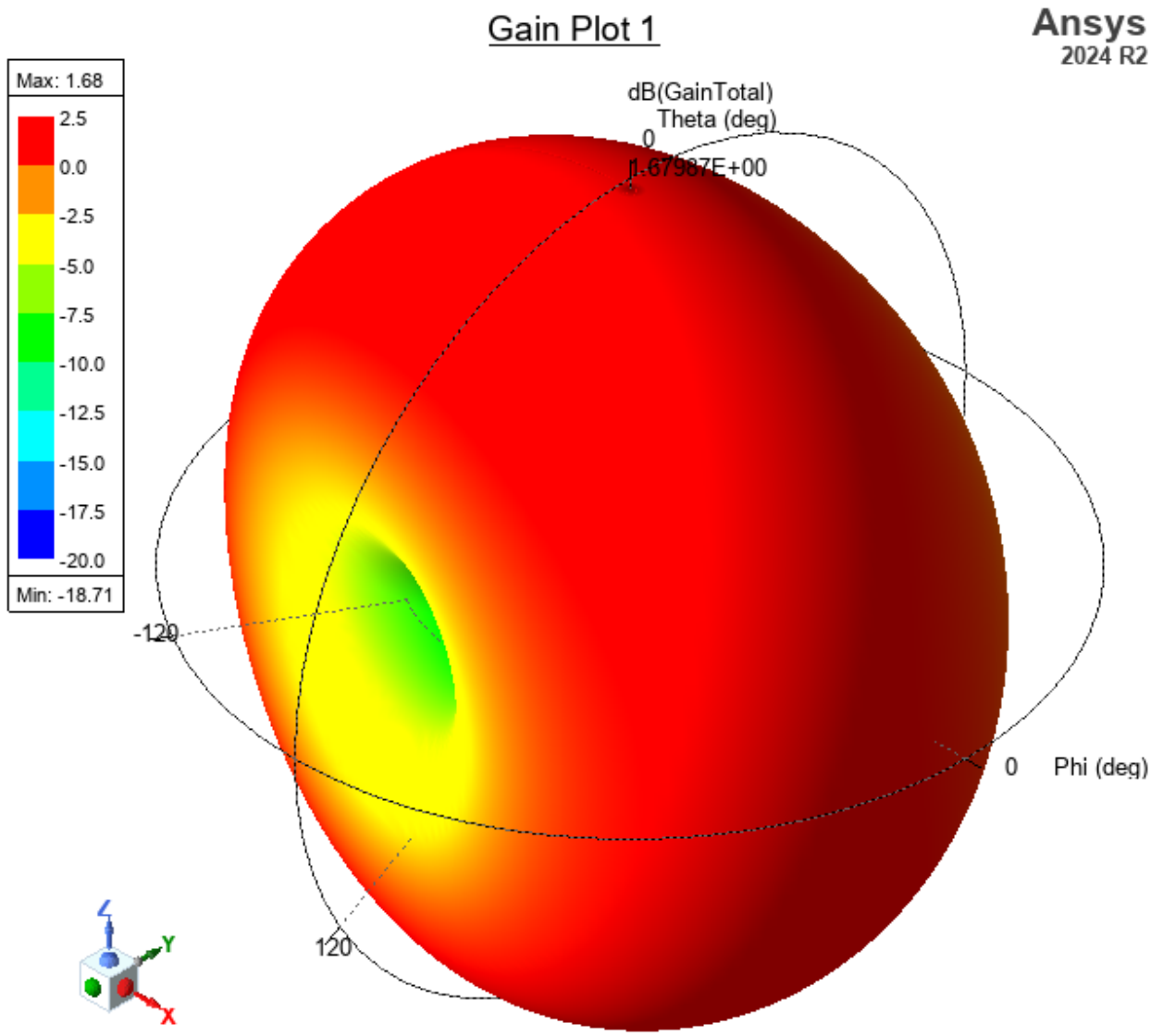


Figure 6: Gain Plot 1 – Total Gain (dB) vs. Phi and Theta

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

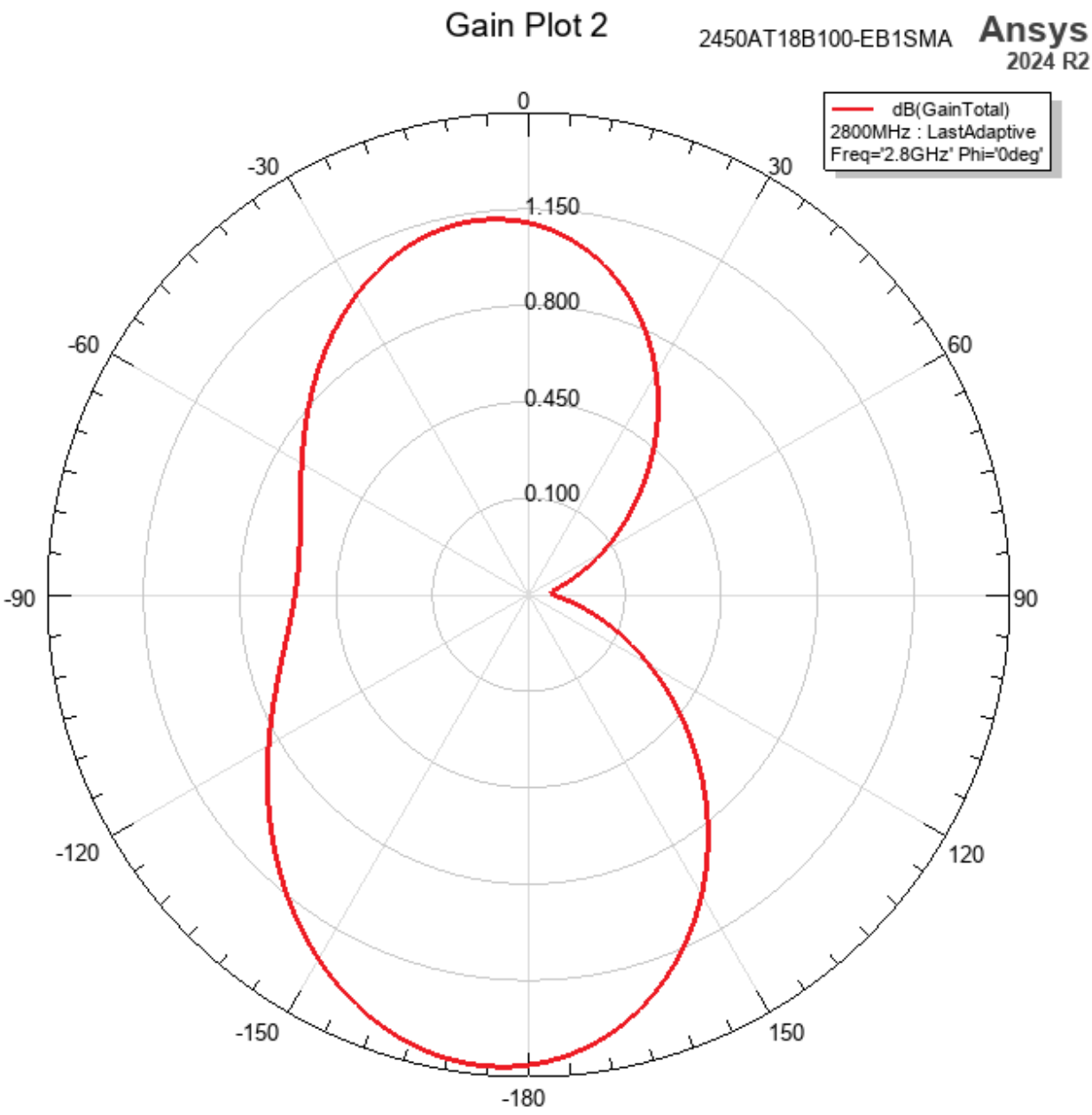


Figure 7: Gain Plot 2 – Total Gain (dB) vs. Theta, Phi = 0°

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

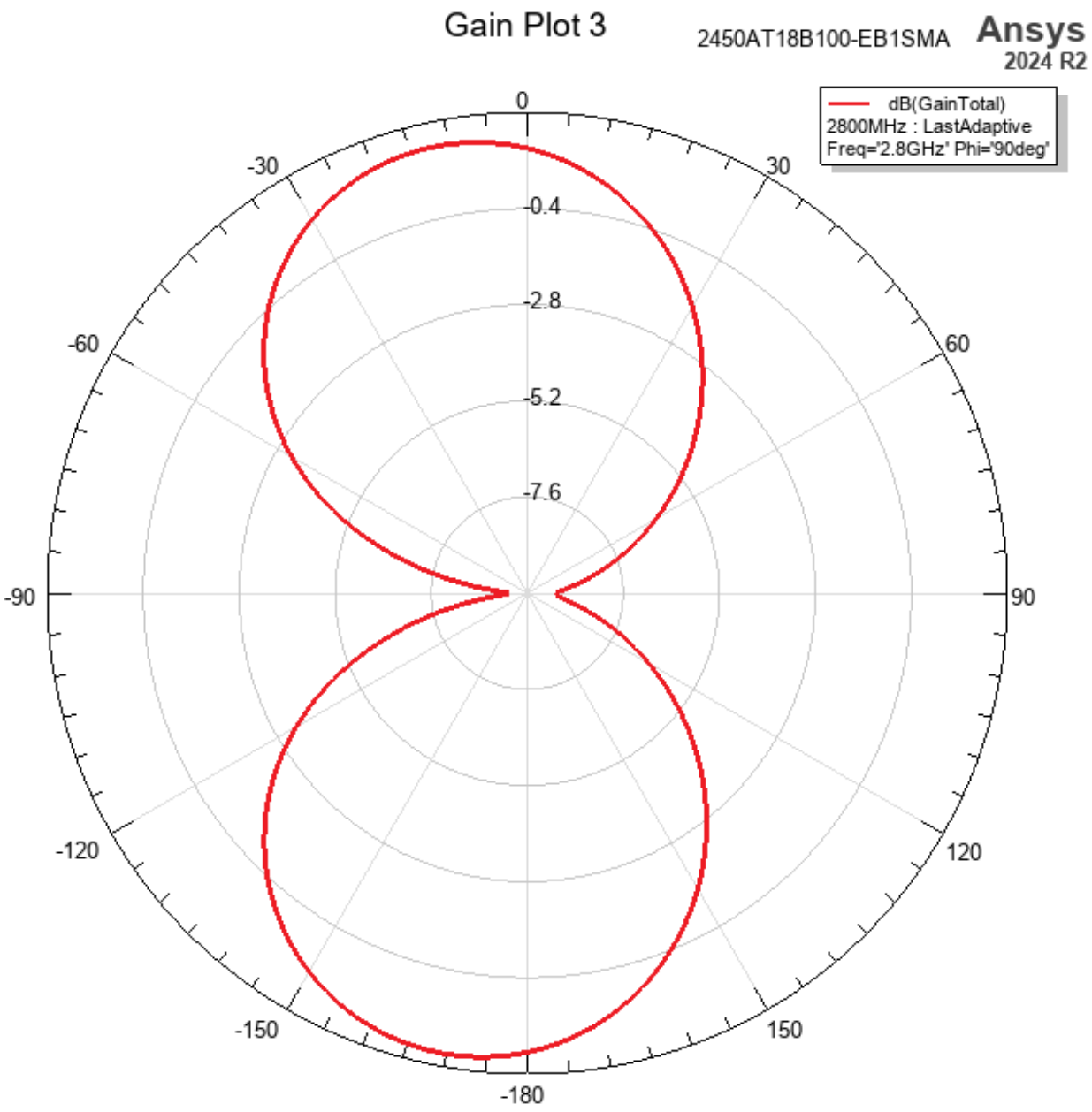


Figure 8: Gain Plot 3 – Total Gain (dB) vs. Theta, Phi = 90°

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

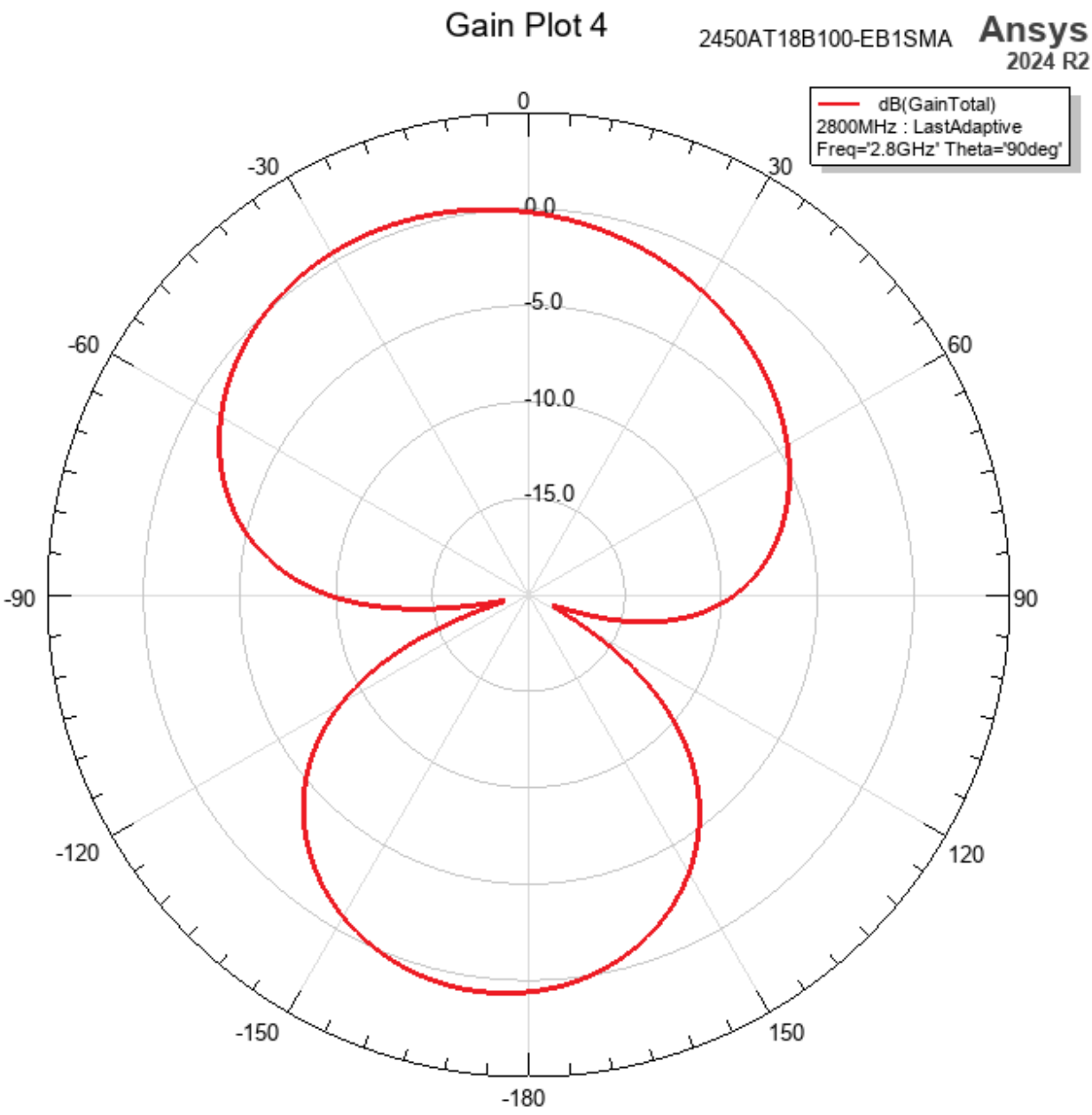


Figure 9: Gain Plot 4 – Total Gain (dB) vs. Phi, Theta = 90°

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

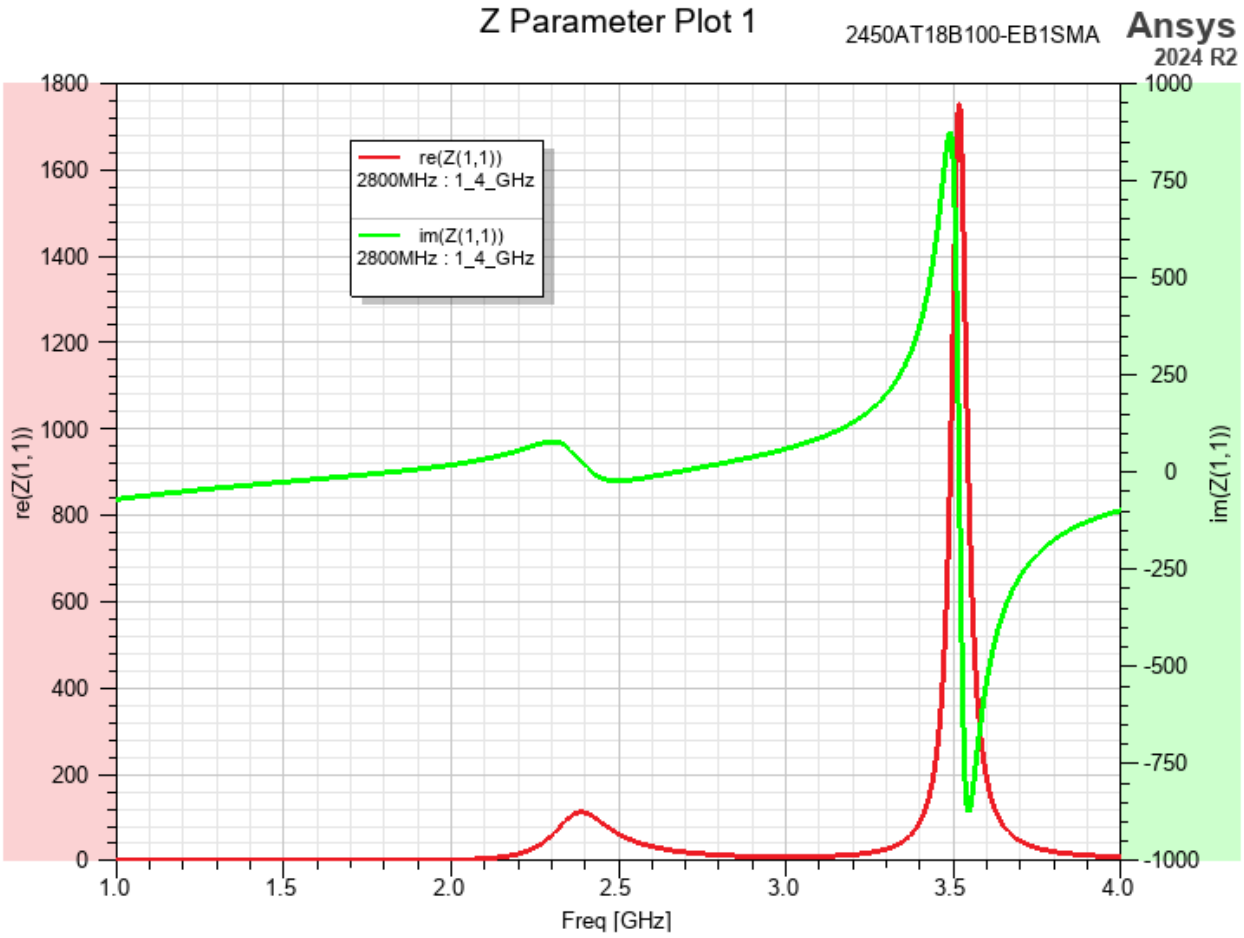


Figure 10: Z Parameter Plot 1 (real and imaginary)

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

Overlaying Reports on the Model Geometry:

To overlay any of the gain plots on the model geometry, right-click in the **Modeler** window and choose **Plot Fields** > **Radiation Field** from the shortcut menu. In the *Overlay Radiation Field* dialog box that appears, select the checkbox in the **Visible** column for one or more of the available gain plots and click **Apply**. Adjust the **Transparency** or **Scale** as desired and click **Apply** again. Click **Close** when finished. Figure 11 below shows the first gain plot overlay:

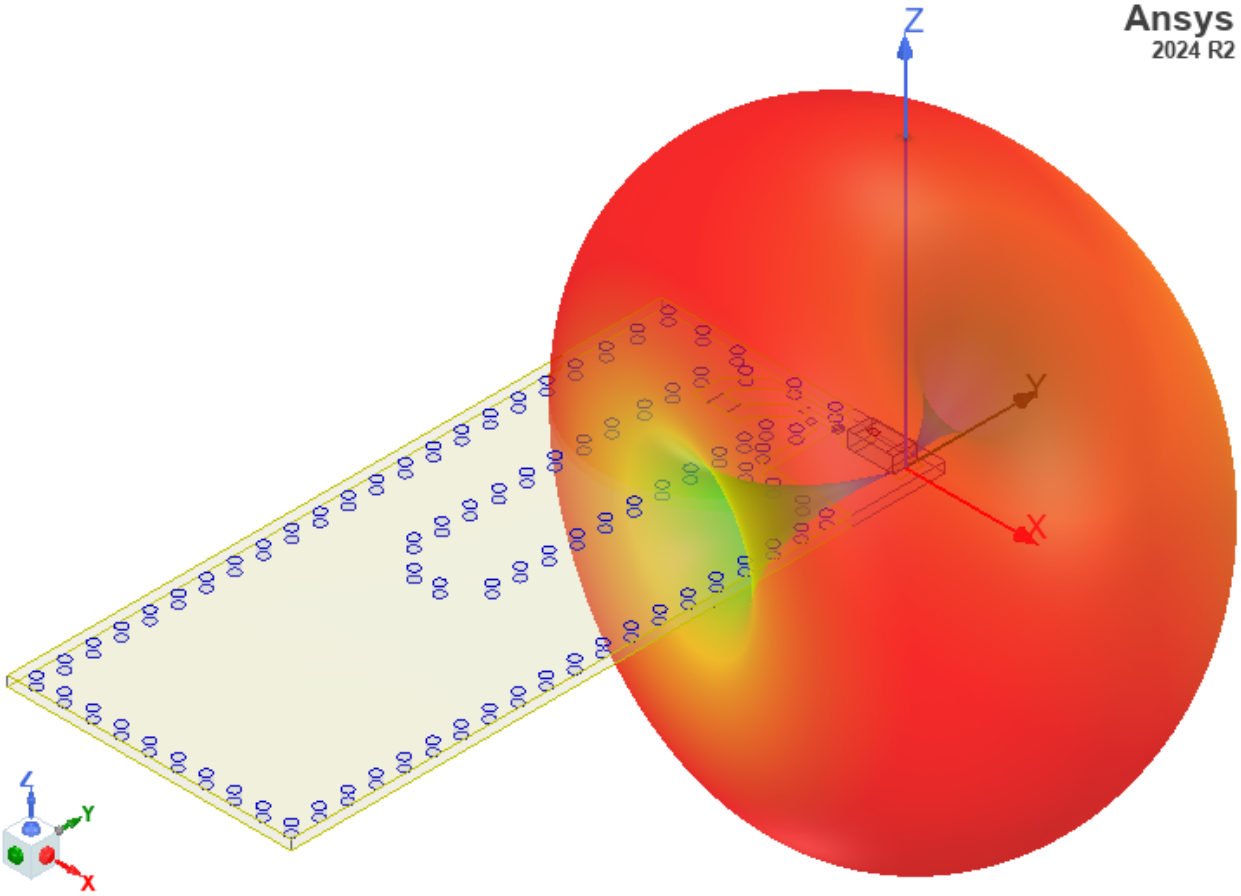


Figure 11: Gain Plot 1 Overlaid on the Model Geometry

Potter Horn Antenna

Abstract

This example shows how to model a Potter Horn Antenna [1]. One design uses an FE-BI boundary, and another uses PML for its open region.

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

The horn is parametrized to facilitate the simulation of different geometry suited for various desired frequencies and gains. A frequency sweep from 28Ghz to 32Ghz is defined.

The model, *Potter Horn Antenna.aedt*, is available in the ...Examples/HFSS/Antennas folder under the Ansys Electronics Desktop software installation path. A PDF version of this topic is included with the project file in the same location.

Simulation Time: (meshing, adaptive solution, frequency sweep, and optimization in 2025 R1 using 6 cores and automatic HPC settings)

- **Horn_Param_FEBI:** 1 minute: 54 seconds, 791 MB max. memory per process
- **Horn_Param_PML:** 1 minute: 49 seconds, 1.67 GB max. memory per process

Model and Setup Details

The Potter Horn is a dual-mode feedhorn, which provides an excellent radiation pattern with suppressed sidelobes (at least 30 dB) and symmetrical beamwidths. The port at the small end of the model is excited at two modes, the fundamental TE_{11} mode and the TM_{11} second mode. The waves travel through a cylindrical guide (**A** in **Figure 1**) followed by taper (**B**). There is a step increase in diameter at the end of taper B. The two modes reach this step discontinuity with a relative amplitude and relative phase angle. The relative amplitudes of the two modes are controlled by the step, which converts a small percentage of the incident power from the TE_{11} mode to the TM_{11} mode, causing the E-Plane sidelobes to cancel. As the waves propagate through the phasing section (**C**), the relative phase angle is reduced prior to entering the flare (**D**). The length of section C is chosen so that the TE_{11} and TM_{11} modes are in phase when they reach the aperture.

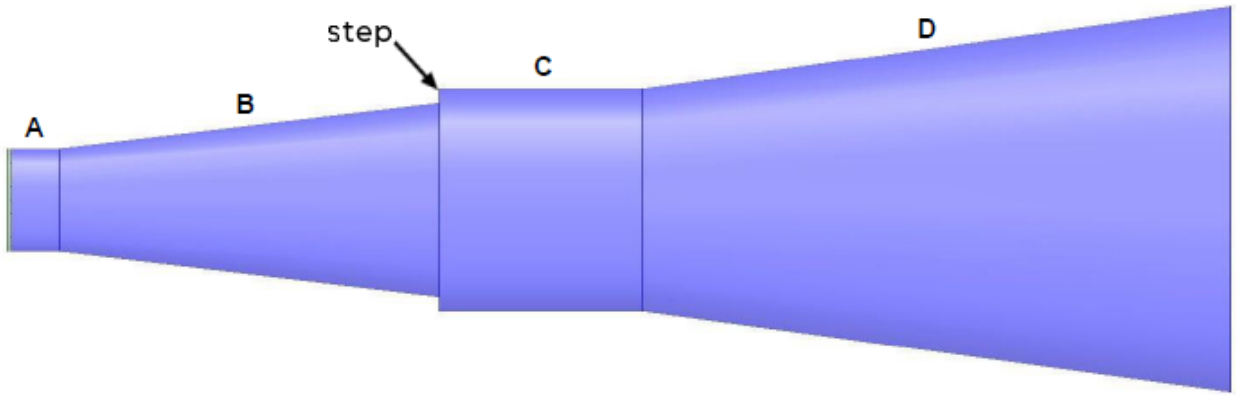


Figure 1: Potter Horn Diagram

The horn has w*λc wall thickness, and it is defined as aluminum. The port is internal to the solution region and is capped by a PEC object.

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

Details about the geometry and parameters can be seen in the History Tree and via **HFSS** > **Design Properties** (see Figure 2). This Potter Horn example has a very small flare angle (approximately 0.42°), as dictated by variables rC and rD:

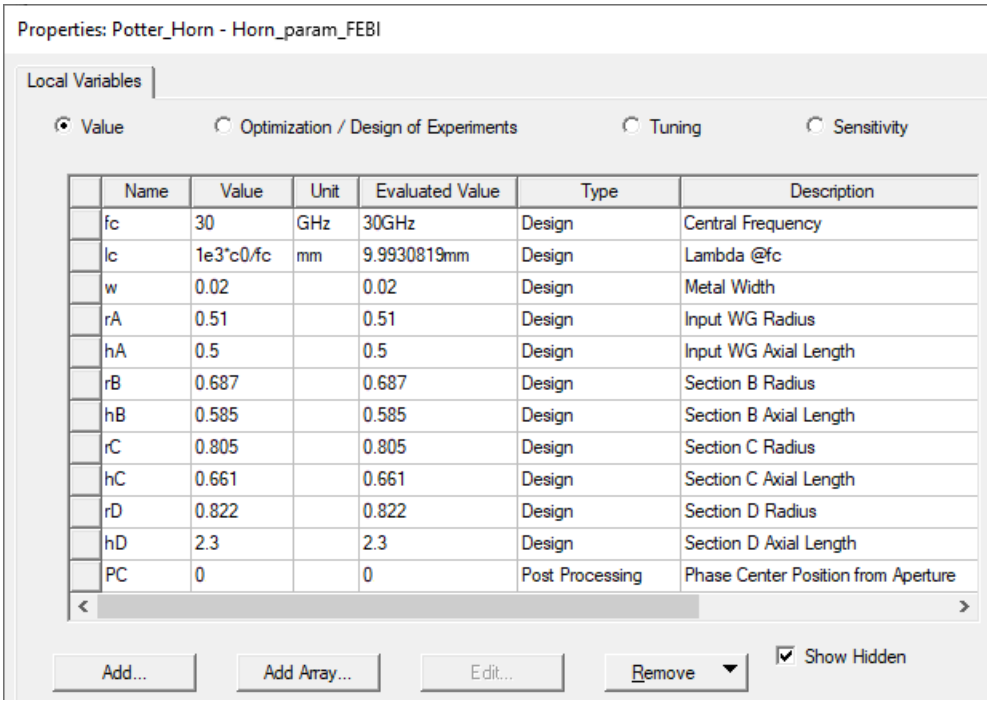


Figure 2: Horn Parameters

A Post Processing variable (PC) is defined to vary the position of the Far-field radiation coordinate system (Aperture_PC) along the z-axis for the phase center location calculation. The value of PC, the bottom variable in Figure 2, will change once the optimization analysis is complete.

This horn has been analyzed for the next Directivity values: 13, 15 and 20 dBi. These designs start with an input waveguide radius, rA=0.51λc and input Axial Length, hA=0.5λc. Starting with this dimension, an optimization within Ansys Optimization tools, quickly produces a design very close to the desired gain that radiates with low cross-polarization and very symmetrical radiation pattern. **Table 1** shows Potter Horn Design Parameters (λc normalized):

Table 1: Parametric analysis of the Potter Horn

Gain (dBi)	hD	rD	hC	rC	hB	rB	X-Pol Boresight (dB)	10 dB Beamwidth (º)
13.0	2.3	0.822	0.661	0.805	0.585	0.687	-38	79.2
15.0	4.1	1.02	1.5	0.961	3.062	0.833	-36	60.2
20.0	6.2	1.963	2.076	1.117	4.057	0.982	-37.5	33.6

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

To set up any of these horns you only need to change the parameters described in **Figure 2** and change the analysis setup to the desired frequency sweep range and increment.

Figure 3 shows the analysis setup for the Potter Horn example (**HFSS** > **Analysis Setup** > **Add Solution Setup** > **Auto**). **Figure 4** shows the frequency sweep setup (**HFSS** > **Analysis Setup** > **Add Frequency Sweep**). And **Figure 5** shows the initial mesh settings that were specified (**HFSS** > **Mesh** > **Initial Mesh Settings**):

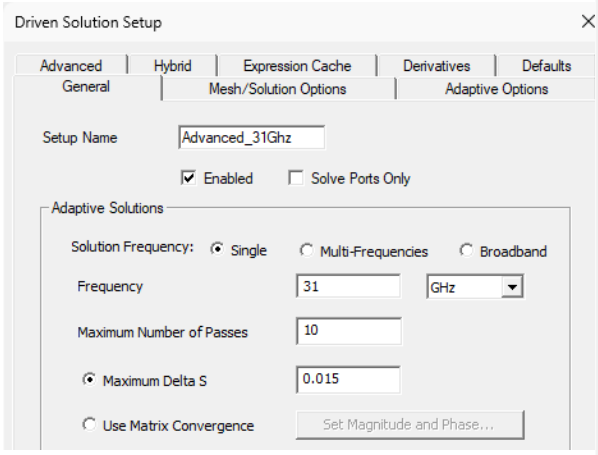


Figure 3: Analysis Setup

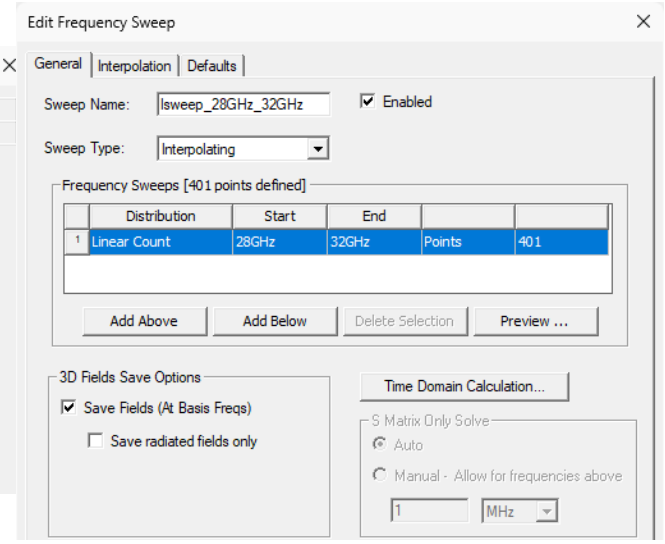


Figure 4: Frequency Sweep Setup

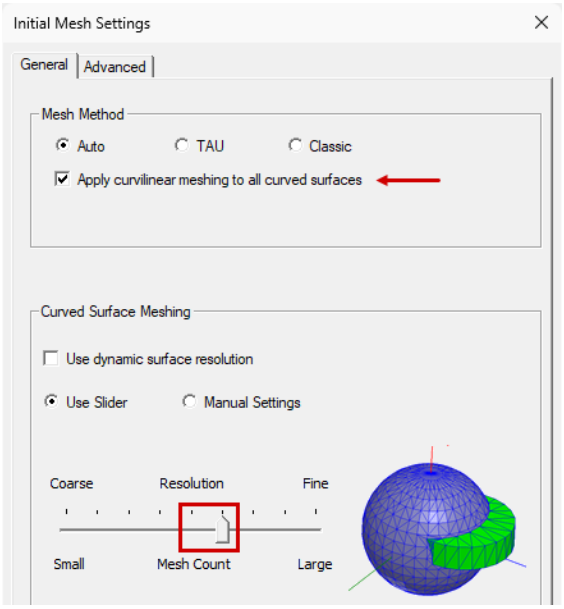


Figure 5: Initial Mesh Settings

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

Analysis Setup Notes:

- Either an FEBI- or PML-based absorbing boundary is a good choice for accurate directive antenna analysis.

  Both setups are automated. In the Modeler window, right-click and choose **Create Open Region** from the shortcut menu. Using PMLs is more efficient (faster) if multiple antennas or sources are present.

  FEBI is mainly used when combined with hybrid boundaries (IE, PO, SBR+) to model electrically large environments, such as the antenna platform or radome.

- Advanced/Single-frequency adaptive solution is used here because the design is narrow band.

  The convergence criterion for delta S is 0.015. This value is a bit lower than the default of 0.02, since the return loss was expected to be very low (mainly below -30dB), and the additional accuracy has a negligeable effect on simulation time.

- In the Initial Mesh Settings: The Curvilinear option and finer than default curved surface meshing is used for increased accuracy for critical curved surfaces.

  For highly complex geometry with many more curved faces (such as a non-simplified CAD model), curved surface meshing based on surface deviation can reduce the meshing complexity of non-critical and small sized curves.

- Mesh seeding on the face of the absorbing boundary (to approximately λ/5) increases the accuracy of the far field pattern computation for a directive antenna. Only the face in the direction of the main beam is selected.

  When only interested in S parameters or analyzing a less directive antenna, this mesh refinement is not needed.

Postprocessing

After solving (**Simulation** > **Analyze all**), you can view different post-processing results. Look in the Project Manager under *Results* and double-click on the different predefined results (**Figure 5**):

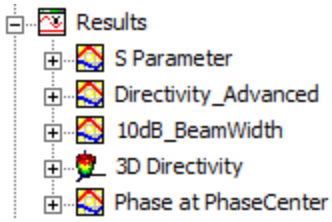


Figure 6: Post-Processing Results

In **Appendix 1** you will find the FEBI version's predefined results obtained for a 13 dBi horn at 31 GHz (the adaptive solution frequency). S Parameters are plotted for the specified sweep

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

frequency range of 28 to 32 GHz. Additionally, E-Field and 3D Directivity overlays are included. You can run the PML version and double-click the plots listed in the Project Manager for this version for comparison.

If you want to calculate the position of the phase center, please use the methodology defined in the Help Section: *Determining Phase Center Using Optimetrics*.

Once you simulate the horn alone you will be able to export it as a 3D Component (**Draw** > **3D Component Library** > **Create 3D Component**) and simulate it as the feed of a reflector or quasi-optical system.

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

Appendix 1:

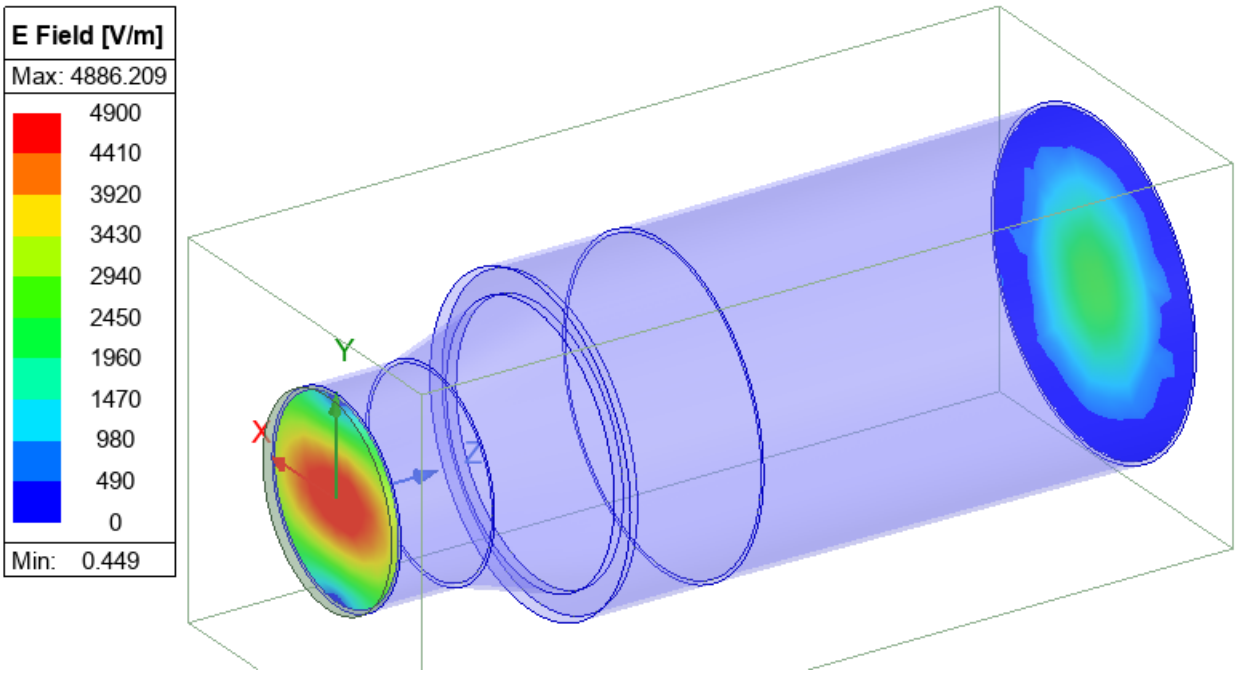


Figure 7: E-field @ Port 1 and Radiation Boundary (Aperture End) – FEBI Solution

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

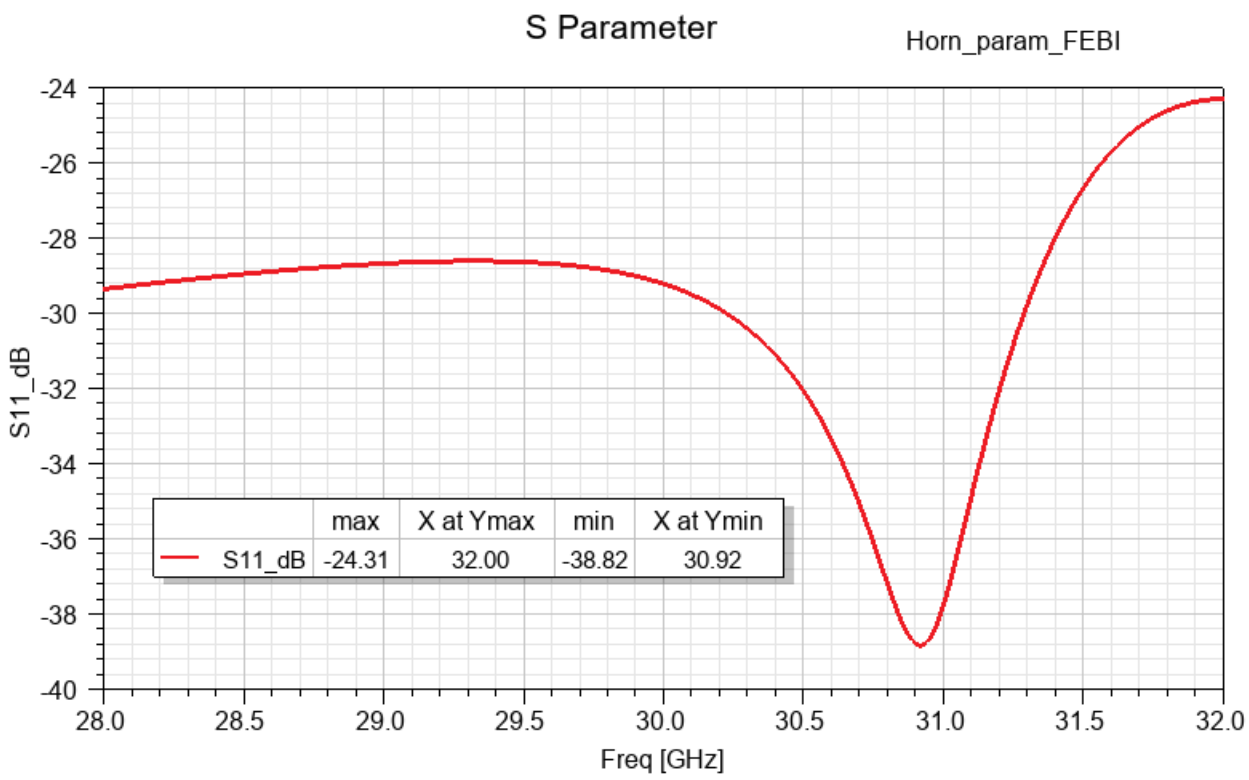


Figure 8: S Parameter vs. Frequency – FEBI Solution

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

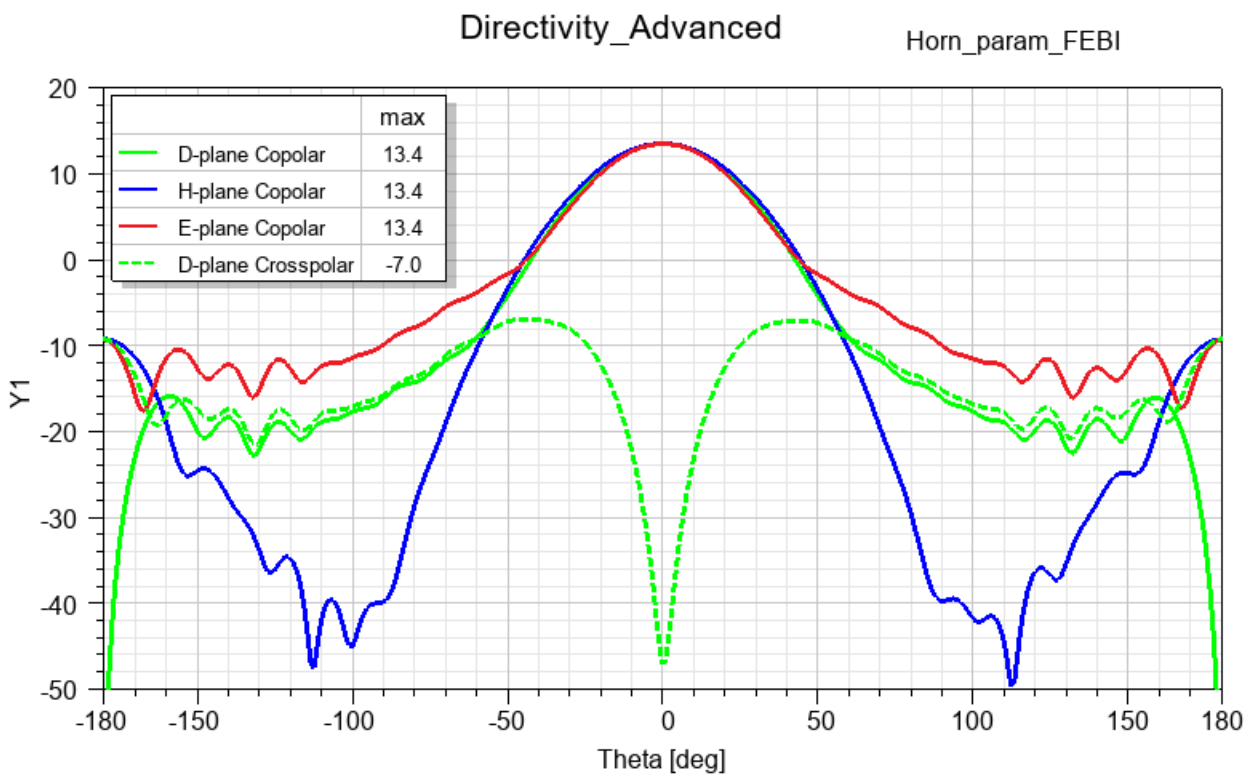


Figure 9: Directivity – FEBI Solution

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

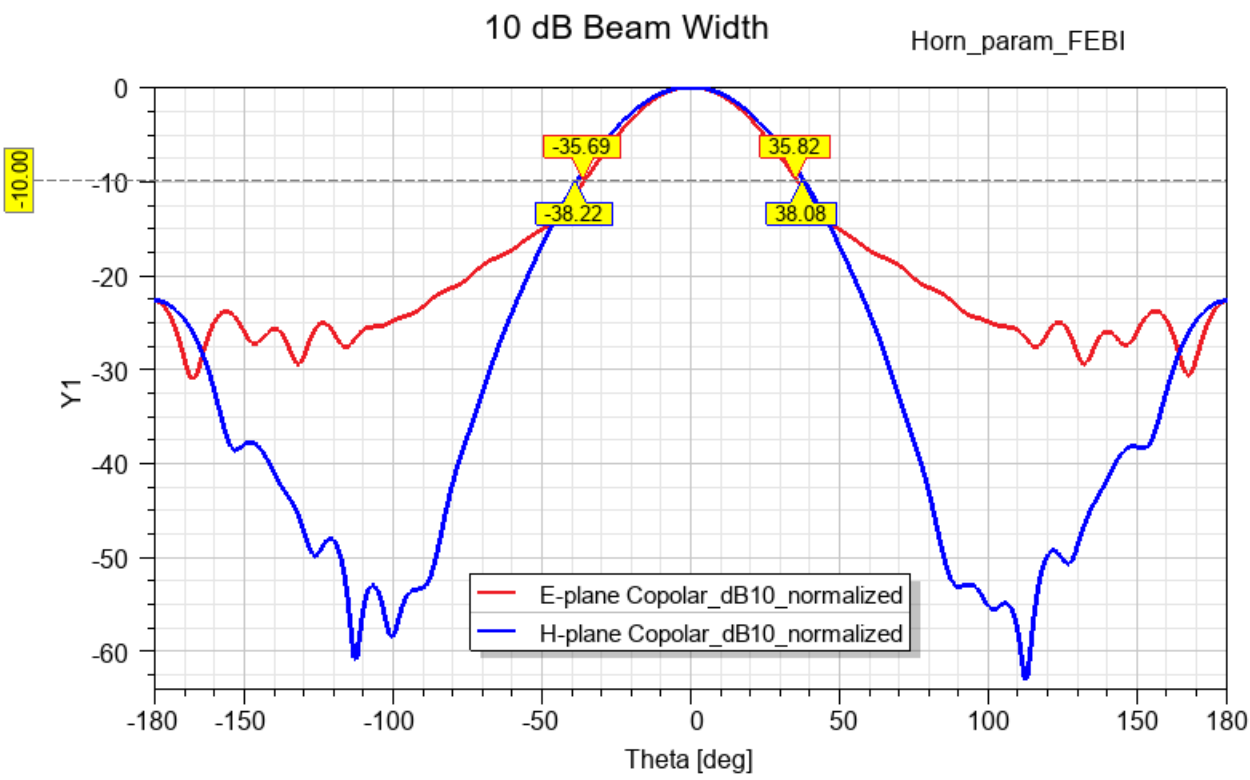


Figure 10: 10 dB Beam Width – FEBI Solution

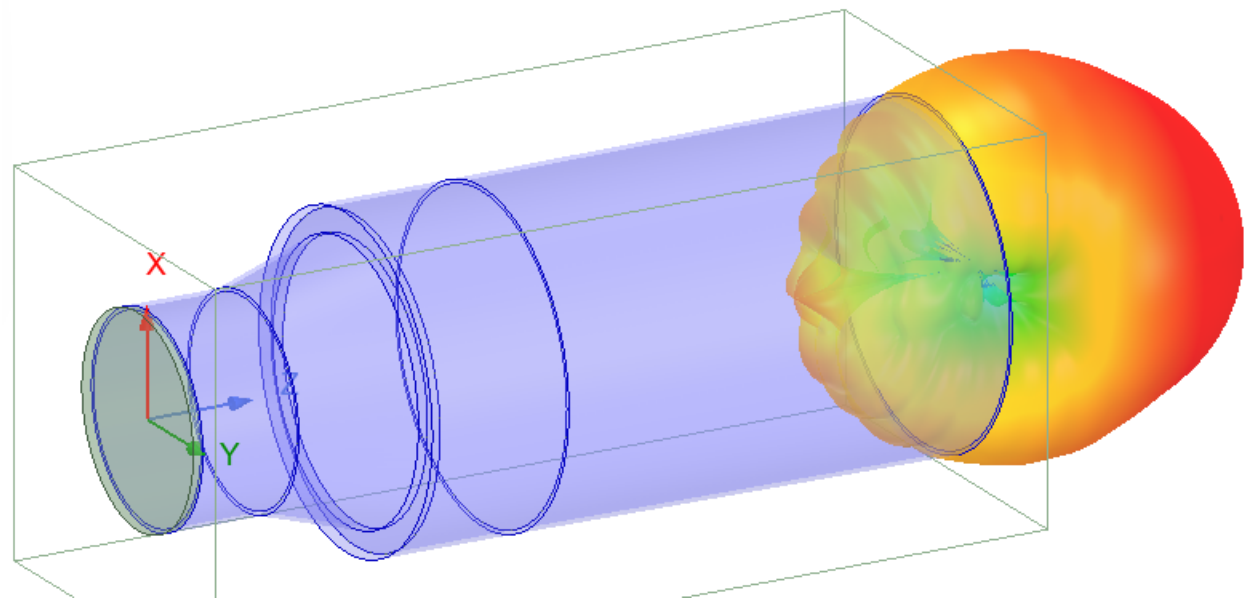


Figure 11: 3D Directivity Overlay – FEBI Solution

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

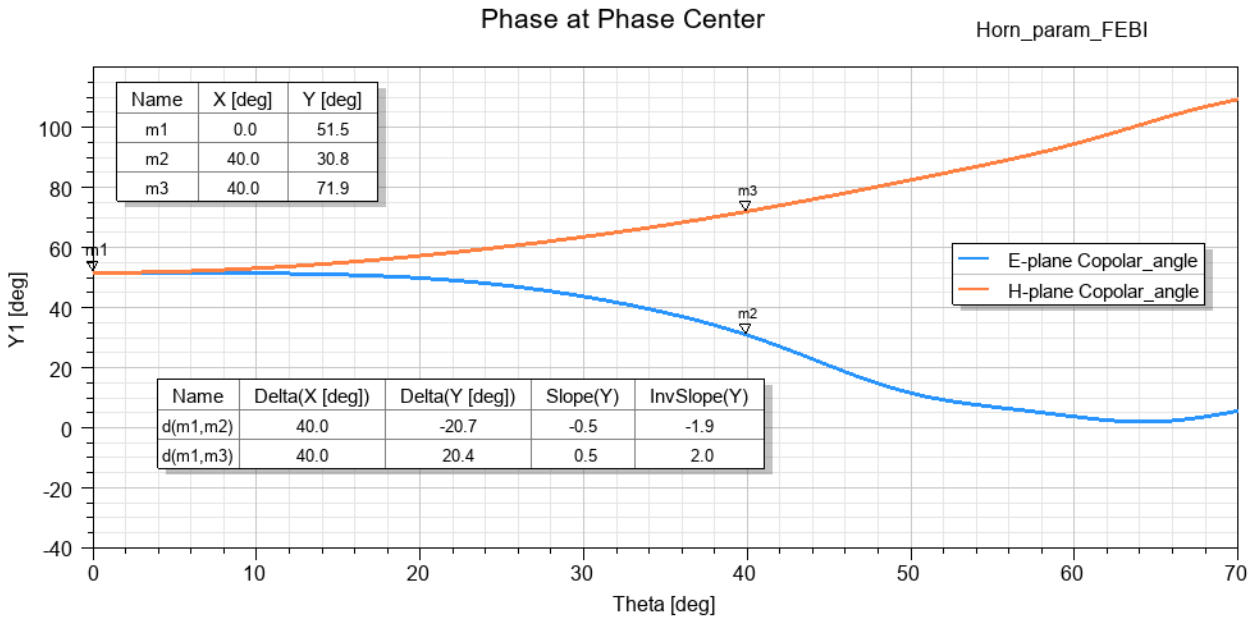


Figure 12: Phase at Phase Center – FEBI Solution

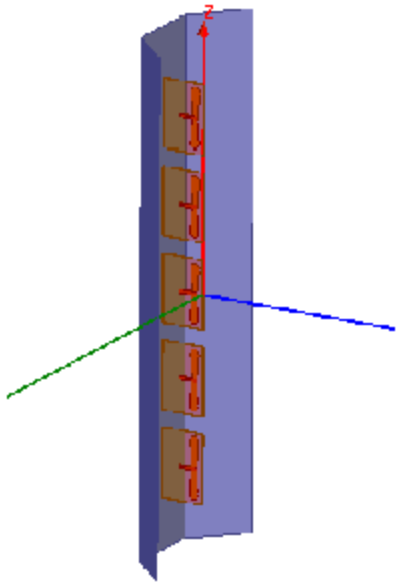
References:

[1] P. Potter, "A new horn antenna with suppressed sidelobes and equal beamwidths," Microwave Journal, pp. 71-78, June 1963.

Small Array of Planar Flared Dipole Antennas

Description: The model consists of a five element array of flared dipole antennas with a common trough reflector. A postprocessing variable has been defined, which controls the element-to-element source phases, allowing the resulting beam to be easily steered.

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



Model Description: The elements use a Teflon substrate with 1.6 mm thickness. The excitations are lumped ports. A radiation boundary is defined on the faces of a vacuum box that is not shown in the preceding image. The postprocessing variable is named **phase_shift**.

Setup: Adaptive Solution at 1.9 GHz.

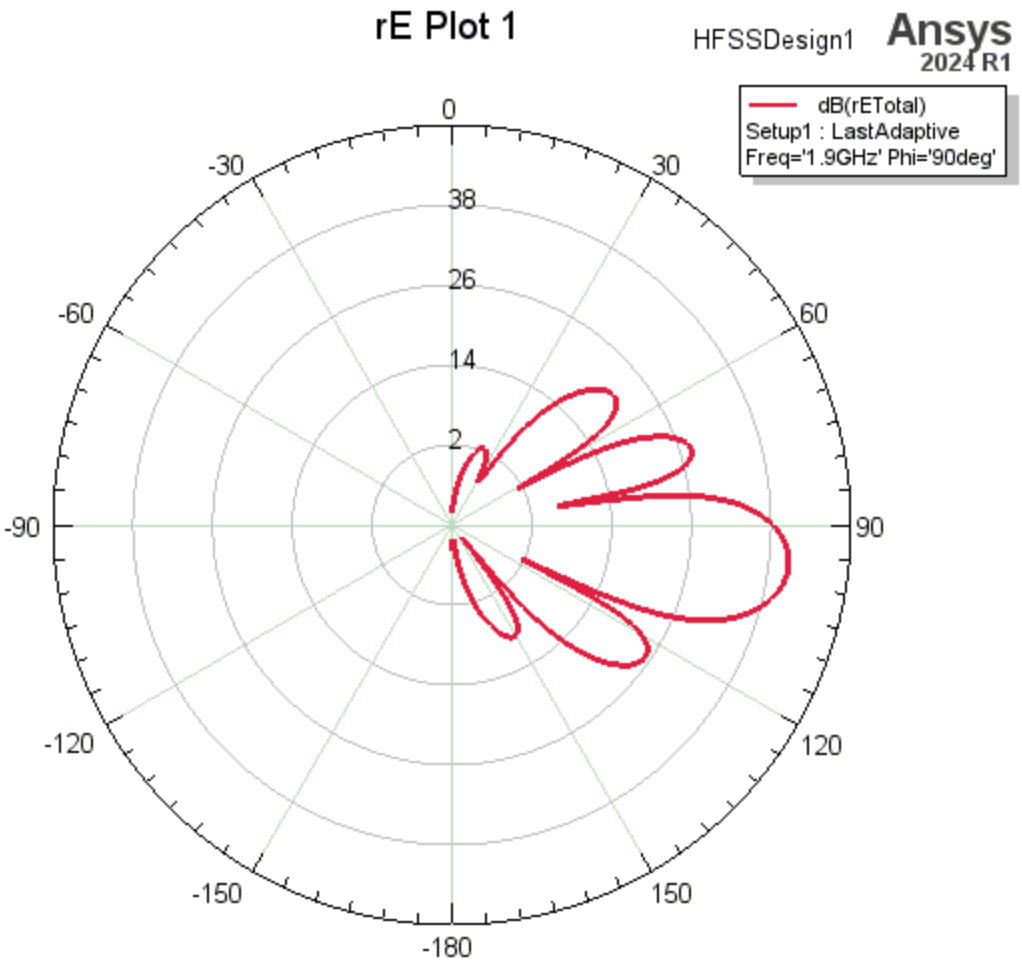
> **Note:**
>
> To view a port or boundary, select the desired item in the Project Manager. It is then highlighted in the Modeler window, and the properties are displayed in the docked *Properties* window. Selecting an object in the History Tree will also display its properties.

Postprocessing Small Array of Planar Flared Dipole Antennas

After solving, you can view solution data by right-clicking on **Setup1** and selecting **Profile** to display the *Solution* dialog box. You can also view the *Solution* tabs for **Convergence**, **Matrix Data**, and **Mesh Statistics**.

To view the $\phi = 90°$ radiation pattern, double-click on **rE Plot 1** under *Results* in the Project Manager.

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



The relative element-to-element phase shift, in degrees, for this array is controlled by the **phase_shift** postprocessing variable. To verify, go to **HFSS** > **Fields** > **Edit Sources**, where you will see the phase values defined as multiples of the variable **phase_shift:**

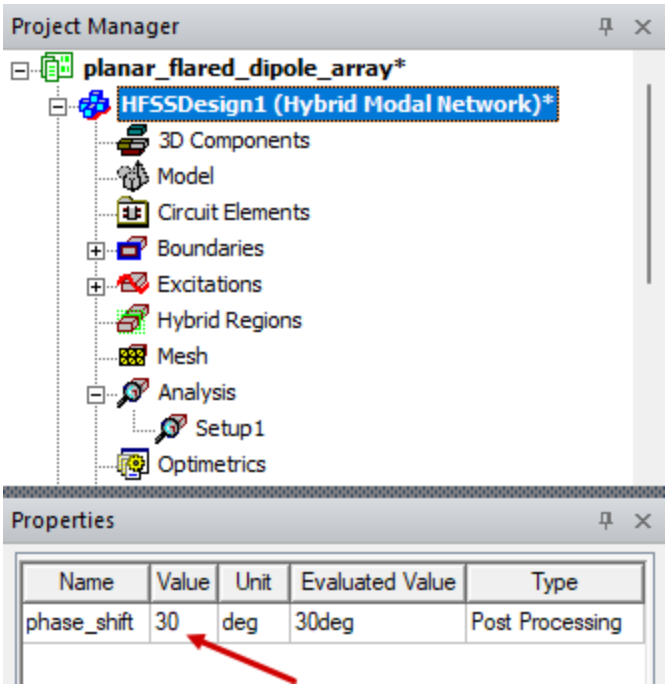
[HFSSDesign1] Edit post process sources

Spectral Fields | Source Context

	Source	Type	Magnitude	Unit	Phase	Unit
1	1:1	Port	1	W	0	deg
2	2:1	Port	1	W	-1*phase_shift	
3	3:1	Port	1	W	-2*phase_shift	
4	4:1	Port	1	W	-3*phase_shift	
5	5:1	Port	1	W	-4*phase_shift	

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

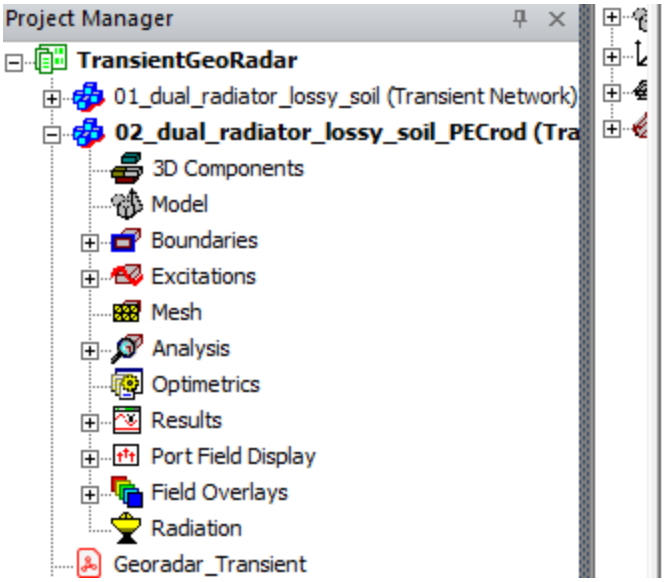
The easiest way to adjust the **phase_shift** variable is to click on the design name in the Project Manager (**HFSSDesign1**). You will see the design variable displayed in the docked *Properties* window. You can change the value of **phase_shift** in this window. The plot shown here is for a setting of 30°.

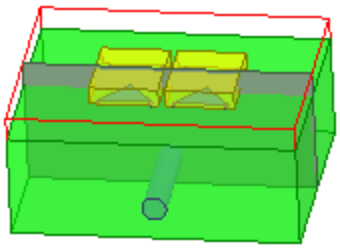


Transient Simulation of Ground Penetrating Radar

This example demonstrates a transient simulation setup in Ansys HFSS. The project models a ground penetrating radar to detect a PEC rod buried in a lossy soil environment. The project consists of two designs: 01_dual_radiator_lossy_soil models lossy soil environment. 02_dual_radiator_lossy_soil_PECrod models the lossy soil environment when the PEC rod is buried. Note that the geometry of the two designs is identical. However, the rod in the first design is assigned soil (the same as environment). The reason is to have identical geometry to be able to import the mesh from the first design into the second design. Therefore, we remove any effect of possible different mesh. That means in comparing the results of the two design, any difference is associated to different material properties of the PEC rod (what radar would detect). The Georadar_Transient pdf provides details for using the model.

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





The Ground Penetrating Radar (GPR) design consists of two cavity backed bow-tie type antennas, one for transmission and one for reception, Figure 1. Each corner of the antenna is loaded with a 200 Ohm resistance model by an RLC boundary condition, Figure 2. This lump resistance helps improve the bandwidth of the antenna input impedance. The antenna is also enclosed in a rectangular conducting cavity to reduce direct coupling and unwanted radiation into air.

The solution type for this example is set as Transient with Network Analysis as Driven Option, Figure 3. By implementing a transient simulation, we can apply a time domain input signal (e.g. Gaussian Pulse) as either TDR or Broadband Pulse. In addition, excitations are identical in all active ports and are simulated one at a time to facilitate the data collection. To properly model the electrical properties of the ground soil the effect of the moisture contents of the earth needs to be accounted for. These are recorded as frequency dependent properties affecting the permittivity and conductivity.

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

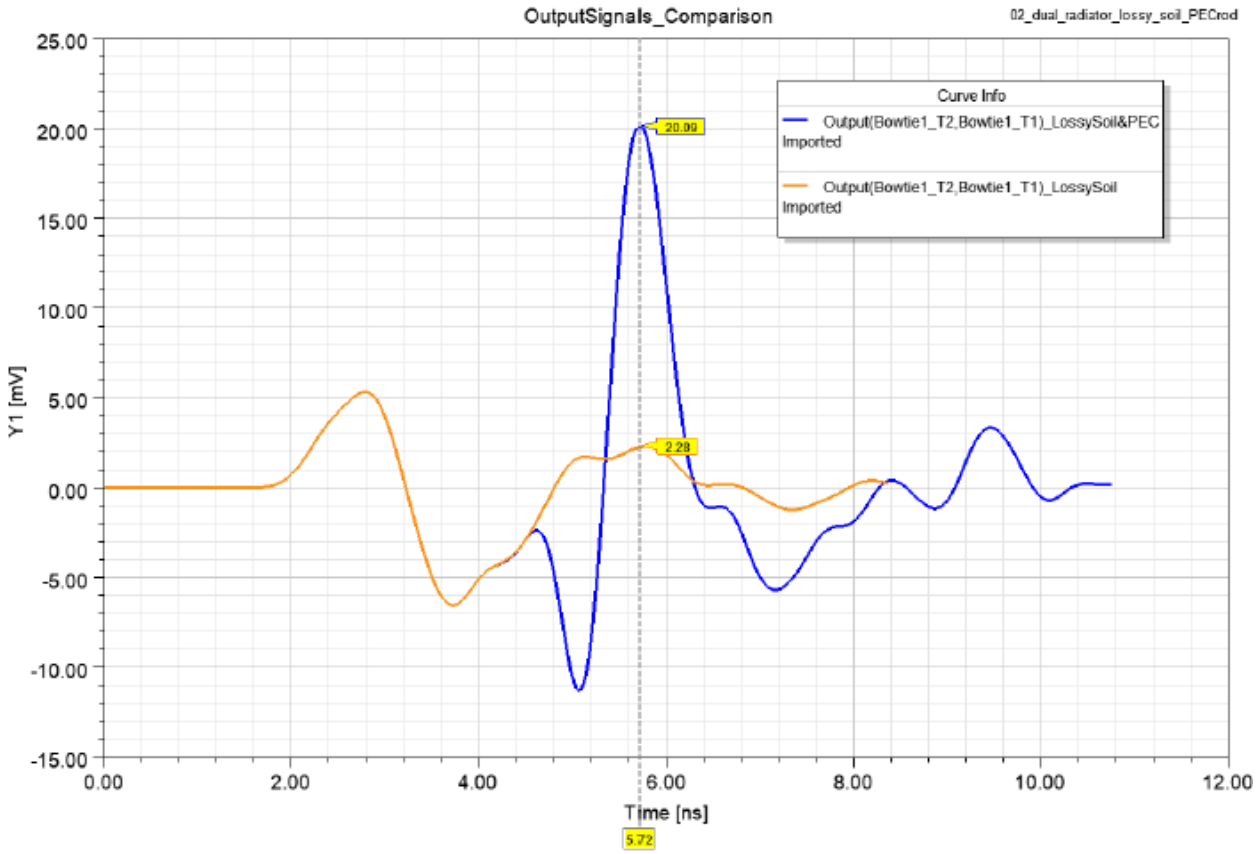
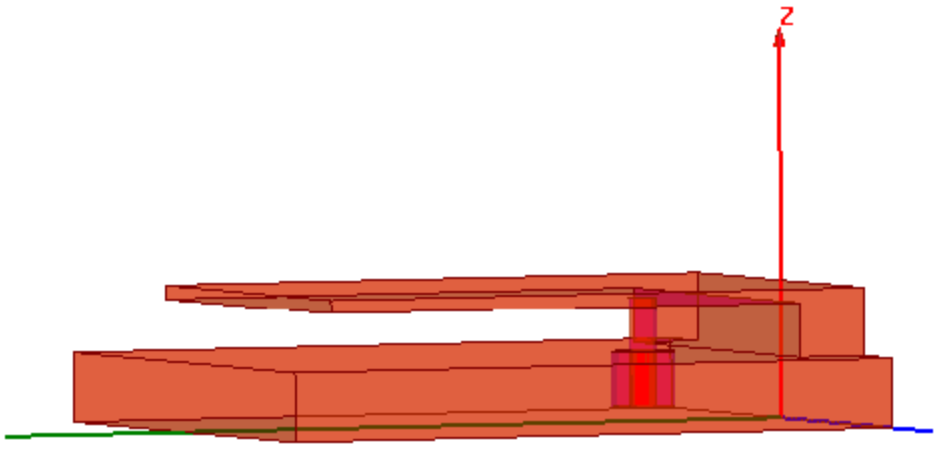


Figure 10. Output Signal showing the detection of the PEC buried in the soil

Tune a Coax Fed Patch Antenna

Description - A coax fed quarter wave patch antenna will be tuned using the Adjoint Derivatives. The antenna is shorted on one end. The design variable, feed_pos, controls the location of the coax feed along the x direction. Derivatives of the S parameter w.r.t. this variable will be computed.

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



Model - The patch, shoulder, and ground plate are united into a single PEC object. The air filled coax has an internal port that is capped by the PEC ground plate. The variable feed_pos controls the value of X for the coax location in millimeters. The surrounding air box with radiation boundary is not shown.

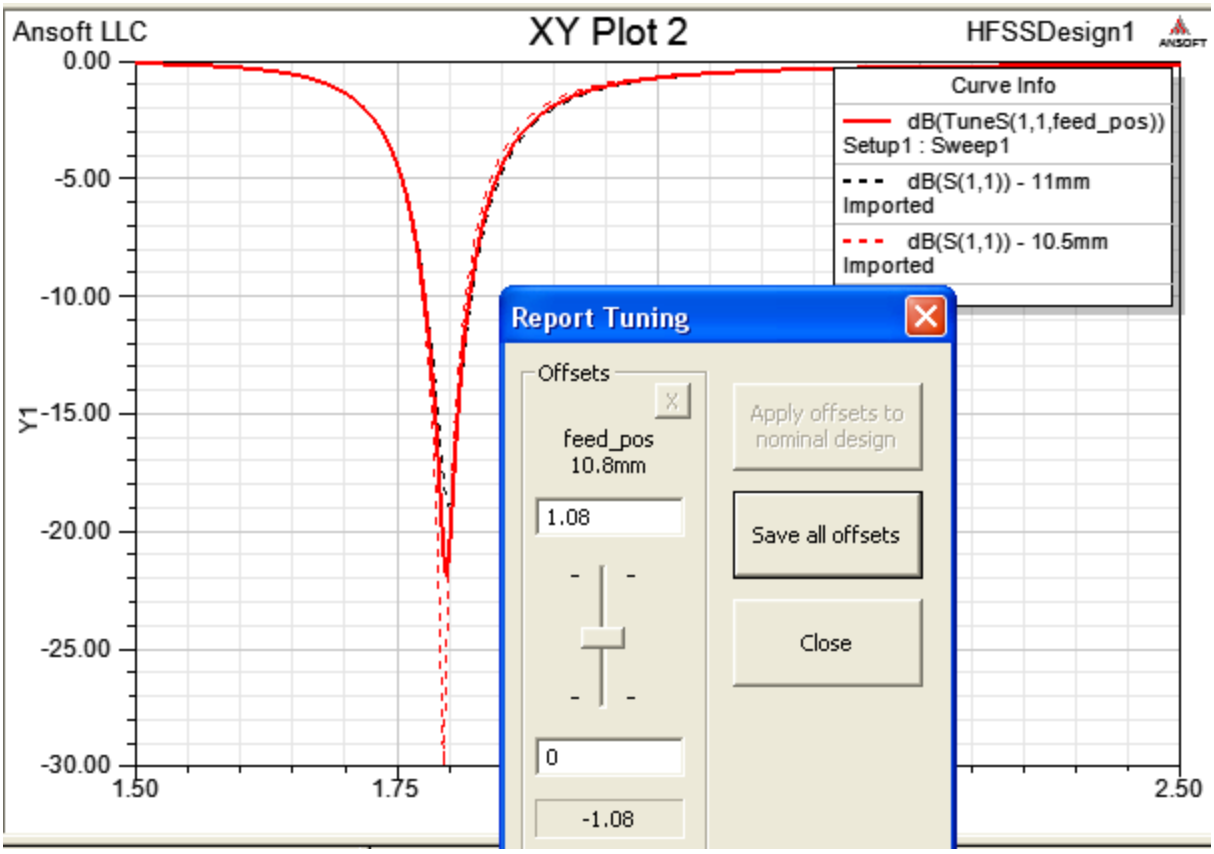
Setup - Adapt at 1.8 GHz., and do an interpolating sweep from 1.5 to 2 GHz. The derivative of the S parameter data w.r.t. feed_pos about the default value of 10.8 mm is computed. This feature is enabled in the **Derivatives** tab of the **Setup1** window.

Post processing for tuning a Coax Fed Patch Antenna

After solving, you can view solution data by right-clicking on Setup1 and selecting **Profile** to display the **Solution** dialog. You also view the **Solution** tabs for **Convergence**, **Matrix Data**, and **Mesh Statistics**.

To view the derivatives you can create a new plot by right-clicking on Results and choosing **Create Modal Solution Data Report>Rectangular Plot**. In the **New Report Setup** window, select feed_pos in the derivative box and select the desired quantities to plot.

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



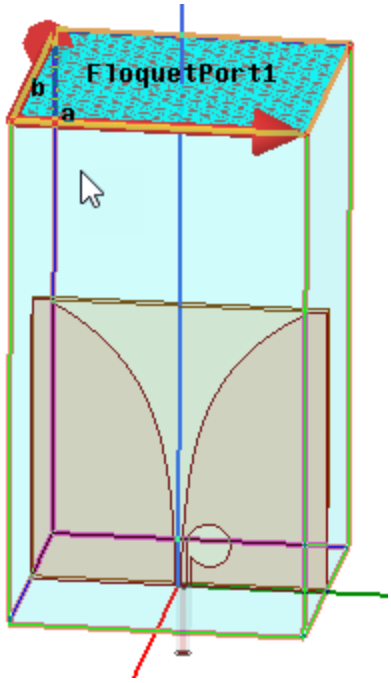
To tune the design, double-click XY Plot 2 under Results to open the plot shown. In a separate project, the response for feed_pos = 10.5 and 11 mm was computed and the results from each was imported into this plot. By right-clicking on Results and choosing **Tune Reports** you can open the **Report Tuning** dialog shown. As you adjust the slider, the plot is recomputed using the derivatives. Adjust the Offset to -0.3 and +0.2 to compare to the imported feed_pos = 10.5 and 11mm results.

Unit Cell of a Phased Array

Description - A unit cell of an infinite phased array of vivaldi antennas is simulated using linked boundaries and a Floquet port.

Model - the antenna is fed by a coax line with a wave port. The upper face of the unit cell is terminated in a Floquet port. The sides are two pairs of Primary and Secondary (Lattice pair) boundaries. The substrate ε_r = 6 and is 1.27 mm thick. The conducting traces are 2D objects

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



with PerE boundaries.

Setup - Adapt at 4.5 GHz with an interpolating sweep from 2 to 5 GHz.

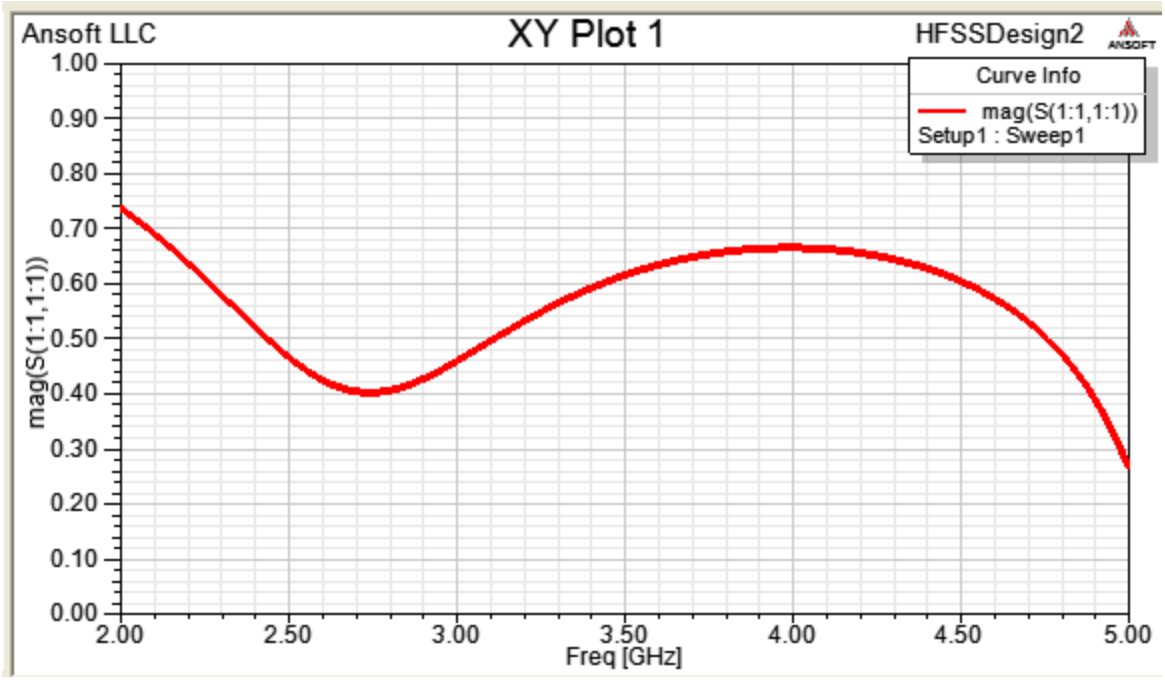
> **Note:**
>
> To view a port or boundary, select the desired item in the Project Tree. It is then highlighted in the Model window and the properties will be displayed in the Properties window.

Post Processing

After solving, you can view solution data by right-clicking on Setup1 and selecting **Profile** to display the **Solution** dialog. You also view the **Solution** tabs for **Convergence**, **Matrix Data**, and **Mesh Statistics**.

To view a plot of active S parameter seen at the feed, look in the **Project** tree and double-click on XY Plot1. To view the modes present on the Floquet port click on the desired mode under **Port Field Display>Floquet Port 1** in the **Project** tree, and a vector plot of the mode will be displayed.

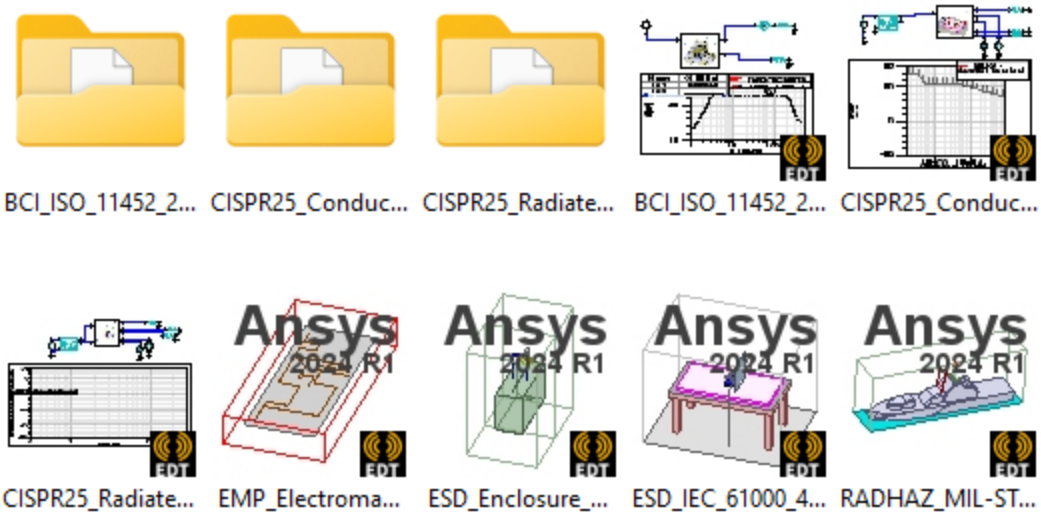
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This design was analyzed in “Analysis of Periodic Structures via a Time-Domain Finite-Element Formulation with a Floquet ABC,” L.E.R. Peterson et al., IEEE Trans, AP, March 2006, pp 933-944. You will see the plot computed here agrees nicely with Fig. 9b in the reference.

EMI EMC Examples (Electromagnetic Interference, Electromagnetic Compatibility)

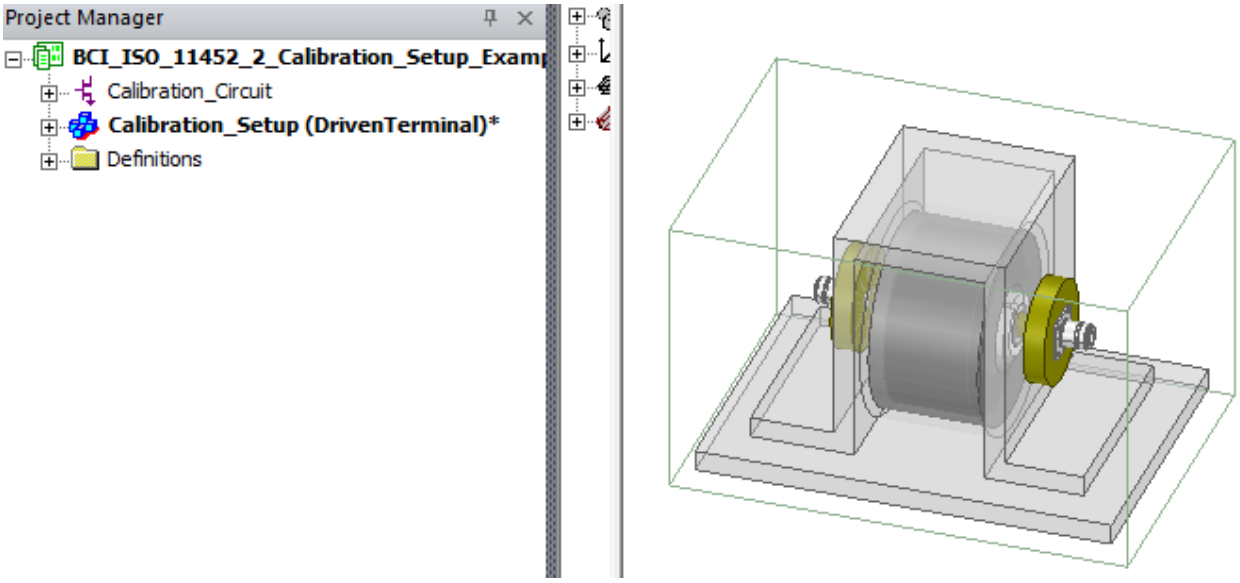
Click graphics for descriptions of the following HFSS EMI EMC examples:



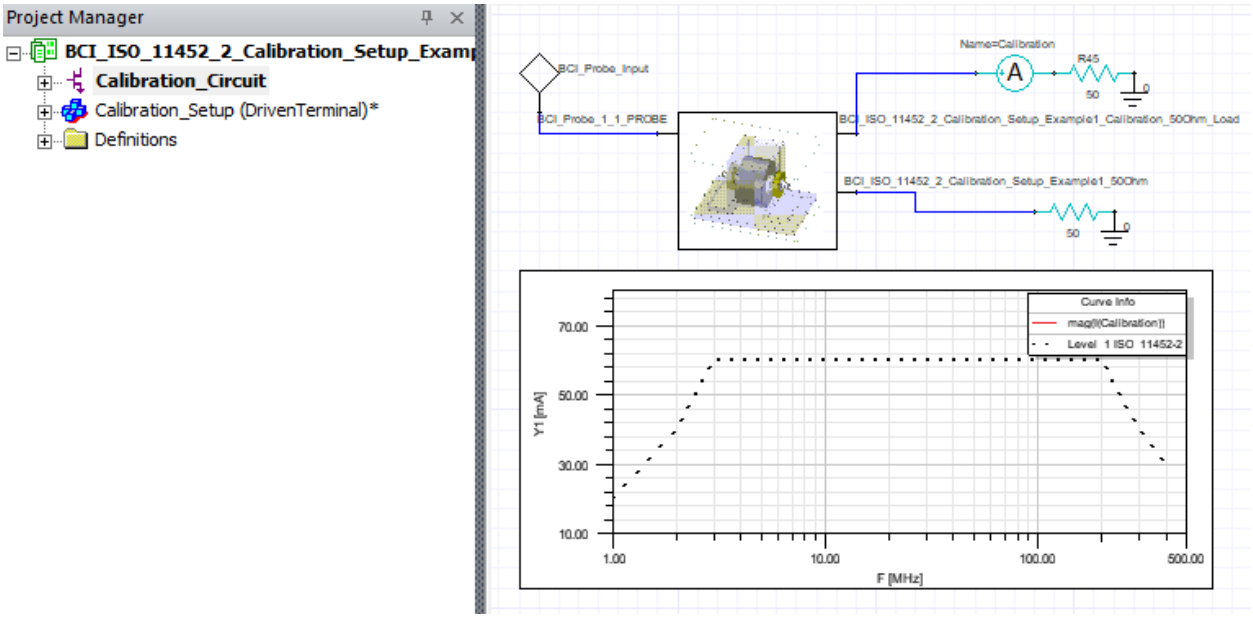
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BCI ISO11452-2 Calibration Test

Description - This example includes the ISO 11452-2 calibration test fixture.



An optimization setup is included in the circuit schematic to calibrate the input signal to achieve the current level [mA] according to Level ISO 11452-2 standard.

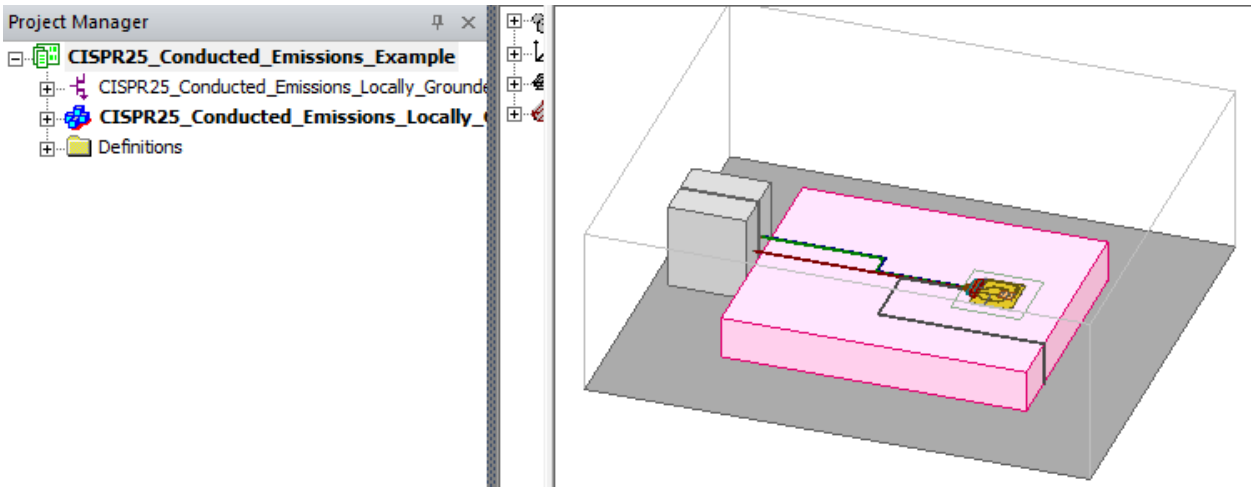


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CISPR25 Conducted Emissions Example

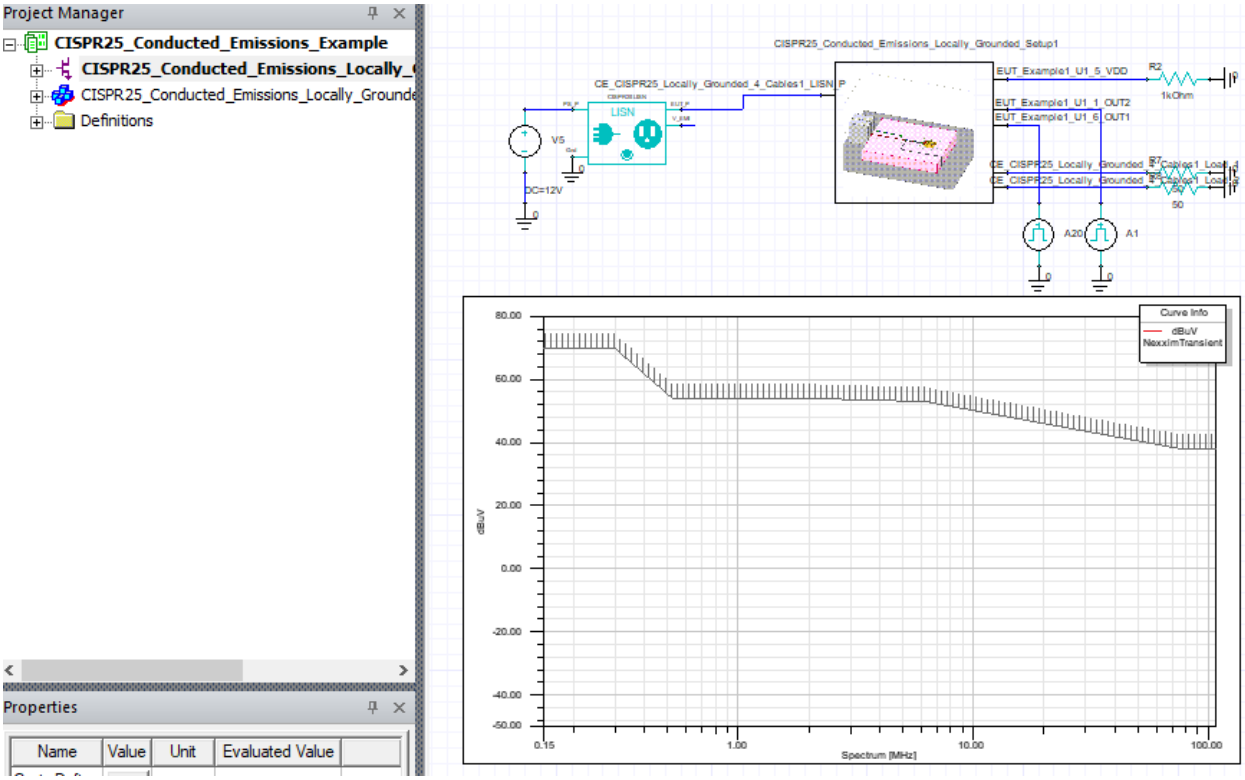
Description - This project shows how to simulate the conducted emissions from a dummy PCB according to CISPR25:2008

The CISPR25:2008 test bench model can be edited to manually change the number of cables according to the EUT.



The circuit schematic is included.

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

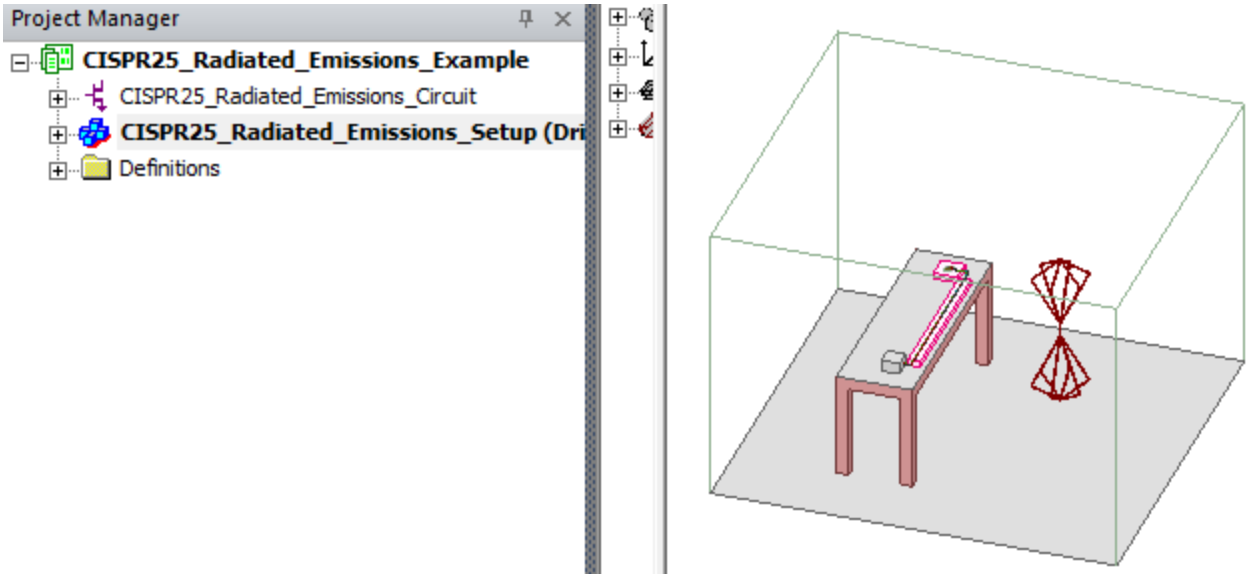


CISPR25 Radiated Emissions Example

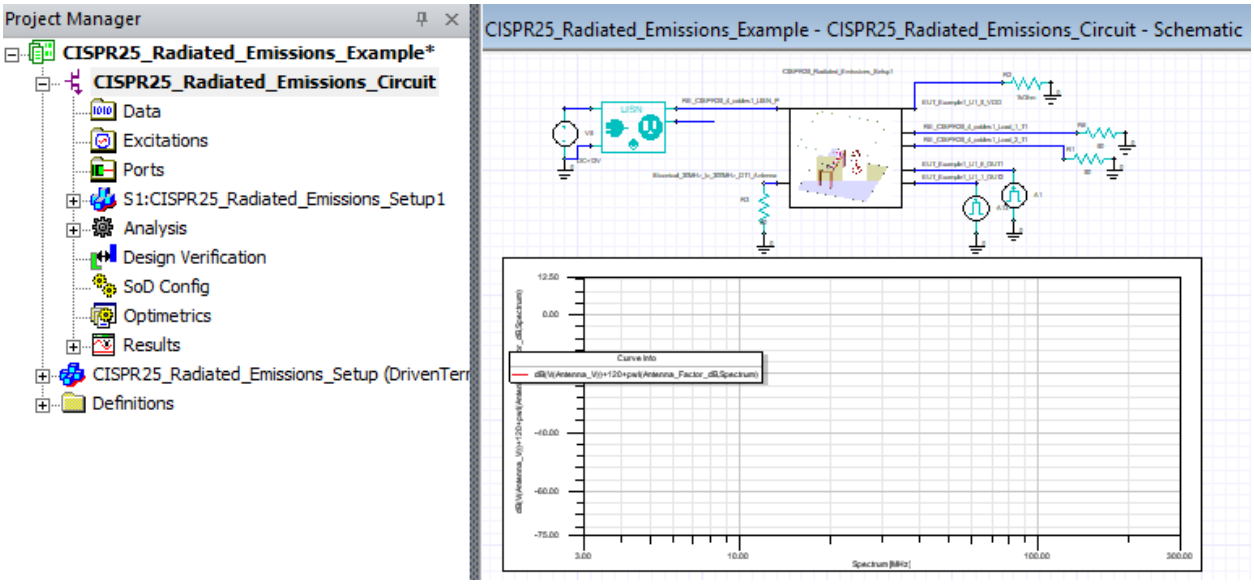
Description -This project shows how to simulate the radiated emissions from a dummy PCB according to CISPR25:2008

The CISPR25:2008 test bench model can be edited to manually change the number of cables according to the EUT

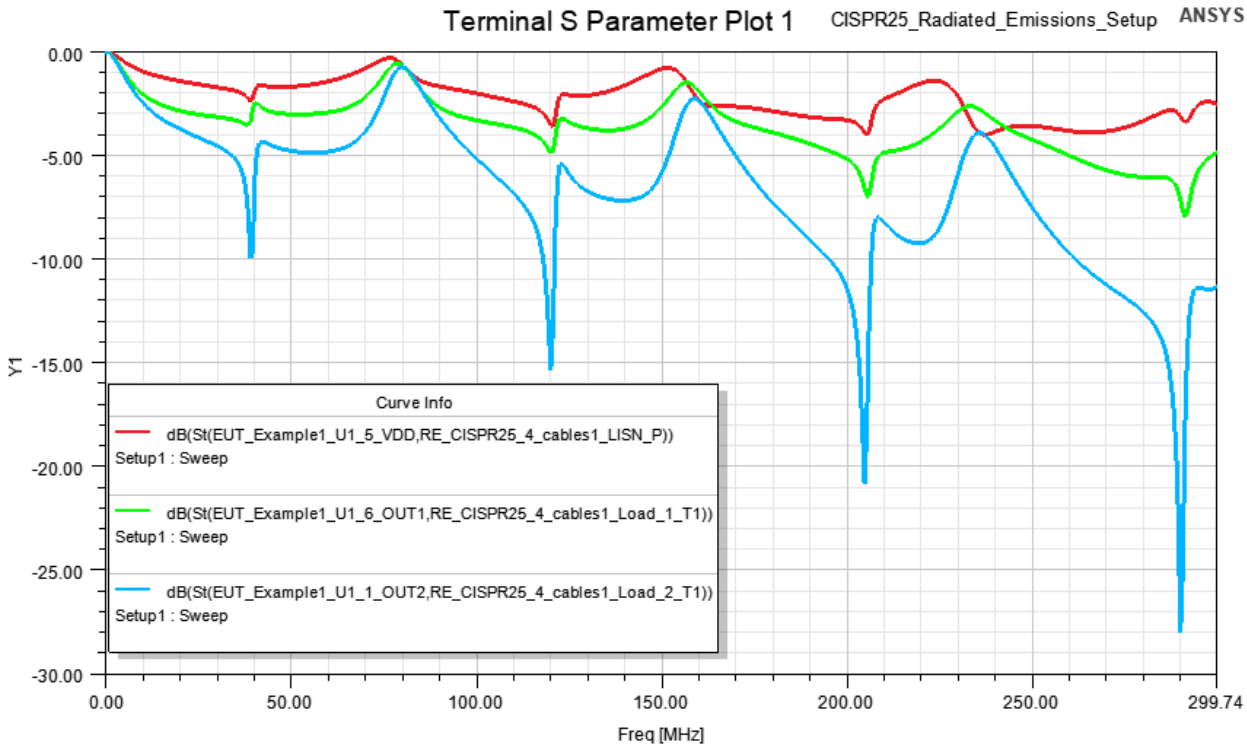
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



An antenna factor is applied in the circuit schematic to calculate the radiated emissions. The circuit schematic is included.



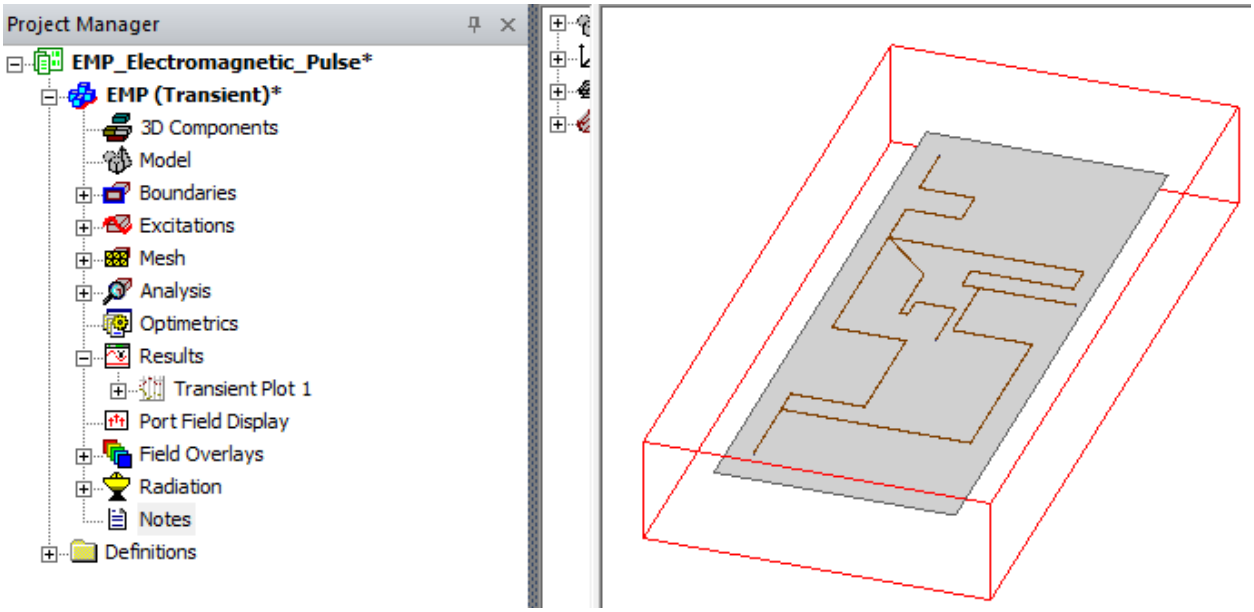
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



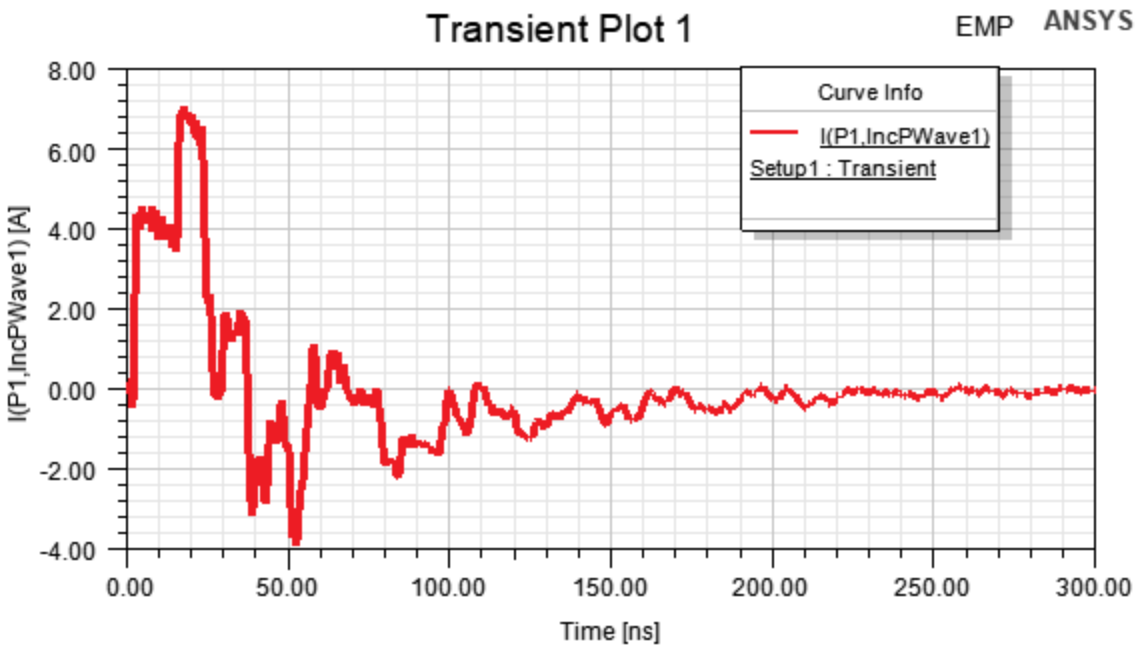
EMP Electromagnetic Pulse (HFSS Transient) Example

Description - This project is based on information from the following paper: Petrache, E. et al "An experimental test for the validation of time domain codes for the analysis of transient field coupling to transmission line networks." It calculates the induced voltage on resistors connecting a conductor representing cables on a vehicle to a ground plane due to an electromagnetic pulse

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



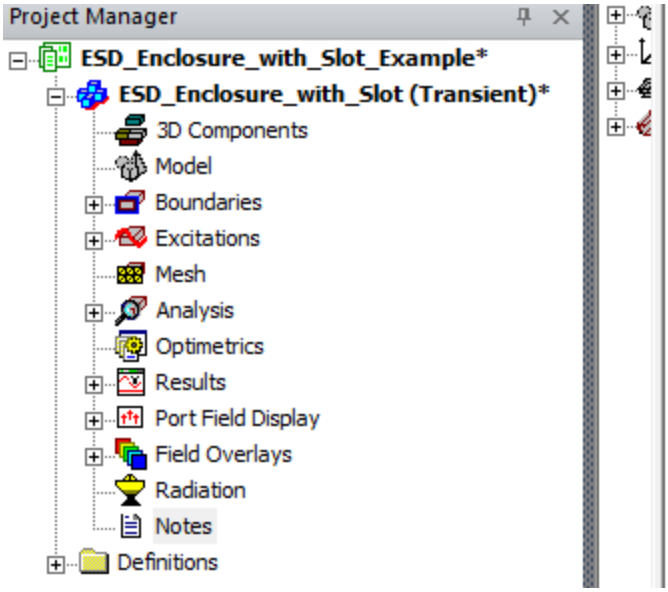
Running the simulation produces the following plot.

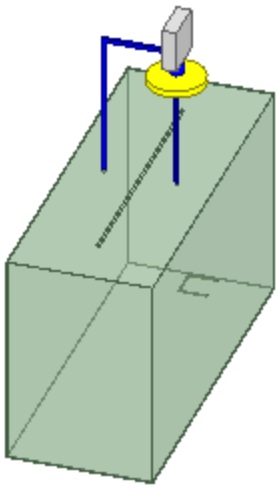


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ESD Enclosure with Slot (HFSS Transient) Example

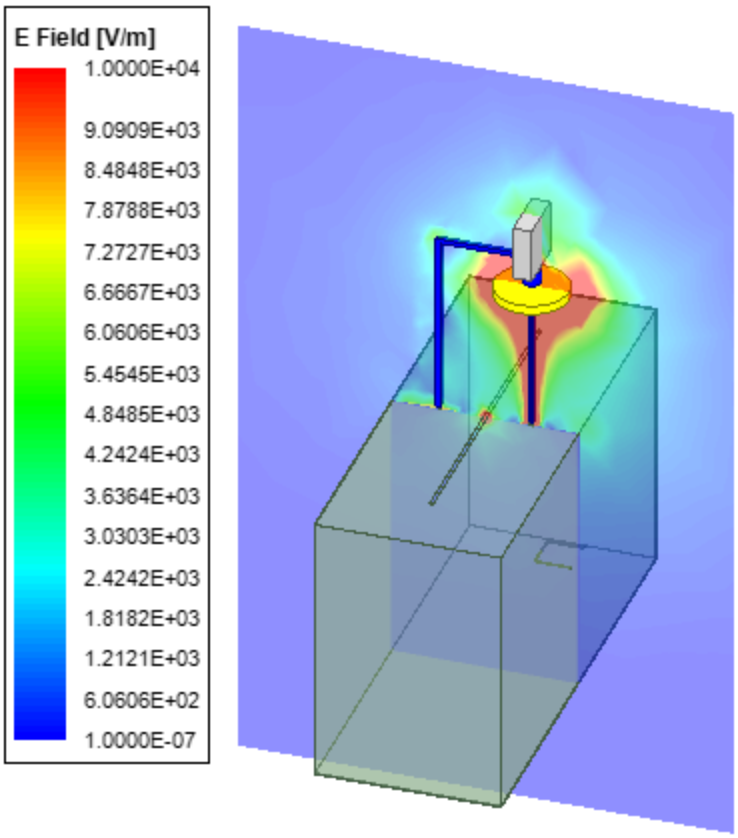
Description - This project shows the design of a ESD gun and the voltage induced in a loop inside of an enclosure with a slot. Model and measurements based on the following paper: F. Centola, D. Pommerenke, Wang Kai, T. Van Doren and S. Caniggia, "ESD excitation model for susceptibility study," 2003 IEEE Symposium on Electromagnetic Compatibility. Symposium Record (Cat. No.03CH37446), Boston, MA, USA, 2003, pp. 58-63 vol.1.doi: 10.1109/ISEMC.2003.1236564





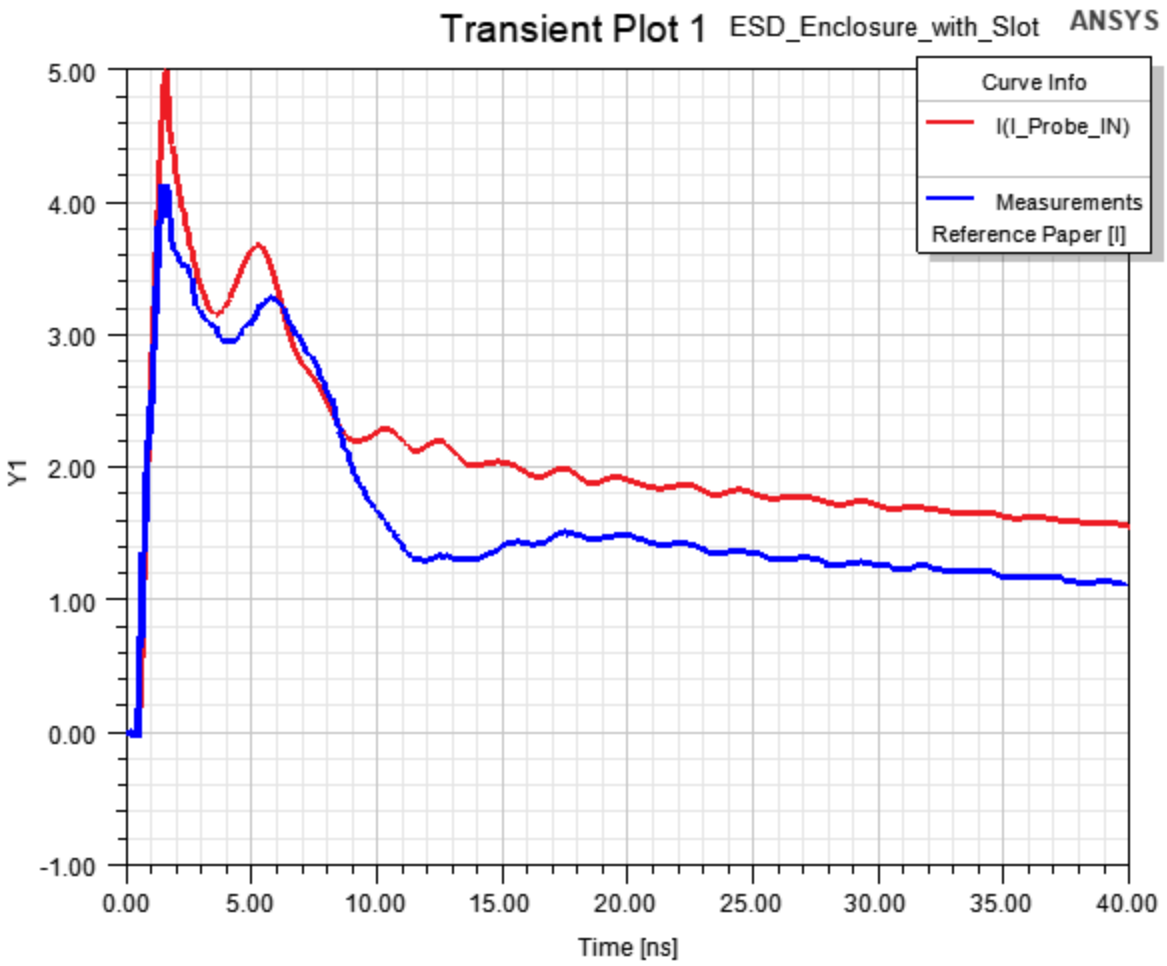
Running the simulation produces the following field plot.

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



The project also includes reports.

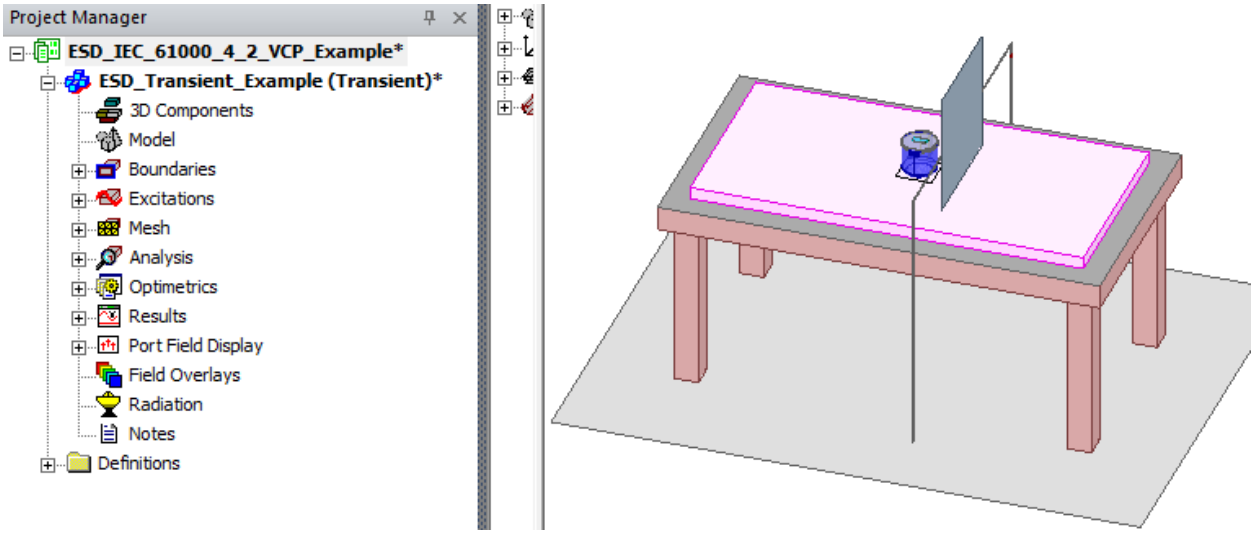
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



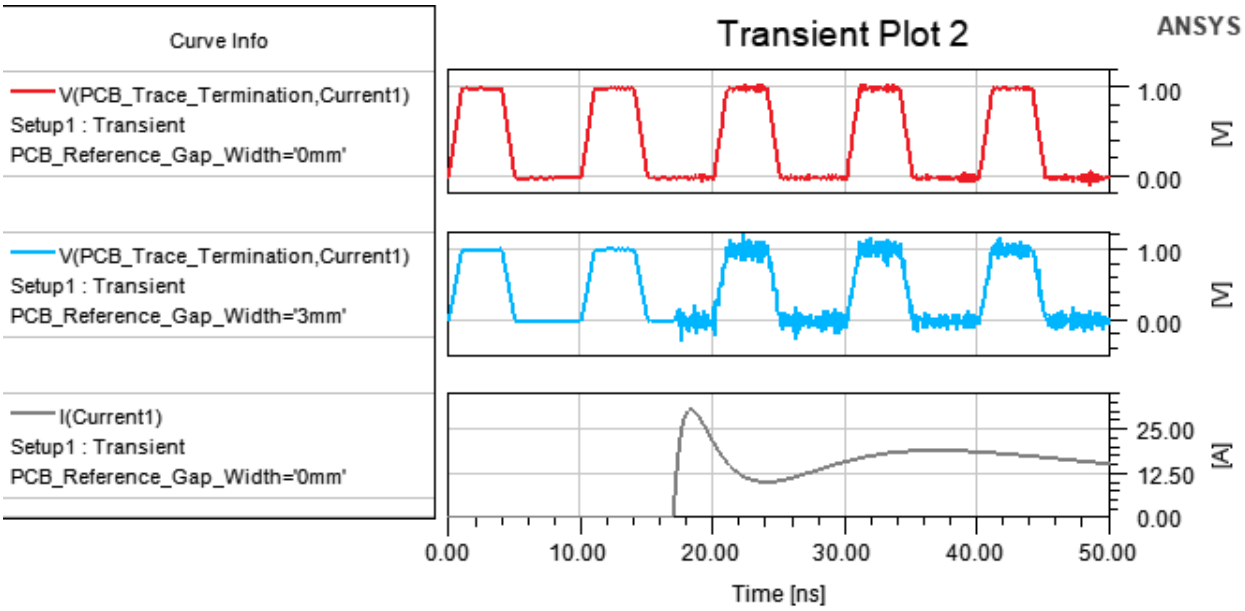
ESD IEC 6100000-4-2 VCP (HFSS Transient) Example

Description - This example shows how a 100MHz clock transmitted by a microstrip inside the jar is affected by the ESD. The variable "PCB_Reference_Gap_Width" controls the width of a gap in the reference plane under the microstrip. When the variable is 0mm (no gap) the induced noise on the clock signal is smaller when compared to a gap of 3mm. The setup includes a parametric sweep of the PCB reference gap width.

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



The project also includes reports.

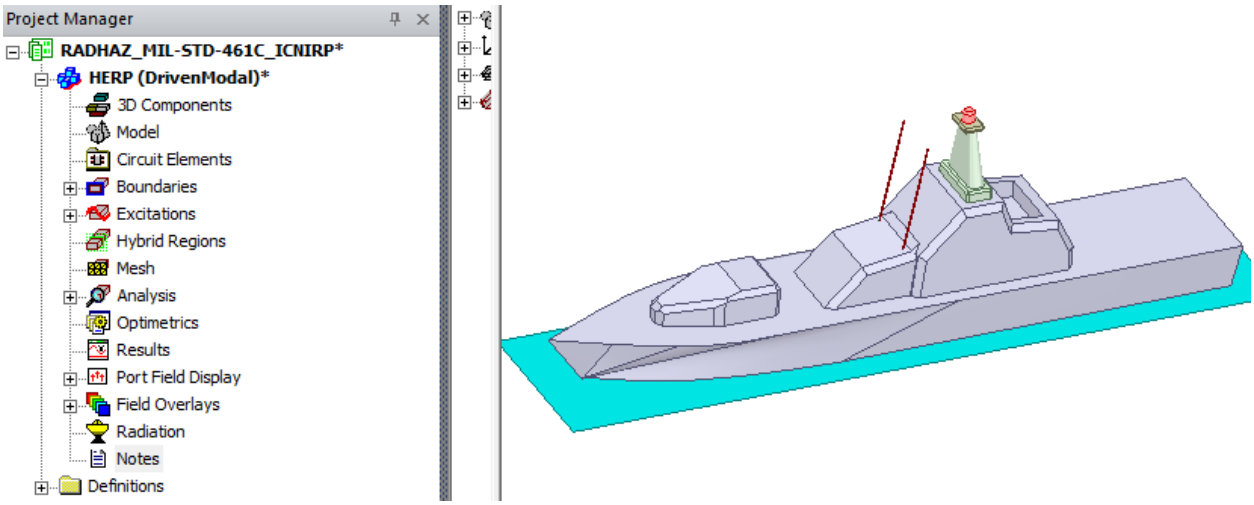


Radiation Hazards MIL STD461C ICNIRP Example

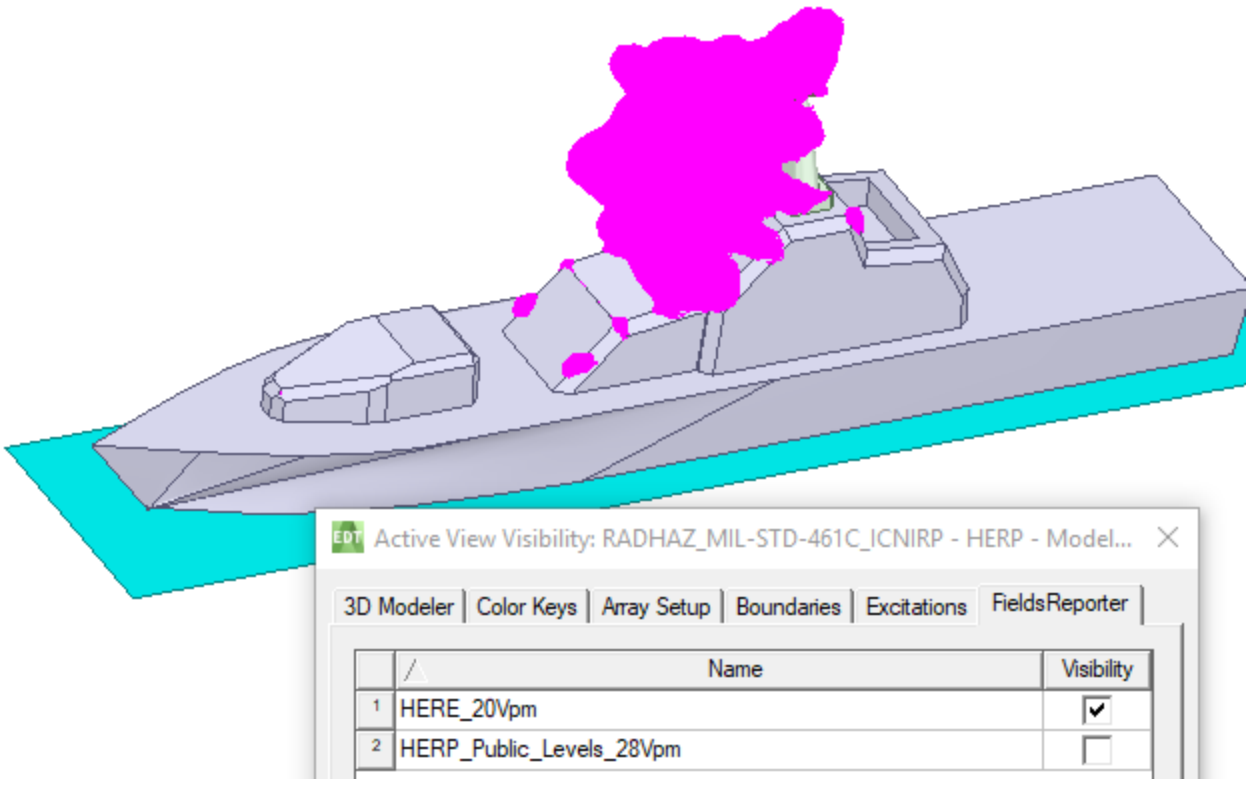
Description - This example shows how to calculate Radiation Hazards due to the radiation of two antennas placed on a ship. The limit for HERE (Hazards of Electromagnetic Radiation to Equipment) according to MIL-STD-461C is represented by an isosurface of 20V/m isosurface. Inside the volume of this isosurface no equipment shall be installed.

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

The HERP (Hazards of Electromagnetic Radiation to Personnel) level of 28V/m according to ICNIRP is also created. A post-processing variable "Tx_Power" control the antenna Tx power.

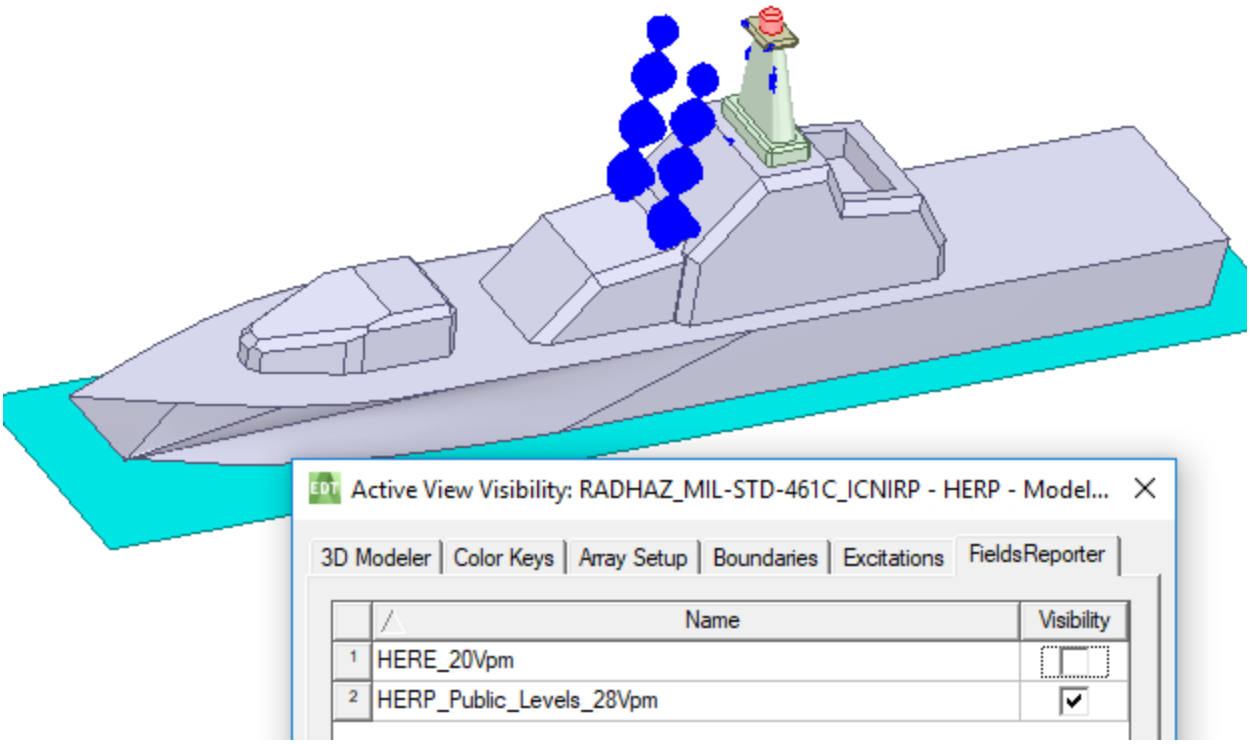


The project includes a Field Overlay for the Hazards of Electromagnetic Radiation to Equipment.



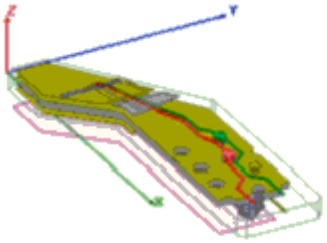
It also includes a Field Overlay for Hazards of Electromagnetic Radiation to Personnel.

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

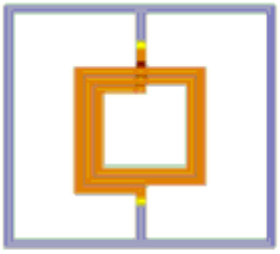


Integrated Circuits in HFSS

Click the graphics below for examples of modeling integrated circuits in HFSS:



Package



Spiral Inductor

The first model is an example project included with the Ansys Electromagnetics Suite installaion. It is located in the **...Examples\HFSS\Signal Integrity** subfolder and is named **package.aedt**.

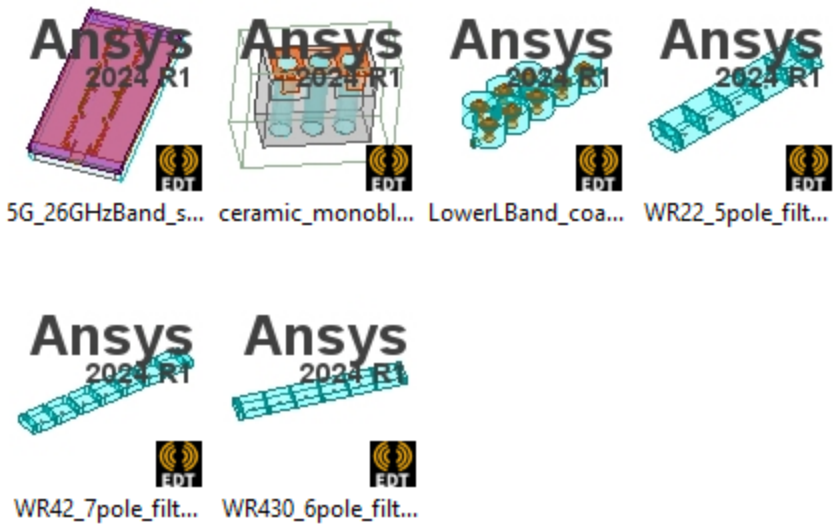
You can build the second model by following the steps in the **Silicon Spiral Inductor** getting started guide.

Also see: Application Specific Modeling Guide: Spiral Inductors on Silicon Substrate

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

Filter Examples

The following figure shows the project files installed with the Ansys Electromagnetics Suite and located in the **...Examples\HFSS\Filters** subfolder:



These filter examples are not currently described in the help. See the PDF files that accompany the projects (located in the same folder).

Metamaterial Examples

Click graphics for descriptions of the following HFSS Metamaterial examples:

Cylindrical Hyperlens

Abstract

Imaging applications are often presented with a difficult resolution barrier intrinsic to wave phenomena: the diffraction limit. While overcoming this limitation may seem impossible with a naive glance at Maxwell's equations, in fact some further consideration shows it can be overcome through a variety of means. This example demonstrates one such method, a

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

“hyperlens,” which has been realized at ultraviolet frequencies. This HFSS simulation predicts the hyperlens can resolve objects with separation of less than one-third the incident wavelength, well below the diffraction limit.

Simulation Time / RAM: (6-core machine with 64GB RAM in 2025 R1)

- **Hyperlens Imaging:** 1 minute: 34 seconds with 1.6 GB max. memory used per process
- **No Hyperlens:** 1 minute: 47 seconds with 1.33 GB max. memory used per process

Introduction

Imaging systems are essential technology in many industries, including medicine, semiconductors, defense, academic research, among others. While many of these systems are quite sophisticated, there are a few applications where optical imaging has yet to penetrate due to the diffraction limit hindering resolution performance.

For example, some sub-cellular processes of interest in medical research cannot be imaged in vivo due to their small size (much smaller than optical wavelengths) and the requirement for high vacuum inside electron microscope chambers. There are some existing sub-diffraction optical imaging methods, such as stimulated-emission-depletion microscopy, but the high optical power applied quickly leads to photobleaching and is limited to specific fluorophores.

Additionally, near-field scanning optical microscopy is a well-established commercial technology, but it requires relatively slow raster scanning of the sample, precluding it from real-time imaging.

Recent developments in electromagnetic metamaterials research have given us a few sub-diffraction imaging solutions at multiple portions of the spectrum [1-3]. One such technology of interest has been called the “hyperlens.” The hyperlens is constructed from a hyperbolic metamaterial, a special class of metamaterial that has dispersion relation described by a hyperboloid in three dimensions, in contrast to the usual ellipsoid. In addition to imaging, hyperlenses can also be applied to optical nanolithography and ultra-high-density optical data storage [2]. This example project shows one realization of a hyperlens from the literature which can achieve sub-diffraction resolution in the ultraviolet region.

Full-wave electromagnetic solvers like Ansys HFSS give engineers and researchers crucial information when designing and characterizing subwavelength electromagnetic devices, such as hyperlenses. This information includes parametric analysis, device optimization, and field plots.

Hyperbolic Metamaterials and Hyperlenses:

The dispersion relation for electromagnetic waves in spherical coordinates [4] is given by:

$$\frac{k_r^2}{\varepsilon_\theta} + \frac{k_\theta^2}{\varepsilon_r} - \left(\frac{\omega}{c}\right)^2$$

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

When $\varepsilon_r = \varepsilon_\theta > 0$, the conventional diffraction limit constrains the resolution of any imaging process, since only waves with $\kappa_\theta \leq \eta(\omega/c)$ will propagate. In other words, large spatial frequencies which carry the high-resolution image information, will not reach an imaging detector in the far-field. However, consider the case where:

$\varepsilon_r < 0$ and $\varepsilon_\theta > 0$

In this case, the dispersion relation becomes a hyperboloid,

$$\frac{k_r^2}{\varepsilon_\theta} - \frac{k_\theta^2}{|\varepsilon_r|} = \left(\frac{\omega}{c}\right)^2,$$

and arbitrarily large κ_θ can propagate. Such a medium, at optical frequencies, can be easily realized by modern nanofabrication methods, as in Refs. [4,5]. Fabricating the medium in a curved geometry then allows for magnification, so that as the large spatial frequency waves propagate through the medium, they also separate until they can be resolved by a conventional microscope after they exit the medium.

Model Construction and Setup Details

This example project shows simulation of a typical hyperlens imaging experiment similar to Ref. [4]. After setting up the HFSS project, the first step was to construct the layers of the hyperlens. The hyperlens consists of alternating pairs of metal and dielectric, in this case Ag and Al_2O_3. In principle, the hyperlens can be a truly three-dimensional device if fabricated in a hemispherical geometry. However, for this example it is sufficient to make it cylindrical and restrict the y-dimension with appropriate boundaries to save some computational resources. Also note that the experimental device in Ref. [4] is cylindrical, as we modeled here.

The layers were drawn by placing a series of concentric cylinders centered at the origin, subtracting each cylinder from the last one, then using a box command to subtract out the unwanted portion above the xy plane. The “object” being imaged is a Cr mask with two openings, separated by a subwavelength distance, on the inner radius of the hyperlens.

You can see the object history for the mask in the “Model > Solids > Cr_365nm” and "...Quartz_ 365nm" branches in the History Tree. After drawing the hyperlens and Cr mask, a box was then added to represent the quartz substrate. The hyperlens/mask structure was subtracted from the box and then the *Separate Bodies* command was used so that the portion near the center of the hyperlens could be deleted from the final structure.

A region was then added so that an incident beam and boundaries could be applied.

To give the model material properties, custom materials were defined using the permittivity values for a 365 nm wave-length given in Ref. [4]. Since the permittivity for Cr was not given, a value calculated from the commonly cited Lorentz-Drude model was used [6]. It is important to

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

note that the time dependence convention adopted by HFSS requires that the imaginary part of ε and μ be negative for passive media.

Finally, because of the cylindrical geometry, the resulting “image” fields propagating away from the hyperlens have a cylindrical constant phase surface. Therefore, the image target, where we read out the fields, should be curved appropriately to mimic the phase compensation performed by a microscope objective. To do this, a curved line was added, away from the hyperlens, to later plot the fields along it.

Since this is essentially a two-dimensional problem, and the desired polarization determines that H should be oriented along y, Perfect H boundaries were applied to the ±Y faces of the model so that the computational domain can be reduced. Radiation boundaries were applied to the remaining outer faces.

To excite the model, a normally-incident Gaussian beam was added to the +Z face. Given that the operating wavelength for this device is 365 nm, the beam width was set so that it was not too small to show effects of diffraction but not larger than the total width of the model along the X direction. The final geometry is shown below. The two included designs are identical except for the absence of the hyperlens in the second deisgn.

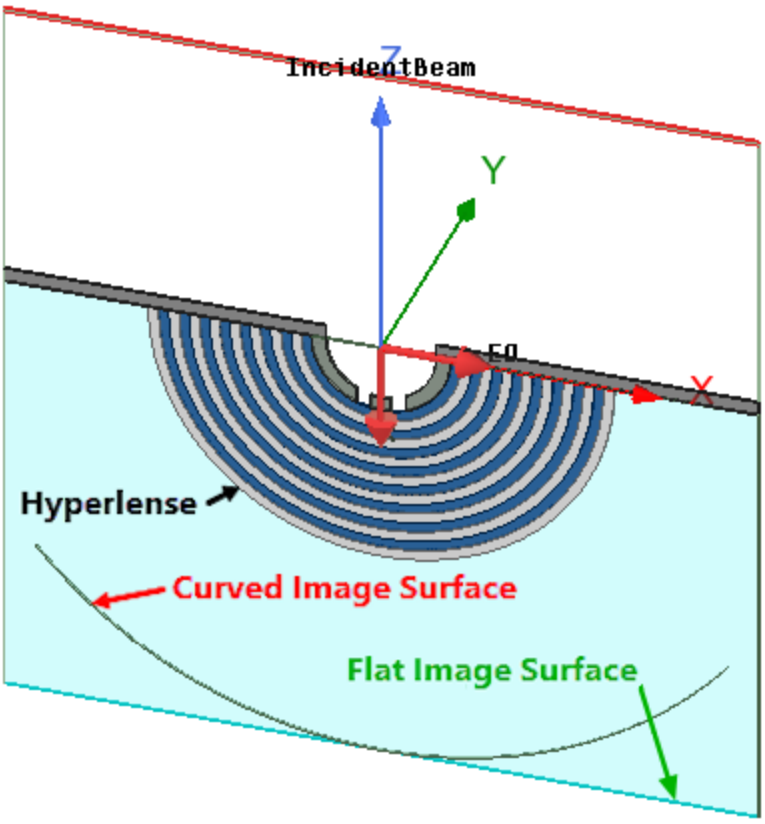


Figure 1: Design 1 – *Hyperlens Imaging*

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

The *Curved Image "Surface"* and *Flat Image "Surface"* objects are actually polylines. These non-model polylines are the specified geometry along which reported quantities are plotted. For the curved polyline, a dummy sheet object has also been created having a 10nm thickness (using the *t_model* variable). The original polyline was copied and given a thickness extending in a direction normal to the +Z axis and the arc itself (+/- Y in this case). The sheet is a model object, and its purpose is to force the mesh of the quartz boxes to conform to the target polyline. That is, solid element faces are generated along the sheet faces. This technique improves the smoothness of plot traces by eliminating the need to interpolate for results within the solid element interiors. The curved polyline, and therefore the sheet derived from it, is segmented to produce a consistant and regular mesh.

Before setting up the analysis, a length mesh operation was added to the hyperlens and Cr mask so that the fields inside the hyperlens can be well-resolved when later plotted. Since HFSS will adaptively mesh the model during analysis, this step is optional. However, since the fields inside the hyperlens layers are of some interest, it is nice to have a smooth picture of them when viewing the field overlay plots.

Since it is crucial for this model to accurately represent curved surfaces, it can be helpful to adjust some of the settings in the *Initial Mesh Settings* dialog box shown below. The relevant settings are the **Apply curvilinear meshing to all curved surfaces** check box and the *Curved Surface Meshing* slider:

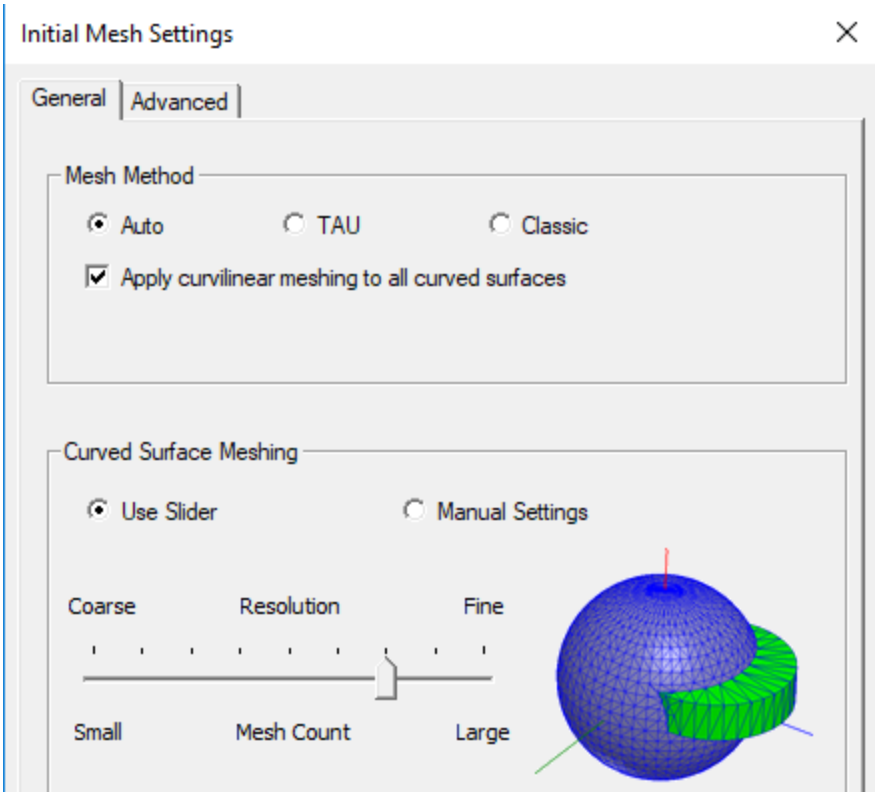


Figure 2: *Initial Mesh Settings* Dialog Box

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

It is worth noting that this slider will affect the number of elements in the model and, in turn, wil affect the memory requirements. However, in this case, the model is relatively small, using <1GB of memory for meshing, So there is little reason for concern.

The only remaining step was to add a solution setup with the frequency determined by the operating wavelength of 365 nm. The frequency can easily be determined from the wavelength by the relationship: $f = c/\lambda$.

Viewing Results with the Hyperlens

After analyzing the model, the results of interest can be easily extracted using the Results and Field Overlays branches of the Project Manager window. In this model, it is informative to view how the fields propagate through the hyperlens structure and into the Quartz substrate. Additionally, the resulting image is obviously of interest, so that we can determine if the Cr object is resolved. Fields overlays were plotted for both the magnetic field magnitude and the real part of the y-component of the magnetic field. The overlays have been assigned to the back face of the air region so that the curved image target polyline and its derived sheet object are visible from the front view while the overlay is displayed:

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

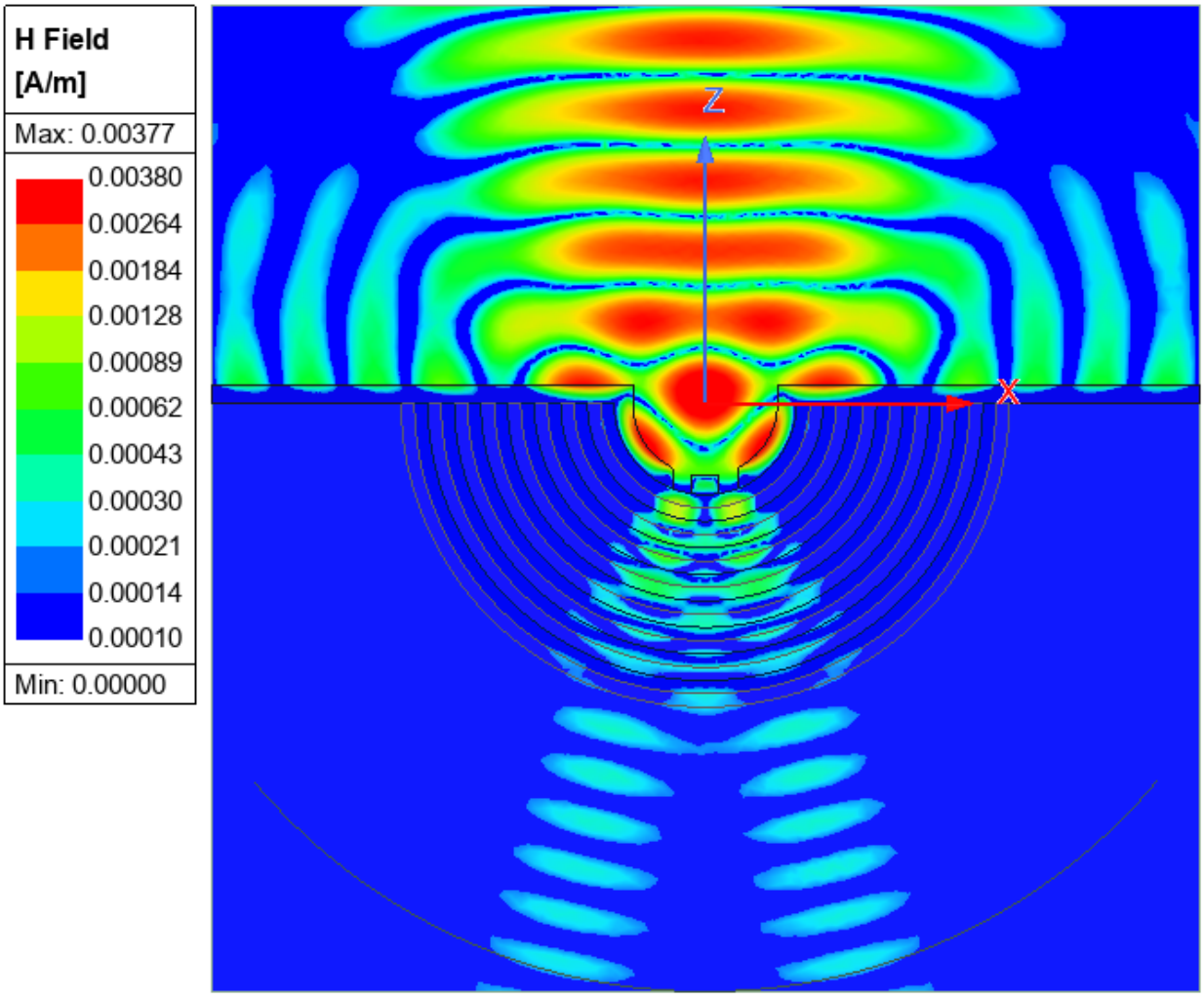


Figure 3: H Field Overlay – *Hyperlens Imaging* Design

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

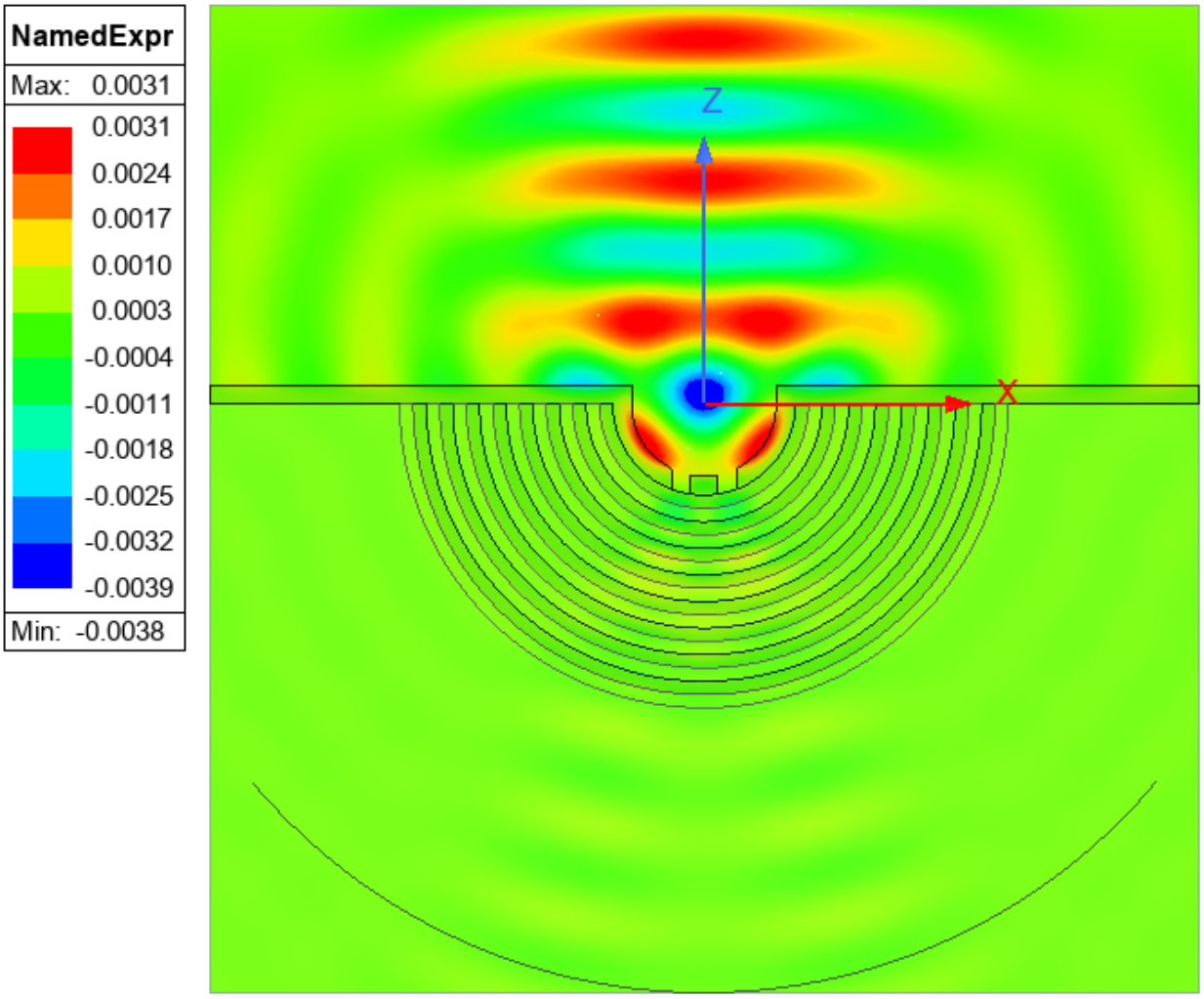


Figure 4: Real Part of Y-Component of H Field – *Hyperlens Imaging* Design

Animating this plot with respect to the phase gives interesting insight into how the waves propagate through the hyperlens.

The resulting complex E magnitude plotted along the previously defined curved image target is shown below. From this plot you can determine that the hyperlens effectively resolves an object having features with only 120 nm center-to-center separation, less than one third the incident wavelength and well below the diffraction limit.

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

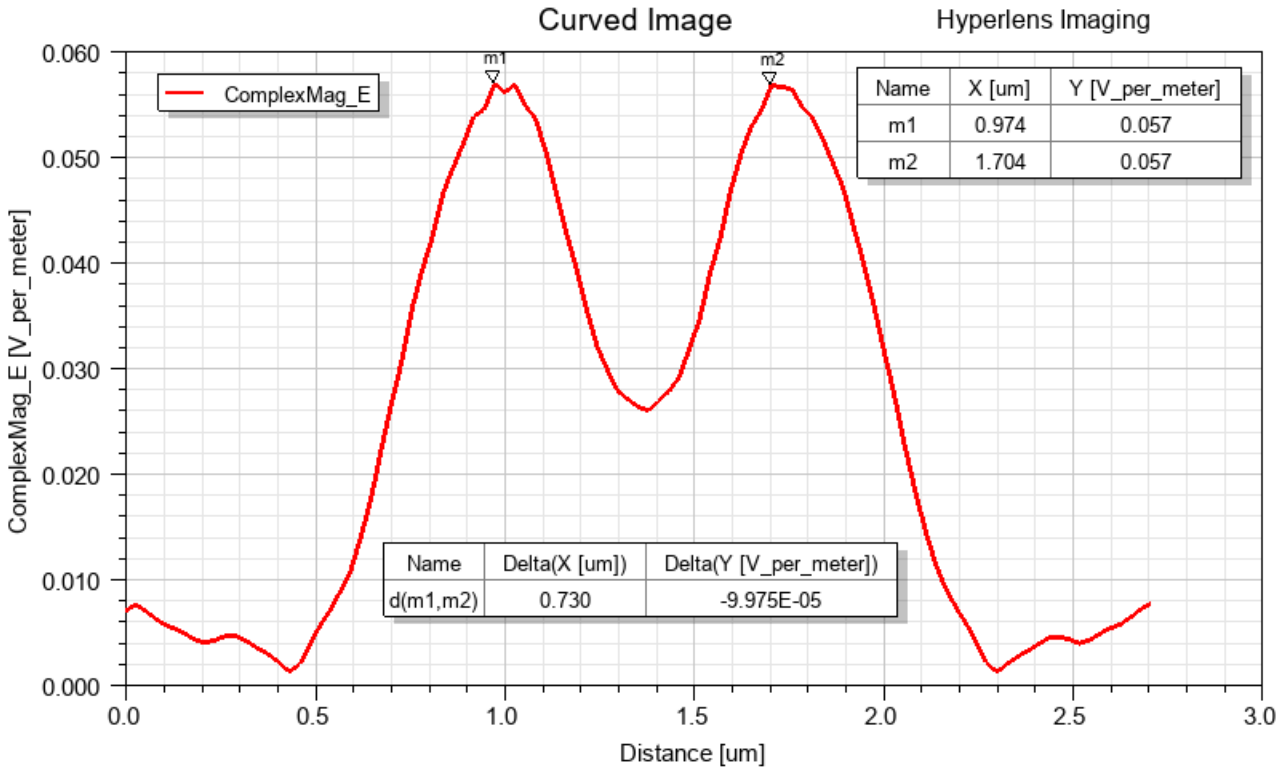


Figure 5: ComplexMag_E Plot – *Hyperlens Imaging* Design, Curved Image Target

Additionally, there is a predefined ComplexMagE plot for the flat image target:

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

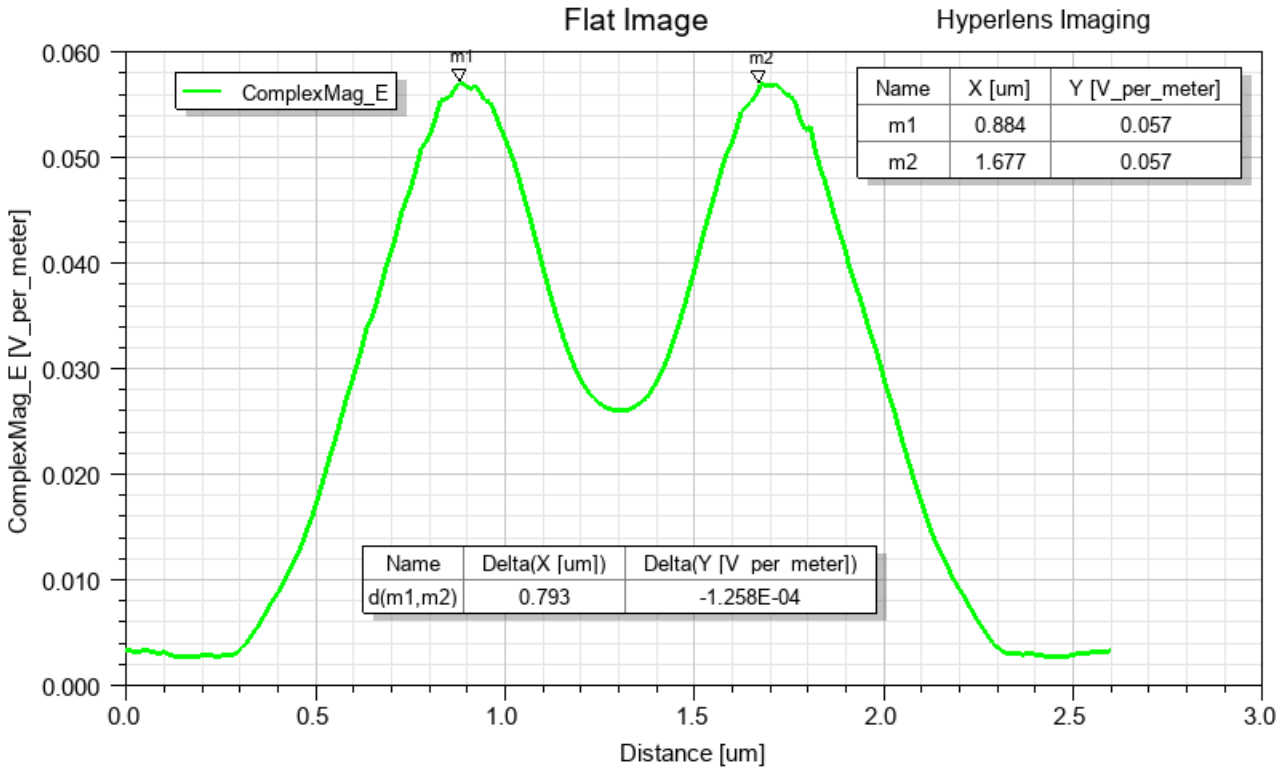


Figure 6: ComplexMag_E Plot – *Hyperlens Imaging* Design, Flat Image Target

Figure 6 clearly shows effective resolution of the small object, as also observed in the overlay in Figure 3 and the plot in Figure 5.

Comparing Results with and without the Hyperlens

A second model, without a hyperlens, is also included in the example project. It is used as a baseline comparison to illustrate the impact of the hyperlens design on field distribution and resolution. Below, the results with and without a hyperlens are compared side-by-side:

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

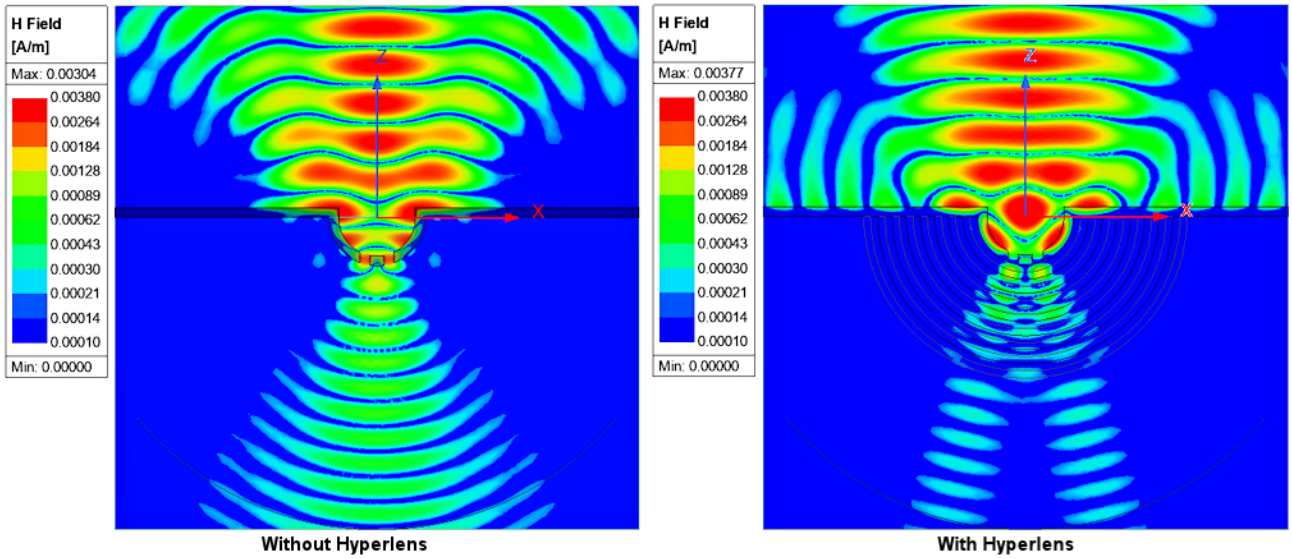


Figure 7: Comparison of H Fields without and with a Hyperlens

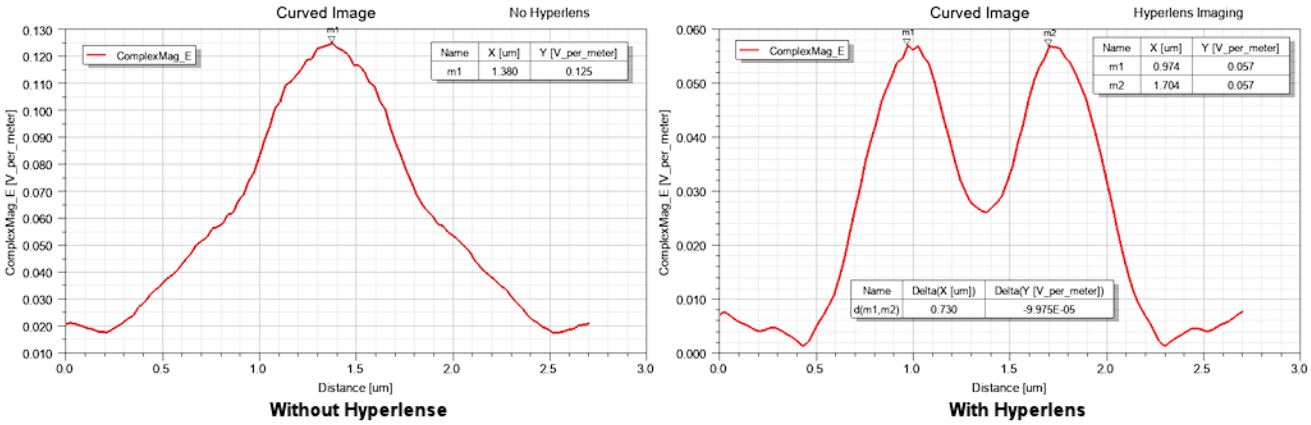


Figure 8: Comparison of ComplexMag_E Plots without and with a Hyperlens (**Curved** Image Target)

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

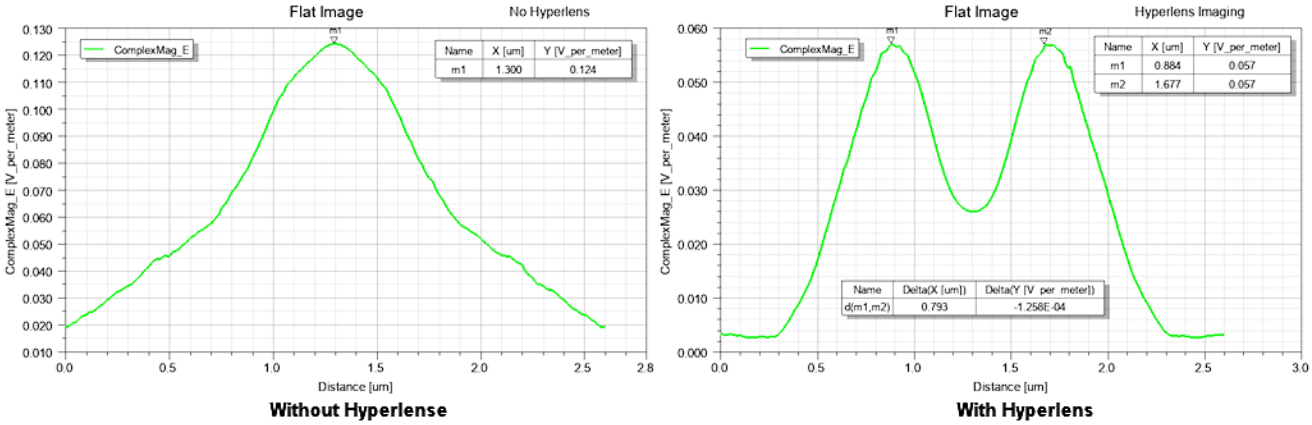


Figure 9: Comparison of ComplexMag_E Plots without and with a Hyperlens (**Flat** Image Target)

Observations:

- The impact of a hyperlens is dramatically observed for both the *Curved Image* results (Figure 8) and the *Flat Image* results (Figure 9). In both cases, the two openings are seen as two peaks in the complex E magnitude traces. The plots from the *No Hyperlens* design show a single peak complex E magnitude at the center of the target, and the two separate openings are not detected.
- Note that the traces in Figure 8 (Cylindrical Images) are roughly symmetrical in shape and are centered at a *Distance* of approximately 1.35 um. Whereas, the traces in Figure 9 (Flat Images) are centered at a *Distance* of about 1.30 um. The flat target object starts at the far left edge of the model geometry and continues to the far right edge. The curved target object begins and ends at some distance inside the left and right extents of the geometry. Nonetheless, the curved object is longer than the straight one. Use the tool **Modeler** > **Measure** > **Objects** and click on each of the polylines that represent image surfaces to determine their lengths. The *Flat_Image_Surface* polyline object is 2600 nm long, and the *Curved_Image_Surface* object is about 2705 nm. The distances from the left endpoint of these objects to their midpoints differ by 52.5 nm (or about 0.052 um). The plots clearly reflect this distance variation.

References:

[1] X. Zhang and Z. Liu, Nat. Mater. 7(6), 435 (2008)
[2] D. Lu and Z. Liu, Nat. Commun. 3, 1205 (2012)
[3] W. Adams, M. Sadatgol, and D. Ö. Güney, AIP Adv. 6, 100701 (2016)
[4] H. Lee, Z. Liu, Y. Xiong, C. Sun, and X. Zhang, Opt. Express 15(24), 15886-15891 (2007)
[5] Z. Liu, H. Lee, Y. Xiong, C. Sun, and X. Zhang, Science 315(5819), 1686 (2007)
[6] A. Rakić, A. Djurišić, J. Elazar, and M. Majewski, Appl. Opt. 37, 5271-5283 (1998)

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

Split Ring Resonator

Abstract

The advent of metamaterials has represented a paradigm shift in how engineers design new electromagnetic devices and even media with unusual properties. One canonical example of a metamaterial is the split ring resonator, the first to experimentally verify the possibility of a negative refractive index. This example shows how to simulate a split ring resonator using metamaterials with a negative refractive index near 11 GHz using Ansys HFSS.

The retrieved material parameters are extracted from the first design as datasets and then used in two other designs to create equivalent and simpler models. In addition to the vacuum region, the simplified design contains only a monolithic cube of equivalent material, with the detailed geometry (substrate and conductors) of the split ring resonator excluded.

This example model document is available from the HFSS online help and from the PDF included with the example model in the software installation (accessible from the Project Manager when this example project is open).

Simulation Time / RAM: (meshing, adaptive solution, and frequency sweep in 2025 R1, 6 cores with 64 GB RAM)

- **1_Original:** 5 minutes: 20 seconds with 2.76 GB max. memory used per process
- **2_Complex_Equiv_Matl:** 42 seconds with 206 MB max. memory used per process
- **3_Equiv_Matl_Using_Loss_Tangent:** 40 seconds with 206 MB max. memory used per process

The equivalent material approach is much faster and, if used with a more complex design, the benefit in RAM and solution time will be even greater.

Introduction

The split ring resonator is the classic example of a metamaterial that can achieve negative refractive index. First verified experimentally in 2001 [1], much of the motivation for development of such materials can be attributed to the prospect of a “perfect lens” [2]. However, many other interesting applications have been identified in the last decade.

Since metamaterials are usually complex composite structures, analytical solutions to their scattering properties quickly become unfeasible. Therefore, numerical simulation is a crucial step in the design process. In this example, we show how to model and analyze a split ring resonator [3] in Ansys HFSS. The example shows some of the most important points of metamaterial simulation and characterization, including the boundary and port definitions, solution setup, and even effective parameter retrieval.

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

Model Construction and Setup Details

Design *1_Original*:

Only the first design (*1_Original*) contains the split ring resonator substrate and conductor geometry. For the two subsequent designs, a simple cube of equivalent material, with parameters extracted from the first design, is substituted for the split ring resonator assembly.

Using the geometric parameters in Ref. [3], the geometry of the split ring resonator was modeled in HFSS. Given the relative simplicity of the structure, there are several valid ways to draw it. In this case, we first started with a box to represent the FR4 substrate. Next, polylines were drawn to trace out the metallic pieces on both sides of the substrate, which were then thickened to the desired conductor dimensions by setting the *Cross Section* to “Rectangle” and specifying the desired variables for the width and height of the traces.

After drawing the substrate, resonator, and wire structures, a region was added to define the boundaries of the computational domain. This region is important so that the correct unit cell size along x and y is defined, and so that the ports can be placed sufficiently away from the near-fields induced on the structure. This region size ensures that the scattering parameters are calculated properly. The resulting geometry is shown below:

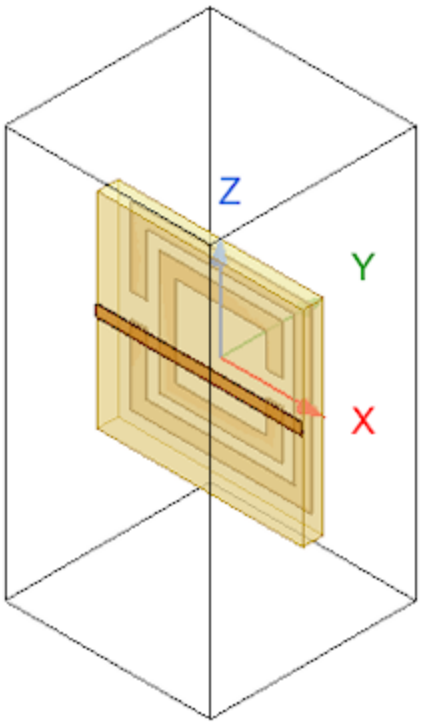


Figure 1: Design *1_Original* Geometry

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

Coupled Lattice Pair Boundaries and Floquet Port Excitations

After drawing the geometry and defining the materials, the next step was to assign boundaries and excitations. Since we modeled a single unit cell, which is to be situated in a periodic lattice, coupled *Lattice Pairs* are the correct choice for the boundaries on the ±x and ±y faces of the model domain:

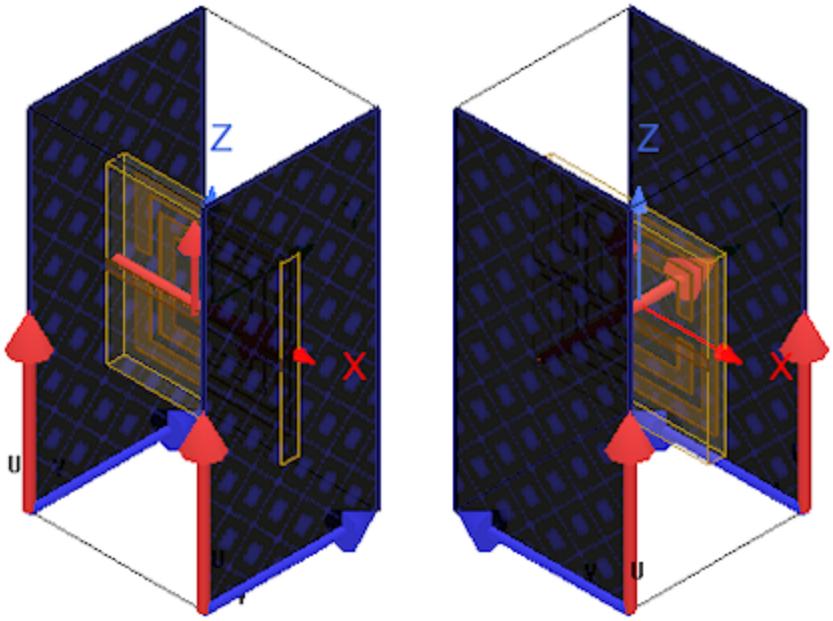


Figure 2: Coupled Lattice Pair Boundaries

Floquet ports were then added as excitations to the ±z faces of the model domain:

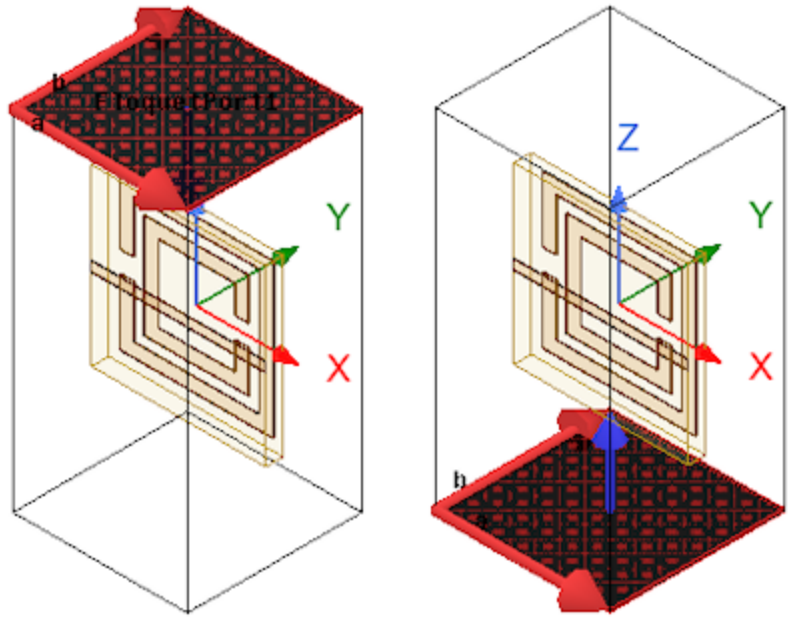


Figure 3: Floquet Port Excitations

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

A crucial setting worth noting for the Floquet ports in this case is the de-embedding. Since accurate calculation of the S-parameter phase is important for effective parameter retrieval, the ports must be de-embedded to the surface of the unit cell, as indicated by the blue arrows in the above figures.

Mesh Settings and Operations:

To ensure higher quality elements, the *Mesh Method* has been set to **TAU** for all three designs. Additionally, because the thin solid conductors in the first design (*1_Original*) are more problematic than the simple box features of the other two designs, the following mesh operations were defined for the first design only. These operations provide the desired element sizes and help to ensure the success of the TAU mesh:

- Length-Based, On Selection: 0.1 mm Maximum Length
- Surface Representation Priority for TAU: High (assigned to all conductors)

Setup:

All three designs have a single adaptive solution frequency of 11 GHz specified and an interpolated frequency sweep from 1–20 GHz in 0.05 GHz steps. Only the convection criteria differ. Design *1_Original* allows more adaptive passes (20 maximum versus 15 for the other two) and a relaxed Delta S value (0.02 versus 0.01 for the other two). The differing convergence criteria ensure successful solving of the more complex geometry of the first design. For all three solutions, two consecutive converged steps must be achieved.

Effective Material Parameter Retrieval in HFSS

Perhaps the most important and difficult step in metamaterial analysis is retrieval of the effective material parameters from the frequency-dependent S-parameters obtained by the simulation or scattering experiment. Thanks to the Output Variables functionality, it is possible to perform this step in HFSS without having to export any data to an external script, though one may do so if desired.

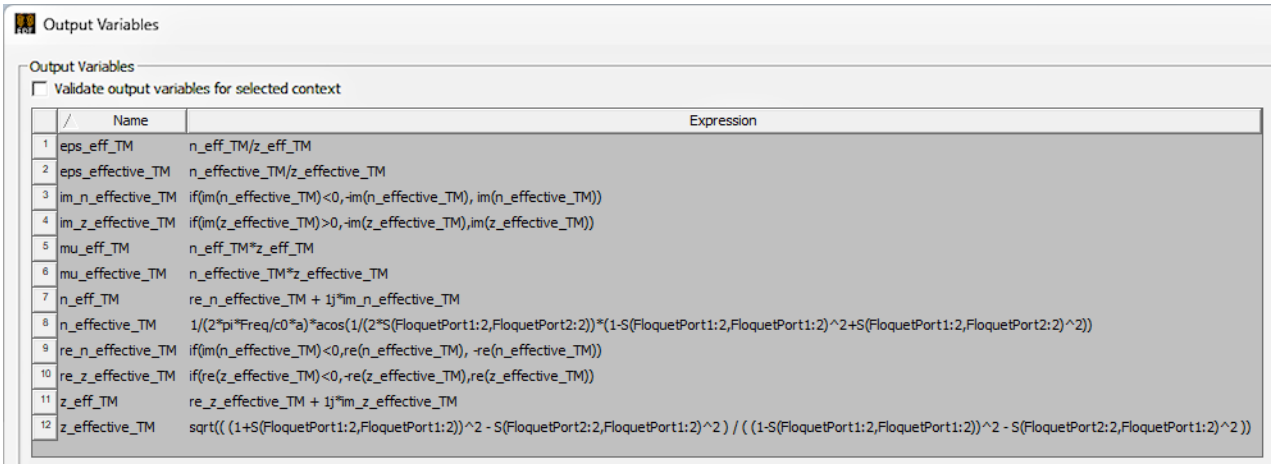
Ref. [3] gives the relevant equations needed for performing the effective parameter retrieval, specifically Equations 9 and 10:

$$n = \frac{1}{kd}\cos^{-1}\left[\frac{1}{2S_{21}}(1 - S_{11}^2 + S_{21}^2)\right]$$

$$z = \sqrt{\frac{(1+S_{11})^2 - S_{21}^2}{(1-S_{11})^2 - S_{21}^2}}$$

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

In the *Output Variables* window for this example project (**HFSS** > **Results** > **Output Variables**), you will find the equations for the effective parameters used in HFSS. They are also shown below:



	Name	Expression
1	eps_eff_TM	n_eff_TM/z_eff_TM
2	eps_effective_TM	n_effective_TM/z_effective_TM
3	im_n_effective_TM	if(im(n_effective_TM)<0,-im(n_effective_TM), im(n_effective_TM))
4	im_z_effective_TM	if(im(z_effective_TM)>0,-im(z_effective_TM),im(z_effective_TM))
5	mu_eff_TM	n_eff_TM*z_eff_TM
6	mu_effective_TM	n_effective_TM*z_effective_TM
7	n_eff_TM	re_n_effective_TM + 1j*im_n_effective_TM
8	n_effective_TM	1/(2*pi*Freq/c0*a)*acos(1/(2*S(FloquetPort1:2,FloquetPort2:2))*(1-S(FloquetPort1:2,FloquetPort1:2)^2+S(FloquetPort1:2,FloquetPort2:2)^2))
9	re_n_effective_TM	if(im(n_effective_TM)<0,re(n_effective_TM), -re(n_effective_TM))
10	re_z_effective_TM	if(re(z_effective_TM)<0,-re(z_effective_TM),re(z_effective_TM))
11	z_eff_TM	re_z_effective_TM + 1j*im_z_effective_TM
12	z_effective_TM	sqrt(((1+S(FloquetPort1:2,FloquetPort1:2))^2 - S(FloquetPort2:2,FloquetPort1:2)^2) / ((1-S(FloquetPort1:2,FloquetPort1:2))^2 - S(FloquetPort2:2,FloquetPort1:2)^2))

Figure 4: List of Defined Output Variables (All Designs)

> **Important:**
>
> Parameter retrieval is a complicated process, and some care must be taken when defining these equations for your projects. There are some conditional statements that are required to avoid discontinuities and to make the resulting effective parameters physically meaningful and passive. As an example, view the results below (Figures 5–8) for the retrieved parameters that were calculated before applying the required conditions, that is, naively applying Equations 9 and 10 from Reference [3].

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

Retrieved Material Parameters (before Applying Necessary Conditions):

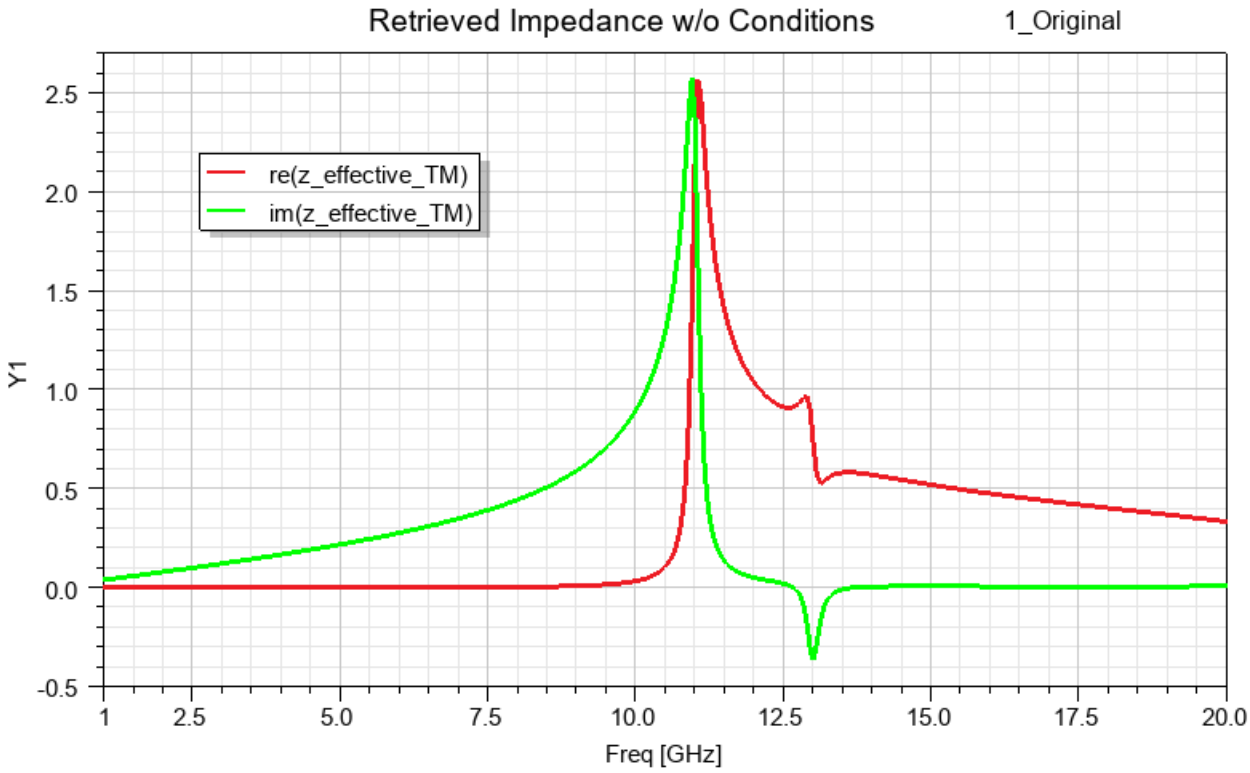


Figure 5: *Retrieved Impedance* Plot without Required Conditional Statements

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

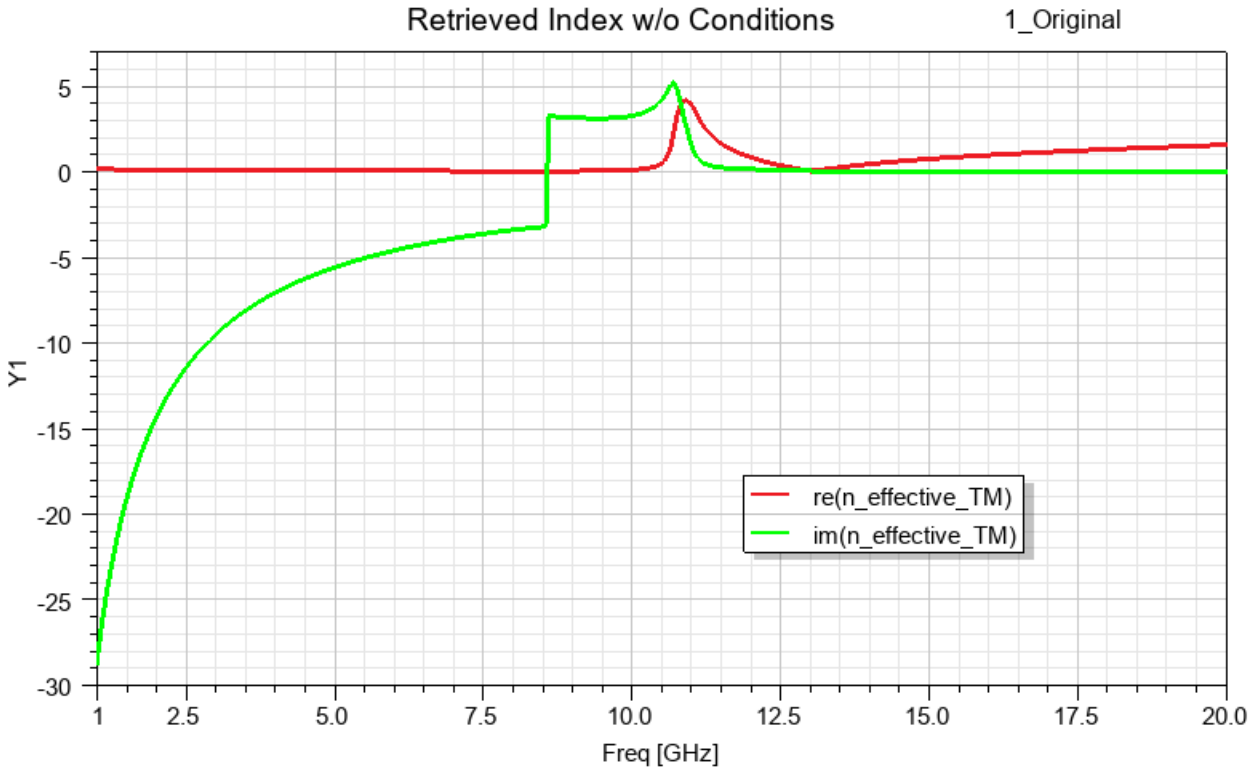


Figure 6: *Retrieved Index* Plot without Required Conditional Statements

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

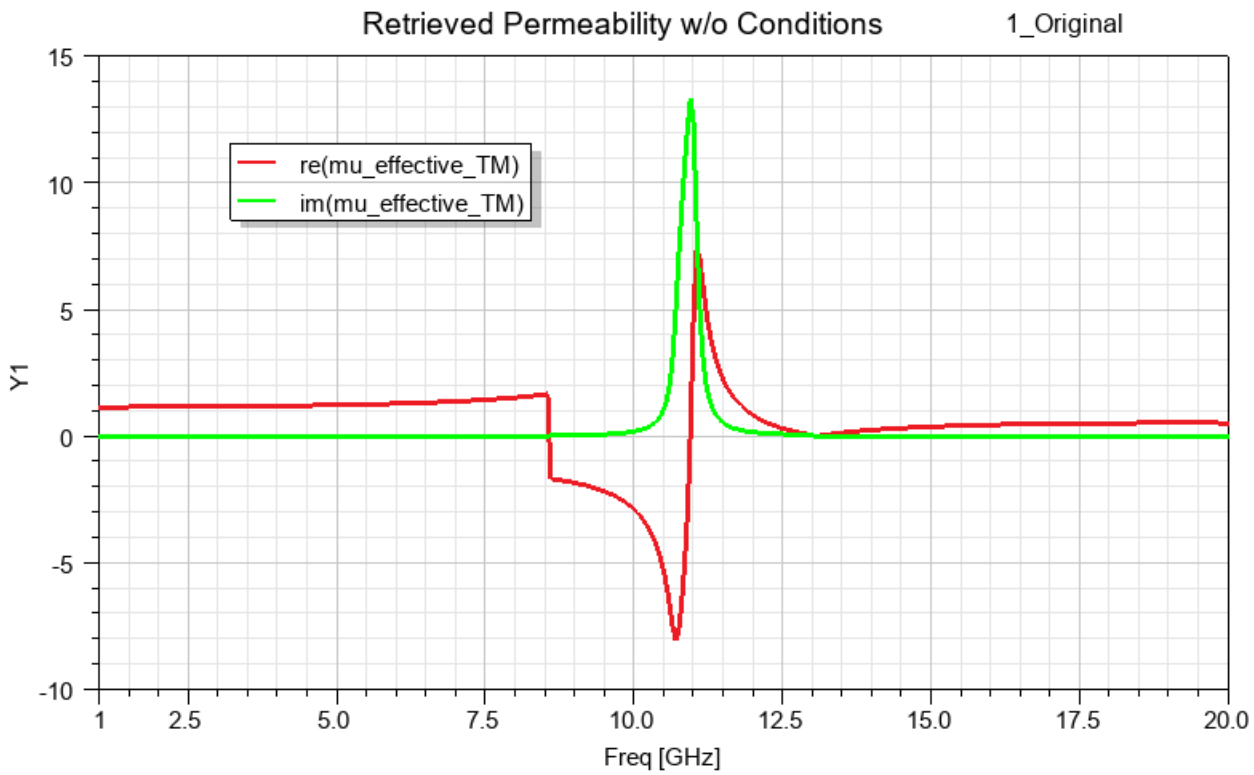


Figure 7: *Retrieved Permeability* Plot without Required Conditional Statements

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

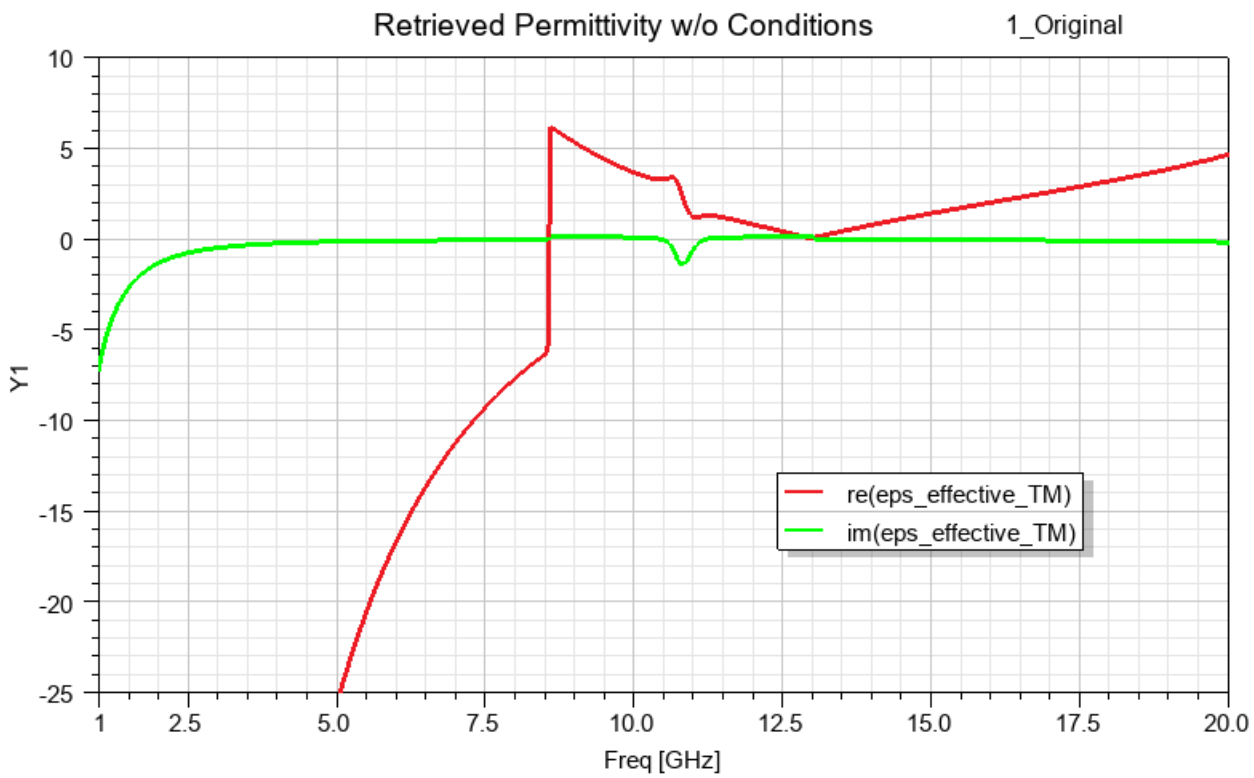


Figure 8: *Retrieved Permittivity* Plot without Required Conditional Statements

Note the unphysical discontinuities in the preceding retrieved index, permeability, and permittivity plots. Additionally, there is clearly an issue with the signs of the retrieved index and impedance. These clues indicate that we have not calculated physical and passive material parameters.

Next, variables with the conditional statements required to correct the retrieved parameters were used as the basis of a second set of plots. The final results showing the correct retrieved parameters follow (Figures 9–12).

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

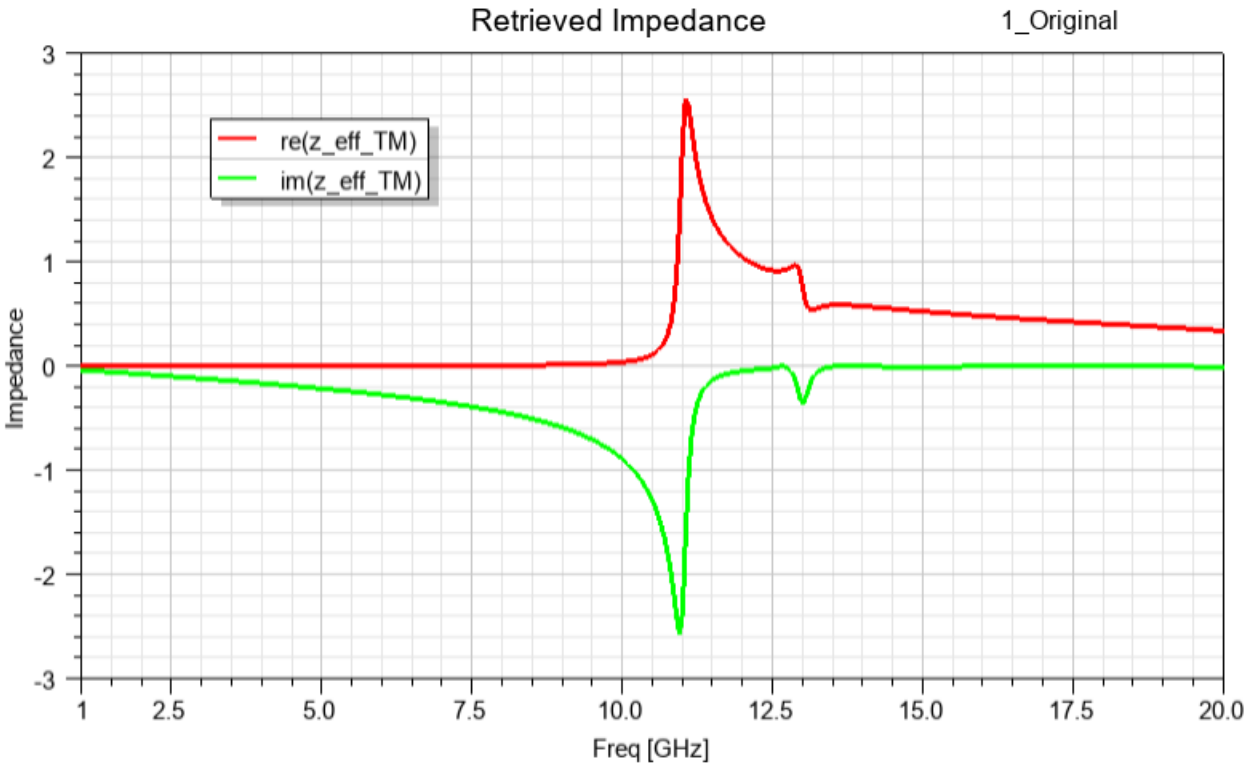


Figure 9: *Retrieved Impedance* Plot after Applying Conditional Statements

The following figure confirms the negative sign of the real part of the refractive index (red trace) near resonance:

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

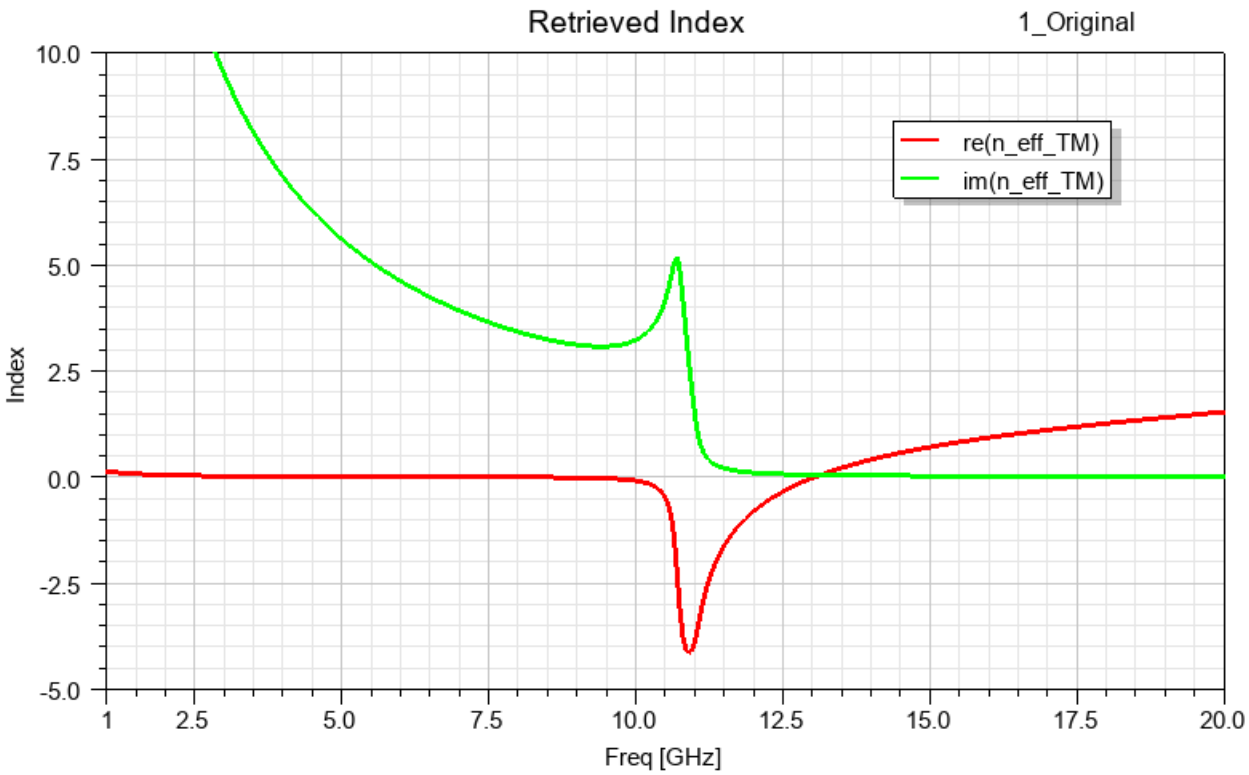


Figure 10: *Retrieved Index* Plot after Applying Conditional Statements

Also, the magnetic resonance of the structure is clearly visible from the following retrieved permeability figure:

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

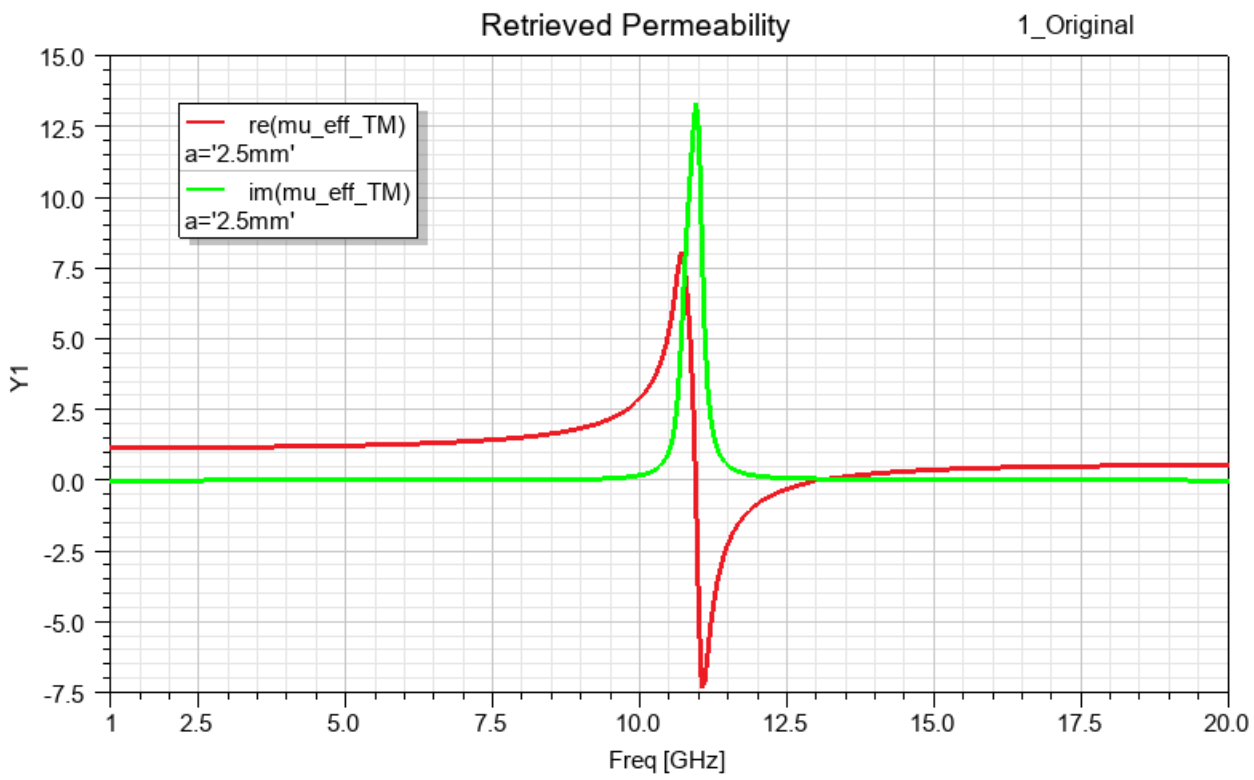


Figure 11: *Retrieved Permeability* Plot after Applying Conditional Statements

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

It may appear strange that there is an antiresonance in the negative, imaginary part of the retrieved permittivity (the green trace in the following figure). However this can be attributed to the fact that the homogenization limit has not quite been reached. More details are given in the text of Reference [3]:

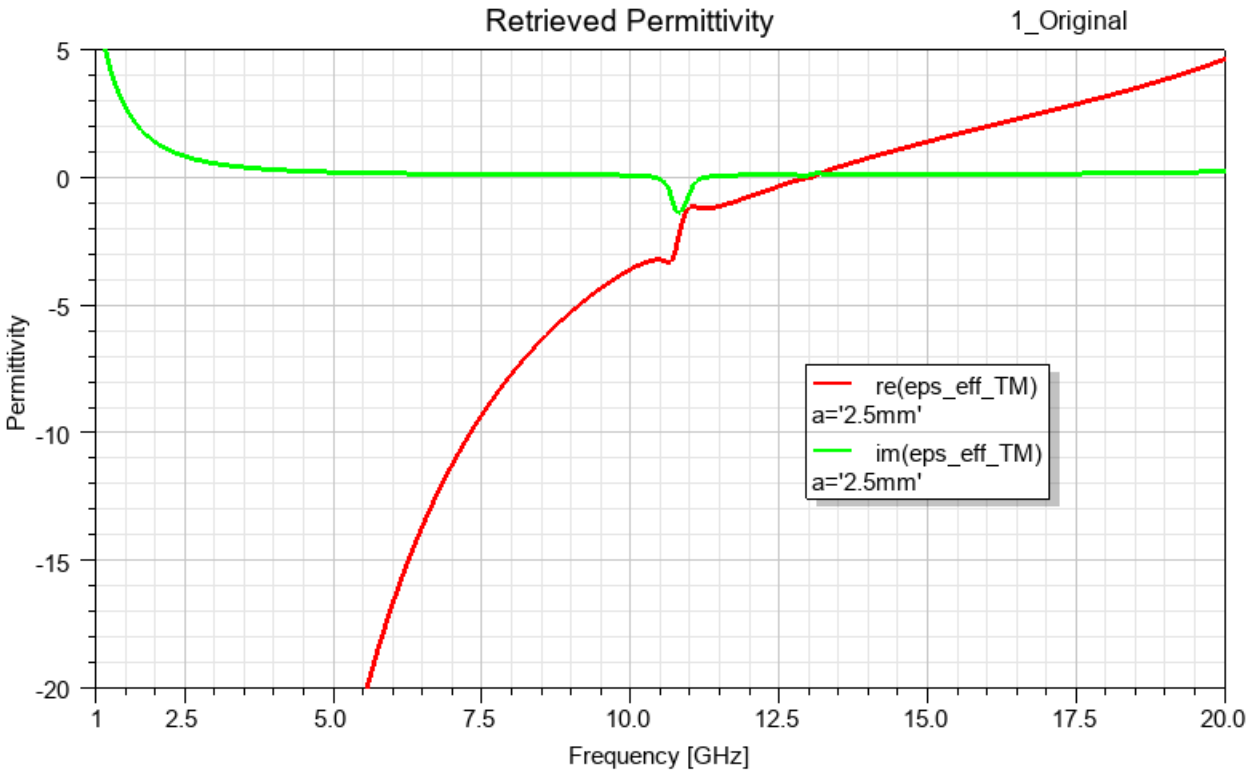


Figure 12: *Retrieved Permittivity* Plot after Applying Conditional Statements

Constructing an Equivalent Material from the Retrieved Properties:

1. Right-click inside one of the retrieved parameter plot windows and choose Export from the shortcut menu. Use the default CSV format.
2. Import the CSV as a project dataset so that it is available to all designs in the project:
    a. From the menu bar, click **Project** > **Datasets**.
    b. In the *Datasets* dialog box that appears, click **Import**.
    c. Browse to and select the exported CSV file and click **Open**.

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

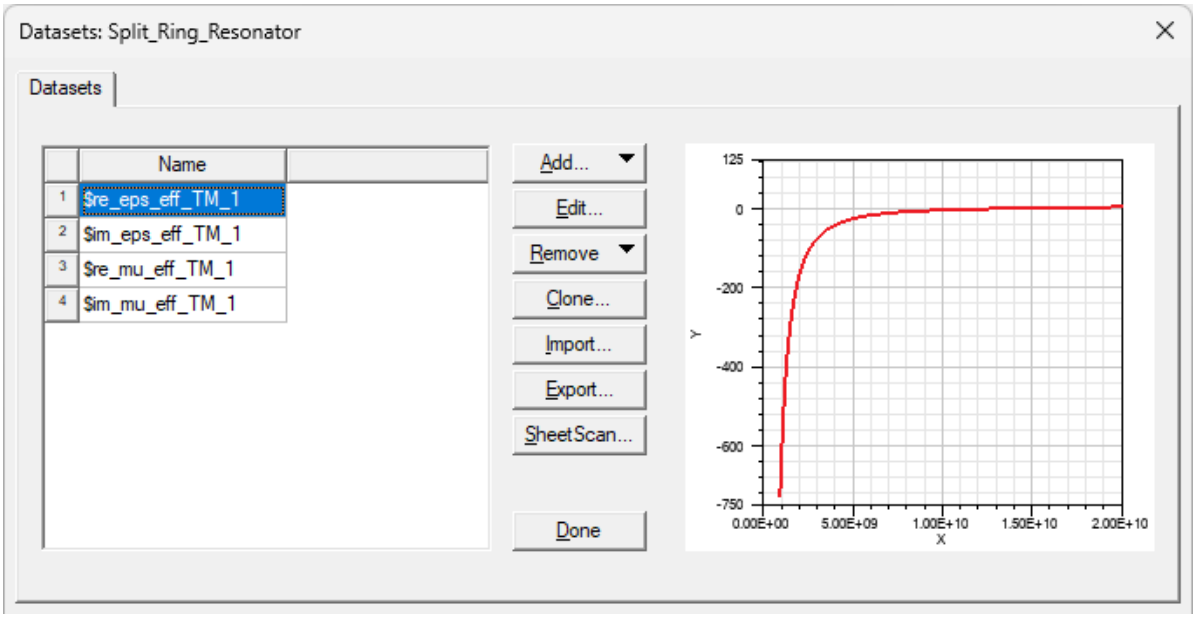


Figure 13: *Datasets* Dialog Box

As an illustration of the two different ways to define equivalent material properties, two equivalent designs have been built:

- Design *2_Complex_Equiv_Matl* uses a complex equivalent material based on a complex definition for permittivity and permeability.

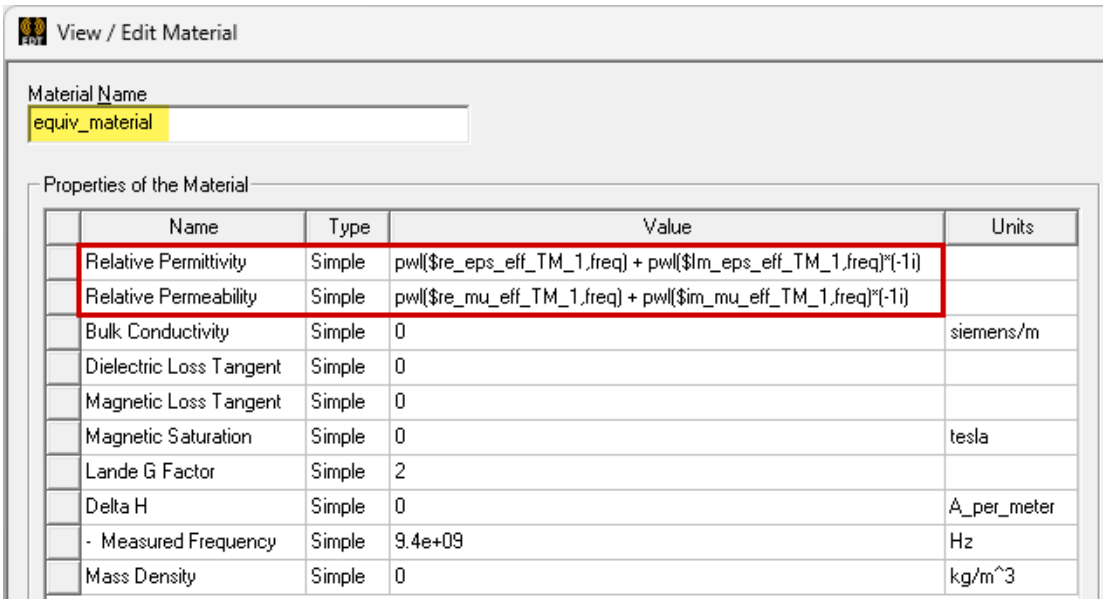
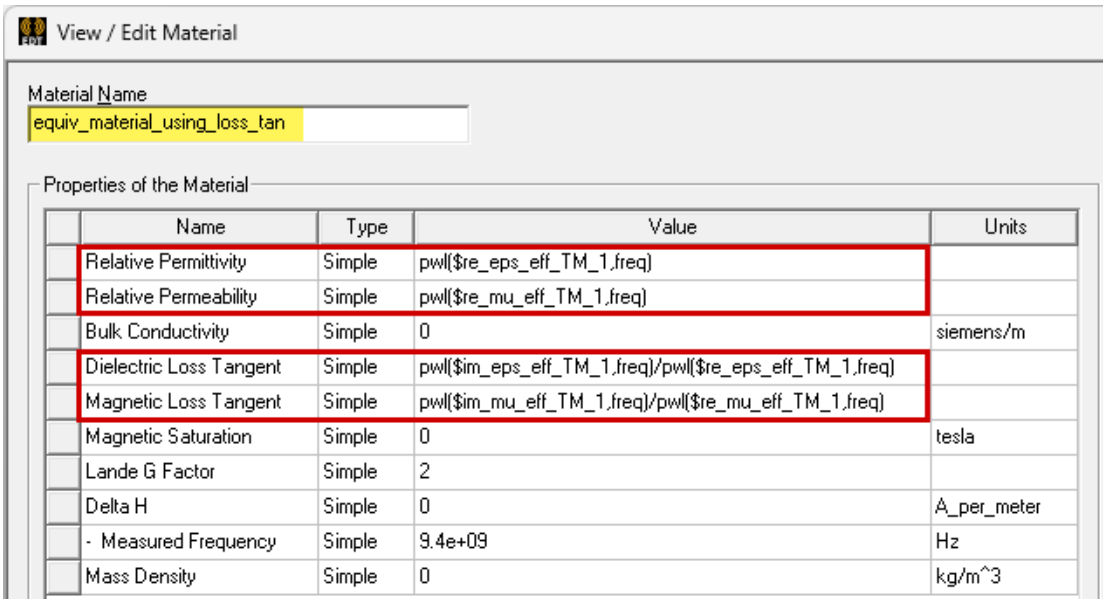


Figure 14: Complex Equivalent Material Properties – Used in Design 2

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

- Design *3_Equiv_Matl_Using_Loss_Tan* uses a material based on real values for permittivity and permeability but with the dielectric and magnetic loss tangents defined.



	Name	Type	Value	Units
	Relative Permittivity	Simple	pwl($re_eps_eff_TM_1,freq)	
	Relative Permeability	Simple	pwl($re_mu_eff_TM_1,freq)	
	Bulk Conductivity	Simple	0	siemens/m
	Dielectric Loss Tangent	Simple	pwl($im_eps_eff_TM_1,freq)/pwl($re_eps_eff_TM_1,freq)	
	Magnetic Loss Tangent	Simple	pwl($im_mu_eff_TM_1,freq)/pwl($re_mu_eff_TM_1,freq)	
	Magnetic Saturation	Simple	0	tesla
	Lande G Factor	Simple	2	
	Delta H	Simple	0	A_per_meter
	- Measured Frequency	Simple	9.4e+09	Hz
	Mass Density	Simple	0	kg/m^3

Figure 15: Equivalent Material with Loss Tangent Properties – Used in Design 3

Alternative Metamaterial Designs:

Now that we have the proper metamaterial properties, they can be used in designs *2_Complex_Equiv_Matl* and *3_Equiv_Matl_Using_Loss_Tan*. Both of these designs substitute a monolithic cube of metamaterial for the detailed geometry of the split ring resonator substrate and conductive traces.

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

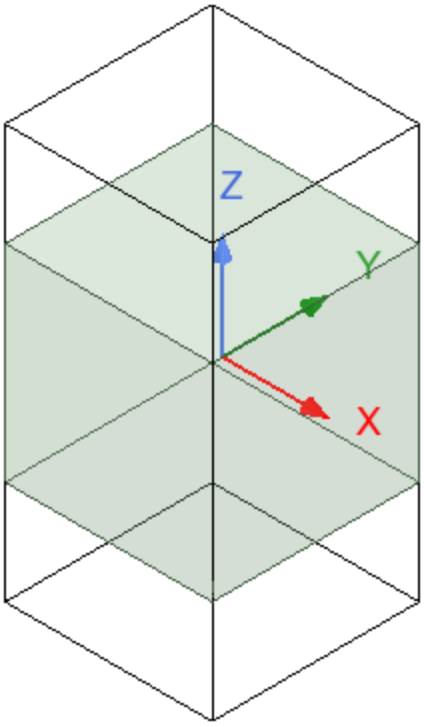


Figure 16: Metamaterials Model Geometry – Used in Designs 2 and 3

Post Processing

For each design, the *Results* branch of the Project Manager lists the available predefined plots. All three designs contain plots of the retrieved metamaterial properties—impedance, index, permeability, and permittivity. Additionally, each includes an S Parameters plot (S12). For this plot, trace data from the second and third designs has been copied to the first design's plot to directly compare S12 results between the original split ring resonator design and the equivalent metamaterial designs. After analyzing the setup, double-click any of these entries to display the associated plot. The retrieved material parameter plots have already been covered in the description of the retrieval process.

In this section we will look at a mesh overlay on the second design, to show the relative coarseness of the mesh that provides good, converged results when using simplified geometry and metamaterials:

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

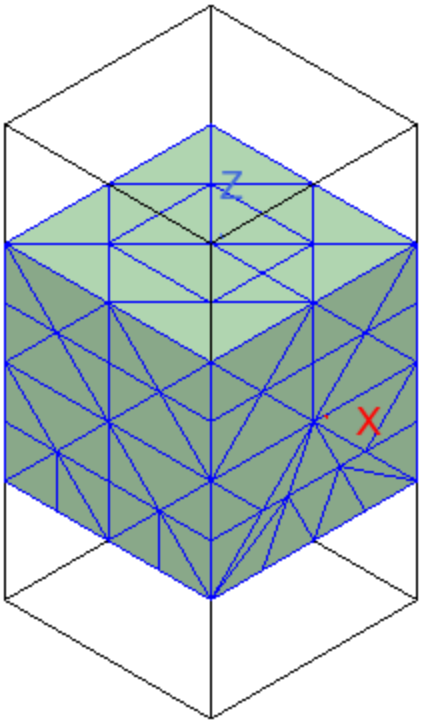


Figure 17: Equivalent Material's Mesh Overlay – Design 2

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

We will also look at comparative S Parameter results from all three designs to show how closely the metamaterials match the results of the original resonator geometry and conventional materials:

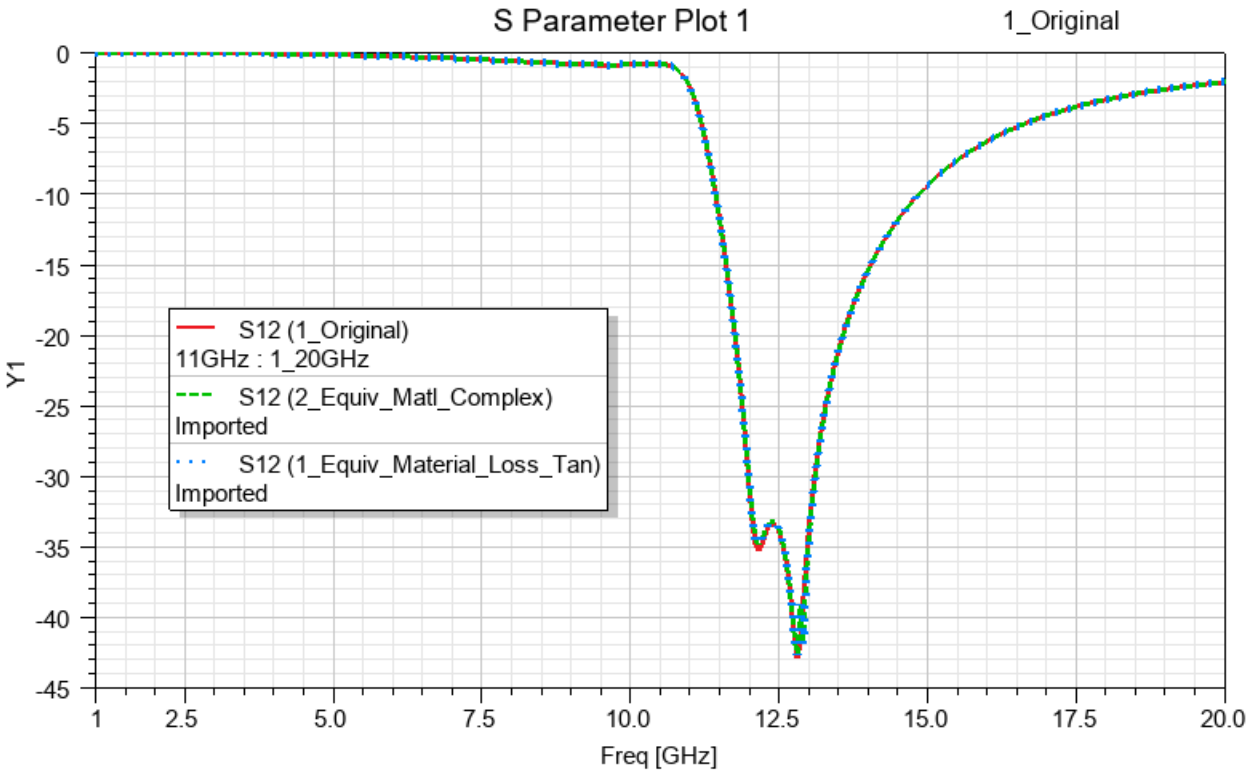


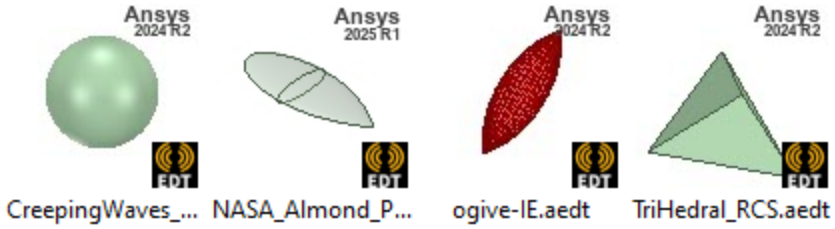
Figure 18: Comparison of S Parameter Results of all Three Designs

References:

[1] R. A. Shelby, D. R. Smith, S. Schultz, Science 292 (5514), 77-79 (2001).
[2] J. B. Pendry, Phys. Rev. Lett. 85, 3966-3969 (2000).
[3] D. R. Smith, D. C. Vier, Th. Koschny, C. M. Soukoulis, Phys. Rev. E 71, 036617 (2005).

RCS Examples (Radar Cross Section)

Click graphics for descriptions of the following HFSS RCS examples:

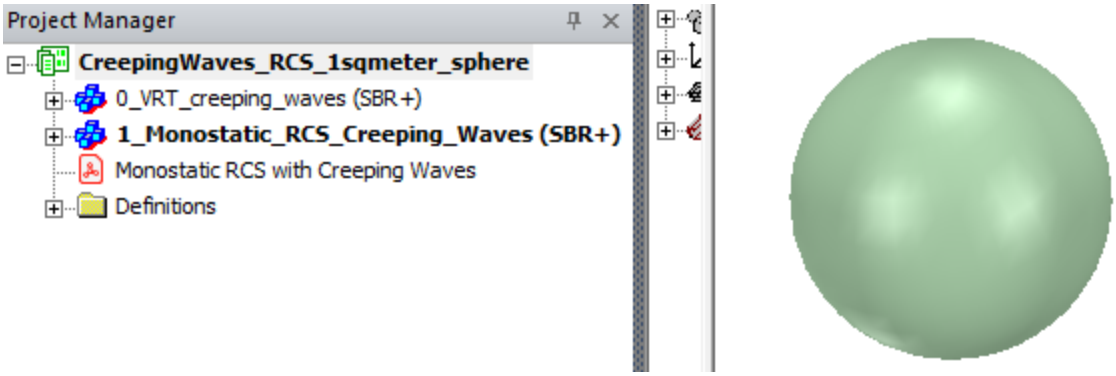


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Monostatic RCS of a PEC Sphere Using SBR Solver with Creeping Waves

Description - The simulated geometry is a perfectly conducting sphere of radius 0.56419 meter. Monostatic RCS value of such a sphere in the optical region is 1 square meter. The term creeping wave refers to fields that propagate along smoothly curved surfaces. Finite fields in the shadow of a curved surface are due to the propagation of a creeping wave. Creeping waves decay quickly as they propagate, since they progressively shed energy to space along their direction of travel. As the curved structure increases in electrical size, the shadow region fields become weaker. Hence, while creeping waves keep propagating on a continuous smooth surface, such as a sphere, the currents they paint on a surface are important only up to a certain traveled distance. The goal of this example is to show how to simulate the Monostatic RCS of a sphere vs. frequency using SBR+ solver and compare the results simulated with and without enabling CW option.

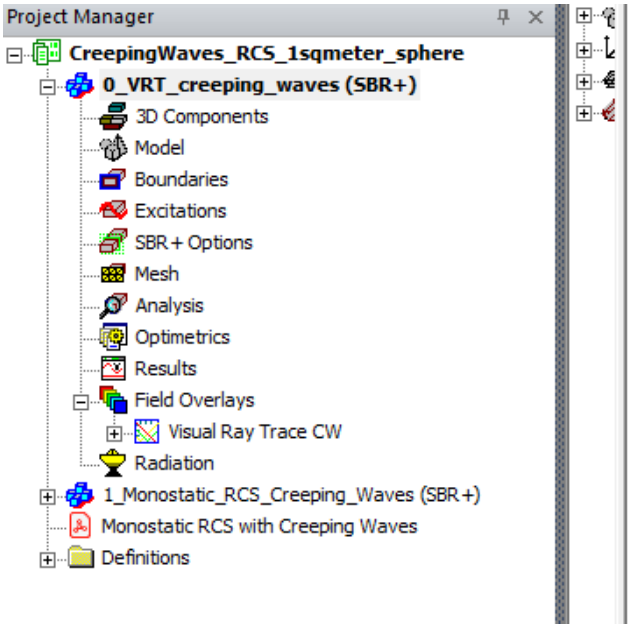
This example demonstrates how to use Creeping Waves (CW) option for RCS simulations of a conducing sphere in HFSS SBR+ design type. The project “CreepingWaves_RCS_1sqmeter_sphere” contains two designs: “0_VRT_creeping_waves” to visualize the CW rays and choose the proper CW setup options; “1_Monostatic_RCS_Creeping_Waves” to simulate the Monostatic RCS values. The Monostatic RCS with Creeping Waves pdf includes details and guidance on using and interpreting the model.

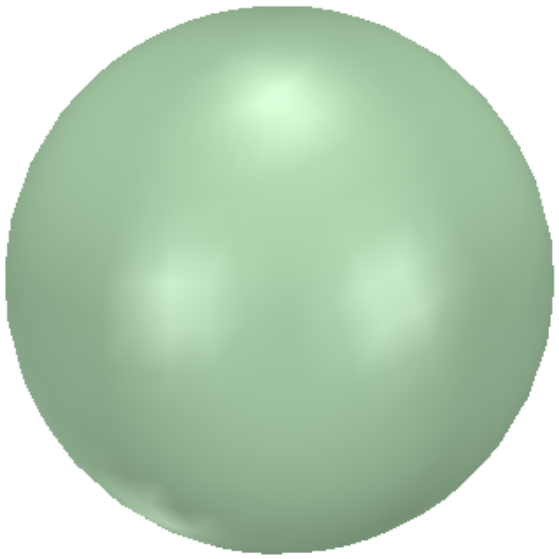


Design “0_VRT_creeping_waves”

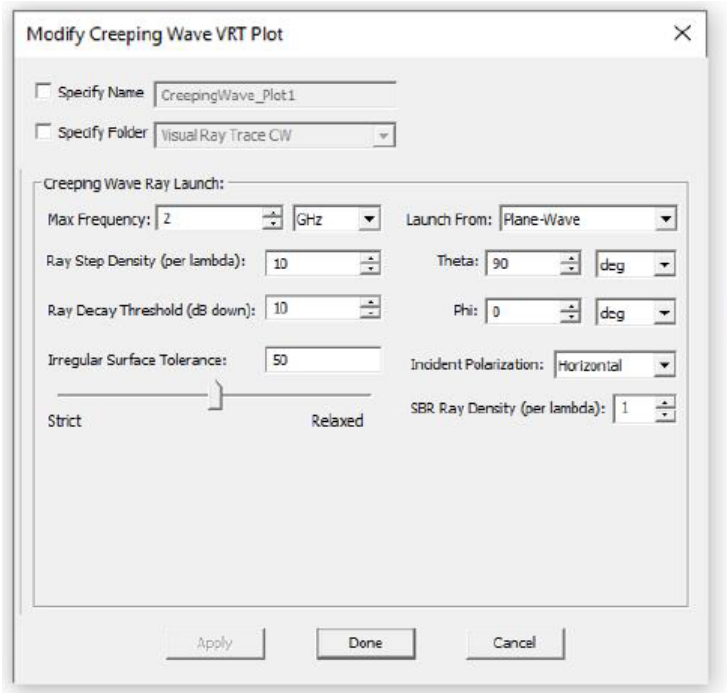
Design “0_VRT_creeping_waves”. VRT (Visual Ray Tracing) of creeping waves analysis is recommended to help configure the creeping wave settings. Note that there is neither excitation nor setup present in this design – VRT analysis and setups are completely different from the RCS solution setup. To add VRT plot, right click on Field Overlays in the Project Manager, choose **Plot VRT > Creeping Waves**.

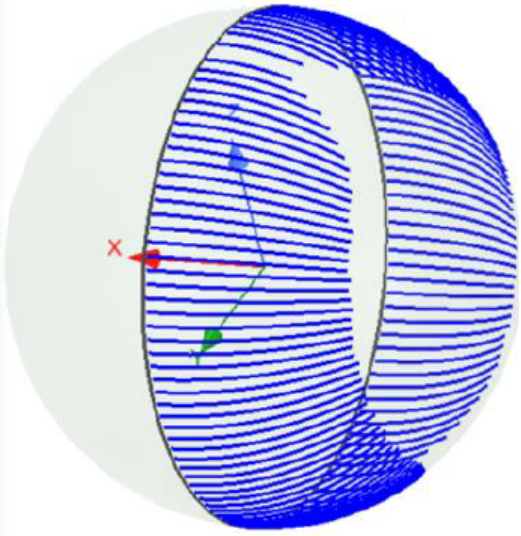
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





You are encouraged to investigate how different options change the CW rays. Selecting Visual Ray Trace CW and CreepingWave_Plot1 entries in **Project Manager** window, one can edit and investigate options in the **Properties** window. See more details about CW rays in Appendix 1 of the Monostatic RCS with Creeping Waves pdf included in the Project.

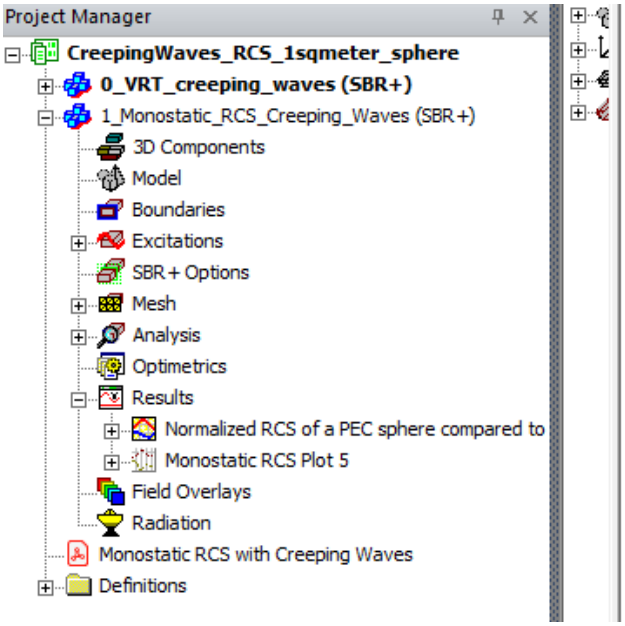


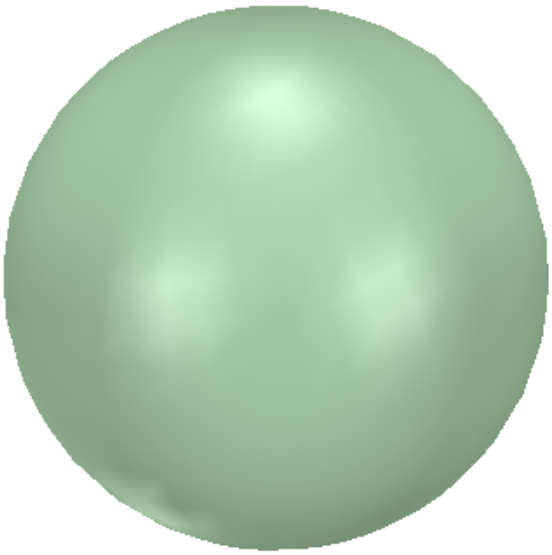


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

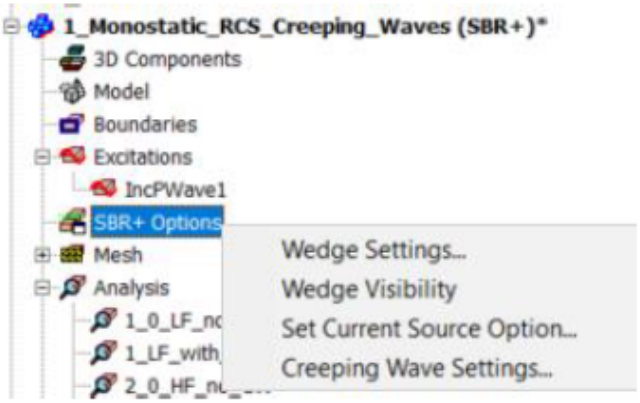
1_Monostatic_RCS_Creeping_Waves

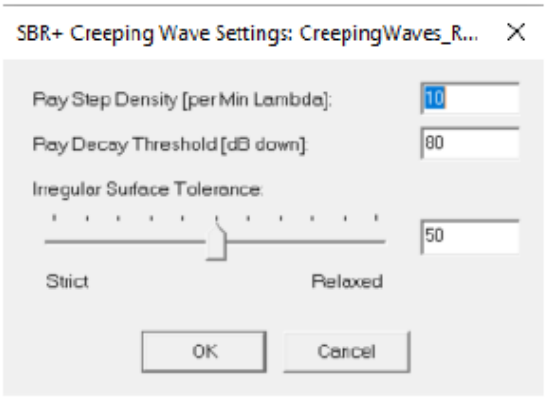
This model has 4 Analysis setups for Monostatic RCS of the sphere to cover two frequency ranges and to compare results with and without CW option enabled. CW options are part of SBR+ options and set throughout the design.





Since enabling the CW options adds simulation time, you should consider the results from VRT CW.

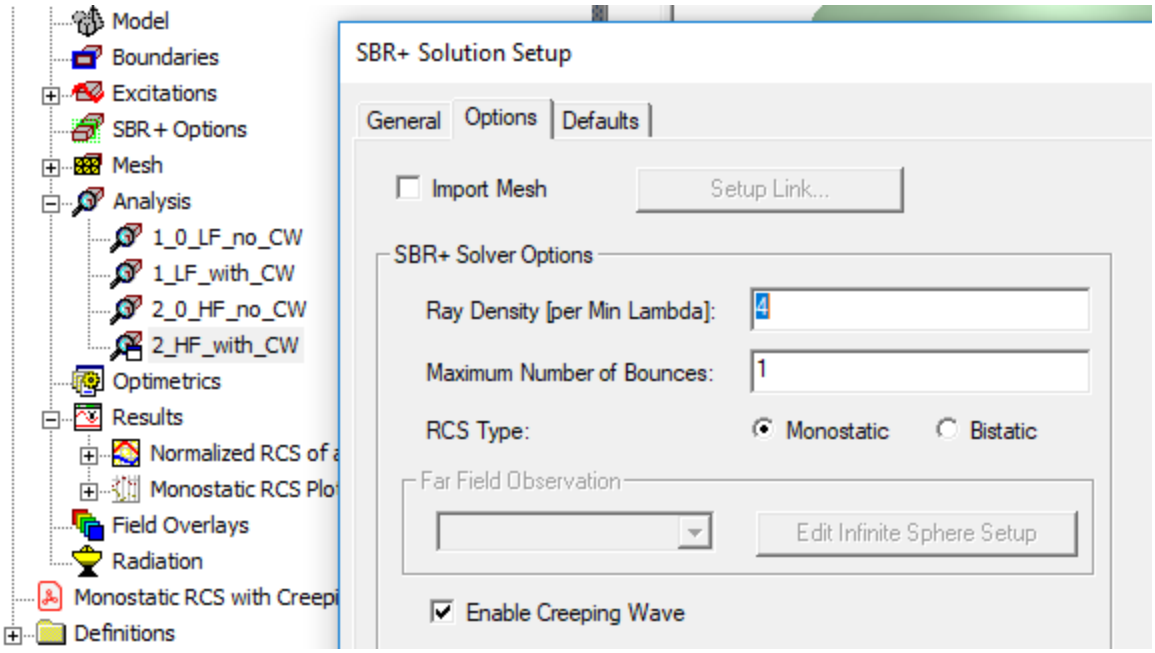




The same design might have several solution setups with and without creeping waves options. CW option is enabled in Options Tab of solution setup (see Fig.2). Setup 1_0_LF_no_CW: frequency from 0.05GHz to 2GHz, CW is not enabled Setup 1_LF_with_CW: frequency from

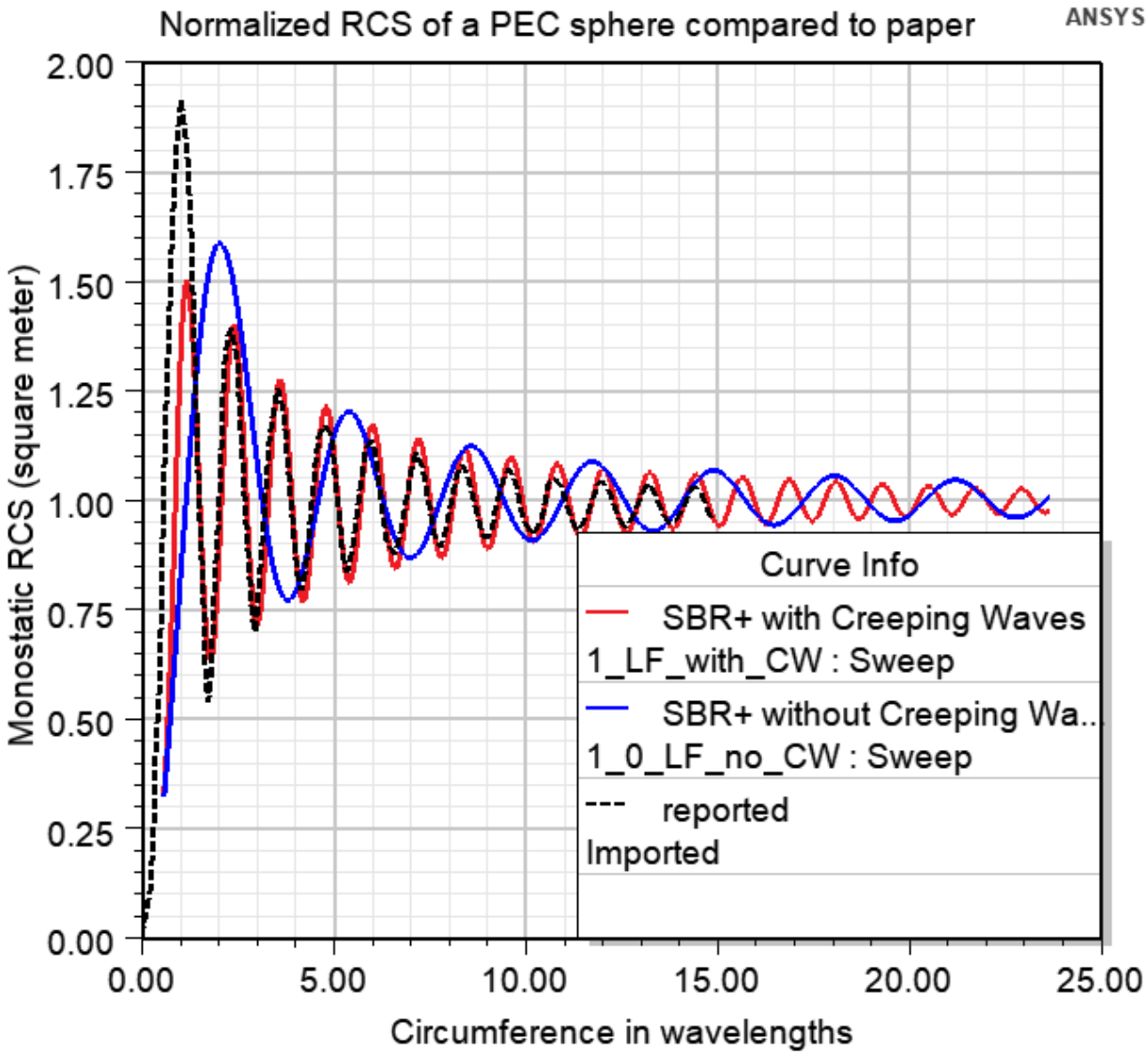
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0.05GHz to 2GHz, CW is enabled Setup 2_0_HF_no_CW: frequency from 1GHz to 10GHz, CW is not enabled Setup 2_HF_with_CW: frequency from 1GHz to 10GHz, CW is enabled.



The project includes reports.

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



NASA Almond Parametric

Abstract

This example is an HFSS design for reference results of a NASA almond monostatic RCS (radar cross section) analysis. The design is solved using the Integral Equation Method and is parameterized. A discrete sweep includes the adaptive solution frequency and two other frequencies. Two design are include, identical except for the polarization of the incident wave. The first design is for horizontal (HH) polarization, and the second, vertical (VV). The results match the reference publication[1] at 1.19, 7, and 9.92 Ghz for vertical and horizontal polarization.

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

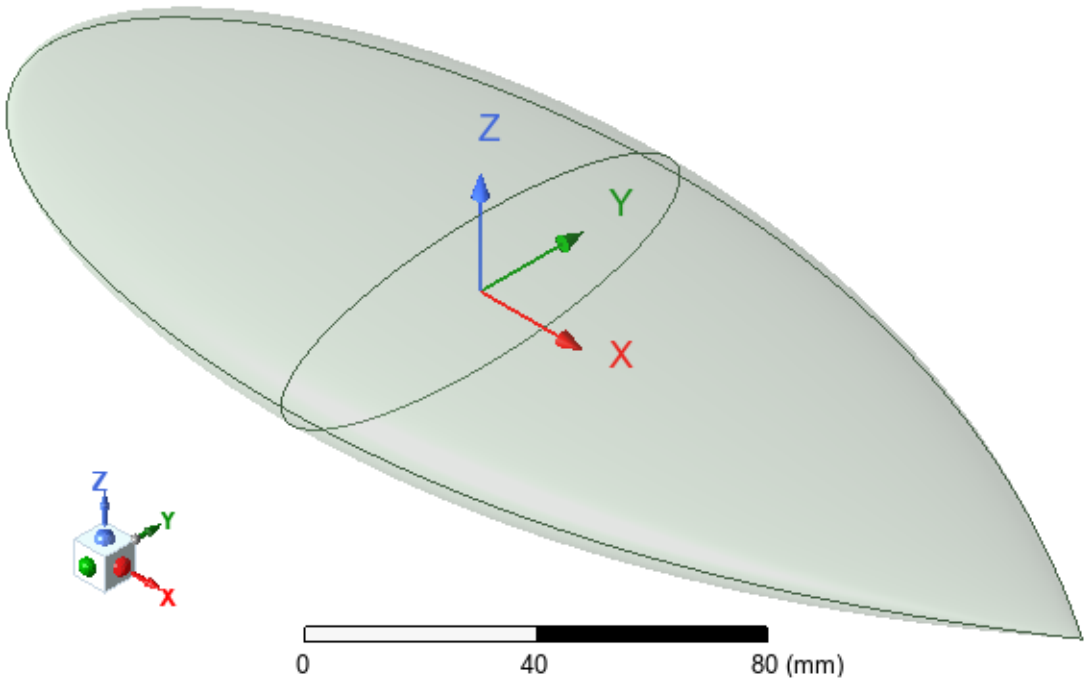


Figure 1: NASA Almond Geometry – Global Coordinate System

Simulation Time: (meshing, adaptive solution, and sweep in 2025 R1 using12 cores)

- **HH Polarization:** 4 minutes: 48 seconds, 1.54 GB max. memory per process
- **VV Polarization:** 4 minutes: 13 seconds, 1.48 GB max. memory per process

Model and Setup Details

The NASA almond is a solid constructed using two Equation Based Surfaces. Design variable *"d"* is defined to provide a scalable model.

Each surface is solidified using the following operations:

The two volumes are united.

Excitations:

- Plain Incident Wave – Spherical input format, used for monostatic RCS analysis (**HFSS** > **Excitations** > **Assign** > **Incident Wave** > **Plane Wave**). Polarization is horizontal for the first design (*NASA_Almond_HH_Parametric*) and is vertical for the second design (*NASA_Almond_VV_Parametric*).

Hybrid Region:

- IE Region, defined to solve using Full Wave BEM method (**HFSS** > **Hybrid** > **Assign Hybrid** > **IE Region**).

Mesh Operations: (Default initial mesh settings are used)

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

- Model Resolution (*ModelResolution1*): *Length* = 0.05 mm (**HFSS** > **Mesh** > **Assign Mesh Operation** > **Model Resolution**) – Set to avoid meshing excessively small details
- Surface Approximation (SurfApprox1): Settings are as shown in Figure 2 (**HFSS** > **Mesh** > **Assign Mesh Operation** > **Surface Approximation**) – Mainly used to control the mesh on true surface geometry (not segmented). Surface deviation controls the distance between the mesh and the true surface. Typically λ / 300 is chosen.

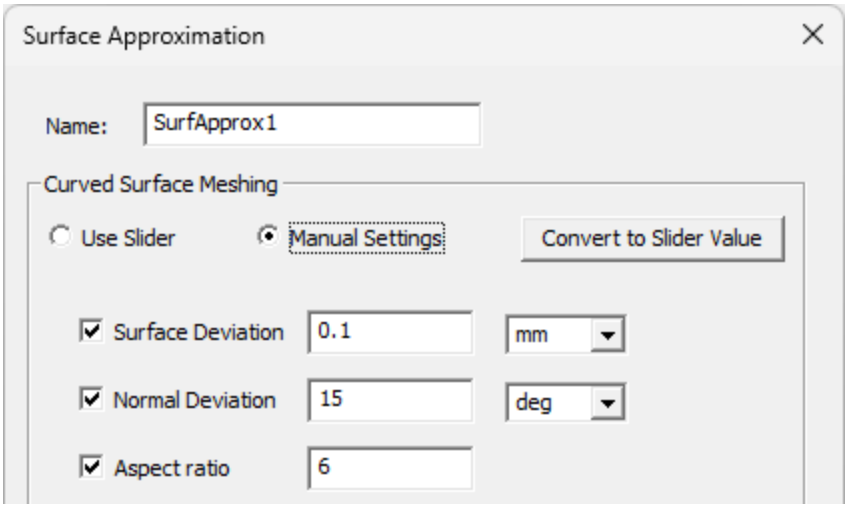


Figure 2: Surface Approximation Settings

> **Note:**
>
> The mesh can be imported into Hybrid (IE/SBR+) designs if there is an available source design and if no geometric changes are needed. For this example, you could choose to import the mesh from the HH design (horizontal polarization) into the VV design (vertical polarization). However, for this example the benefit of doing so would be minimal since adaptive meshing is limited to only two passes.

Setup: (*9p92GHz*)

- 9.92 GHz single adaptive solution frequency, default settings, *Save Fields* enabled
- *Maximum Numper of Passes* = 2
- Discrete sweep including three frequencies: 1.19, 7, and 9.92 GHz, *Save Fields* enabled

Based on the field accuracy , adaptive meshing will refine the most critical area. In this example, the current distribution is simple, and a single pass is sufficient to produce fair results. However, you can see that the results will slightly change from the first to the second pass. Generally speaking, the use of adaptive meshing is recommended for accurate and reliable results.

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

Postprocessing

After solving (**Simulation** > **Analyze all**), you can view different postprocessing results. Look in the Project Manager under *Results* and double-click on the different predefined reports. Each design also has to Jsurf overlays, one at 1.19 GHz, and one at 9.92 GHz. The first design (horizontal polarization) also includes a predefined mesh overlay. Figure 3 shows the available reports and overlays for the first design:

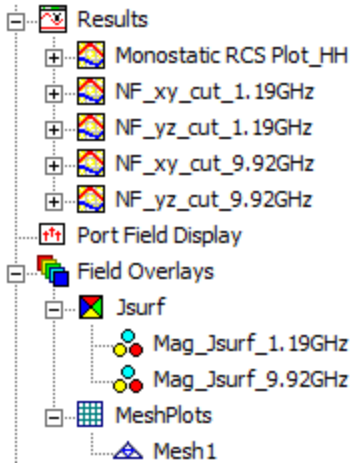


Figure 3: Predefined Reports and Field Overlays Listed in the Project Manager

Predefined Reports:

Figures 4 and 5 show the monostatic RCS results for all three frequencies that were solved and for the horizontal (HH) and vertical (VV) polarization, respectively:

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

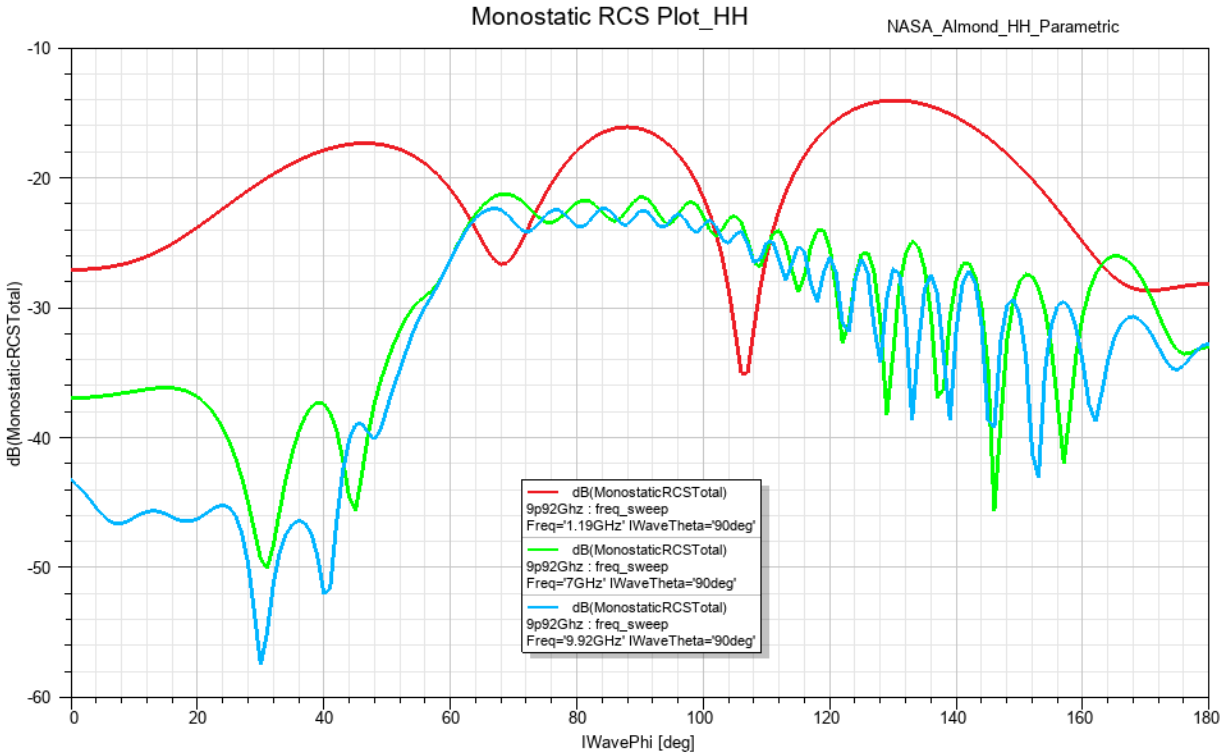


Figure 4: Total Monostatic RCS versus IWavePhi for Horizontal Polarization

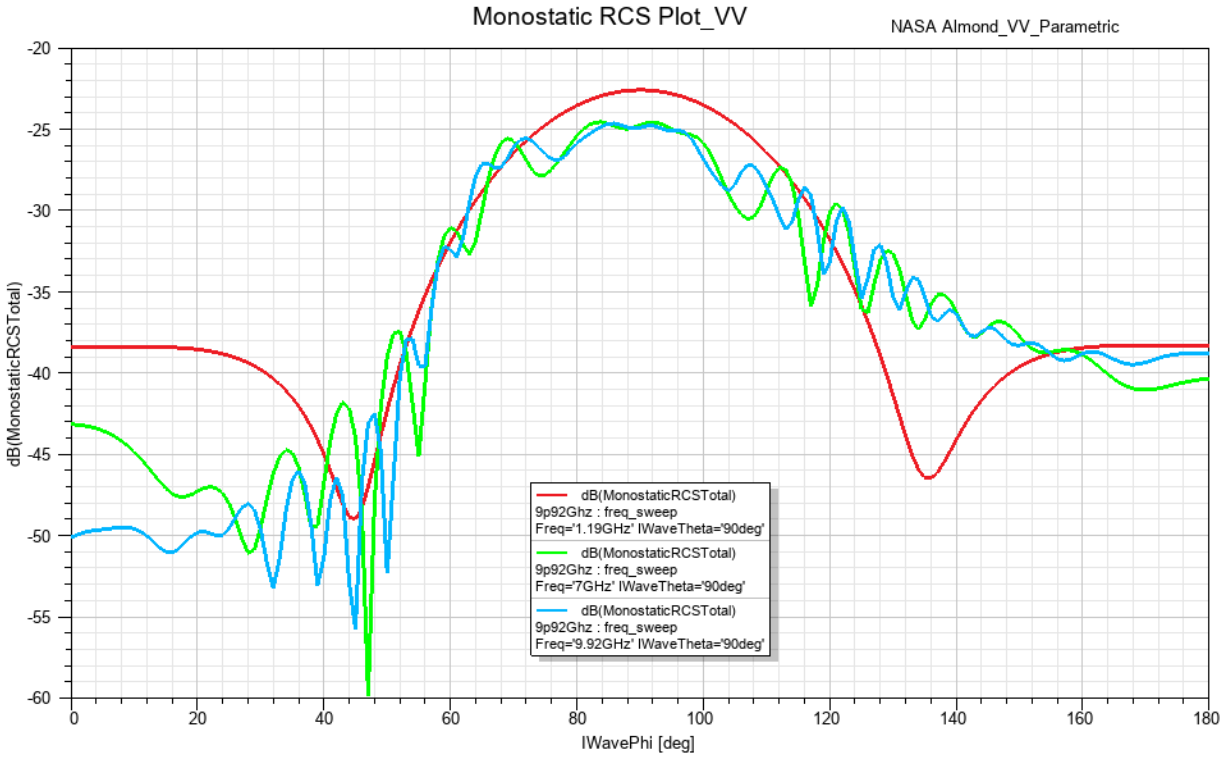


Figure 5: Total Monostatic RCS versus IWavePhi for Vertical Polarization

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

Figures 6 through 13 are near field contours at predefined rectangles that bisect the almond radar target. These figures include results at 1.19 and 9,92 GHz, horizontal and vertical polarization, and the near field radiation setups *NF_xy_cut* and *NF_yz_cut*. Results are also available at 7 GHz if you would like to modify one of the existing near field contour plots or add any plots.

Note:

In the eight contour plots that follow, you can observe nonzero fields within the conducting radar target body. The explanation is as follows:

- In the Finite Element Method (FEM), the solver directly computes the field solution within an air or vacuum region. However, within metallic bodies, the fields are set to zero instead of being computed.
- In the Integral Equation (IE) method (used for this example) or the Shooting and Bouncing Ray (SBR+) method, when creating a cut plane contour plot, the solution is derived from the current on the conducting body's surface. Inside metallic bodies, the total fields should ideally be zero, necessitating the scattered field to be exactly opposite to the incident field to achieve a total field of zero. Due to numerical inaccuracies, the field solution **inside metals** may be nonzero for IE and SBR+ methods, and it should be disregarded.

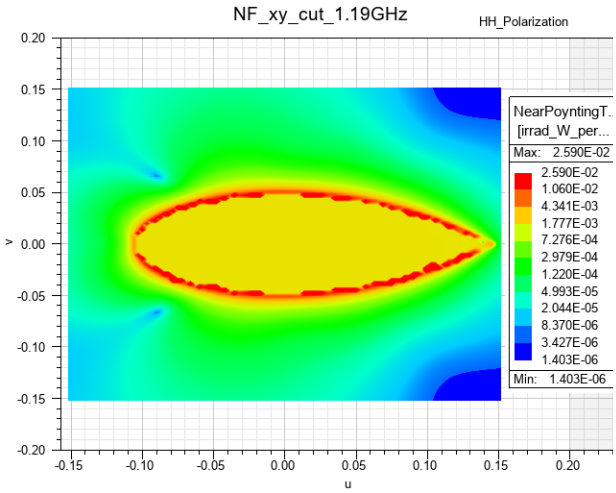


Figure 6: Near Field Radiation Contours at *NF_xy_cut* – 1.19 GHz, Horizontal Polarization

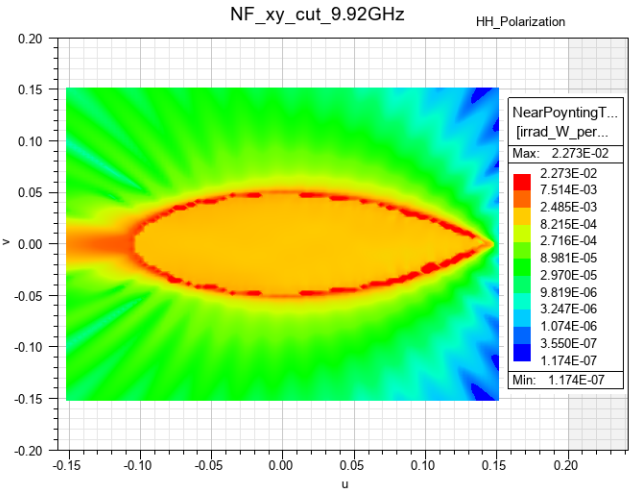


Figure 7: Near Field Radiation Contours at *NF_xy_cut* – 9.92 GHz, Horizontal Polarization

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

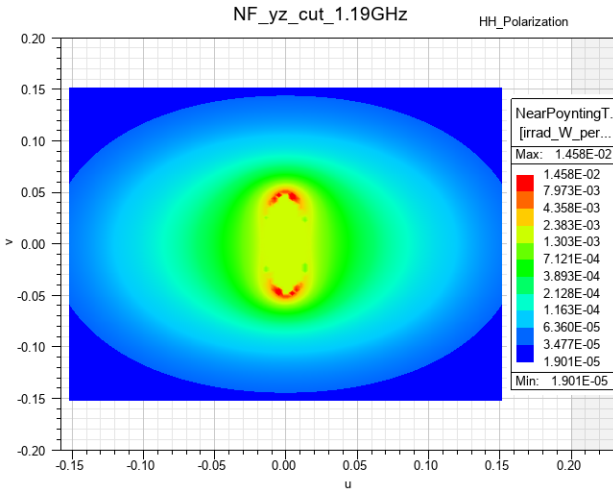


Figure 8: Near Field Radiation Contours at *NF_yz_cut* – 1.19 GHz, Horizontal Polarization

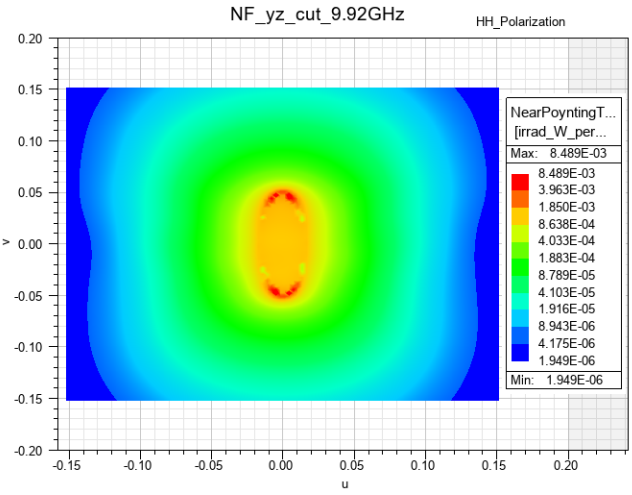


Figure 9: Near Field Radiation Contours at *NF_yz_cut* – 9.92 GHz, Horizontal Polarization

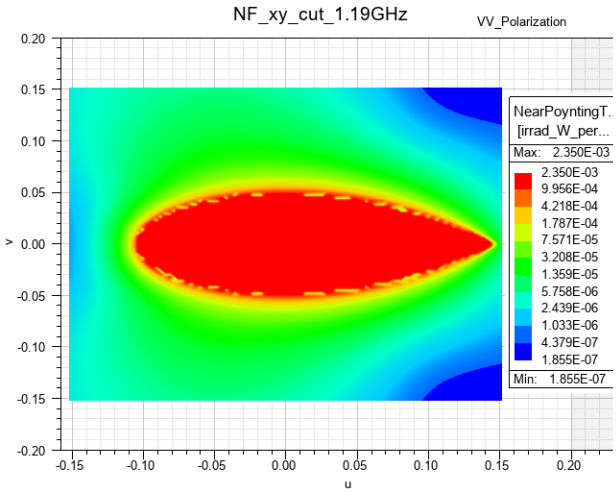


Figure 10: Near Field Radiation Contours at *NF_xy_cut* – 1.19 GHz, Vertical Polarization

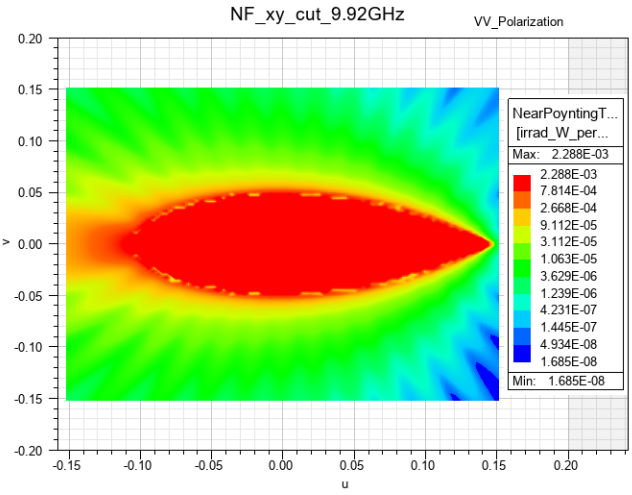


Figure 11: Near Field Radiation Contours at *NF_xy_cut* – 9.92 GHz, Vertical Polarization

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

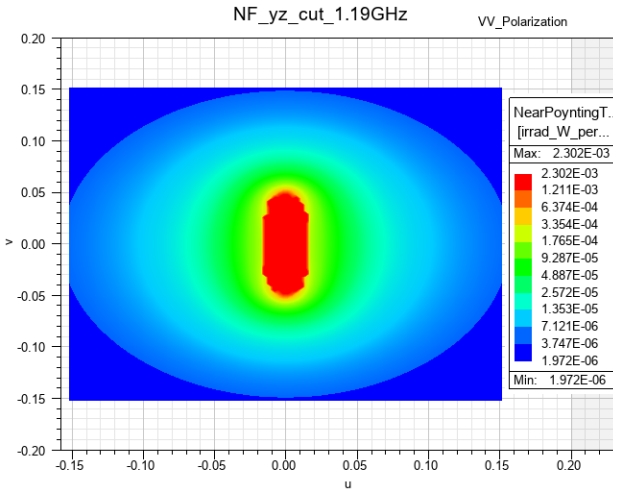


Figure 12: Near Field Radiation Contours at *NF_yz_cut* – 1.19 GHz, Vertical Polarization

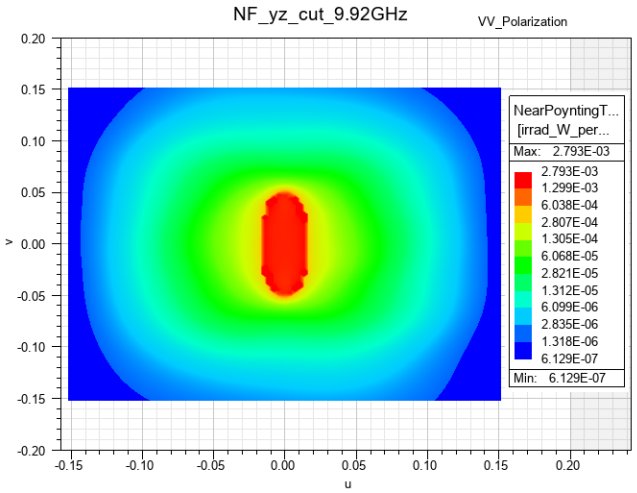


Figure 13: Near Field Radiation Contours at *NF_yz_cut* – 9.92 GHz, Vertical Polarization

Mesh Overlay:

The first design includes a mesh overlay, which is shown in Figure 14. A mesh overlay was not added for the second design, since the model and setups are identical except for the incident wave polarization, and the element counts are the same.

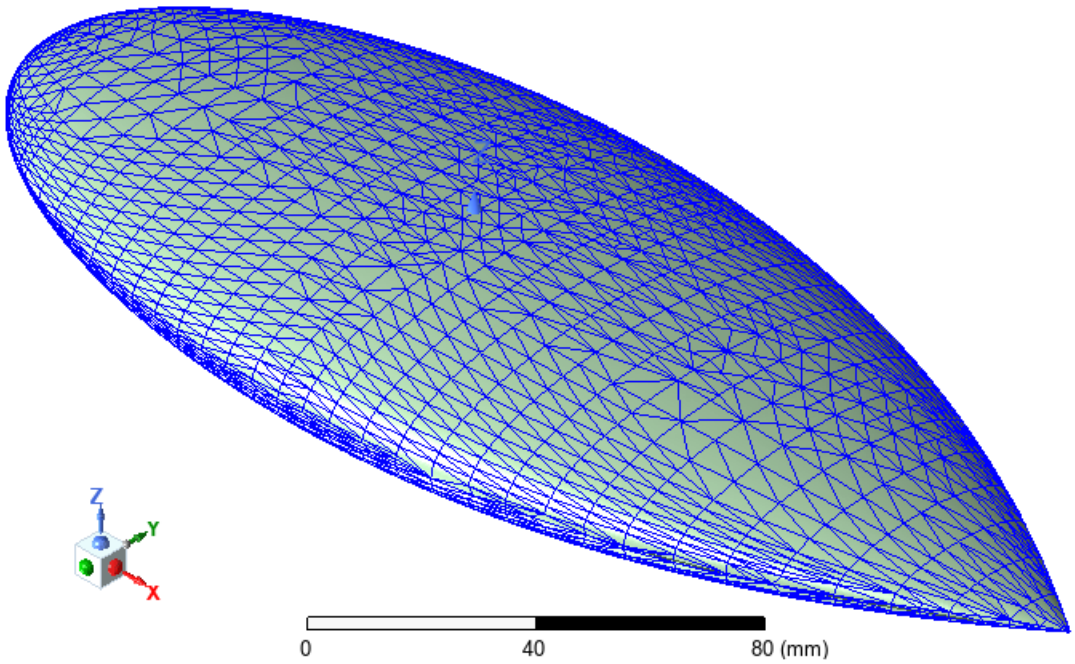


Figure 14: Mesh Overlay

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

Field Overlays:

Each design has two predefined Jsurf magnitude overlays—one at 1.19 GHz and one at 9.92 GHz. See Figures 15 through 18. For each of these figures, the corresponding near field radiation contours are also displayed (**HFSS** > **Fields** > **Plot Fields** > **Radiation Field**). Additionally, fields are available at 7 GHz if you would like to modify one of the existing overlays or add a third one.

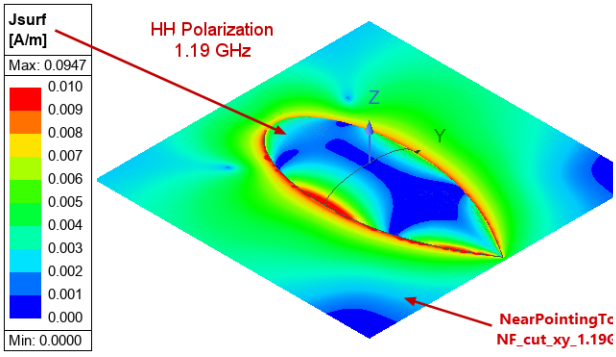


Figure 15: Jsurf Overlay and XY Radiation Contours – 1.19 GHz, Horizontal Polarization

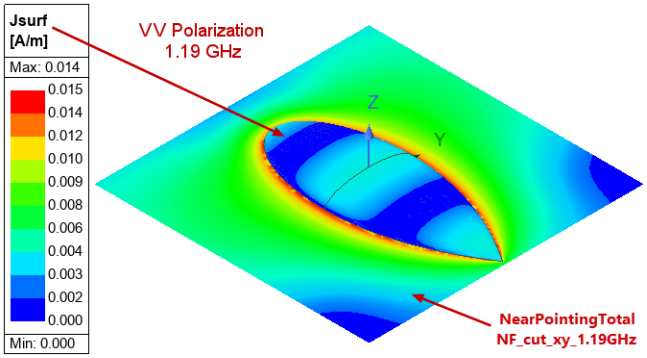


Figure 16: Jsurf Overlay and XY Radiation Contours – 1.19 GHz, Vertical Polarization

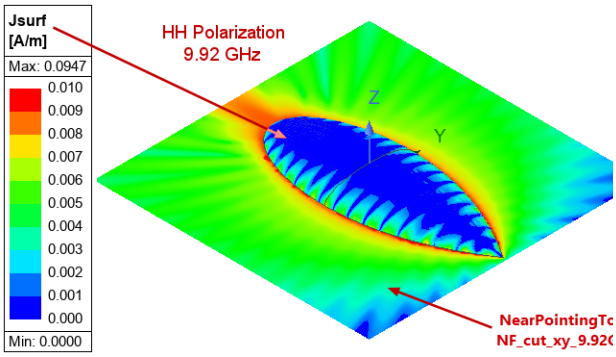


Figure 17: Jsurf Overlay and XY Radiation Contours – 9.92 GHz, Horizontal Polarization

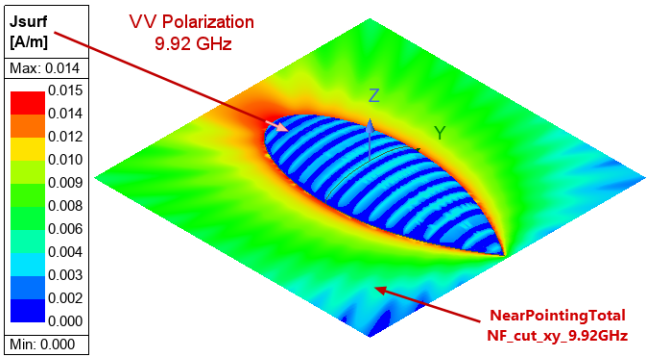


Figure 18: Jsurf Overlay and XY Radiation Contours – 9.92 GHz, Vertical Polarization

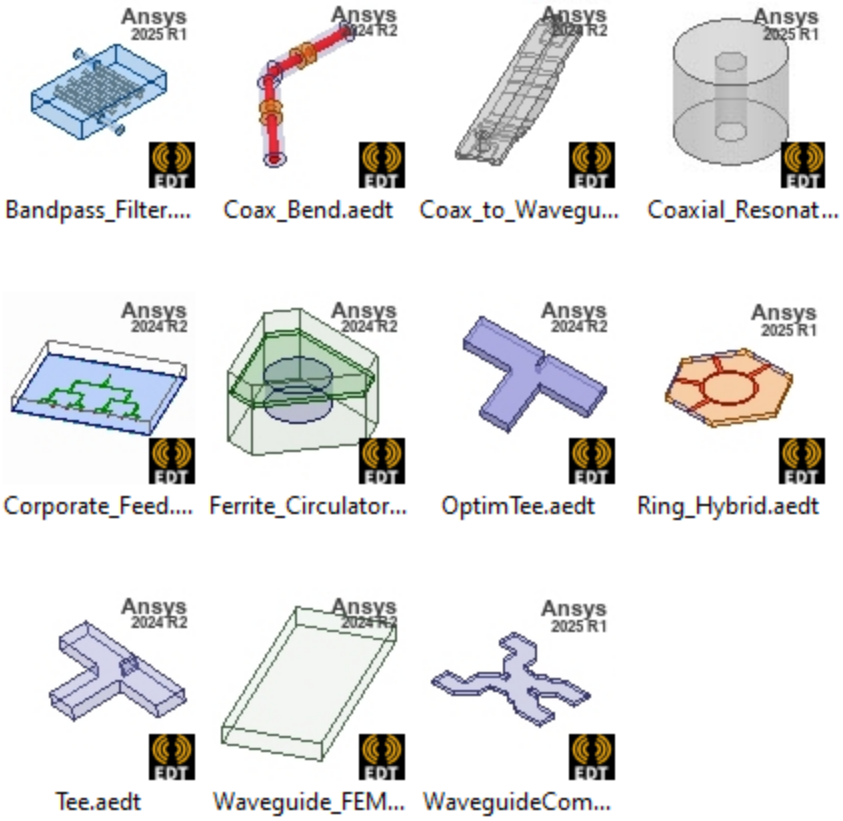
References

[1] "A. C. Woo, H. T. G. Wang, and M. J. Schuh, Benchmark Radar Targets for the Validation of Computational Electromagnetics Programs, IEEE Antennas and Propagation Magazine, vol. 35, no. 1, February 1993, pp. 84–89"

RF Microwave Examples

Click graphics for descriptions of the following HFSS RF/Microwave examples:

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



Coax Bend

Abstract

The setup and simulation of a coaxial cable with a bend is illustrated in this model along with defining variables and parametric sweeps.

Version: 2024 R1

Description

An example model of an air-filled coaxial bend that includes two Teflon supports with inner radius compensation. The model shows how to define a parametric sweep using a variable "**bend_angle**" to control the angle of the top half of the bend and evaluate the change in S-parameters.

Parametric Sweep of a Coax Bend

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

Model: The walls of the coax use the default outer “Perfect E” boundary. The inner conductor is copper. The ports are assigned to the faces of the coax ensuring port 2 will automatically move with the coax body as the angle is changed.

Setup: The simulation is set to adapt at 5 GHz with an interpolating sweep from 0.1 to 5 GHz. The defined Parametric sweep in “bend angle” sweeps from 50 to 90 degrees with steps of 10 degrees.

> **Note:**
>
> To view a port or boundary, select the desired item in the *Project Manager*. It is then highlighted in the *Modeler* window, and the properties are displayed in the docked *Properties* window. Selecting an object in the *History Tree* will also display its properties.

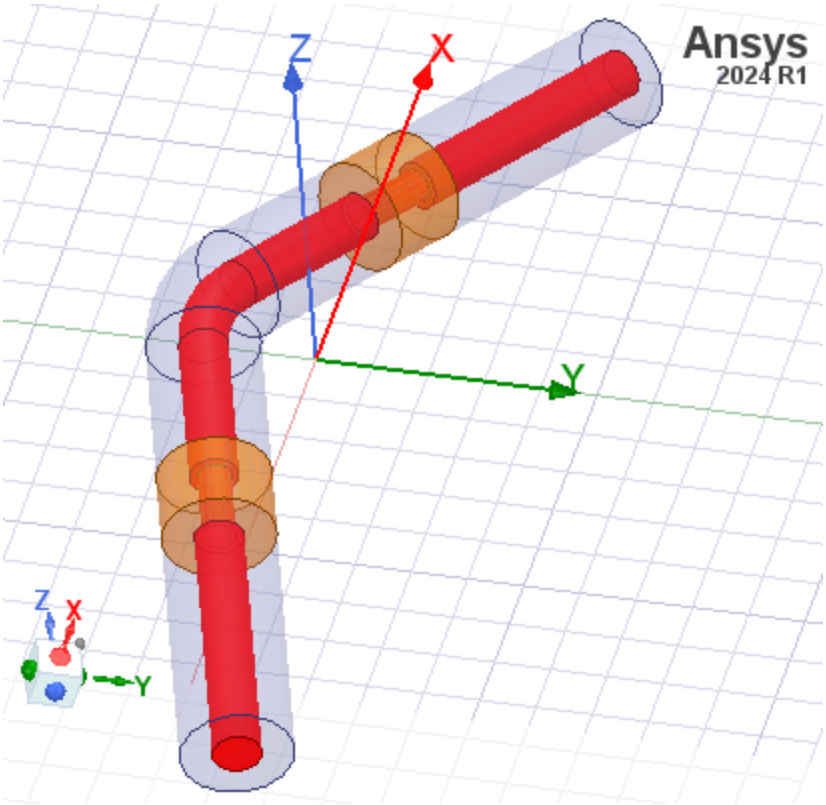


Figure 1: Coax Bend Geometry

Solution Time: 4–5 minutes total (using 4 cores)

- Adaptive Solution and Frequency Sweep: Less than 1 minute
- Parametric Sweep: 3–4 minutes

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

Postprocessing

After solving, you can view the solution data by right-clicking **AnalysisSetup** (under *Analysis* in the Project Manager) and selecting **Profile** to display the *Solution* dialog box. In addition to the **Profile** tab, you also view the **Convergence**, **Matrix Data**, and **Mesh Statistics** tabs. To view the solution data for different *bend_angle* values, click the ellipsis [...] button and select the variation of interest in the **Set Design Variation** dialog box.

To view the S Parameter versus Frequency plot for all five *bend_angle* variations, double-click **S Parameter** under *Results* in the Project Manager.

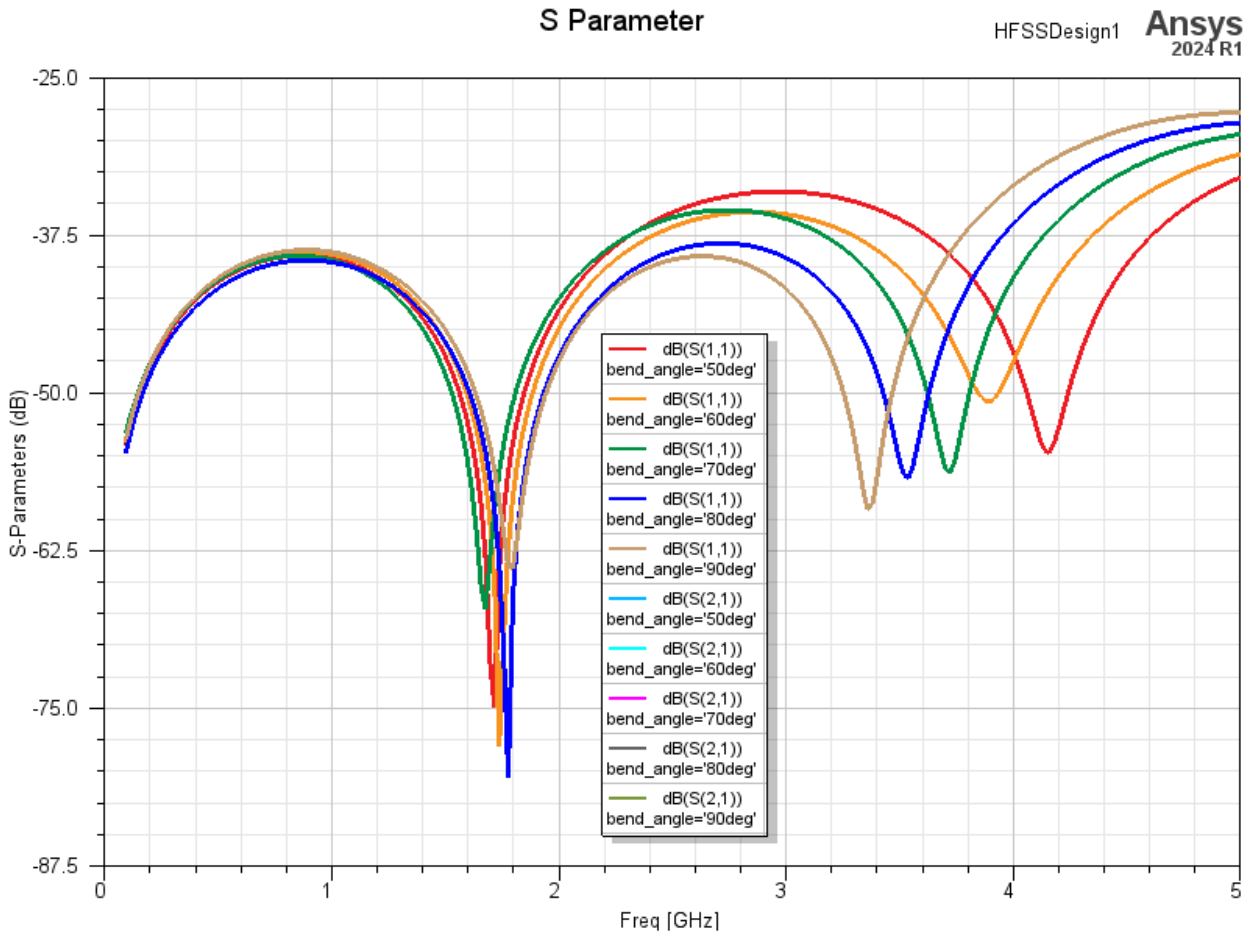


Figure 2: S Parameters vs. Frequency Plot (Y Max has been set to -25 dB)

To see the electric field overlay, double-click **EFieldPlot** under *Field Overlays* > *E Field* in the Project Manager. Animate the overlay to show how the field changes with phase angle by right-clicking **EFieldPlot** and choosing **Animate** from the shortcut menu. In the *Select Animation* dialog box, choose **Animation3 (Swept variable Phase)** and click **Animate**:

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

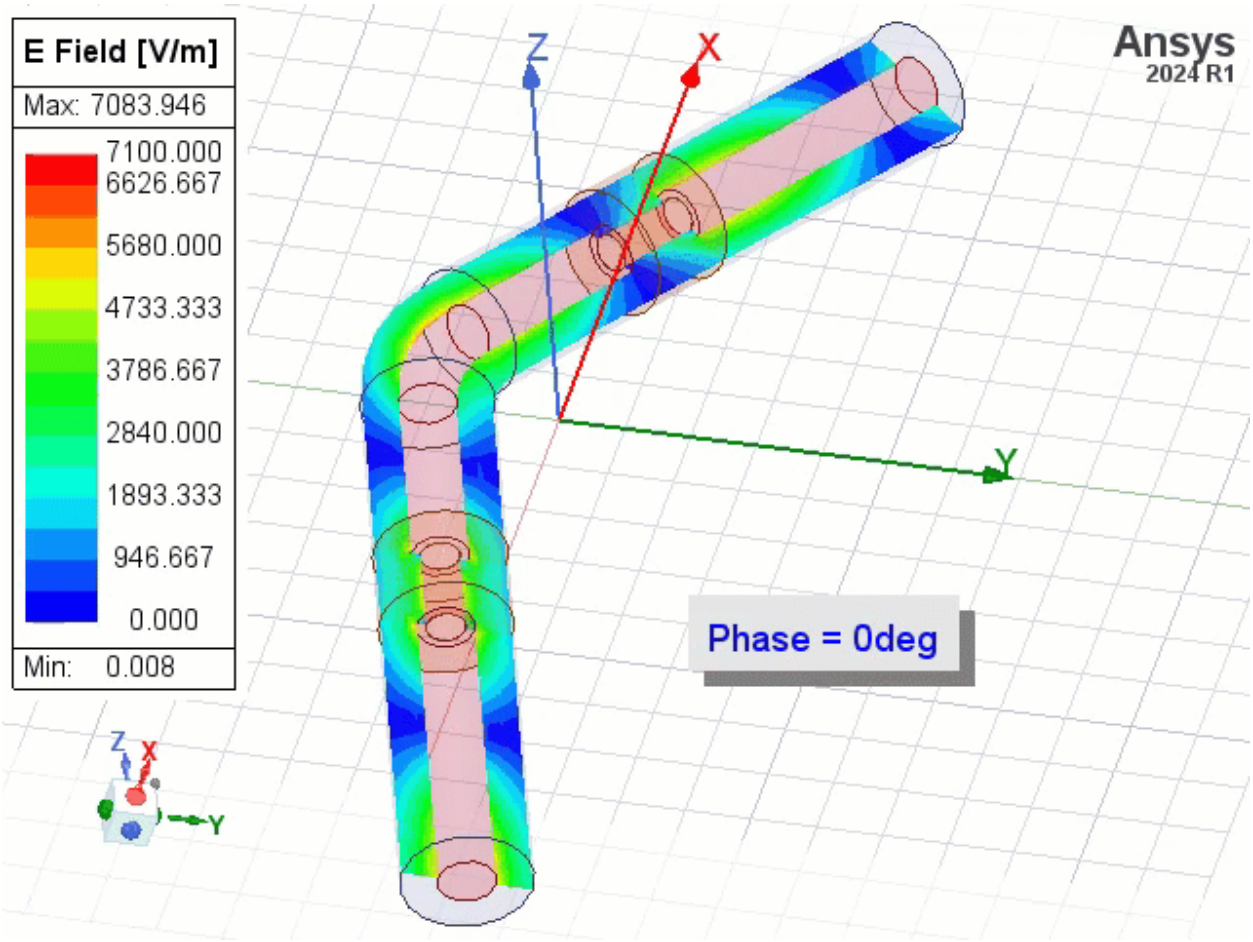


Figure 3: E Field versus Phase Animation

To view all geometric variations right-click **EFieldPlot** again and select **Animate**. This time, in the *Select Animation* dialog box, choose **Animation2 (Swept variable bend_angle)** and click **Animate**.

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

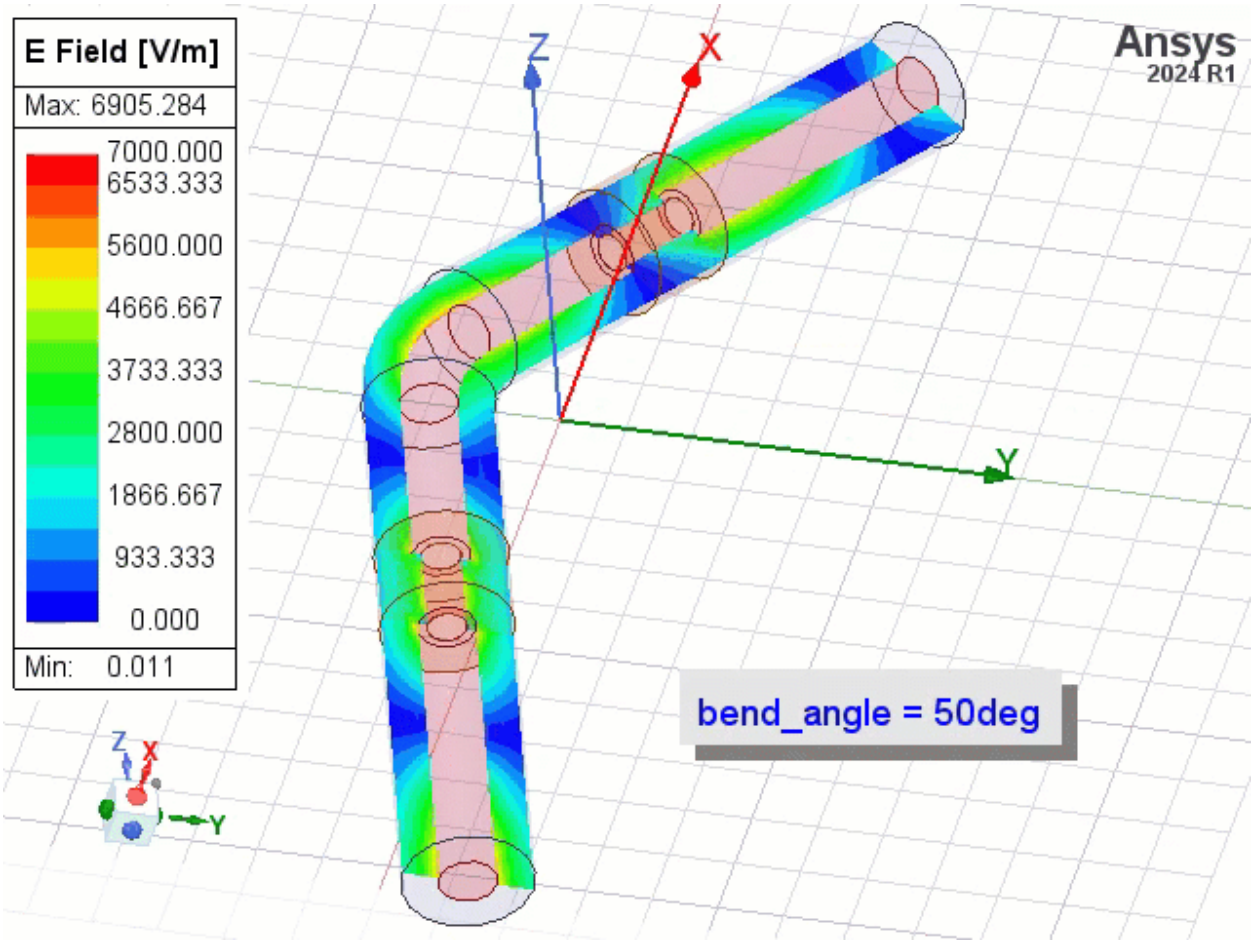


Figure 4: E Field versus Bend Angle Animation

Coax to Waveguide

Abstract

This example project consists of several test designs demonstrating analysis setups of a coaxial-to-waveguide launch utilizing symmetry, back-to-back configurations, and transient analysis. This particular design was the original geometry analyzed by HFSS when first released in 1990. An individual frequency point took 16 hours. Today, in 2023 on a single processor/core, a full frequency sweep, including the creation of the automatic adapted mesh, takes less than 1.5 minutes (<1 minute using a 16-core standard laptop, hardware dependent).

Version: 2024 R2

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

Simulation Time: The back-to-back frequency model solves in less than 1 minute, and the time domain implicit model approximately 8 minutes, on a 16-core i7 laptop with about 700 MB of RAM used.

> **Note:**
>
> Simulation time and memory will vary depending on your available resources. In most cases, High Performance Computing (HPC, parallel resources) can be leveraged to significantly reduce simulation time and the memory footprint required by each CPU.

Model and Setup Details

Boundaries:

- Perfect H Symmetry (*Sym1*) is assigned to the face lying along the XZ plane only for **Design 01** (**HFSS** > **Boundaries** > **Assign** > **Perfect H**). All other designs are full geometry without a symmetry boundary.
- Finite Conductivity (*AluminumWalls*) applicable to all designs except *Design 01*, the original 1990 model. It is based on aluminum material properties, zero surface roughness, and is assigned to all exterior faces of the vacuum object excluding two wave port faces.

Excitations:

- Wave Ports (*1* and *2*), either *Modal* or *Terminal* type, depending on the solution type; *Renormalize all modes* is selected for port *2* of all five designs; single mode and a *Full Port Impedance* of 50 Ω is set for all modal solutions (**HFSS** > **Excitations** > **Assign** > **Port** > **[Modal | Terminal] Wave Port**)

Mesh Setting:

- No mesh operations
- Initial Mesh Settings:
  - Default (centered) slider position for curved surface meshing
  - The **Classic** *Mesh Method* is selected for all five designs:

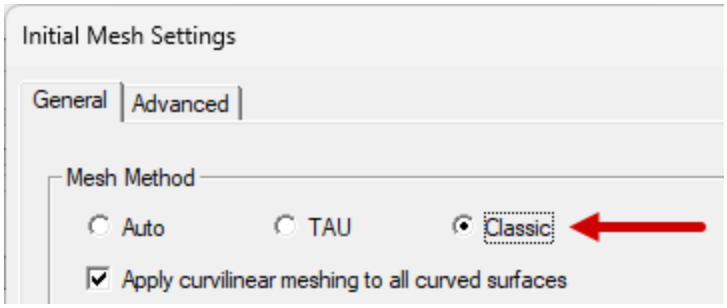


Figure 1: Initial Mesh Settings Dialog Box, Classic Mesh Method Selection

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

Design Variants:

Design *01_Original_Symmetry* consists of a coaxial-to-waveguide launcher, with H-plane symmetry in the XZ plane, as was used in the 1990 test case. The *Impedance multiplier* of the symmetry boundary is set to 0.5 to ensure correct Zo calculation. The solution tolerance, *delta S*, is set to 0.005. This design is a *Modal Network* solution solved in the frequency domain. The *Linear Count* frequency sweep range is set to 5-17 GHz with 601 points.

In addition to the analysis setup named *11GHz,* there is a second one named *11GHz_ wDerivatives*, which includes the calculation of a number of derivatives associated with the geometry of the design. You can use this solution to perform real-time tuning of the design or to understand the relative sensitivity of the design for each parameter analyzed with derivatives.

The symmetry plane is highlighted in the following model image, and the losses are plotted for both solution setups. The traces from the *11GHz_wDerivatives* solution are rendered using short dashes so that the other solution's traces are visible where they overlap:

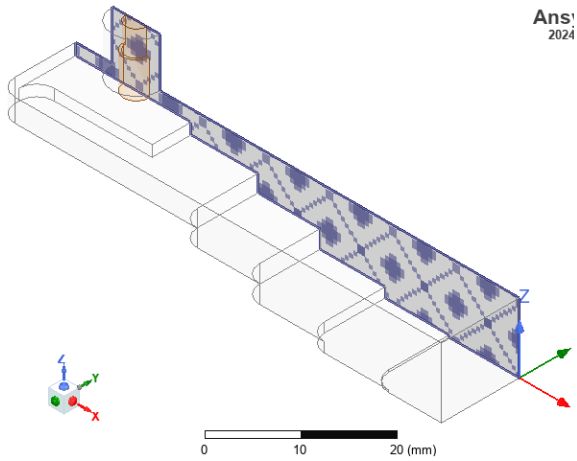


Figure 2: Design *01_Original_Symmetry*

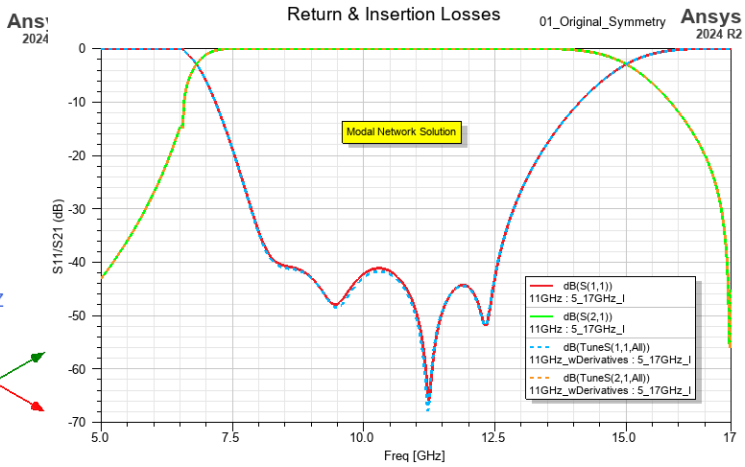


Figure 3: Design 01 – Return and Insertion Losses

Two additional designs (*02_Full* and *03_Back-to-Back*) model the full launcher (without symmetry) and a back-to-back setup of the full launcher, respectively.

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

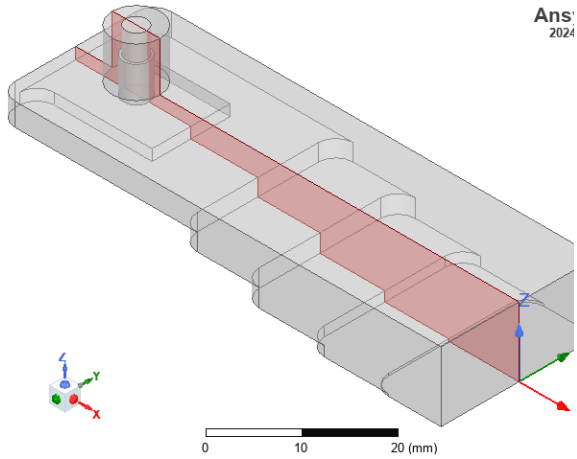


Figure 4: Design *02_Full*

Figure 5: Design *02* – Return and Insertion Losses

In Figure 4, you can see that a sheet object has been added for the purpose of overlaying fields at the mid-plane of the full model. This design is a *Modal Network* solution solved in the frequency domain.

Figure 5 includes trace data copied from the first solution in design *01* for direct comparison of the two sets of S parameter results. Note that the solid green trace (full model return loss) exactly corresponds to the dotted green trace (symmetry model return loss). The dotted trace has a darker shade of green specified to keep it from being obscured by the lighter green solid trace.

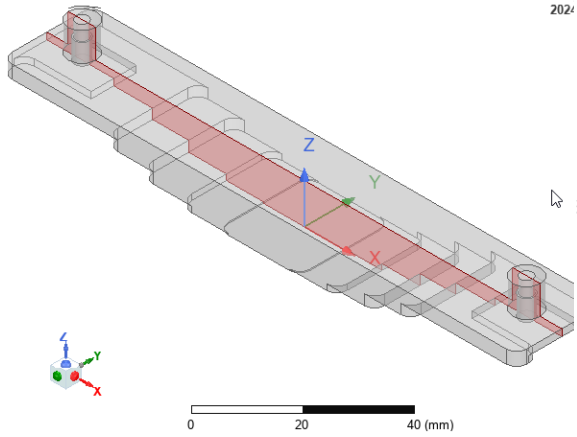


Figure 6: Design *03_Back-to-Back*

Figure 7: Design *03* – Return and Insertion Losses

Again, in Figure 6, you can see that a sheet object has been added for the purpose of overlaying fields at the mid-plane of the back-to-back model. This design is a *Terminal Network* solution solved in the frequency domain.

While the frequency domain solver is the most commonly used solver setup for microwave components, HFSS can also solve in the time domain using a *Transient* analysis setup. This

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

setup uses the same underlying FEM mesh solution with time domain response computed using a hybrid (DGTD) or implicit (FETD) solver.

Two additional designs are defined as *Transient Network* solutions. One has geometry identical to design 02 (*04_Full_Trans*) and the other has geometry identical to design 03 (*05_Back-to-Back_Trans*). Both solutions take advantage of mesh linking to use the existing mesh from the corresponding frequency domain design. (Mesh linking is not necessary but saves time.) Accordingly, the frequency domain solutions must be analyzed before attempting to run the time domain solutions.

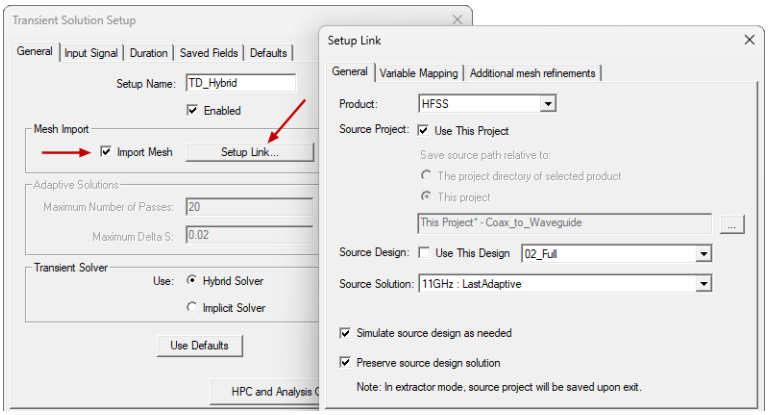


Figure 8: Linked Mesh Setup for Design 04

The linked mesh setup is the same for Design *05_Back-to-Back_Transient* except that the *Source Design* is *03_Back-to-Back*.

Each transient design includes analysis setups for both the *Hybrid* solver and the *Implicit* solver, for comparing the results of the two. Normally, the implicit solver would be favored for this electrically small problem. Aside from the *Setup Name* and *Transient Solver* selection, the setup of the *Implicit Solver* solution is identical to that of the *Hybrid Solver* solution shown in Figure 5 above.

The excitation input signal is defined as a broadband pulse over the range 5–17 GHz, *Target Residual* is set at -50 dB. The number of *Frequencies extracted* is set to match the number of points in the frequency-domain solution sweeps. The target residual for transient solvers has a strong impact on solution time and accuracy for low level coupling. Typically it is set in the range -40 to -60 dB. The input signal setup is identical for all four transient solutions:

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

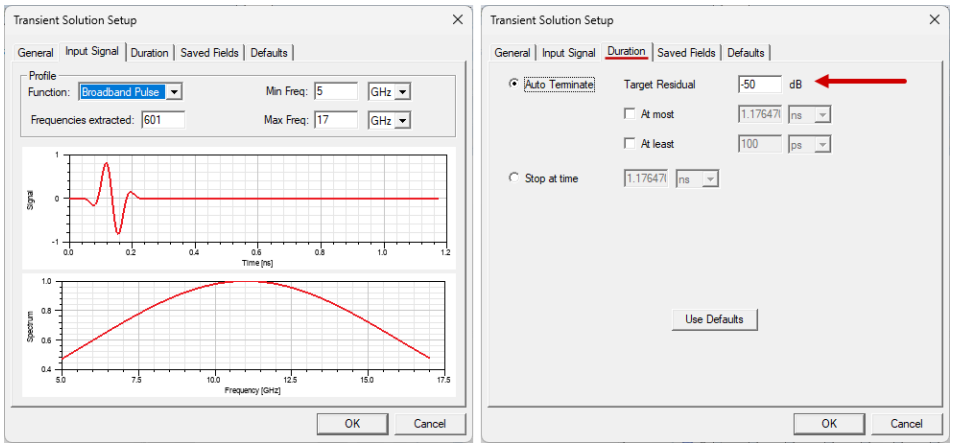


Figure 9: Transient Solution Setup – Target Residual and Input Signal

Postprocessing

After solving each design (**Simulation** > **Analyze all**), you can view different postprocessing results. Look in the Project Manager under *Results* and double-click on the different predefined reports to open them after solving each design.

Predefined Reports:

Screenshots of the predefined reports from the first three design variants are shown alongside the geometry in preceding Figures 3, 5, and 7. For the transient solutions (designs *04* and *05*), geometry screenshots are not included, since they are identical to the geometry in designs 02 and 03, respectively. Transient reports from these designs are shown in the following Figures 10 through 13:

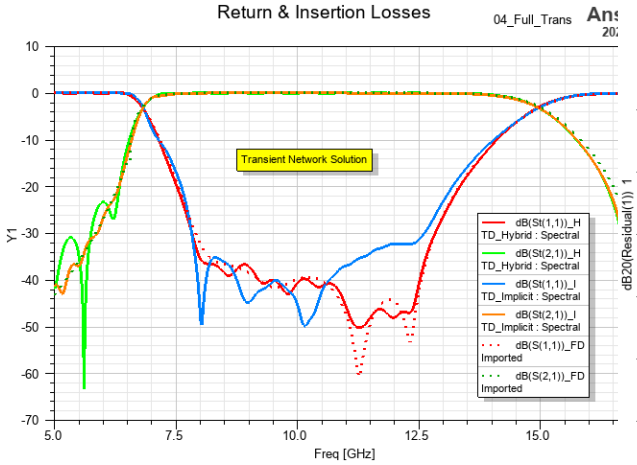


Figure 10: Design *04* – Return and Insertion Losses

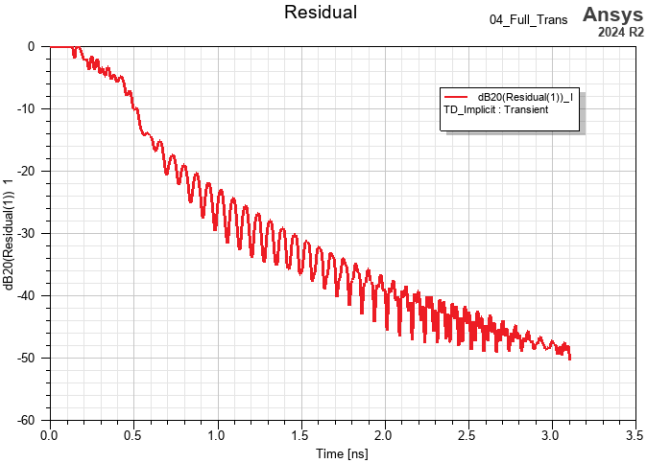


Figure 11: Design *04* – Residual

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

In Figure 10, the return and insertion losses are given for both the Hybrid and the Iterative solver. Additionally, the return and insertion loss curves from Design *02_Full* have been copied to this report (dotted traces) for comparison of the modal network and transient spectral results.

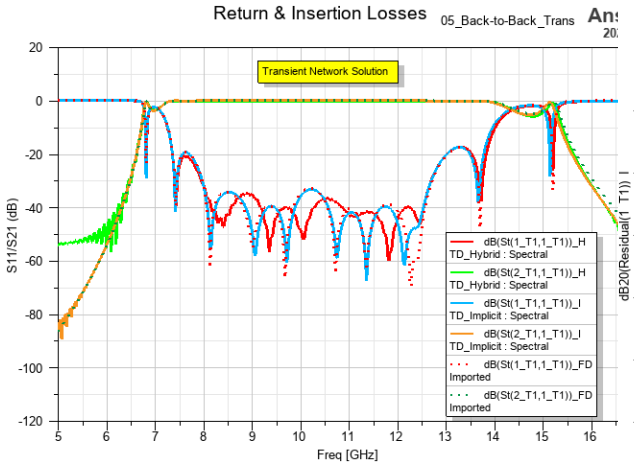


Figure 12: Design *05* – Return and Insertion Losses

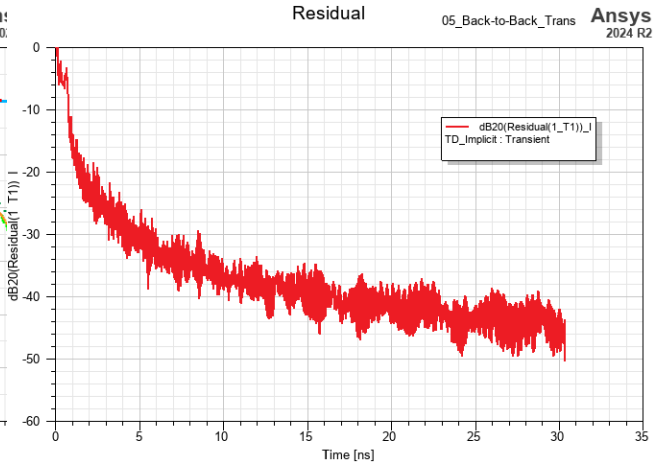


Figure 13: Design *05* – Residual

Again, the return and insertion losses are given for both the Hybrid and the Iterative solver in Figure 12. Additionally, the return and insertion loss curves from Design *03_Back-to-Back* have been copied to this report (dotted traces) for comparison of the terminal network and transient spectral results.

Field Overlays:

E-Field Magnitude and Mesh overlays have been predefined for all five designs in this example model. We will next look at a subset of those overlays. You can phase-animate the fields from the frequency-domain solutions and time-animate the fields from the time-domain solutions.

Meshes for the two transient designs (*04* and *05*) are linked from the source modal network (*02*) and terminal network (*03*) solutions, respectively. The redundant mesh views are not included below. Additionally, we will present side-by-side comparisons of the E-Fields for the corresponding frequency-domain and time-domain solutions.

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

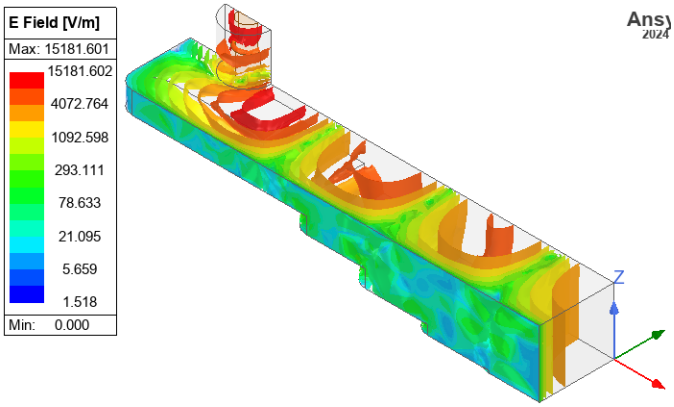


Figure 14: Design *01* – E Fields (Phase = 0°)

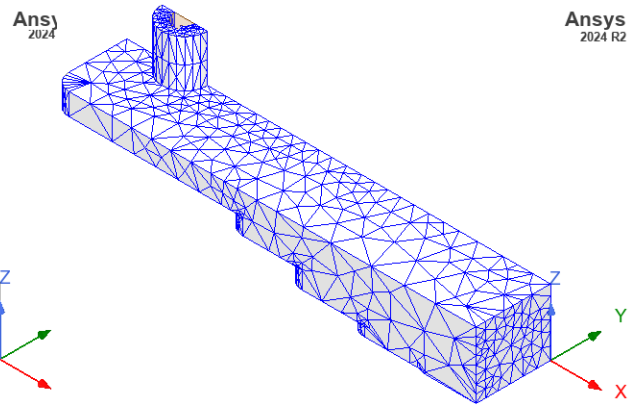


Figure 15: Design *01* – Mesh

In Figure 14, the fields are presented as an iso-surface plot throughout the volume of the vacuum object. All subsequent field overlays show the fields at the sheet object that lies on the XZ plane (mid-plane of the waveguide).

The following two figures show the mesh for the full and back-to-back transient solutions. These meshes were linked from the corresponding frequency-domain solutions (Designs *02* and 03, respectively):

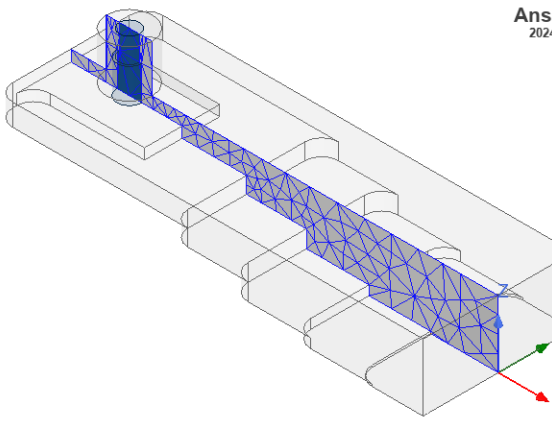


Figure 16: Design *04* – Mesh at Mid-Plane Sheet

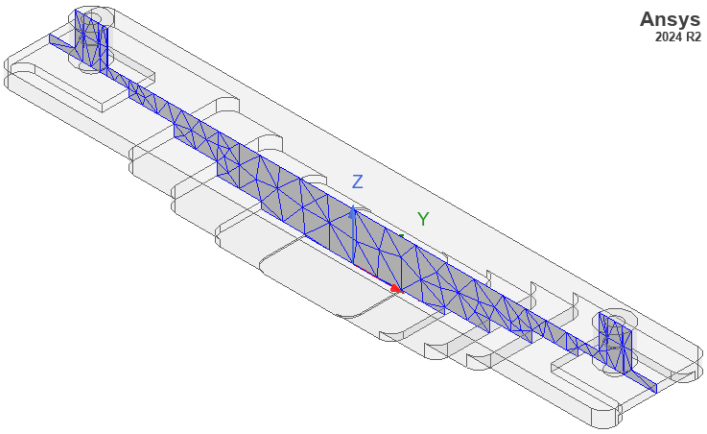


Figure 17: Design *05* – Mesh at Mid-Plane Sheet

For the remaining field overlays, the fields from the time-domain solutions are shown beside the fields from the frequency-domain solutions, for comparing:

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

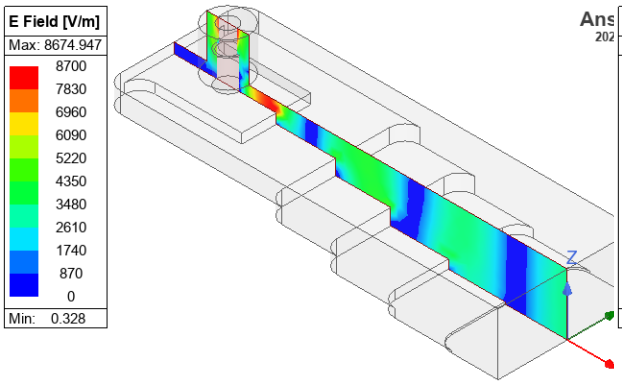


Figure 18: Design *02_Full* – E Field Magnitude, 0° Phase (Frequency Domain)

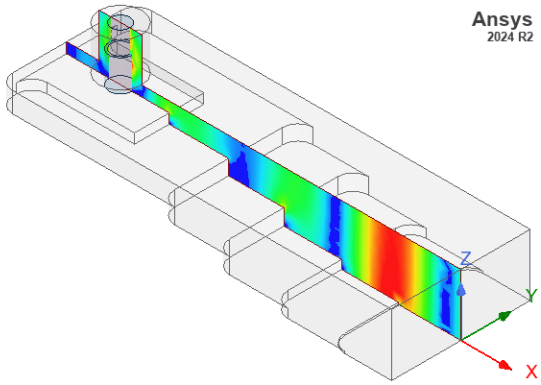


Figure 19: Design *04_Full* – E Field Magnitude at 400 ps (Time Domain)

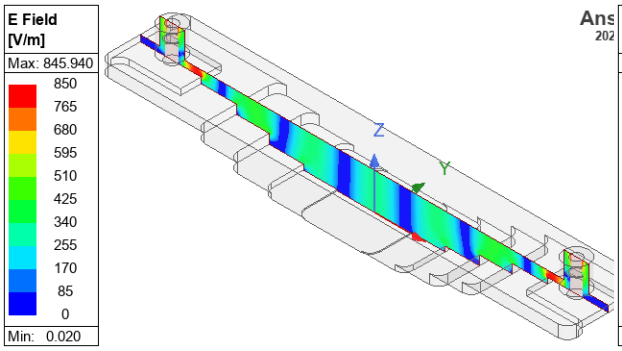


Figure 20: Design *03* – E Field Magnitude, 0° Phase (Frequency Domain)

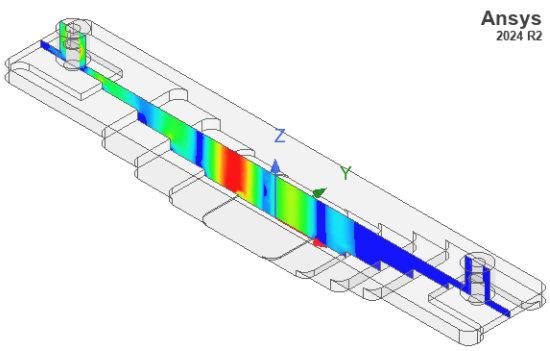


Figure 21: Design *05* – E Field Magnitude at 400 ps (Time Domain)

Coaxial Resonator

Abstract

A coaxial resonator model demonstrating how to use the Eigenmode solver. The Eigen solver computes the resonant frequency and Q of the model. This example was taken from *Microwave Circuit Modelling Using Electromagnetic Field Simulation* (D. Swanson Jr., W. Hoefer).

Simulation Time: (machine with 64GB RAM, 6 cores used, Automatic HPC settings, v. 2025 R1)

43 seconds, 4.61 GB memory used

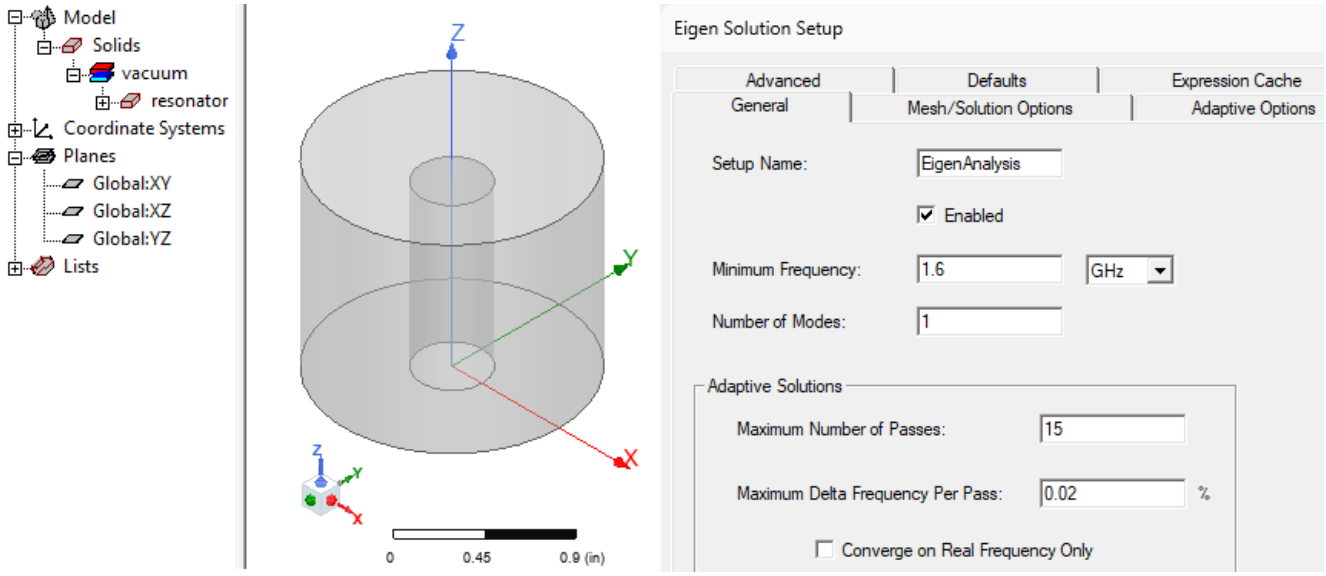
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Simulation time and memory will vary depending on your available resources. In most cases, HPC (parallel resources) can be leveraged to significantly reduce simulation time and the memory footprint required by each CPU.

Description

This example consists of a simple coaxial cavity. The cavity walls are defined to have σ = 6.17 x 10^7 mho/m using a finite conductivity boundary condition. There are no defined sources in an eigen solution, select the number of modes to compute and the convergence criteria. For this model, only the first mode is computed. For maximum accuracy, use curvilinear elements. To verify that this has been set for the model, go to **HFSS** > **Mesh** > **Initial Mesh Settings**, and make sure **Apply Curvilinear Elements** is selected.



Simulation Summary

After solving, you can view the solution data by right-clicking **EigenAnalysis** under *Analysis* in the Project Manager and selecting **Eigenmode Data** from the shortcut menu. This command opens the *Eigenmode Data* tab of the *Solutions* dialog box. You also view the *Profile, Convergence,* and *Mesh Statistics* tabs of this dialog box.

Postprocessing

The resultant resonant frequency and Q values are listed in the *Eigenmode Data* tab of the *Solutions* dialog box.* Alternatively, double-click **Eigenmode Results** under *Results* in the

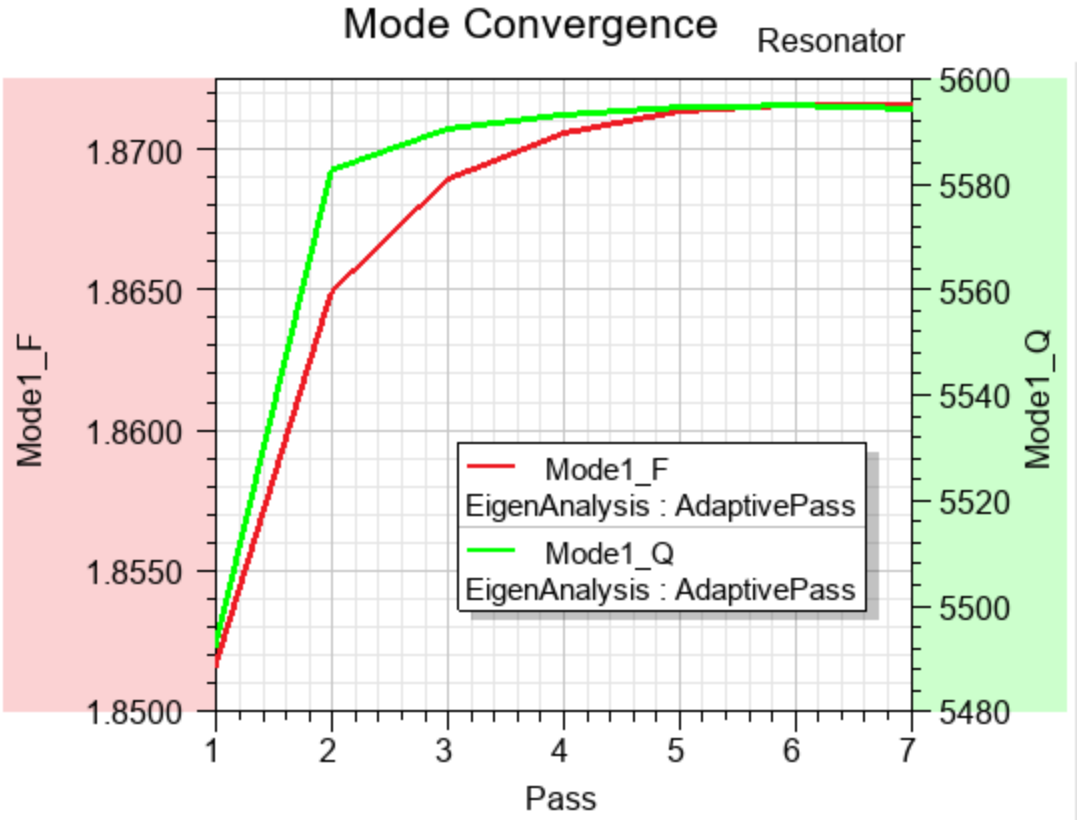
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Project Manager to see these results in a tabular report.

Eigenmode Results Resonator

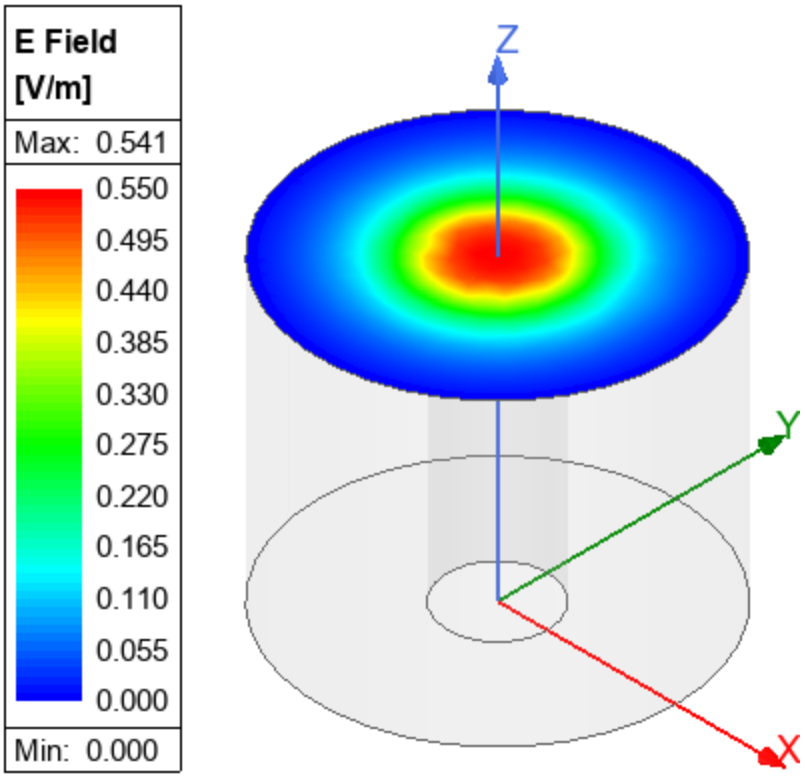
	1
X [mm]	0.000000
Mode1_F	1.871568
Mode1_Q	5594.174855

Under *Results* in the Project Manager, double-click **Mode Convergence** to see a plot of the frequency and Q values for each solution pass until convergence was achieved.



To view a color contour plot of the electrical field results at the top face of the model, right-click E Field under *Field Overlays* in the Project Manager and select **Update Plots**.

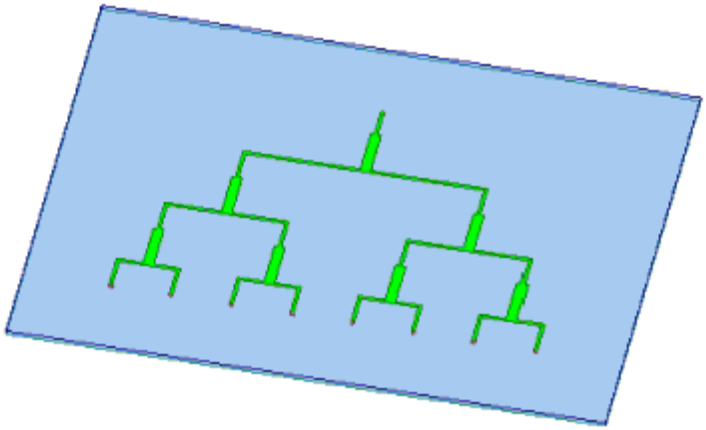
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



* Data computed using a mode matching program is given in the reference. The results presented are f_0 = 1.87 GHz and Q = 5592.

Corporate Feed

Description - A microstrip 8 way corporate feed. The model was originally created in Ansys Electronics Desktop. Export to HFSS was used to create this project. It is a Driven Terminal design.



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

Model - The traces are defined as 2D objects with a finite conductivity boundary of copper. The sources are all lumped ports with a 50 Ohm impedance. There is a surrounding air box with a radiation boundary on the top and sides.

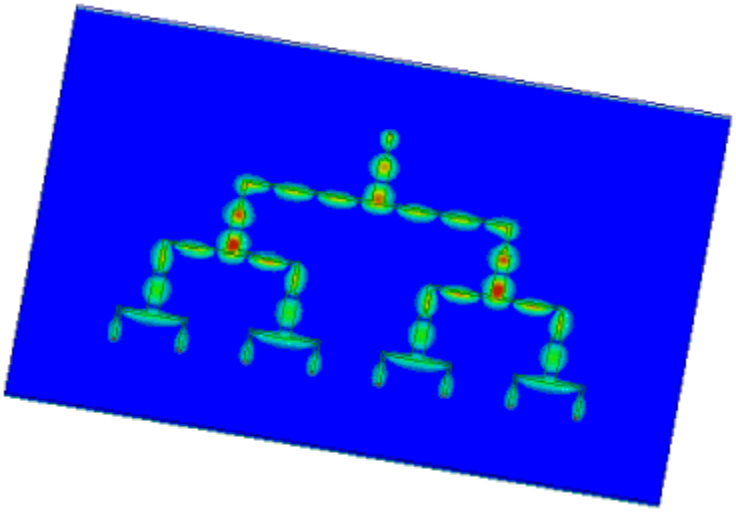
Setup - Adapt at 7.5 GHz and use Mixed Order for the Order of Basis function.

> **Note:**
>
> To view a port or boundary, select the desired item in the Project Tree. It is highlighted in the Model window and the properties will be displayed. Selecting an object in the History tree will also display its properties.

Corporate Feed Post Processing

After solving, you can view solution data by right-clicking on the Setup and selecting **Profile** to display the **Solution** dialog. You also view the **Solution** tabs for **Convergence**, **Matrix Data**, and **Mesh Statistics**.

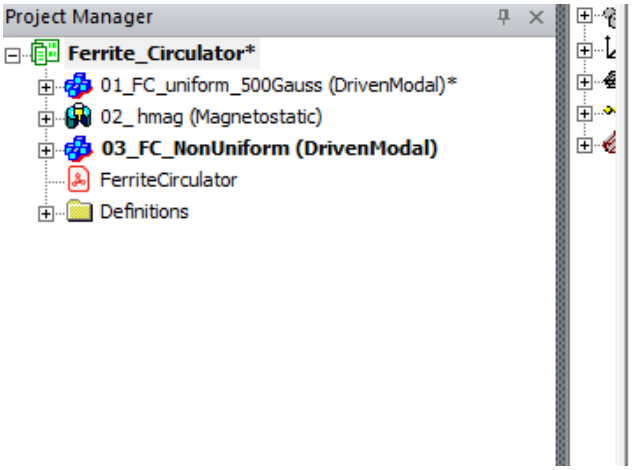
To view the shade plot of the E field, shown below, double-click MagE1 in the Project tree under **Field Overlays>E Field**. To see a phase animation of this field plot, right-click MagE1, and select **Animation**, Then click OK in the dialog.

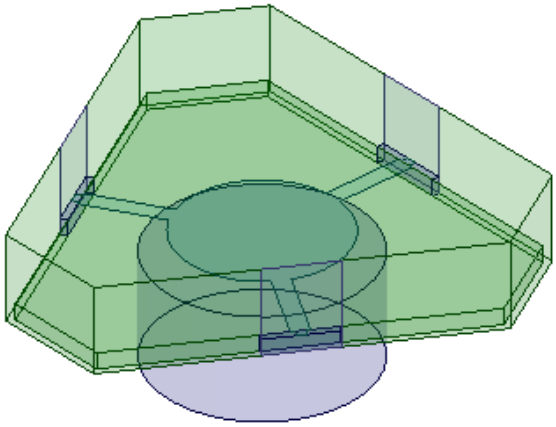


Ferrite Circulator

Description -In this example, a three-port ferrite circulator as shown in fig.1 is analyzed at 8 GHz. This non-reciprocal network transfers power from incident port to the adjacent port in a clockwise or anti-clockwise direction and isolates the third port. The nonreciprocity of the ferrite circulator is achieved by proper magnetic bias fields.

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





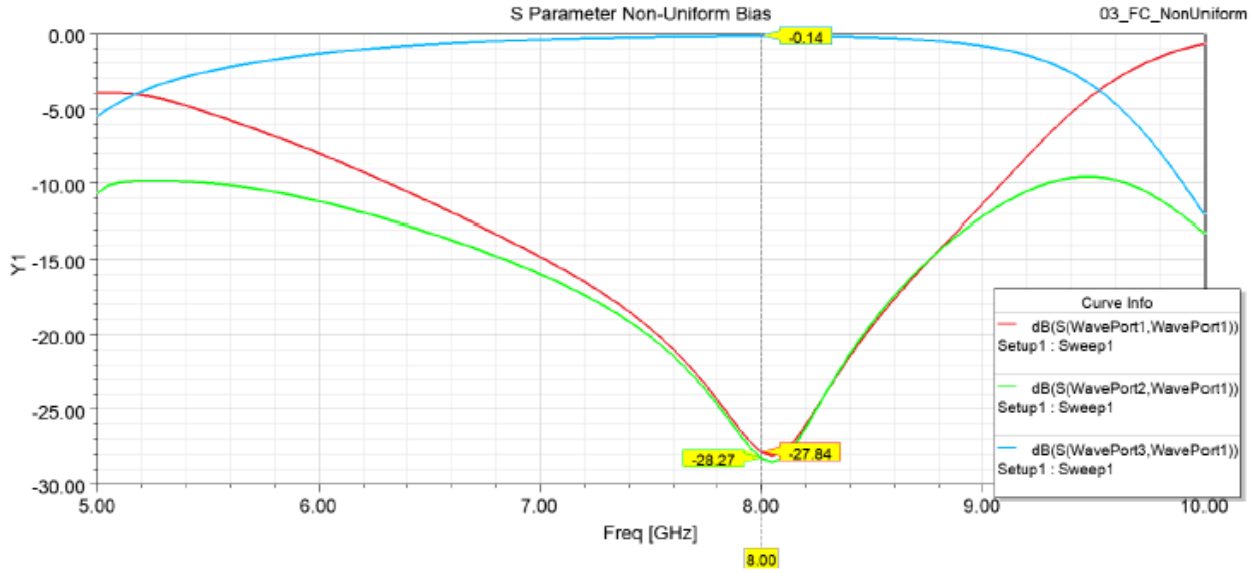
Model - This example demonstrates how to model a ferrite circulator in HFSS with uniform and non-uniform magnetic biasing. The project Ferrite_Circulator.aedtz contains three designs: “01_ FC_uniform_500Gauss” to solve for uniform magnetic bias, 02_hmag(Magnetostatic) to solve magnet with non-linear material in Maxwell to establish non-uniform biasing and “03_FC_ NonUniform” to solve the ferrite circulator in HFSS by linking the fields from Maxwell as a non-uniform magnetic bias on the ferrite.

Most circulator use permanent magnets for the bias field which possibly introduce nonuniform magnetic bias. The design 2_ hmag consists of a magnet of finite size and a ferrite disk. The Maxwell 3D tool based on Maxwell magnetostatics solver is used to solve non-linearity of ferrite material. Note that there are no boundaries and excitation settings required in Maxwell to get magnetic DC bias. However, users can assign currents and different assignments according to the requirement.

The FerriteCirculator pdf in the project gives details for running the simulations and interpreting the results.

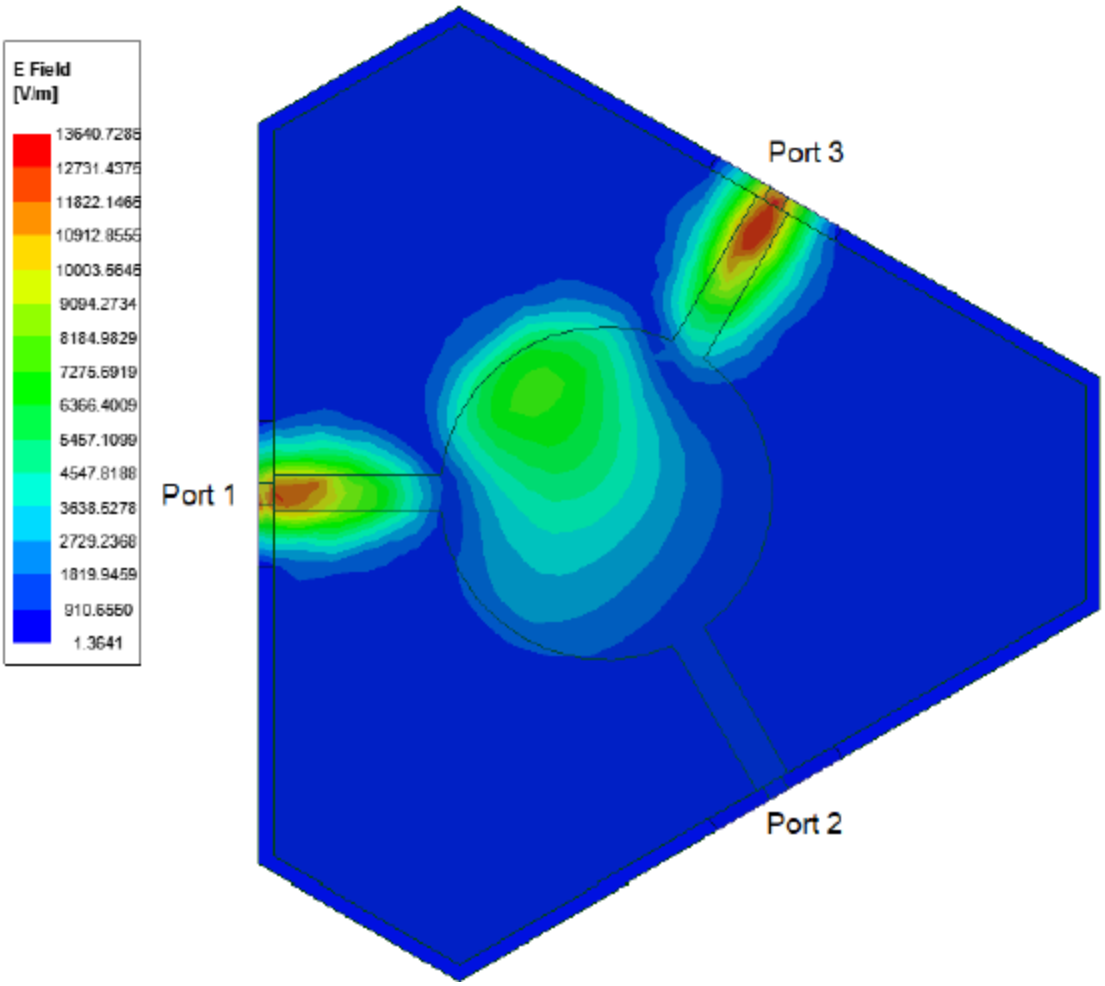
The S parameters plots shown in the following plot show results for a non-uniform biased magnetic field.

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



The electric field plot shows that all the power is being transmitted to port 3 and nothing is going to port 2.

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

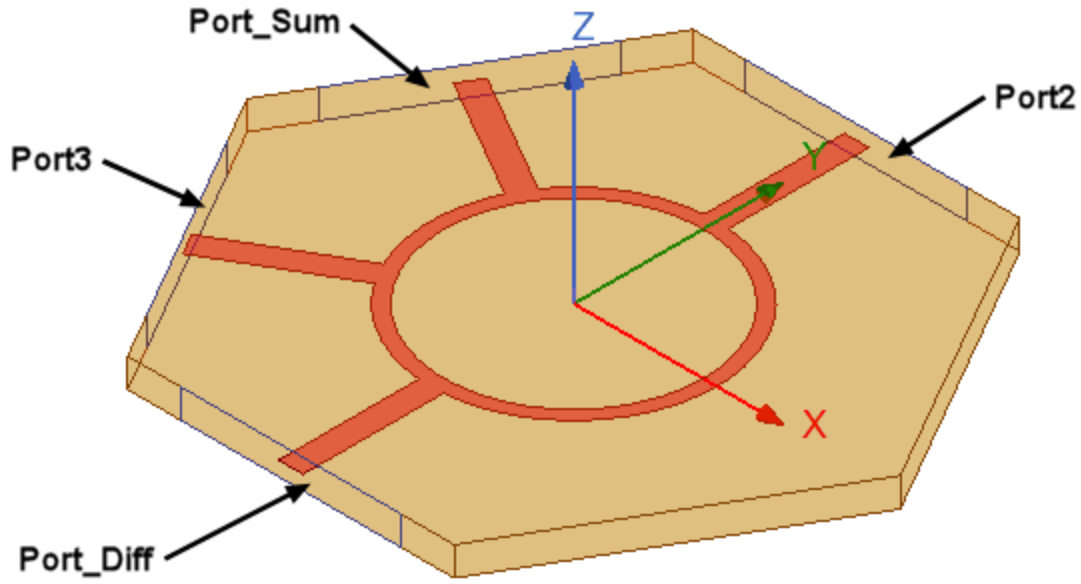


Ring Hybrid

Abstract

This example is a ring hybrid that can be used as a splitter. Power input to **Port_Sum** splits equally and in phase between **Port2** and **Port3**, while power input to **Port_Diff** splits between the same two ports but with a 180° phase shift. This device can be used as a combiner with power input to **Port2** and **Port3**. In this case, the sum of the two inputs will be seen at **Port_Sum**, and the difference at **Port_Diff**.

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



Simulation Time: (machine with 64GB RAM, 6 cores used, Automatic HPC settings, v. 2025 R1)

54 seconds, 359 MB max. memory per process

> **Note:**
>
> Simulation time and memory will vary depending on your available resources. In most cases, HPC (parallel resources) can be leveraged to significantly reduce simulation time and the memory footprint required by each CPU.

Description

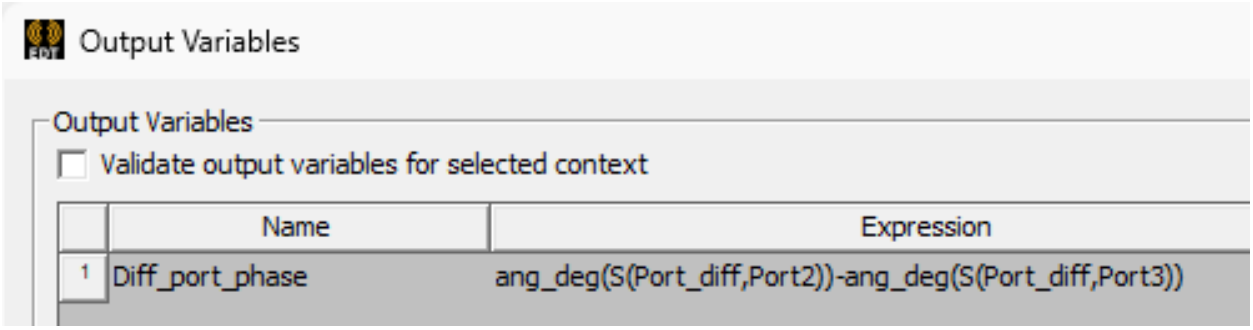
Model:

This ring hybrid is designed to operate at 10 GHz and uses stripline transmission lines. The substrate is 40 mil Duroid 5880™. The ports are defined on four of the faces of the substrate. A default outer boundary (Perfect E) is defined on the remaining faces. The trace is a 2D sheet object, which is also defined using a Perfect E boundary for simplicity.

Setup:

The adaptive solution frequency is 10 GHz with an interpolating frequency sweep from 8 to 12 GHz. The model includes an output variable, **Diff_port_phase** that computes the phase difference between Port2 and Port3 when Port_Diff is excited.

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



Note:

To view a port or boundary, select the desired item in the Project Manager. It is then highlighted in the Modeler window, and its properties are displayed in the docked *Properties* window. Selecting an object in the History Tree also displays that object's properties.

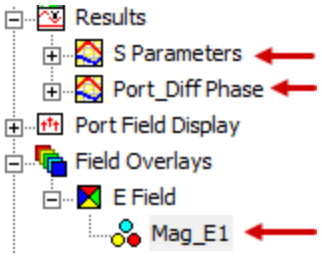
Simulation Summary

After solving, you can view the solution data by right-clicking **Results** in the Project Manager and selecting **Solution Data** from the shortcut menu. In the *Solutions* dialog box, you can view the *Profile, Convergence, Matrix Data,* and *Mesh Statistics* tabs.

Postprocessing

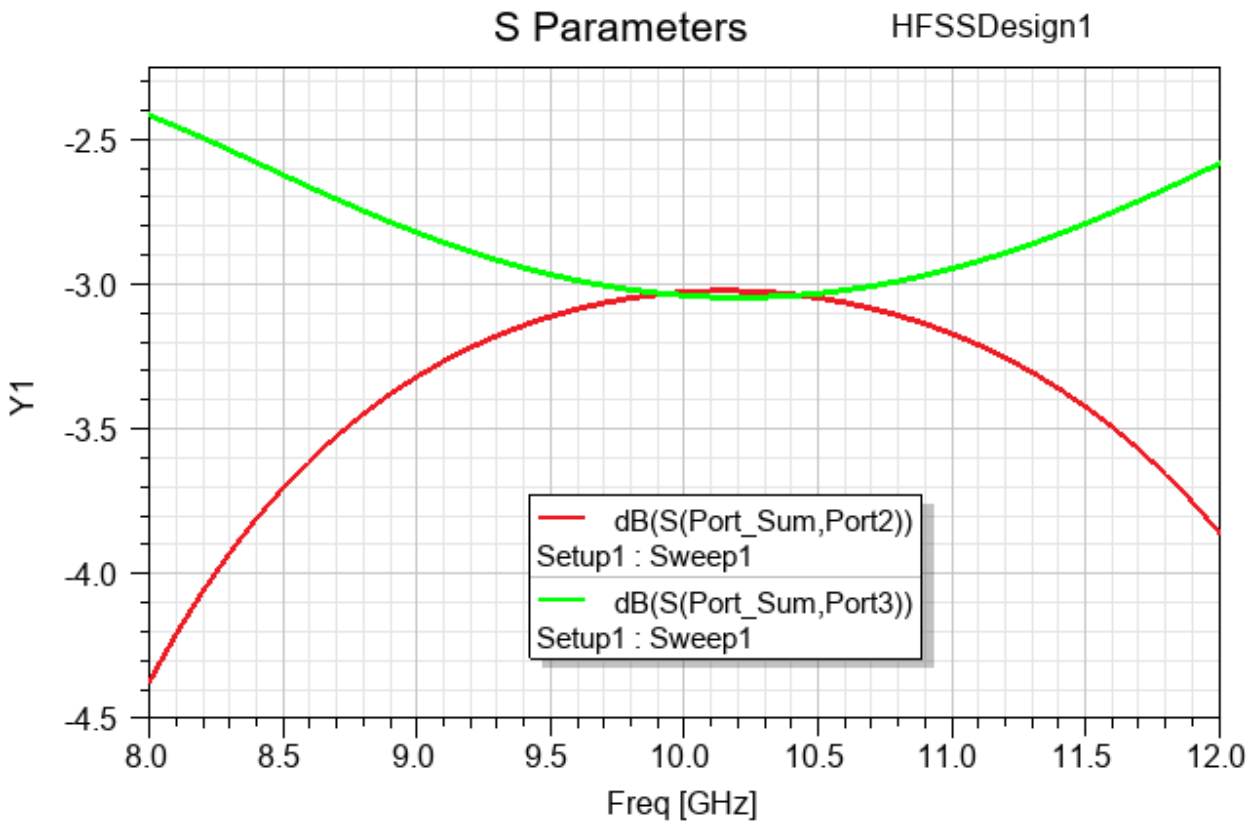
You can generate reports by accessing the **Modal Solution Data Report** menu on the **Results** ribbon tab and choosing the desired type of plot.

Two reports and one field overlay are predefined:

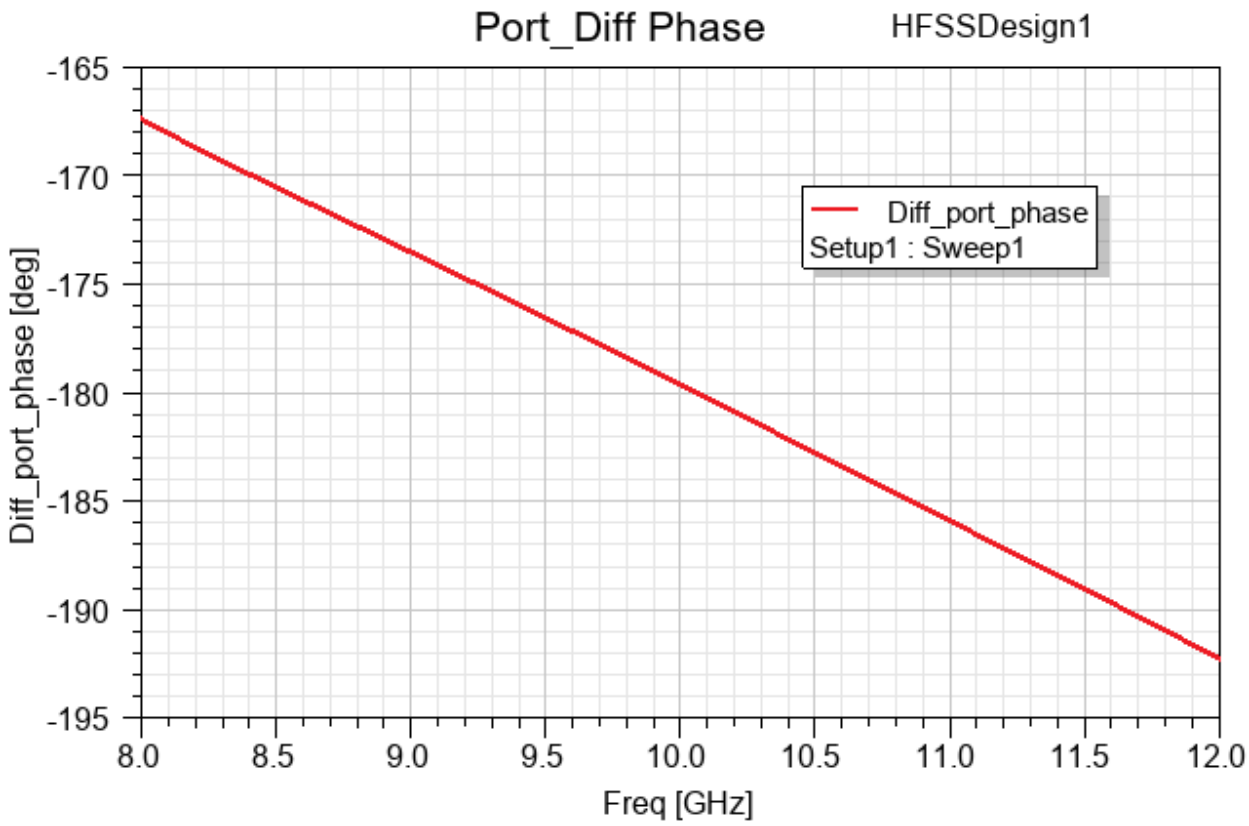


The reports are listed under *Results* in the Project Manager—**S Parameters** and **Port_Diff Phase**. Double-click these reports (shown below) to open and view them:

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



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



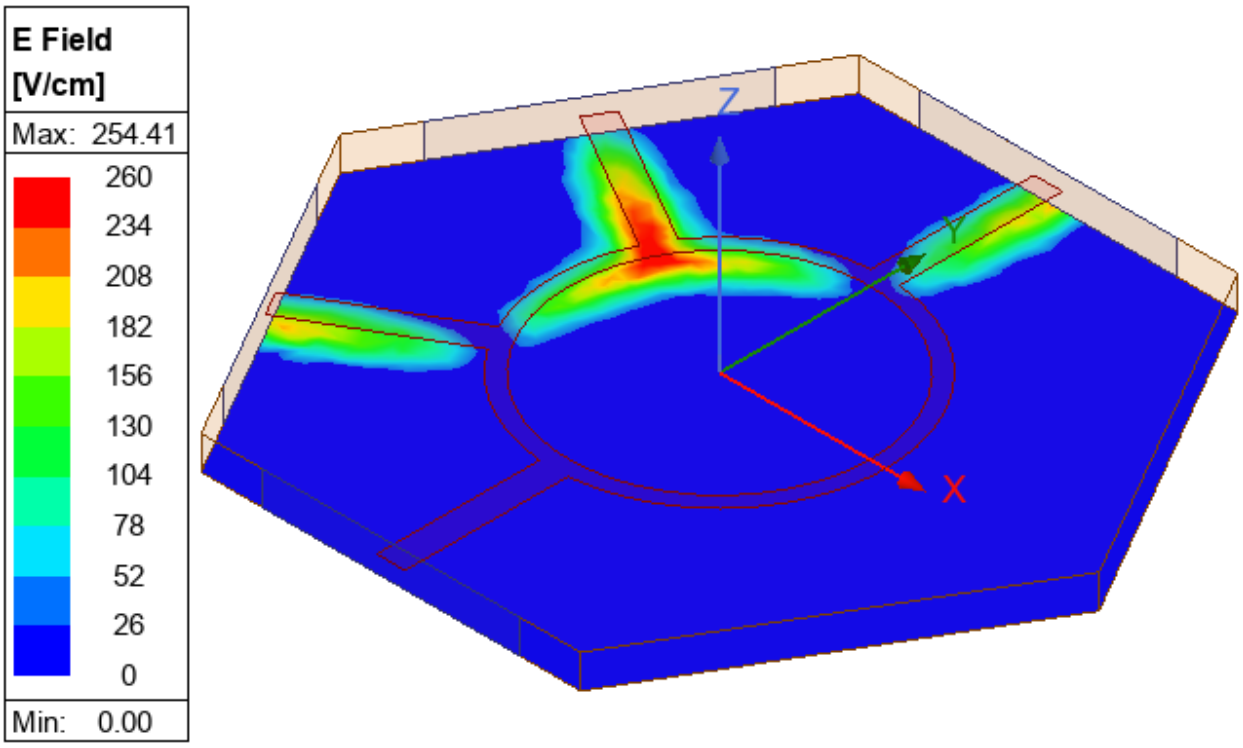
To produce the field overlay shown below, Port2 and Port3 were excited at equal amplitude and phase:

[HFSSDesign1] Edit post process sources

Spectral Fields | Source Context

Source	Type	Magnitude	Unit	Phase	Unit
Port2:1	Port	1	W	0	deg
Port3:1	Port	1	W	0	deg
Port_Diff:1	Port	0	W	0	deg
Port_Sum:1	Port	0	W	0	deg

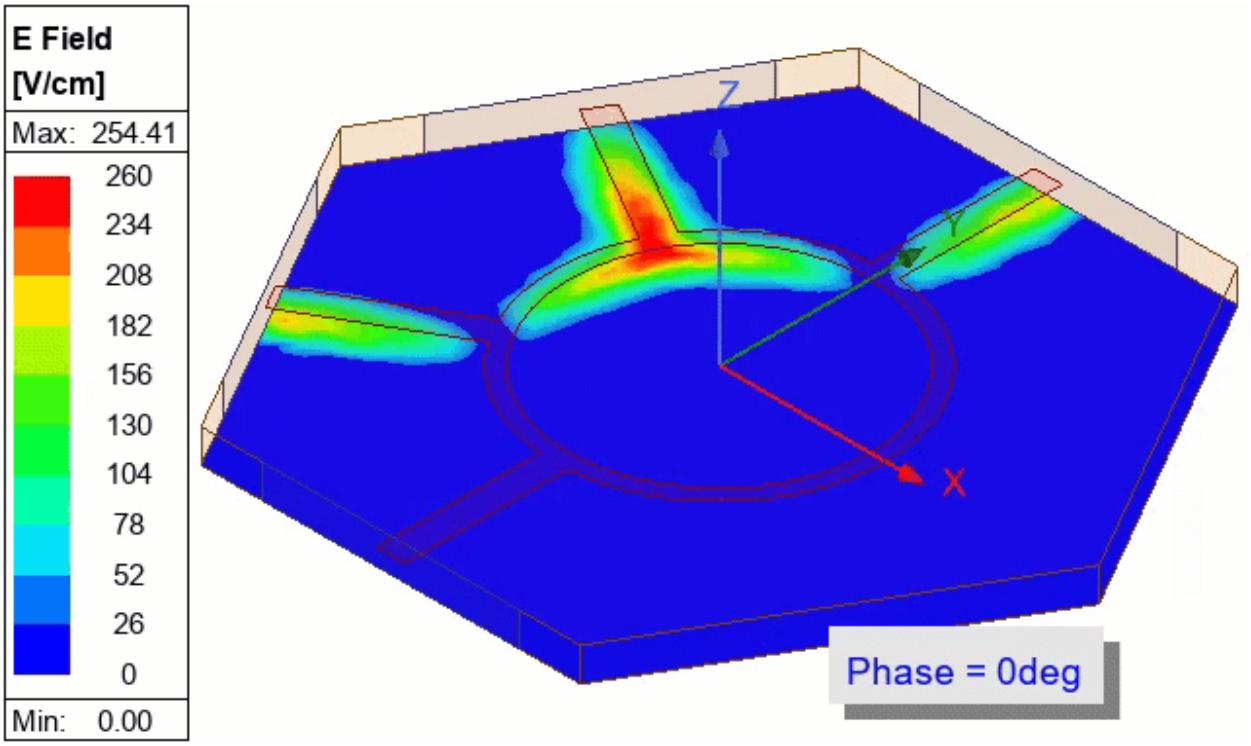
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To see alternative behaviors (such as the effects of a different phase angle or using the ring hybrid as a splitter), change the excitations as desired using **HFSS** > **Fields** > **Edit Sources**.

To view a phase animation of the field overlay, right-click **Mag_E1** and choose **Animate** from the shortcut menu. Then, in the *Select Animation* dialog box, click **Animate** to run the predefined animation setup.

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



Waveguide FEM versus Theory

Abstract

An example of solving a **Rectangular** and **Circular Waveguide** for the propagation constant, frequency cutoff, and wave impedance, by the finite element method (FEM), while making a detailed comparison to theoretical values derived from solving Maxwell's equation for metallic waveguides.

Simulation Time (adaptive solution, frequency sweep, and parametric sweep with 8 cores, 2025 R1):

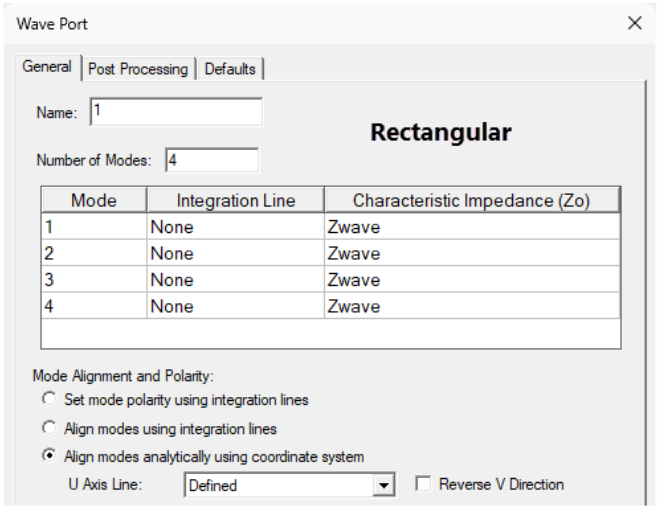
- **Rectangular:** 2 minutes:3 seconds Max. RAM per process = 201 MB)
- **Circular:** 3 minutes:20 seconds (Max. RAM per process = 247 MB)

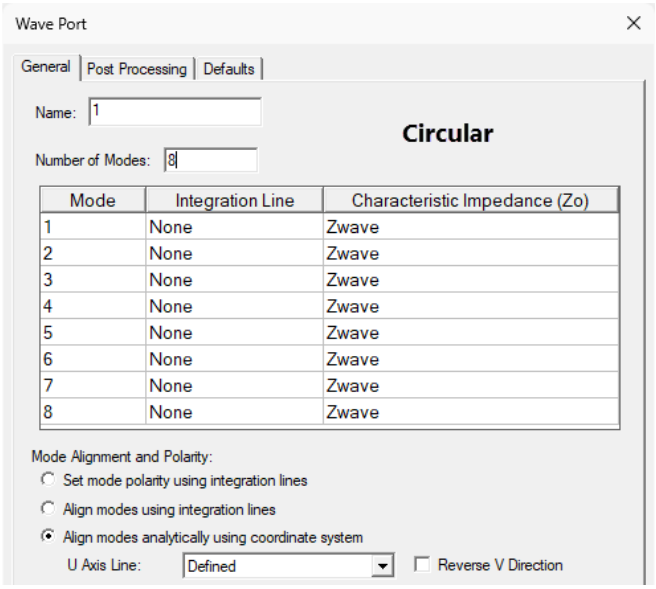
> **Note:**
>
> Simulation time and memory will vary depending on your available resources. In most cases, HPC (parallel resources) can be leveraged to significantly reduce simulation time and the memory footprint required by each CPU.

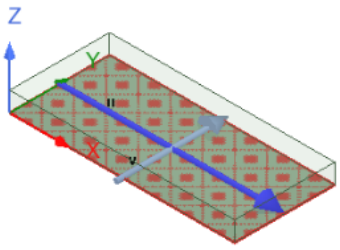
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

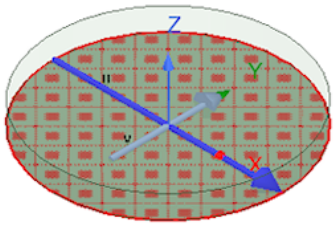
Model and Setup Details

The models are drawn using the box or cylinder primitive, with parameters to control dimensions and solved as a single **Ports Only** solution, as we simply need the port properties for this study. Background **PEC** setting provides the metallic outer boundary, waveguide length is not relevant in this “ports only” example, so is set to a default 2mm. The model is set to solve with required number of modes, **Characteristic Impedance Zo** method is set to **Zwave** to give the wave impedance for comparison with theory. Mode alignment is set analytically as shown below.









Model parameters are defined to scale the waveguide dimensions (**a** and **b** for rectangular, **radius** for circular). Additional parameters demonstrate how to evaluate the cutoff frequencies analytically using the TE/TM mode indices **m** and **n**, as per convention – design equations are shown below. These properties are postprocessing parameters and can be changed to check expected cutoff values for the guide dimensions and mode indices required. They are also used for mode identification in the mode table reports.

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

Rectangular

Properties

Name	Value	Unit	Evaluated Value	Type
_____Waveguide_Properties_____				
a	22.86	mm	22.86mm	Design
b	10.16	mm	10.16mm	Design
er	1		1	Post Processing
ur	1		1	Post Processing
length	2	mm	2mm	Design
____Freq_Cutoff_Calculation__Adjust_m_n_____				
m	1		1	Post Processing
n	0		0	Post Processing
Fcutoff_Mode_mn_GHz	fc		6.5571404	Post Processing

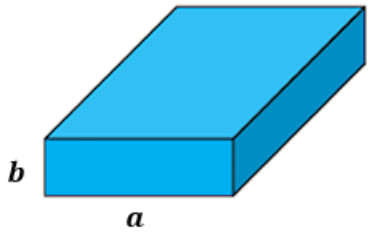
Circular

Properties

Name	Value	Unit	Evaluated Value	Type
_____Waveguide_Properties_____				
radius	10	mm	10mm	Design
er	1		1	Post Processing
ur	1		1	Post Processing
length	2	mm	2mm	Design
____Freq_Cutoff_Calculation__Adjust_m_n_____				
m	1		1	Post Processing
n	1		1	Post Processing
Fcutoff_TE	Fc_TE		8.7849233	Post Processing
Fcutoff_TM	Fc_TM		18.282392	Post Processing

$$f_{cutoff\ m,n} = \frac{c_o}{2 \cdot \pi \cdot \sqrt{\mu_r \varepsilon_r}} \cdot \sqrt{\left(\frac{m \cdot \pi}{a}\right)^2 + \left(\frac{n \cdot \pi}{b}\right)^2}$$

$$f_{cutoff_{lowest}} = \frac{c_o}{2 \cdot a} \qquad c_o = \frac{1}{\sqrt{\mu_o \varepsilon_o}}$$



(TE10 for $a > b$)

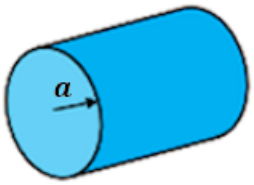
$$f_{cutoff\ TM_{m,n}} = \frac{c_o}{2 \cdot \pi \cdot \sqrt{\mu_r \varepsilon_r}} \cdot \frac{X_{m,n}}{a}$$

$$f_{cutoff\ TE_{m,n}} = \frac{c_o}{2 \cdot \pi \cdot \sqrt{\mu_r \varepsilon_r}} \cdot \frac{X'_{m,n}}{a} \qquad c_o = \frac{1}{\sqrt{\mu_o \varepsilon_o}}$$

where:

$X_{m,n}$ is the nth root of the mth Bessel function, and

$X'_{m,n}$ is the nth root of the mth Bessel function derivative

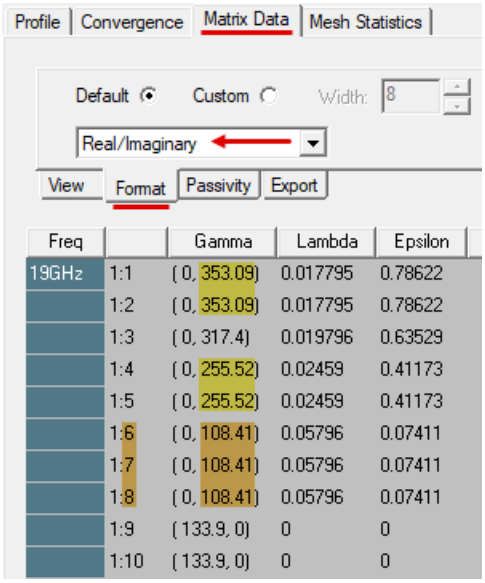


$$f_{cutoff\ TE_{lowest}} = \frac{1.8412 c_o}{2 \cdot \pi \cdot a}$$

$$f_{cutoff\ TE_{lowest}} = \frac{2.4049 c_o}{2 \cdot \pi \cdot a}$$

A single waveport is defined with the number of modes set as required. For this wideband test case, ensure that enough modes are enabled in the port setup to produce accurate results. Run the analysis initially with the frequency sweep disabled. Then, inspect the *Matrix Data* to evaluate the mode Gamma values with the *Format* set to **Real/Imaginary**. For example, with 10 modes set for the circular waveguide, we can see several modes have the same beta values (right/imaginary part of Gamma), indicating degenerate modes. So, if we wish to include mode 6, we should also include modes 7 and 8 (as shown in the preceding *Wave Port* dialog box image for the Circular design).

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



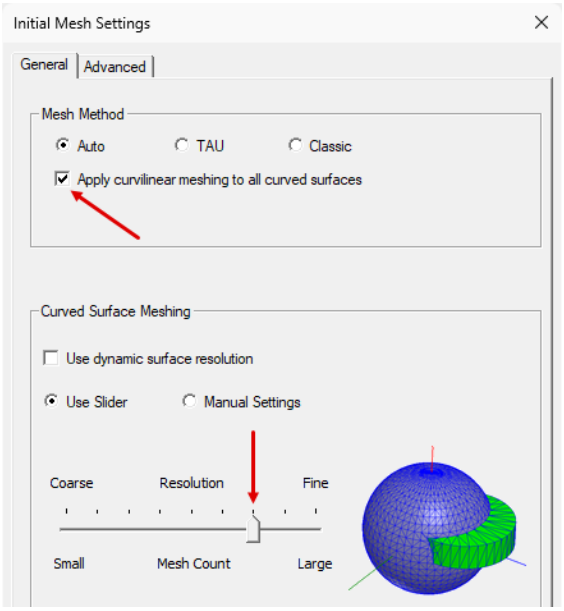
Freq		Gamma	Lambda	Epsilon
19GHz	1:1	(0, 353.09)	0.017795	0.78622
	1:2	(0, 353.09)	0.017795	0.78622
	1:3	(0, 317.4)	0.019796	0.63529
	1:4	(0, 255.52)	0.02459	0.41173
	1:5	(0, 255.52)	0.02459	0.41173
	1:6	(0, 108.41)	0.05796	0.07411
	1:7	(0, 108.41)	0.05796	0.07411
	1:8	(0, 108.41)	0.05796	0.07411
	1:9	(133.9, 0)	0	0
	1:10	(133.9, 0)	0	0

The **analysis setup** and **frequency sweep** are defined based on the frequency span expected to evaluate the desired number of modes:

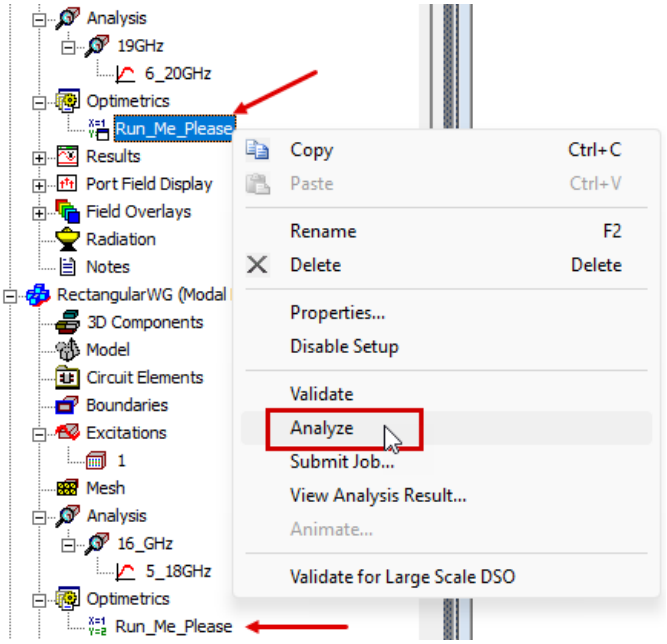
- **Rectangular:**
  - Adaptive Frequency: 16 GHz
  - Sweep: 5–18 GHz in 10,001 points (1.3 MHz increments)
- **CylindricalWG:**
  - Adaptive Frequency: 19 GHz
  - Sweep: 6–20 GHz in 20,001 points (0.7 MHz increments)

Initial mesh settings are adjusted only for the circular waveguide, with curvilinear elements enabled and a finer mesh slider position:

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



A **Parametric** sweep has been defined for each design to look at different combinations of the **m** and **n** indices (n^{th} root of m^{th} Bessel function and its derivative). Right-click and analyze each of these two parametric setups to run the adaptive solution, frequency sweep, and parametric sweep:

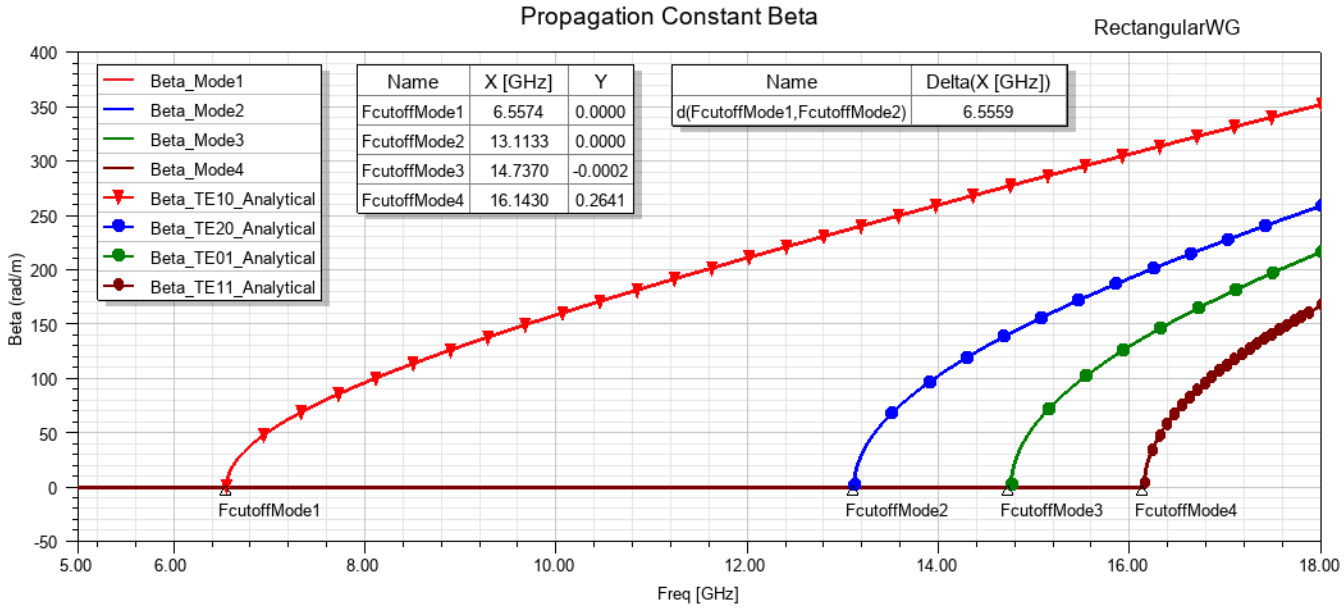


Postprocessing

Rectangular Waveguide:

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

In the Project Manager, expand the *Results* folder in the *RectangularWG* branch to view the predefined reports. The **Propagation Constant Beta** plot shows the phase constant for the first four modes, with analytical calculations overlaid (defined as output variables). Trace markers (**Add Minimum**) are used to identify cutoff, a delta marker shows the single mode range (FcutoffMode1–2).



The **TE/TM Mode** table shows the evaluated cutoff frequencies for the modes as a function of indices **m** and **n**. You can use this table to confirm the FEM cutoff frequencies and to identify the modes.

TE/TM Mode m,n Cutoff Frequencies

RectangularWG

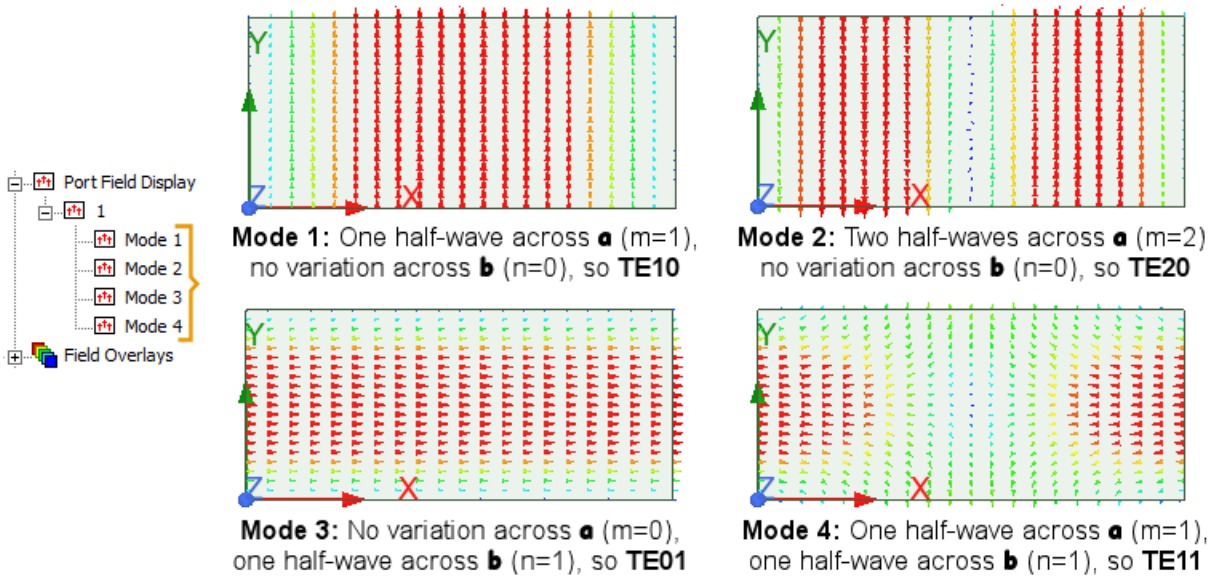
	m	Fcutoff_Mode_mn_GHz Freq='4GHz' n='0'	Fcutoff_Mode_mn_GHz Freq='4GHz' n='1'	Fcutoff_Mode_mn_GHz Freq='4GHz' n='2'	Fcutoff_Mode_mn_GHz Freq='4GHz' n='3'
1	0	0.000000	14.753566	29.507132	44.260698
2	1	6.557140	16.145086	30.226924	44.743775
3	2	13.114281	19.739607	32.290172	46.162687
4	3	19.671421	24.589276	35.463159	48.435257

- **Mode 1:** FcutoffMode1 (6.5574 GHz FEM / 6.5571 theory, 0.005% variance), can either be TE10 or TM01. The port field display (below) identifies it as TE10.
- **Mode 2:** FcutoffMode2 (13.1133 GHz FEM / 13.1143 theory, 0.008% variance), can either be TE20 or TM02. The port field display identifies it as TE20.
- **Mode 3:** FcutoffMode3 (14.7370 GHz FEM / 14.7536 theory, 0.113% variance), can either be TE01 or TM10. The port field display identifies it as TE01.

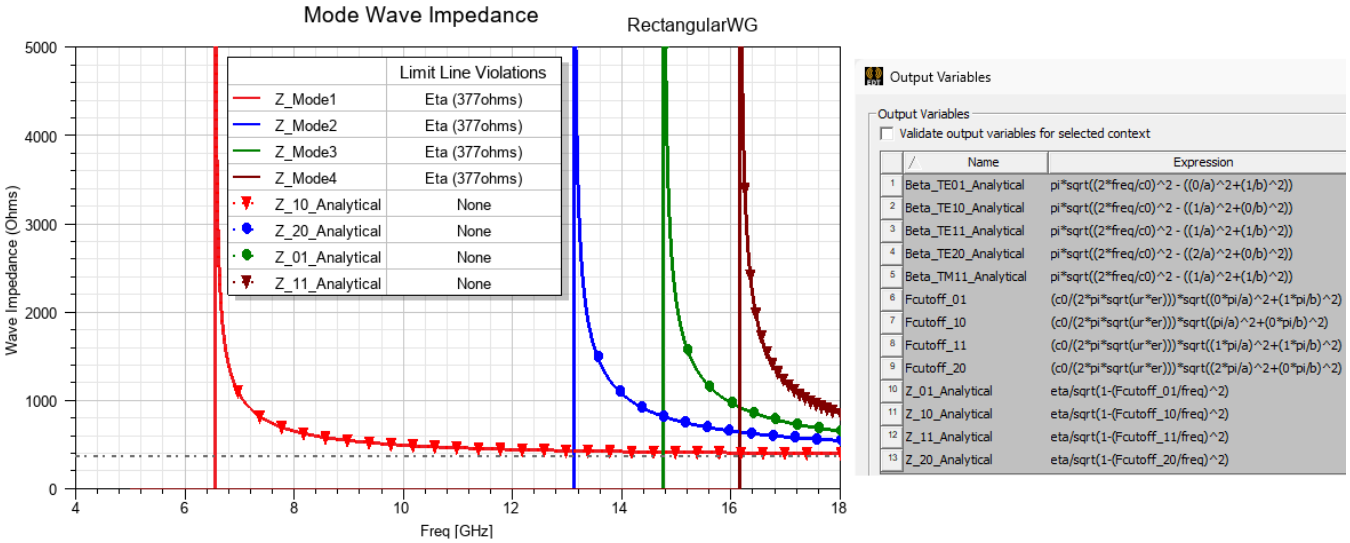
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mode 4:** FcutoffMode4 (16.1430 GHz FEM / 16.1451 theory, 0.013% variance), can either be TE11 or TM11. The port field display identifies it as TE11.

You can make the final mode identification by using the **Port Field Display** for each mode:



The **Mode Wave Impedance** plot shows the wave impedances (Zwave) compared to theoretical values defined using output variables:



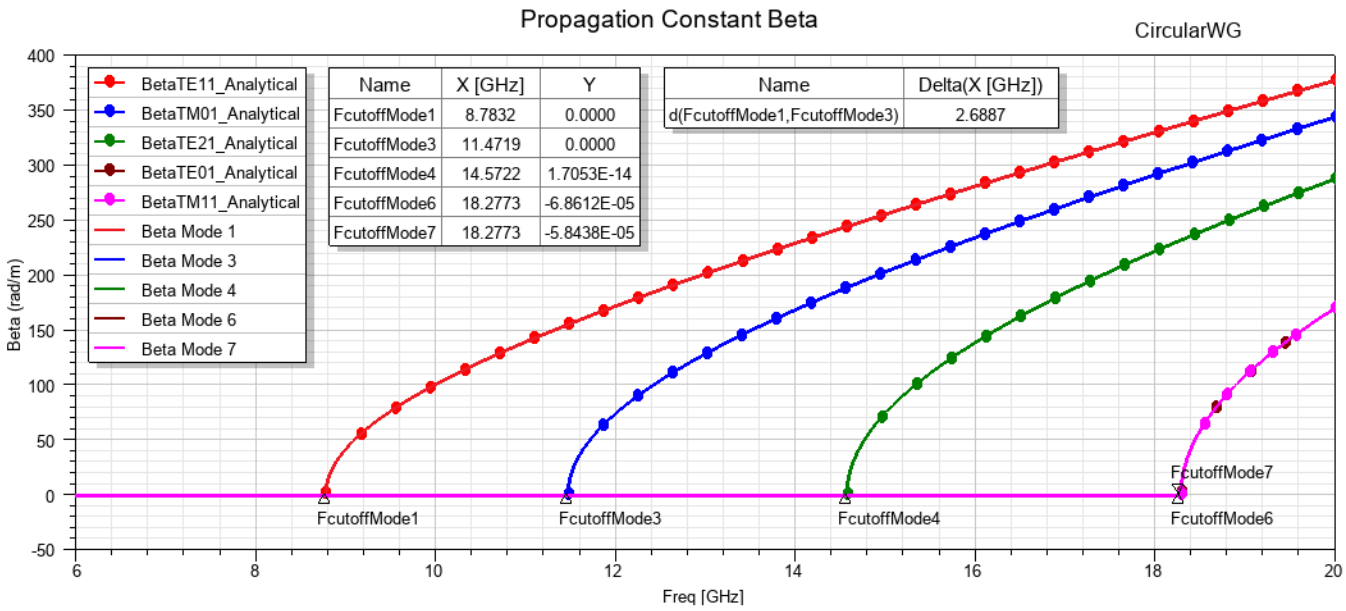
Circular Waveguide:

In the Project Manager, expand the *Results* folder in the *CircularWG* branch to view the predefined reports. The **Propagation Constant Beta** plot shows the phase constant for the first

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

seven modes (ignoring degenerate modes 2 and 5), with analytical calculations overlaid (defined as output variables).

Trace markers (**Add Minimum**) are used to identify cutoff, and a delta marker shows the single mode range (FcutoffMode1–3) = 2.6887 GHz



The **TE Mode** and **TM Mode** tables show the analytical cutoff frequencies for the modes as a function of indices **m** and **n**. You can use these tables to confirm the FEM cutoff frequencies and to identify the modes. Use the **Port Field Display** to confirm the following modes:

TE Mode m,n Cutoff Frequencies — CircularWG

	m	Fcutoff_TE Freq='19GHz' n='1'	Fcutoff_TE Freq='19GHz' n='2'	Fcutoff_TE Freq='19GHz' n='3'
1	0	18.282392	33.473785	48.541128
2	1	8.784923	25.438154	40.729712
3	2	14.572819	31.997276	47.567772
4	3	20.045323	38.243460	54.135321

TM Mode m,n Cutoff Frequencies — CircularWG

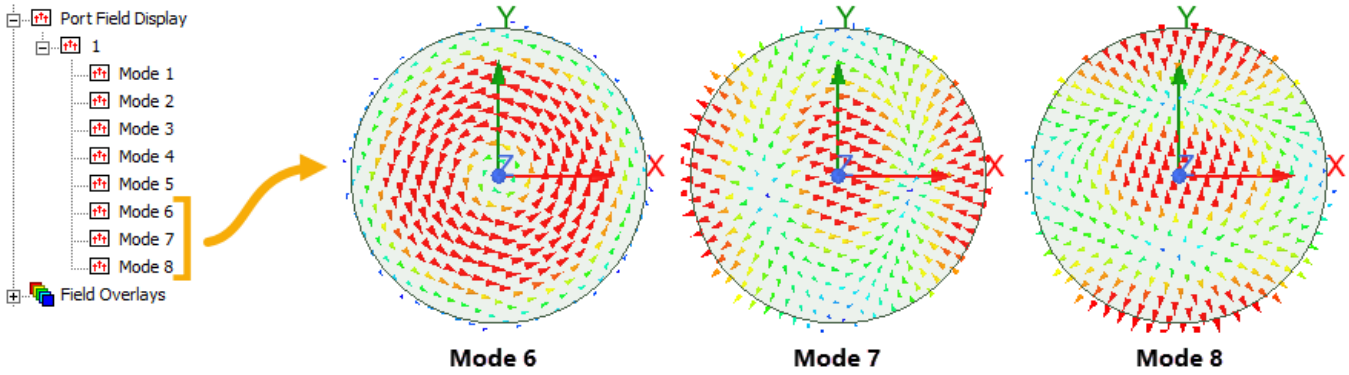
	m	Fcutoff_TM Freq='19GHz' n='1'	Fcutoff_TM Freq='19GHz' n='2'	Fcutoff_TM Freq='19GHz' n='3'
1	0	11.474253	26.338198	41.289923
2	1	18.282392	33.473785	48.541128
3	2	24.503827	40.161577	55.442273
4	3	30.441955	46.573210	62.100015

Modes 1, 3, and 4 are unique in the tables and easy to identify:

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

- **Mode 1:** FcutoffMode1 (8.7832 GHz FEM / 8.7849 theory, 0.02% variance). From the TE Mode table, m=1 and n=1. Therefore, this mode is TE11.
- **Mode 3:** FcutoffMode3 (11.4719 GHz FEM / 11.4743 theory, 0.02% variance). From the TM Mode table, m=0 and n=1. Therefore, this mode is TM01.
- **Mode 4:** FcutoffMode4 (14.5722 GHz FEM / 14.5728 theory, 0.006% variance). From the TE Mode table, m=2 and n=1. Therefore, this mode is TE21.

Identify modes 6 through 8, for which the tables are inconclusive, using the **Port Field Display**:

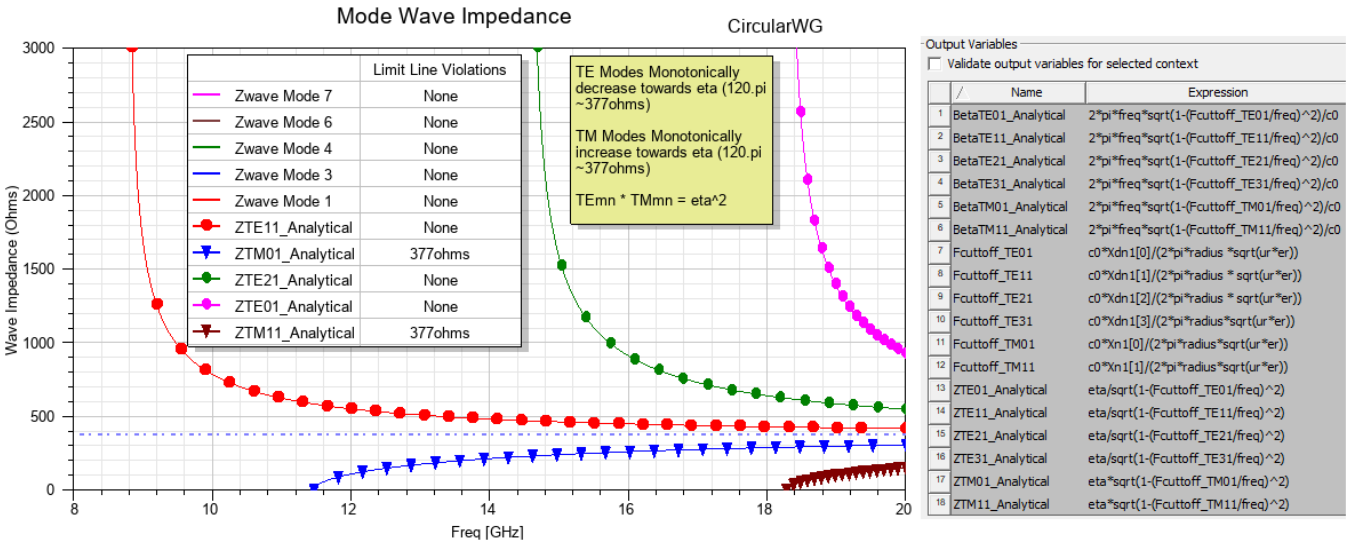


- **Mode 6:** FcutoffMode6 (18.2773 GHz FEM / 18.2824 theory, 0.03% variance), this frequency appears in both mode tables, and it can be either TE01 or TM10. From the port field display, no field variation occurs around the circumference, so m=0. One minimum occurs along the radius, so n=1. Therefore, this mode must be mode TE01.
- **Mode 7** and **Mode 8:** (Same frequency as Mode 6) Two field maxima occur around the circumference, so m=1. One minimum occurs along the radius. Therefore, this mode must be TM11.

> **Note:**
>
> HFSS lists modes in order of decreasing Gamma value. For modes with the same Gamma value (for example, modes 6, 7, and 8), it is possible that HFSS may alter the order in which the modes are numbered due to numerical rounding.

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

The **Mode Wave Impedance** plot shows the wave impedances (Zwave) compared to theoretical values defined using output variables:



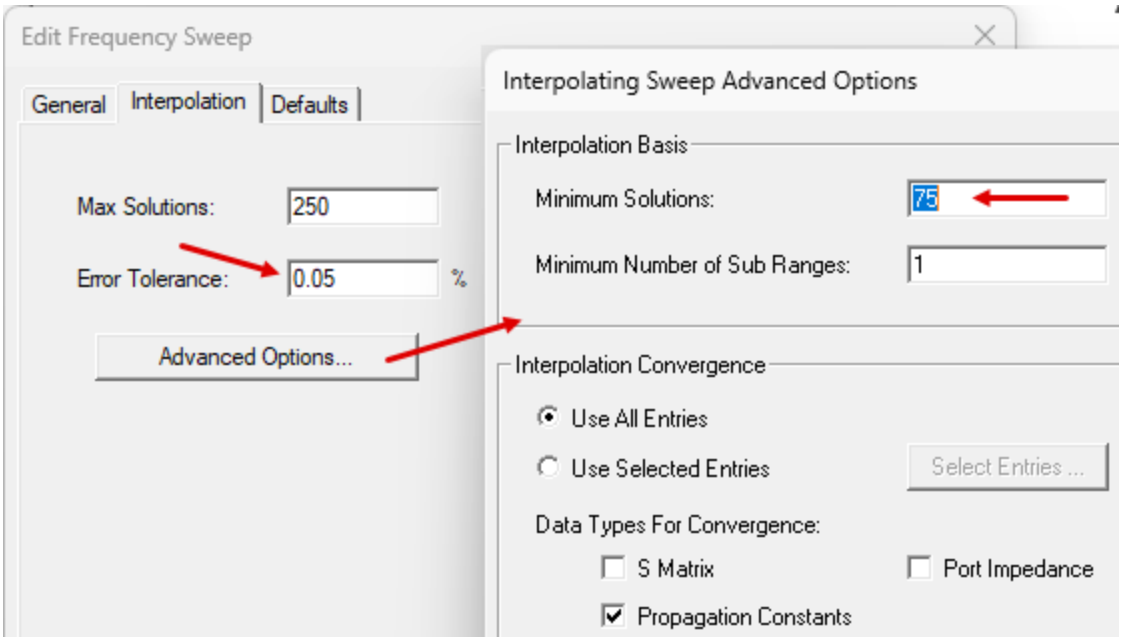
> **Note:**
>
> In the preceding plot, the direction from which the Zwave curve approaches Eta (377 ohms) is helpful in determining whether the curve represents a TE or TM mode.

Interpolating Sweep Setup

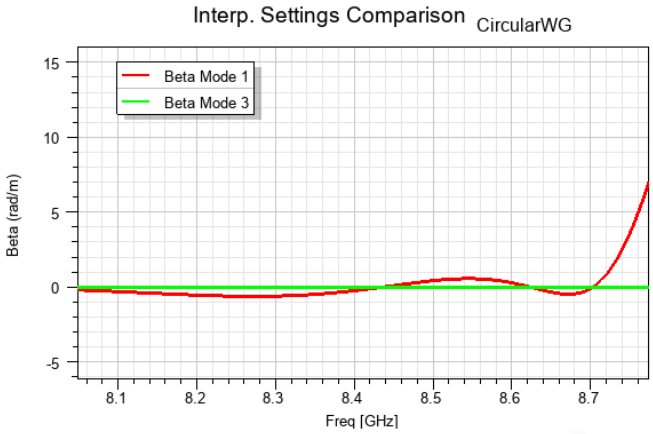
To accurately capture the abrupt change in Beta at cutoff, particularly when comparing it to analytical results, it may be necessary to adjust the interpolating sweep settings to improve fitting fidelity. Adjust these from the *Edit Frequency Sweep* dialog box (**Interpolation** tab).

Decrease the **Error Tolerance** (for example, 0.05%), click **Advanced Options** and increase **Minimum Solutions** (50–75) for better fitting at the zero point:

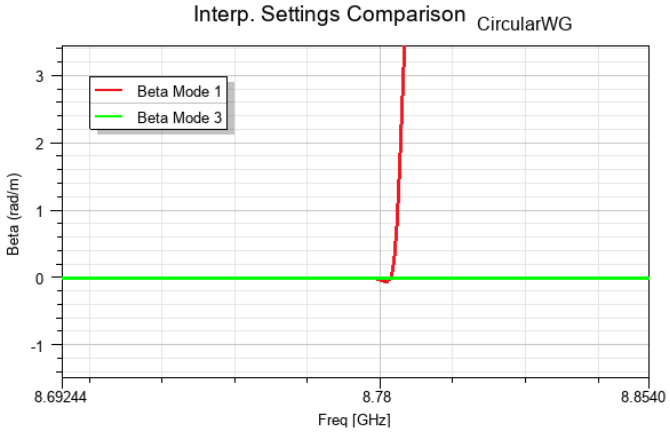
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A comparison of the results from the default and adjusted interpolation settings follows:



Error Tolerance: 0.5% (Default)
Minimum Solutions: 10 (Default = 0)



Error Tolerance: 0.05%
Minimum Solutions: 75

Signal Integrity Examples

Click graphics for descriptions of the following HFSS signal integrity examples:

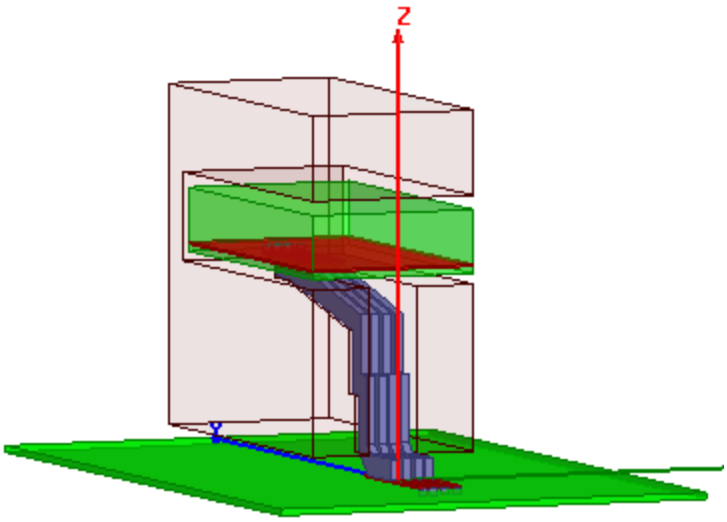
connector.aedt

package.aedt

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

Connector - Terminal Example

Description - a simplified model of a four pin section of a connector. This is a driven terminal design.



Model - the connector is configured with lumped ports on each end of the two inner pins. The two outer pins are each grounded at both ends. The boards are FR4 and the connector body is modified epoxy. A radiation boundary is applied to the surrounding airbox.

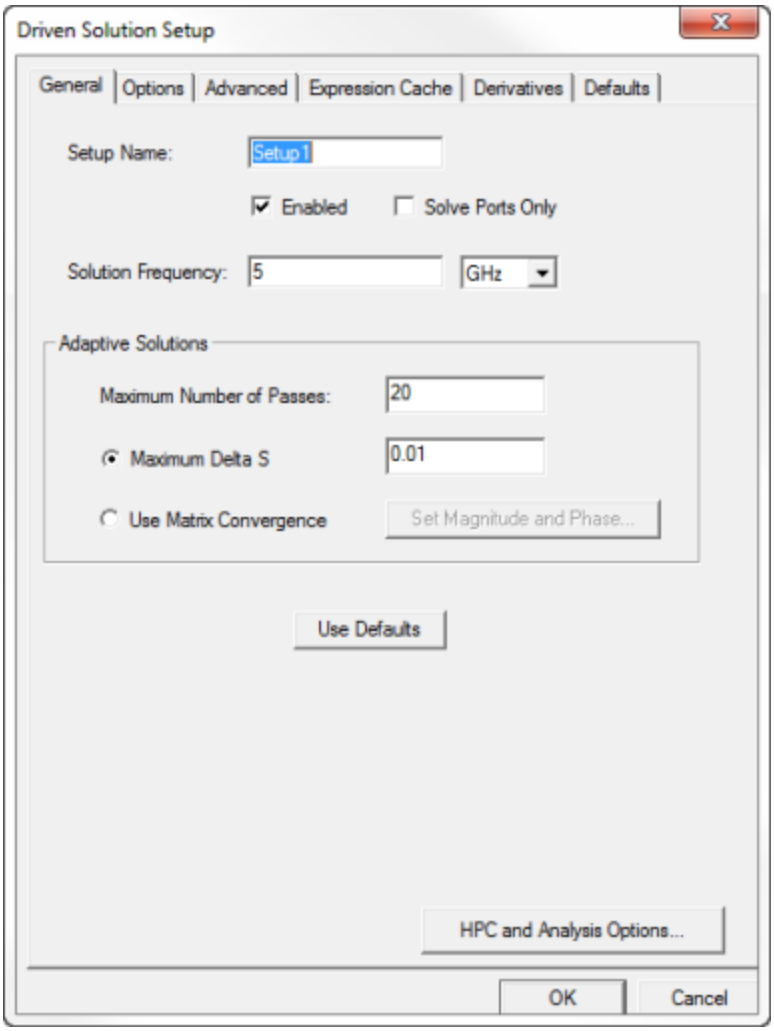
Setup - Driven Terminal Solution with adapt at 5 GHz. An interpolating sweep is also included that has an upper frequency of 5 GHz and uses DC extrapolation at the low end.

> **Note:**
>
> To view a port or boundary, select the desired item in the Project Tree. It is then highlighted in the Model window and the properties will be displayed in the Properties window. Selecting an object in the History tree will also display its properties.

HPC Analysis Setup

You can set an adapt frequency at 5 GHz with an interpolating frequency sweep from 0 to 5 GHz. Since several frequencies are being solved in this design, you can set up an HPC Analysis to distribute the frequencies resulting in efficient simulation.

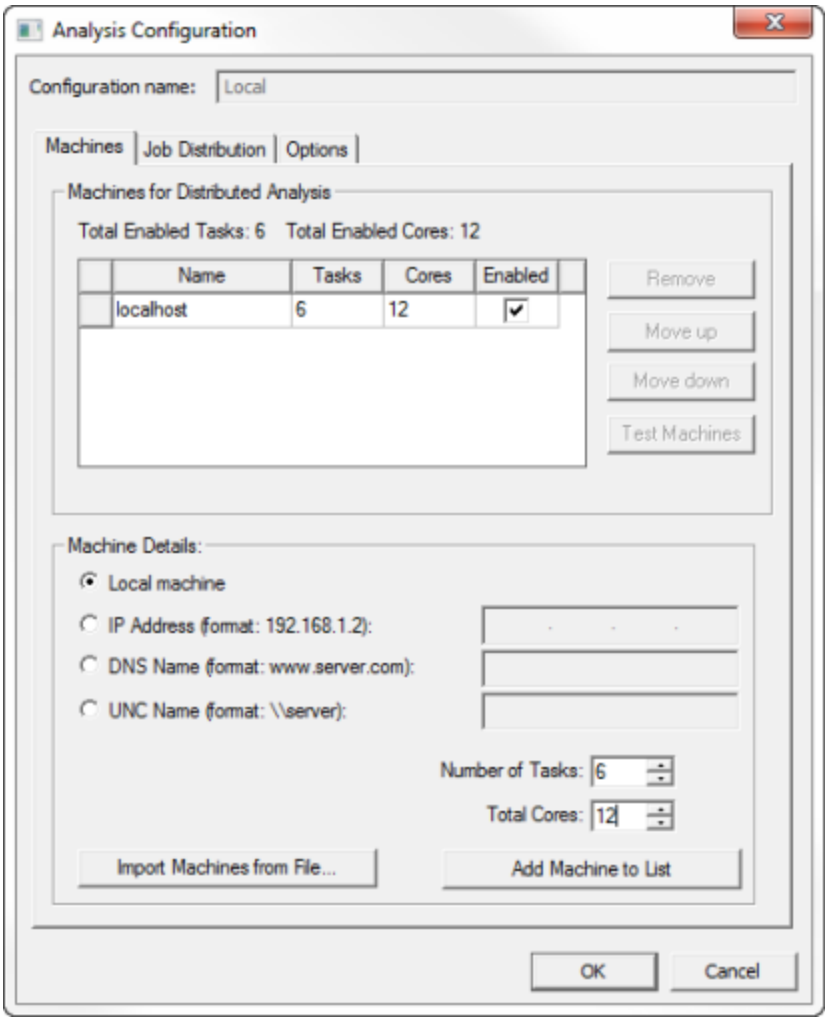
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



During the simulation, adaptive mesh refinement uses the **Total Cores** that are configured in the HPC setup, while the number of cores used to solve each frequency point is determined by **Total Cores**/**Number of Tasks** configured in the **Analysis Configuration**. On the **Driven Solution Setup** dialog box, if you click the **HPC and Analysis Options** button and click **Add**, you can set the **Number of Tasks** and the **TotalCores** on the **Analysis Configuration** window.

Connector Post Processing

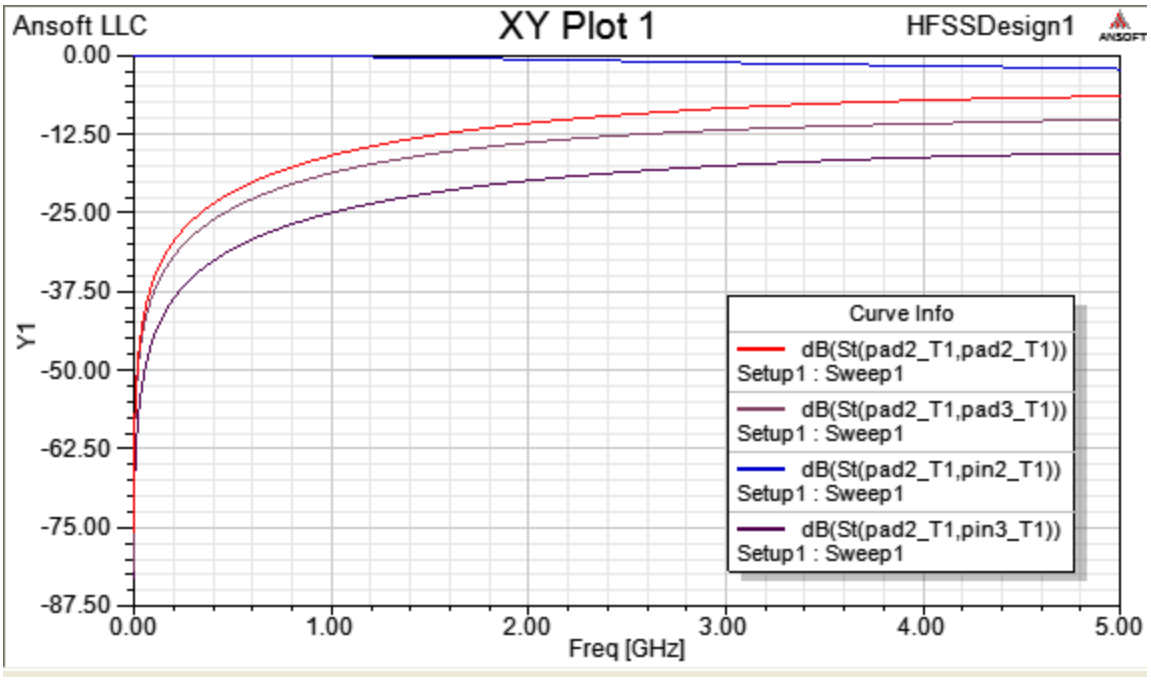
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After solving, you can view solution data by right-clicking on Setup1 and selecting **Profile** to display the **Solution** dialog. You also view the **Solution** tabs for **Convergence**, **Matrix Data**, and **Mesh Statistics**.

To view the S parameter plot show below, double-click on XY plot1 in the Project Tree under Results.

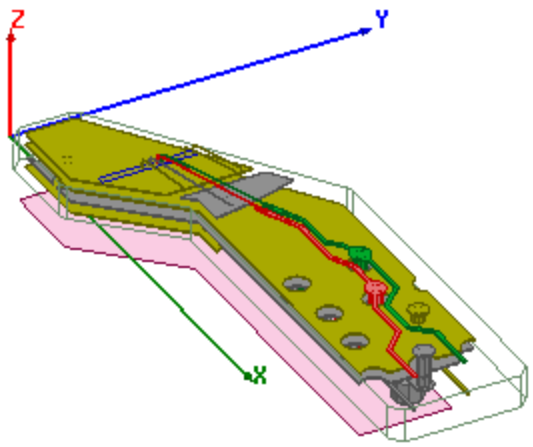
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Package Section

Description - a model of a section of a complex package. The model was created using ALinks from an ANF file and is a driven terminal design.

You can locate this model (**Package.aedt**) in the **Examples\HFSS\Signal Integrity** folder.



Model - the model is two traces of a larger package. At one end are the bond wires that are excited by lumped ports. The opposite ends terminate in ports at the solder balls. The substrate is FR4 and a radiation boundary has been applied to the surface of the surrounding airbox.

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

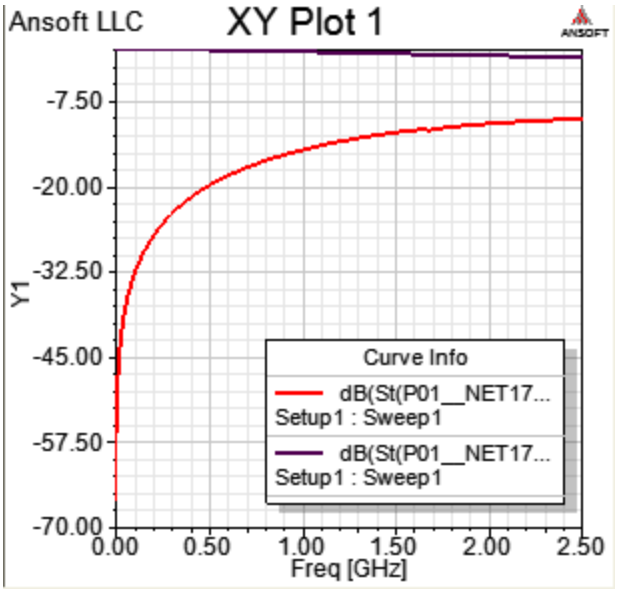
Setup - driven terminal setup with an adapt frequency of 2.5 GHz. An interpolating sweep is also included that has an upper frequency of 2.5 GHz and uses DC Extrapolation at the lower end.

> **Note:**
>
> To view a port or boundary, select the desired item in the Project Tree. It is then highlighted in the Model window and the properties will be displayed in the Properties window. Selecting an object in the History tree will also display its properties.

Package Post Processing

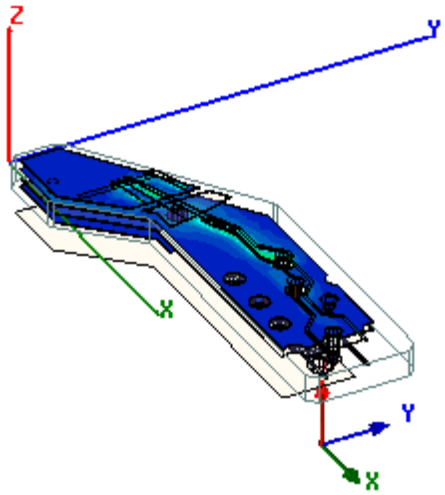
After solving, you can view solution data by right-clicking on Setup1 and selecting **Profile** to display the **Solution** dialog. You also view the **Solution** tabs for **Convergence**, **Matrix Data**, and **Mesh Statistics**.

To view the S parameter plot show below, double-click on XY plot1 in the Project Tree under Results.



To view the shade plot of jsurf, the surface current density, on one of the package planes, right-click Jsurf under Field Overlays in the Project Tree and select update.

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



Transmission Line Examples

Click on the following examples for descriptions of transmission line examples provided with HFSS.

Co-Planar Waveguide (Driven Terminal)

The coplanar waveguide CPW consists of a signal trace sandwiched between two coplanar ground conductors. The width of the signal trace and the gap between the trace and the ground conductors affect the characteristic impedance. Model a short length as shown below and to obtain a longer length of the model you can deembed out of the port.

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

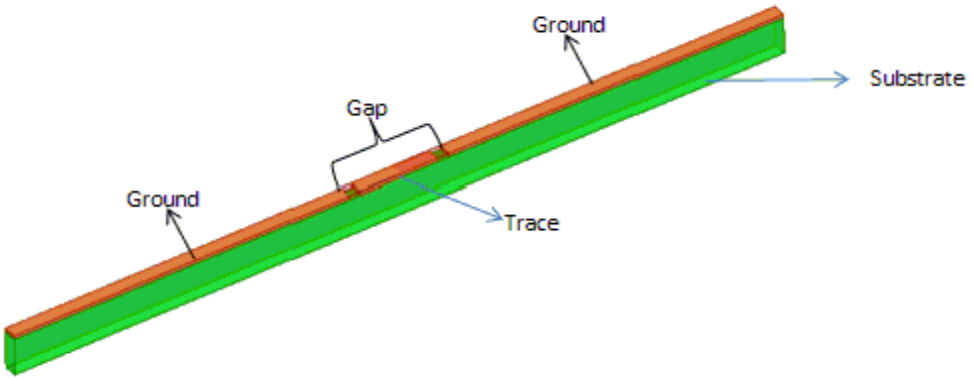


Figure 1 CPW (air box + ports hidden)

Define the ports such that only their faces touch the air box. The edges of the ports should not touch the edges of the air box.

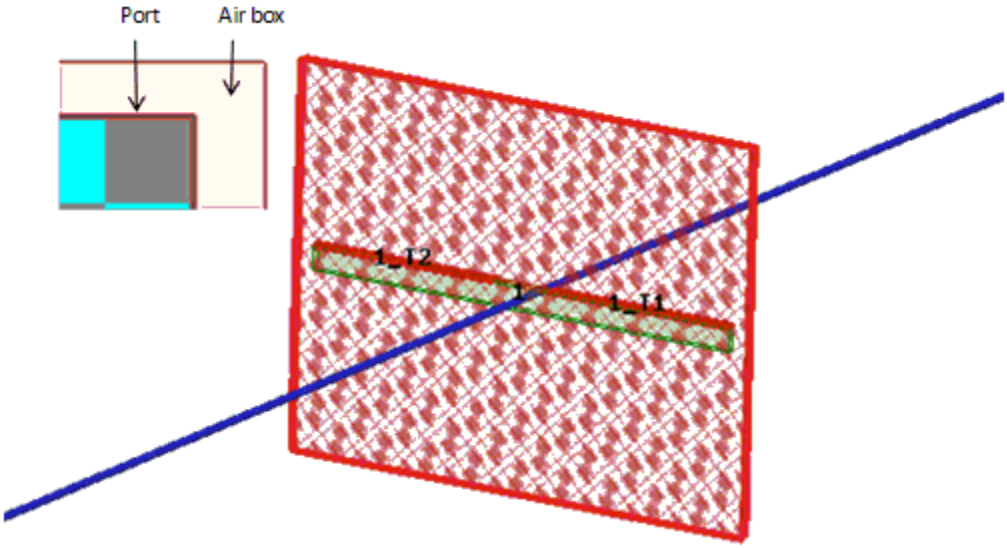


Figure 2 ports with dembeddding

Define the Radiation boundary only along the thickness of the air box. Assigning a radiation boundary on all surfaces of the air box in this model can make the port boundary to be conducting.

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

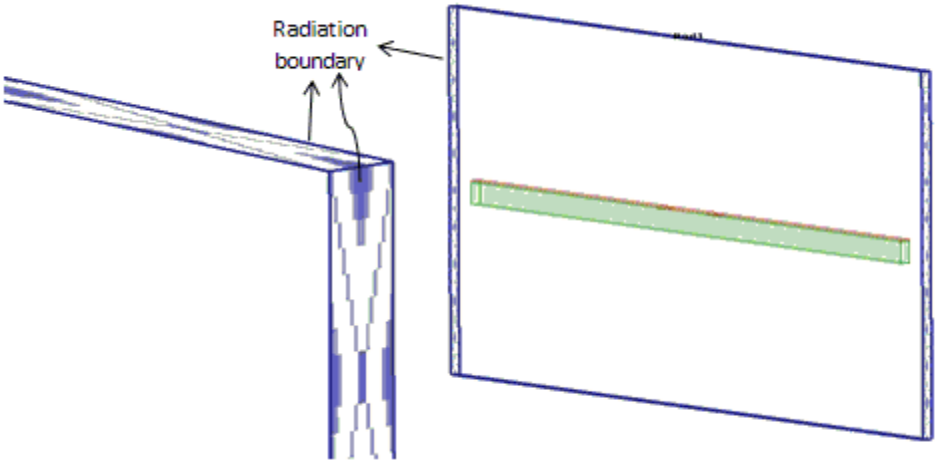


Figure 3 Radiation Boundary

Define a perfect H boundary on the air box. The wave port touches a perfect H boundary and therefore becomes an open circuit.

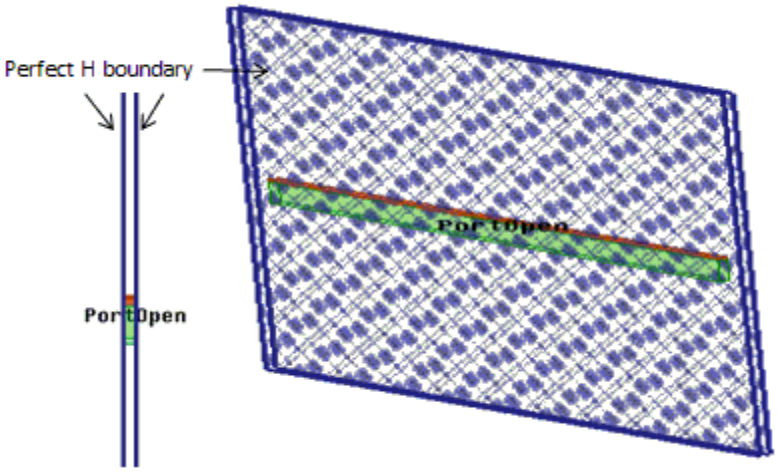


Figure 4 Perfect H boundary

So, the port boundary does not stay as a conductor anymore and almost mimics a “perfect open.” This is because with the application of the perfect H boundary “behind” the wave port, the port boundary becomes an open and will no longer be one of the conductors associated with the port. Now with three conductors namely, the two grounds and the trace, there are two possible modes that this structure can carry. Obviously, for the CPW structure we are interested in the center conductor excited at a voltage with reference to the two sides (or what we arbitrarily call “ground”) conductors at zero potential. Since voltage values can be arbitrary, this same mode

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

could also be considered as the center conductor at “0” volt with the two side conductors at some equal voltage offset from the center conductor. In the terminal framework such a mode can be described as the center conductor labeled “reference” conductor with the two outside conductors considered to be the “terminals.” Then, by placing those two conductors at equal potential with respect to the center conductor they can be defined as differential pair whose common mode is the aforementioned mode of interest.

The Edit Post Process Sources dialog is shown below.

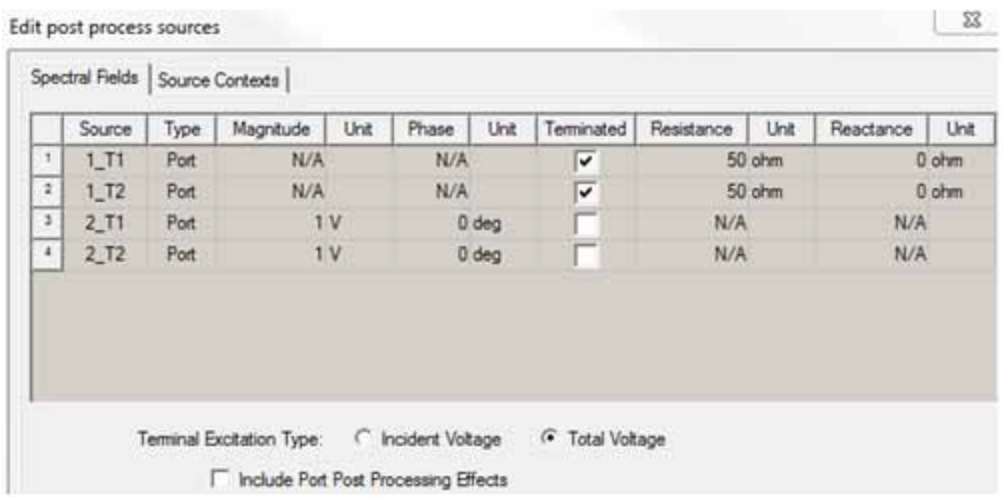


Figure 5 Edit Port Process Dialog

Notice from the plots below how the field gets trapped in the signal trace and dielectric.

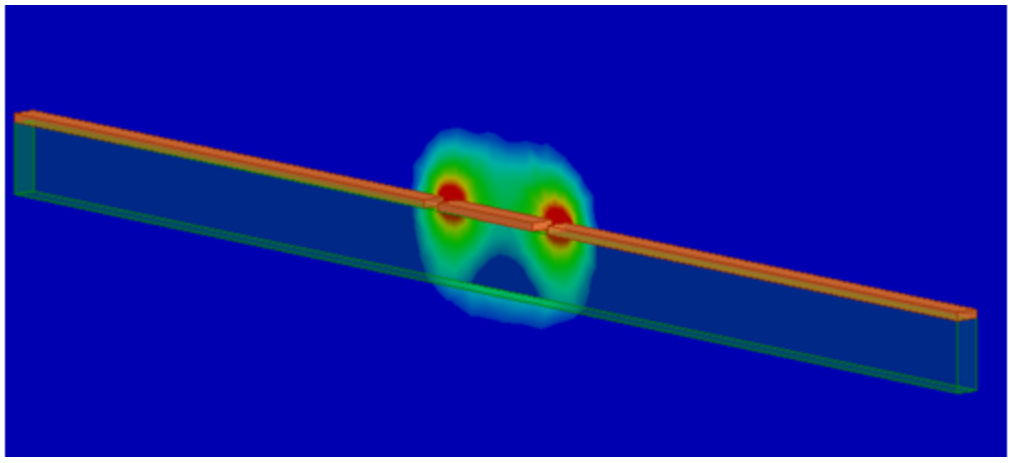


Figure 6 E field Plot

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

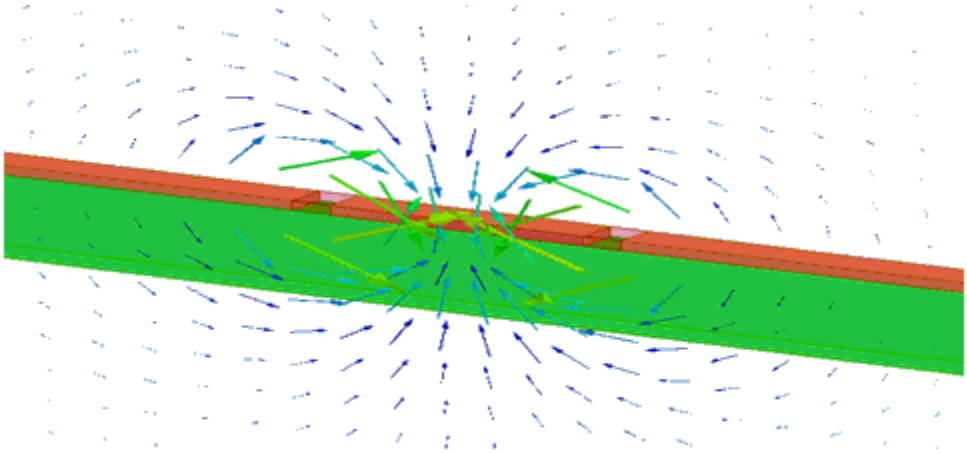


Figure 7: Vector Field

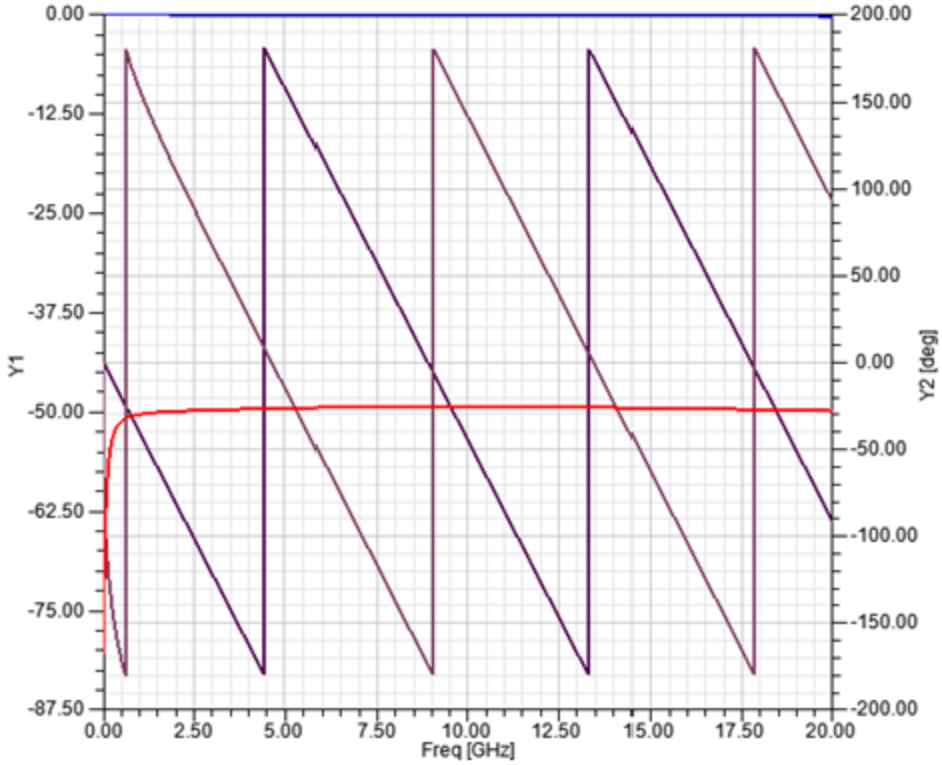


Figure 8 S-Parameter plot for the differential pair (legend shown below)

Curve Info	Y Axis
dB(St(D1,D1)) 5GHz : DC_20GHz	Y1
dB(St(D2,D1)) 5GHz : DC_20GHz	Y1
ang_deg(St(D1,D1)) 5GHz : DC_20GHz	Y2
ang_deg(St(D2,D1)) 5GHz : DC_20GHz	Y2

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

Coplanar Waveguide with Ground (Driven Terminal)

The coplanar waveguide (CPW) with ground plane is shown below.

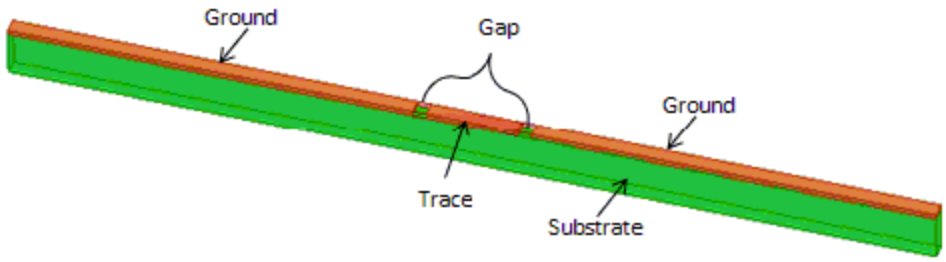


Figure 1: Coplanar Waveguide with Ground

When using HFSS to model a transmission line it is not necessary to model the entire length explicitly. Instead create a short section and to generate the desired length for the model you only need to de-embed the appropriate distance from the wave port. For this model the explicitly solved length was chosen as a dimension typical of the cross-section such as spacing between conductors or a conductor thickness. This is done to keep the 3D mesh as close to a high quality as possible. So for a model of explicit length X and a desired simulation length Y one can de-embed from each port (Y-X)/2.

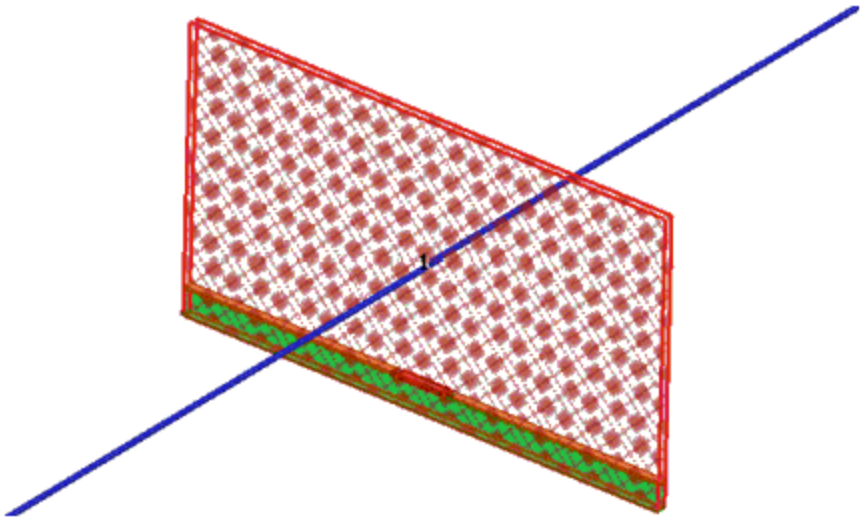


Figure 2: Ports with Deembedding

Deembedding greatly reduces the simulation effort.

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

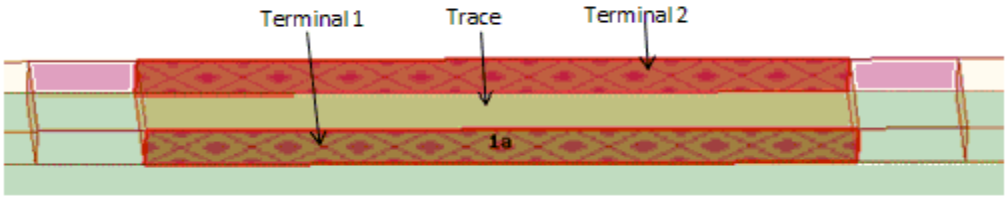
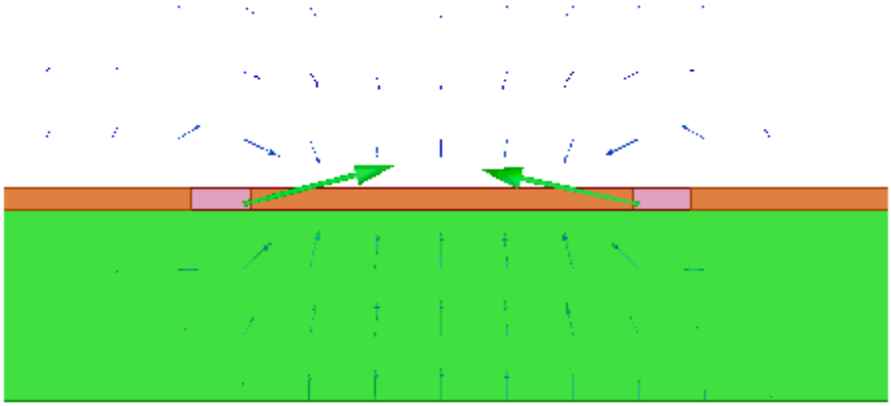


Figure 3: Terminals

The resulting E field and vector fields are shown below.

Figure 4: E Field



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

Figure 5: Vector Field

The S-parameter plot is shown below.

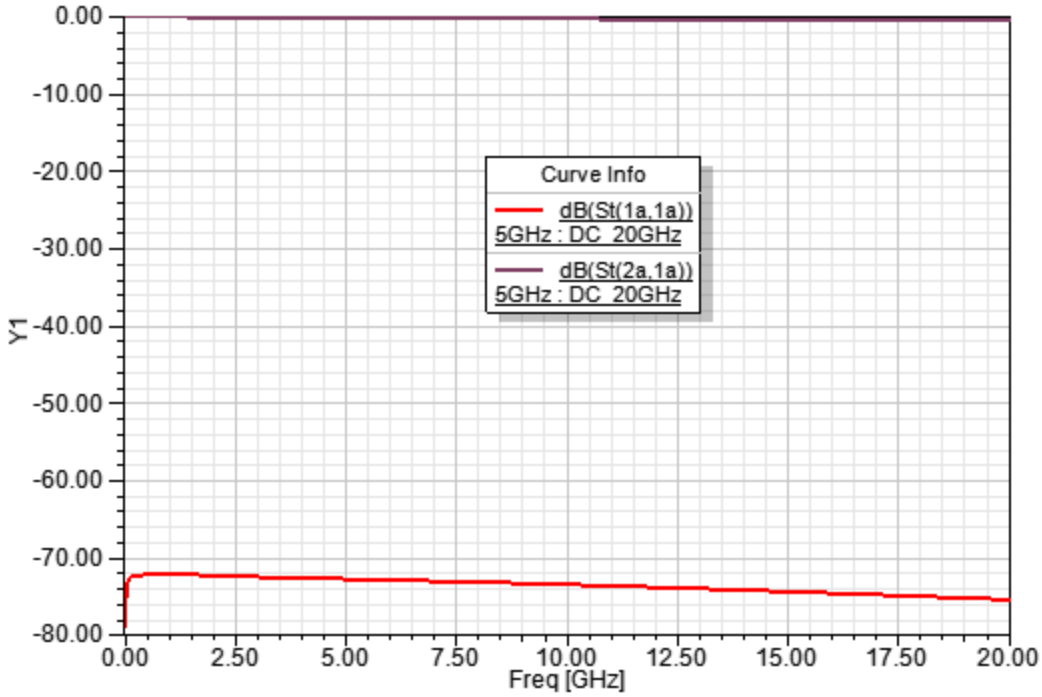
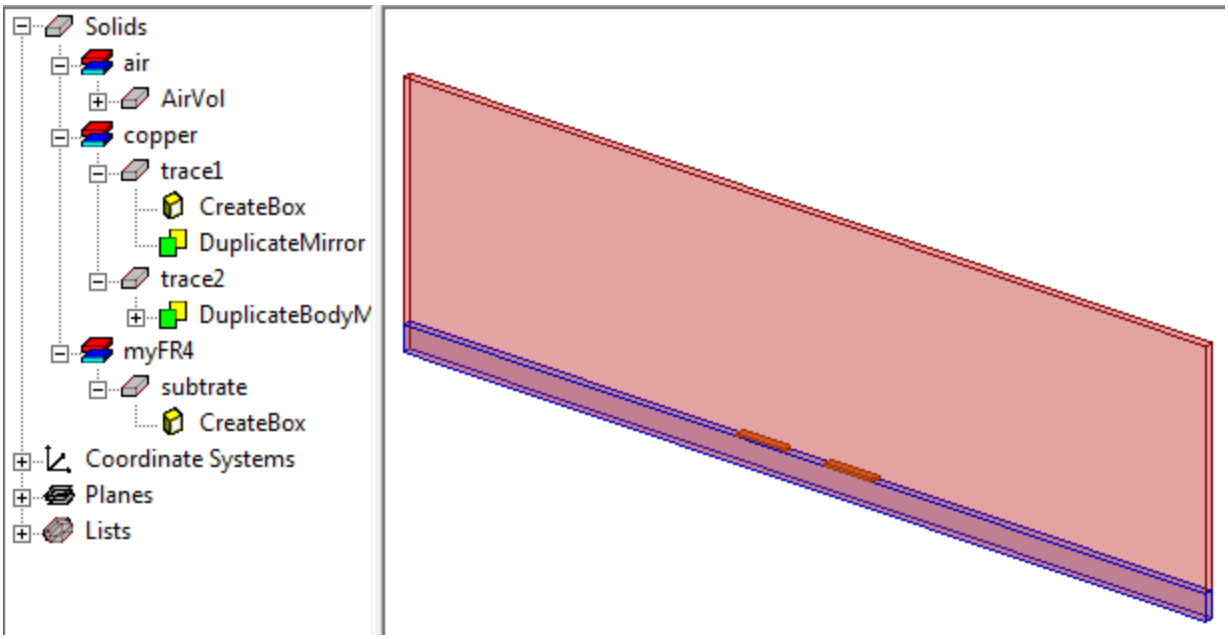


Figure 6: S-parameter Plot

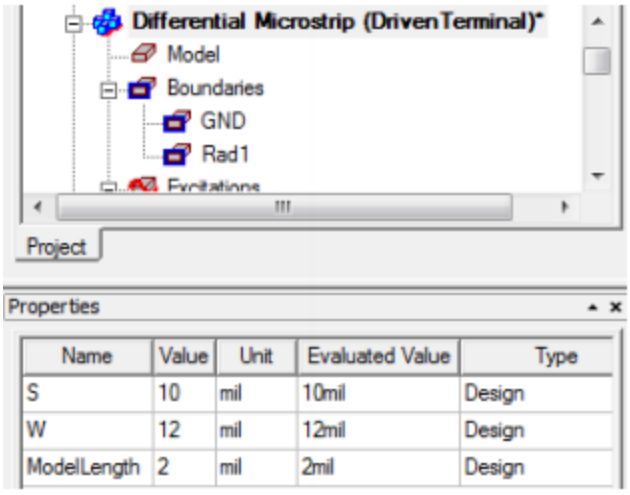
Differential Pair Microstrip

The differential microstrip design comprises two copper traces and an FR4 substrate enclosed in an air-box. The design in the following figure describes an efficient way to model a long transmission line without explicitly drawing the desired length of the microstrip model. HFSS offers a post processing feature called *Deembedding* that can be used to calculate the transmission line characteristics by moving the reference plane of the wave port by the specified deembedding distance.

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



The dimensions of the copper traces, air-box, and substrate are defined by using variables. If you click the design name on the **Project Manager** window, the **Properties** window displays all the design variables.

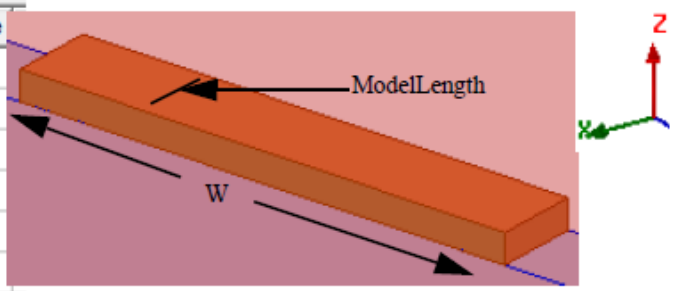


Name	Value	Unit	Evaluated Value	Type
S	10	mil	10mil	Design
W	12	mil	12mil	Design
ModelLength	2	mil	2mil	Design

Double-click **CreateBox** under **trace1** on the history tree to see the dimensions of the copper trace.

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

Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate Sys...	Global		
Position	0mil ,S/2 ,0		0mil , 5mil , 0
XSize	ModelLength		2mil
YSize	W		12mil
ZSize	0.7	mil	0.7mil



In this model, trace2 was created from trace1 by right-clicking **trace1** and selecting **Edit > Duplicate > Mirror.** In the coordinate text boxes of the status bar, starting co-ordinates for the base position was set to 0, 0, 0 and the normal position was set to 0,-1,0.

Name	Value
Command	DuplicateMirror
Coordinate Sys...	Global
Base Position	0 ,0 ,0
Normal Position	0mil ,-1 ,0mil

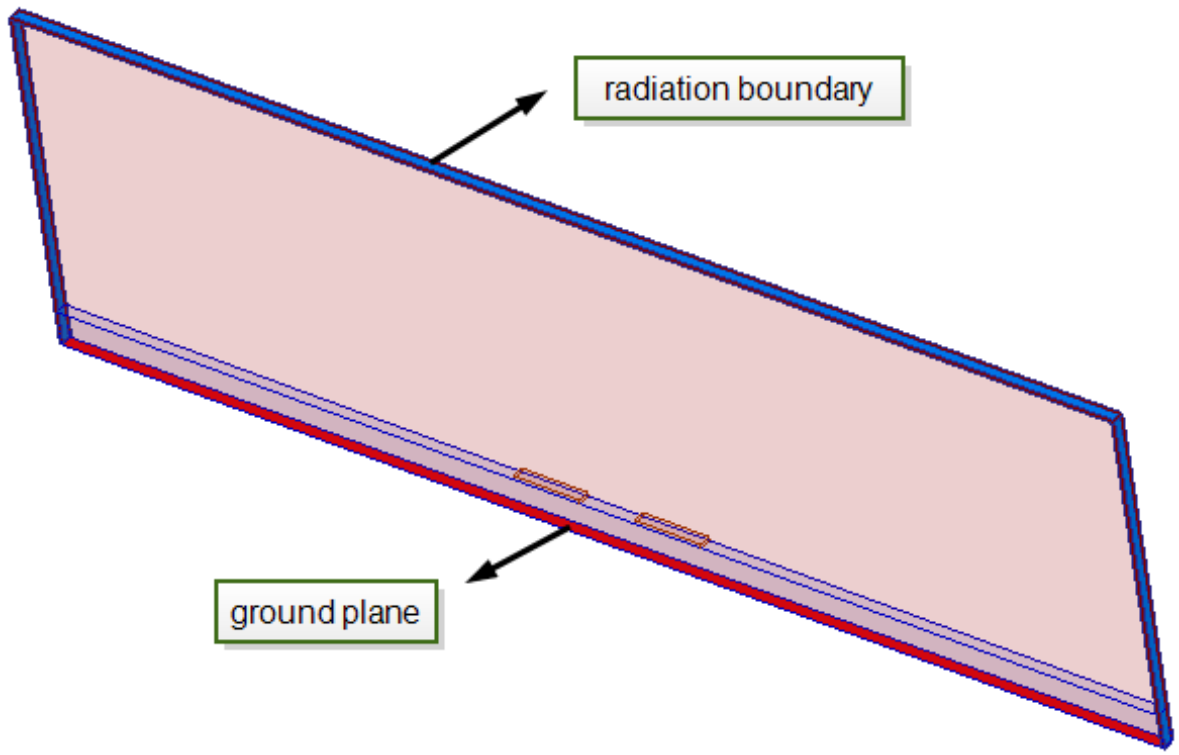
By parameterizing with common variables and taking advantage of the model tree you can create efficient designs in HFSS. Changing the value of one or more common variables ensures individual objects to track with the geometry of the entire model appropriately.

Boundaries and Excitations

On the **Project Manager** window, click **GND** and **Rad1** to see the ground assigned on the bottom face of this design, and the radiation boundary assigned on the top face and along the ZX faces as shown in the following figure.

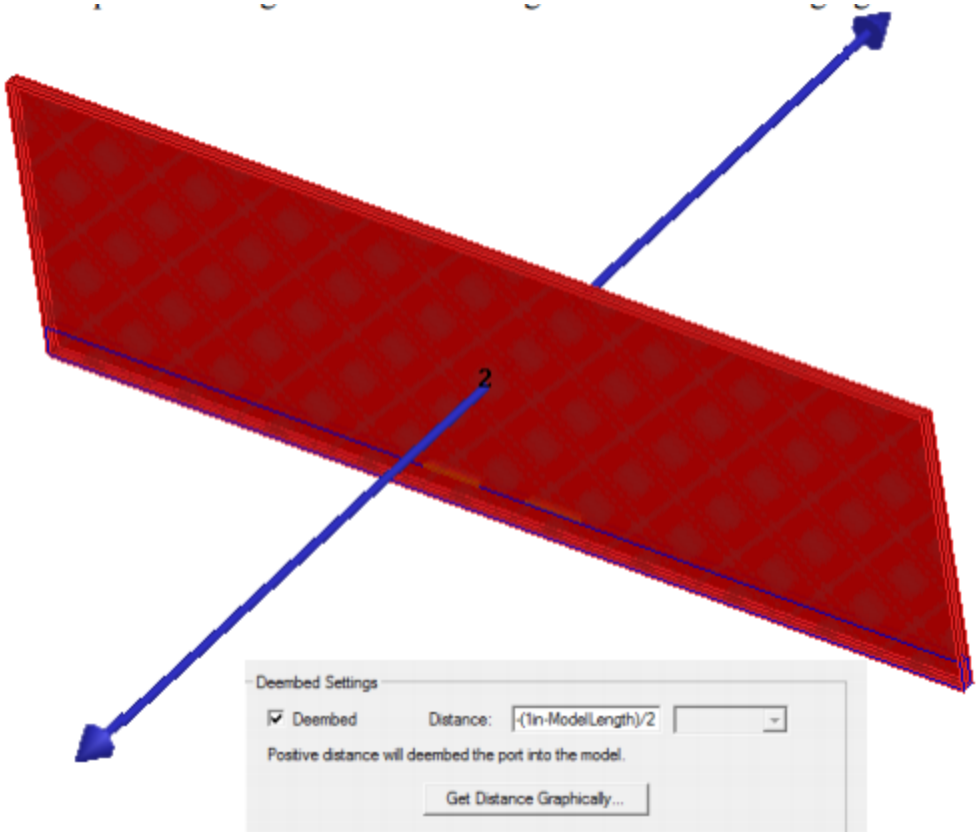
> **Note:** You can also select the **Boundary Display (Solver View)** option from the **HFSS** menu item and select the boundaries, ports, and terminals that you want to see in the design. In the following figure, the ground and the radiation boundary appear highlighted since they are selected on the **Boundary Display (Solver View)** dialog box.

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



Observe how the two ports are assigned in the remaining faces in the following figure.

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



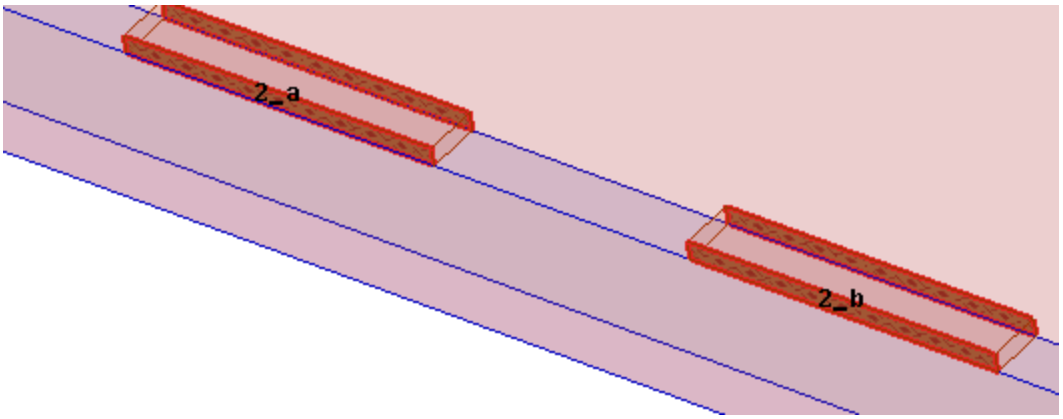
> **Note:**
>
> For guidelines on defining port size, see the section **Assign Wave Ports for Terminal Solutions** in the help.

The ports are deembedded with a negative distance outwards from the structure. The intent is to solve the model of this minimal length and then deembed outwards from the ports using a negative sign for the deembed distance to effectively add the extra length that you want to represent the actual length of the model.

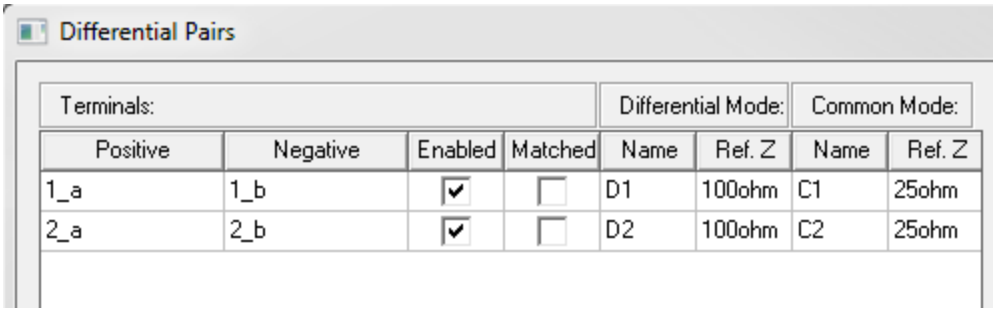
> **Note:**
>
> For more information about modeling long transmission lines, see the **Applications for Deembedding** section in the **Assign Excitation** material in the help.

There are 4 terminals assigned as shown in the following figure.

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



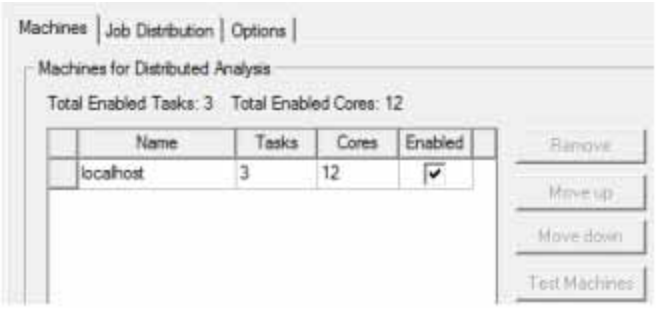
Right-click **Excitations** and select **Differential Pairs** to access the **Differential Pairs** dialog box.



HPC Analysis and Solution Setups

Run this design at an adapt frequency of 20 GHz. Since a parametric sweep (look for **SpacingSweep** under **Optimetrics** on the **Project Manager** window) is defined, this design is a good choice for which HPC can be set up. On the **Solution Setup** dialog box, click the button to open the **HPC and Analysis Options** window. Click **Add** and set the number of tasks and cores.

For example in HPC setup in the following figure, the design was simulated on a machine with 12 cores on it and the Number of Tasks is 3. In such a setup, 3 frequency points are solved in parallel with 4 cores of matrix multiprocessing per frequency point.



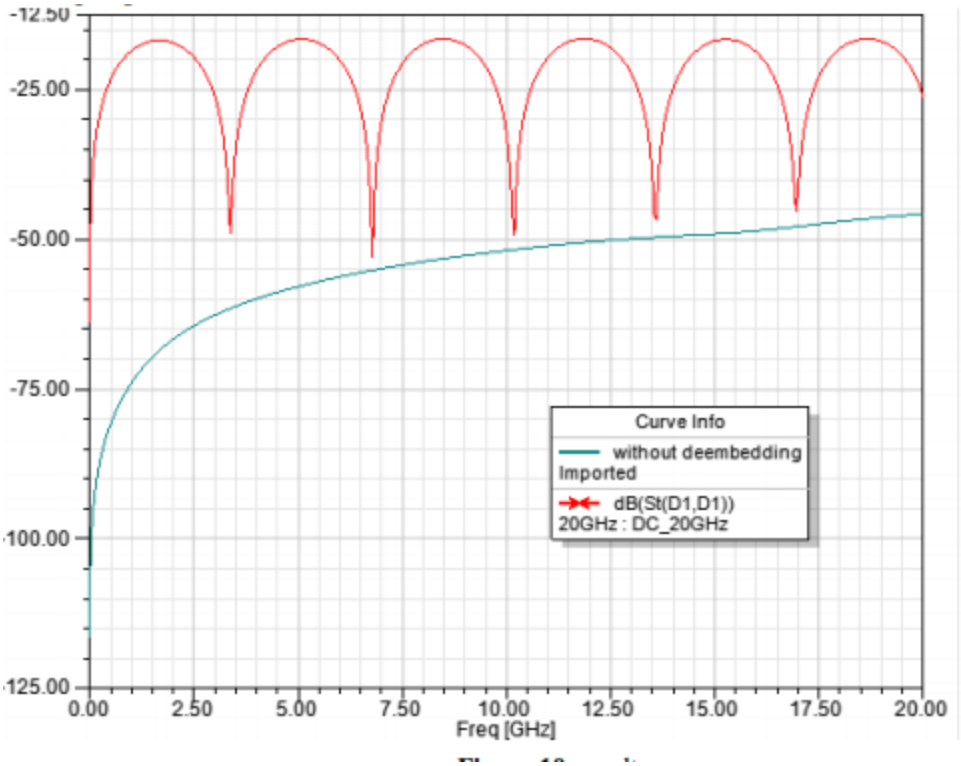
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> For more information about HPC, see **HPC and Analysis Configuration Options** section in the help.

While deembedding simplifies modeling long differential striplines and makes the solution process efficient, the HPC setup further accelerates the simulation process.

Results

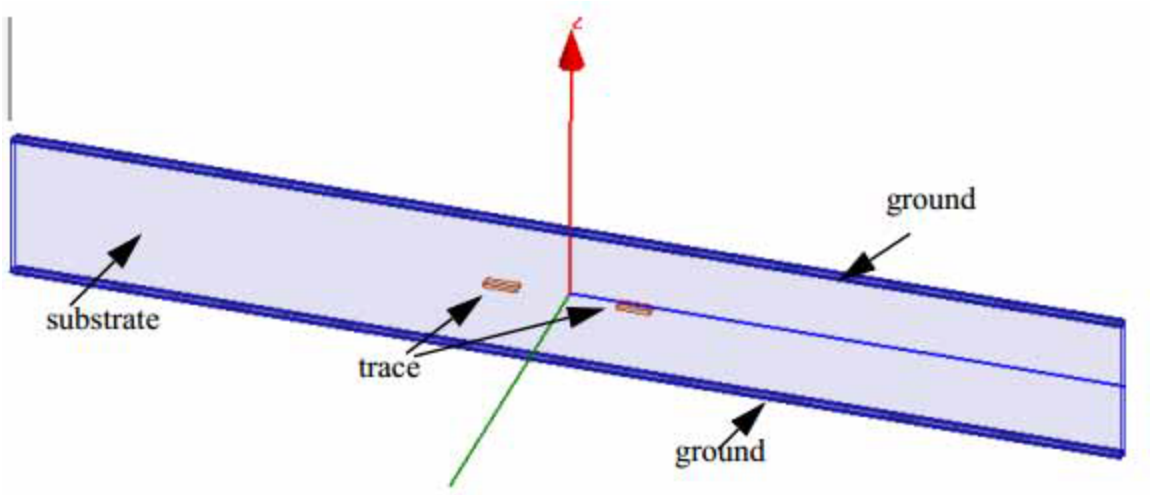
The results of the S-parameter plots with and without deembedding are shown below. The deembedding operation adds the effect of phase delay and additional dielectric and conduction losses to the resulting S-parameter calculated from this model.



Differential Pair Stripline

The following figure shows a differential pair stripline design, where two copper traces are embedded in the substrate, which in turn is sandwiched between two ground conductors. Select **GNDs** on the project tree of the **Project Manager** window to highlight the ground planes assigned on the top and bottom faces of the stripline. These top and bottom ground planes are equipotential surfaces.

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

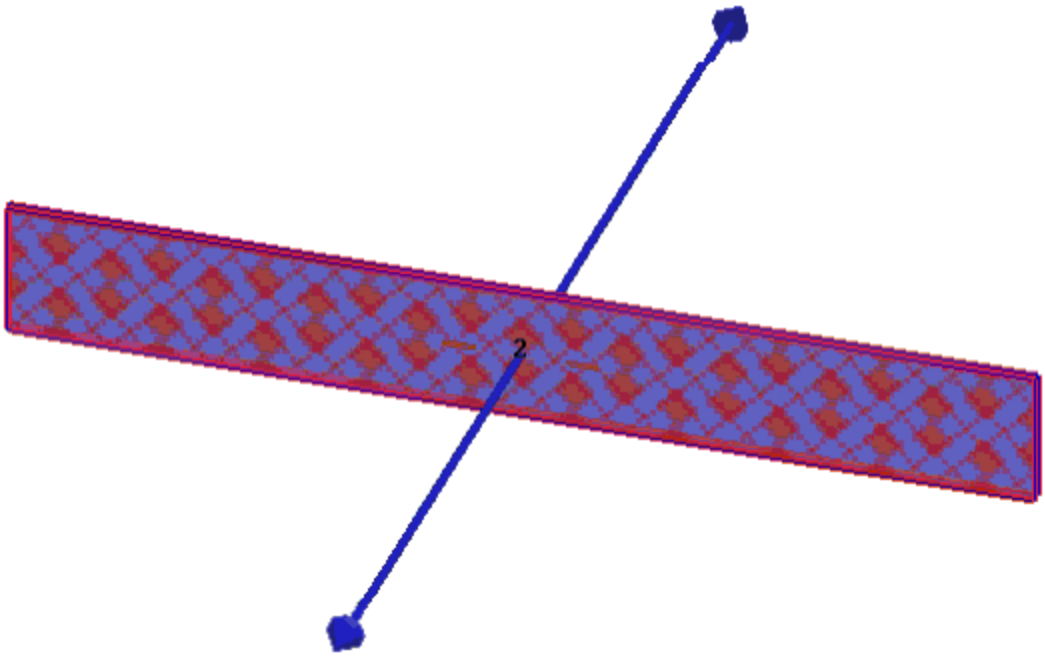


The intent of this design is to draw only minimal lengths of the differential traces containing the two conductors adjacent to each other (and the equipotential ground conductors) without explicitly drawing the actual length throughout the entire trace route. By using a post processing feature of *Deembedding*, transmission line characteristics can be calculated by moving the reference plane of the wave port to desired locations along the trace route, depending upon the specified value of the deembed distance. Such a design is an effective approach to simulate the actual model length. It saves simulation time and uses minimal computational resources.

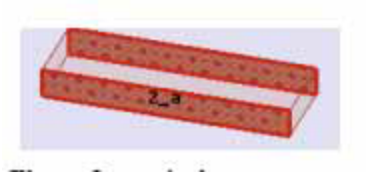
Excitations

To see both excitations at the same time, right-click the **Excitations** option on the **Project Manager** window. Select **List** from the shortcut menu, and click the two wave ports listed in the **Design List** dialog box. The wave ports assigned in the model with the deembedded lengths appear highlighted in the design as shown in the following figure.

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



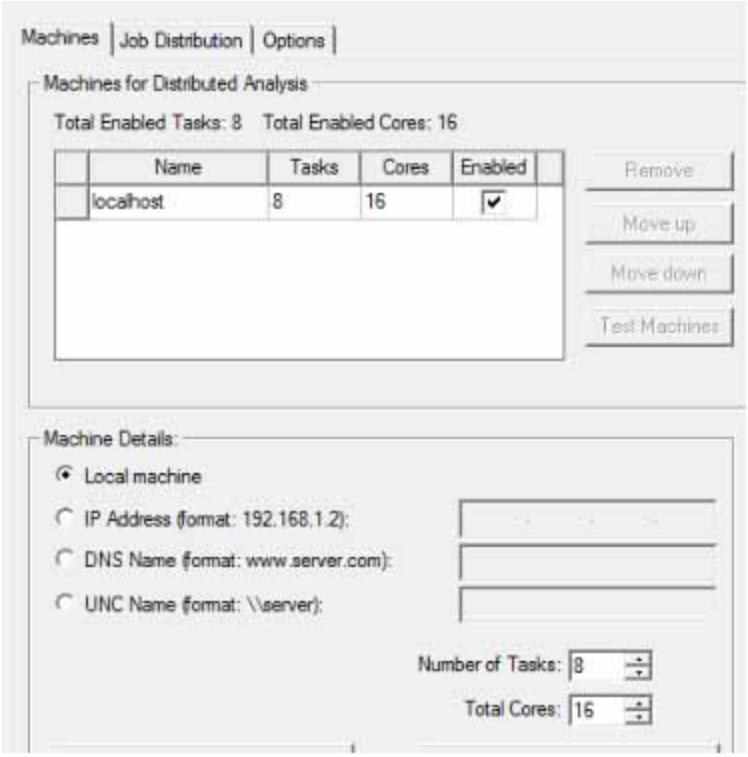
The deembed arrows point outwards from the structure since negative deembedding value of -(1in - ModelLength)/2 was specified for each wave port. The Deembed Distance value is set on the **Post Processing** tab of a wave port dialog box. The main purpose of such a design is to solve the model of minimal length and then, by deembedding outwards from the ports, to represent the actual length of the model. The terminals on a trace are shown in the following figure.



Solution Setup and HPC Analysis

Run this design at a solution frequency of 20 GHz. Since the design has a parametric sweep (with the trace edge-to-edge spacing defined by the variable S), it is a good choice for setting up HPC analysis. From the **Solution Setup** dialog box, click the button to open the **HPC and Analysis** window. Click **Add** to open the **Analysis and Configuration** window, where you can set the number of available cores to use for this design.

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



For example, in the figure above, 16 cores are available on the machine in which the design was simulated and number of tasks is 8. In such a setup, the sweep is run with 8 frequency points solved in parallel by using two cores of matrix multiprocessing for each frequency point. When such an analysis is executed on a single machine, the simulation is very efficient if the machine has enough shared memory to accommodate 8 simultaneous solves. Otherwise the analysis can be performed across multiple machines (that have HFSS installed in them) without requiring any additional HFSS license for each machine

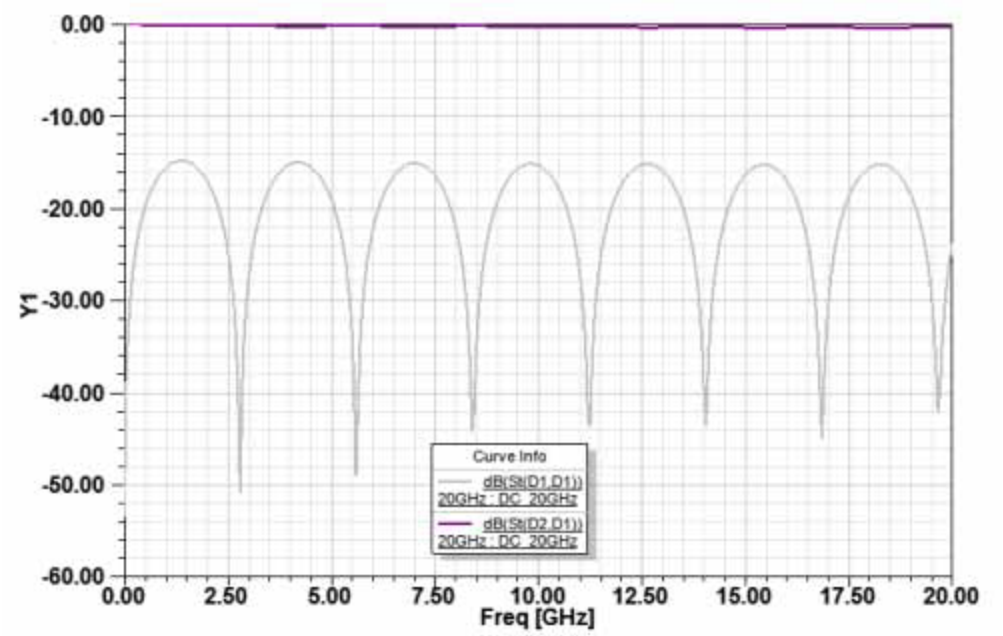
> **Note:**
>
> For more information about HPC, see **HPC and Analysis Configuration Options** section in the help.

While deembedding simplifies modeling long differential striplines and makes the solution process efficient, the HPC setup further accelerates the simulation process.

Results

The following figure shows the S-parameter plots for the stripline.

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



Stripline (Driven Terminal)

The following figure shows the HFSS model of the GSSGSSG Stripline.

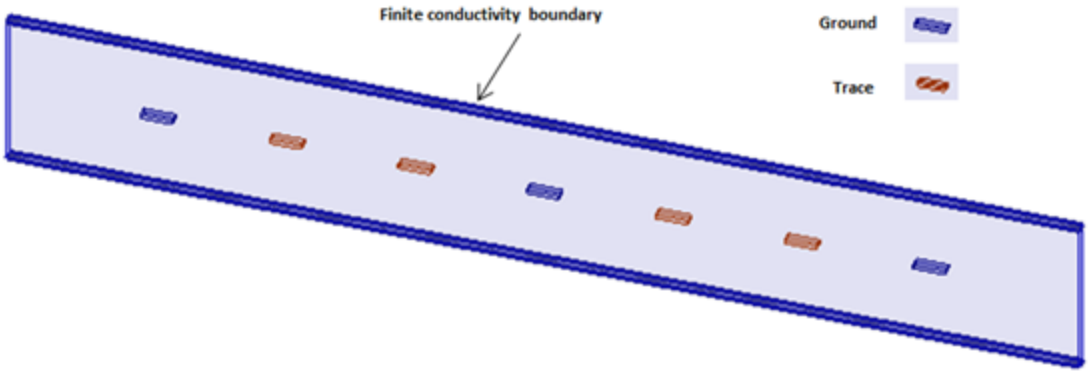


Figure 1-1 GSSGSSG Stripline

The figure illustrates a set of seven striplines composing a GSSGSSG dual differential pair configuration. To reduce the coupling between the differential pairs there are 3 ground conductors. Notice how the terminals are defined in the model shown below.

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

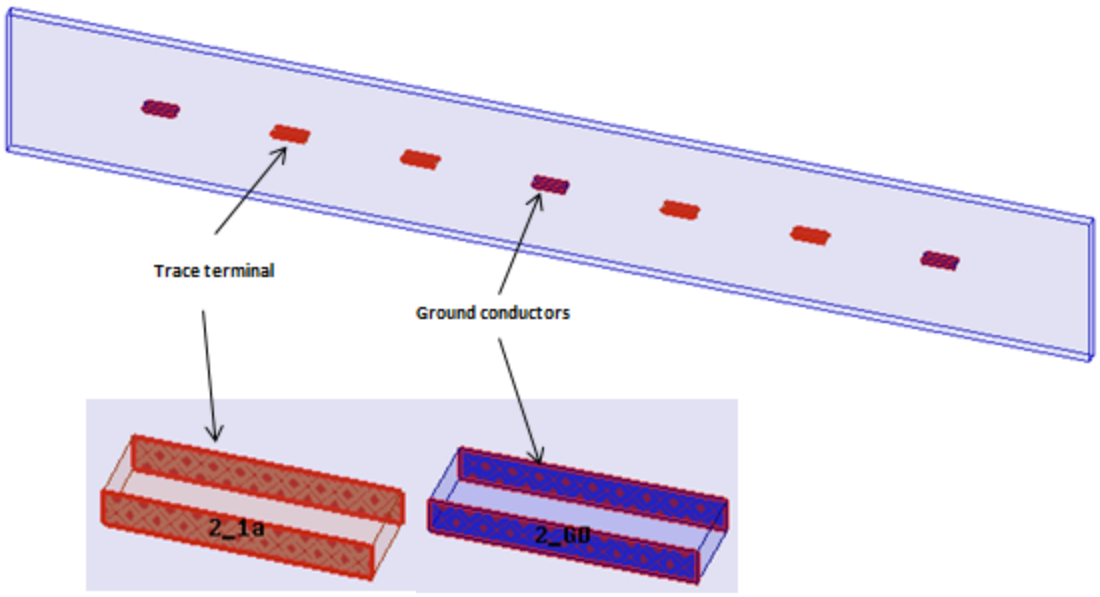


Figure 1-2 Terminals

The grounding is accomplished when you renormalize the impedance of the ground conductors to a very small value like 5e-6 (6 orders of magnitude below the other terminals set to 50 ohms). Notice the contrast in the values of the terminal renormalizing impedance of a trace and a ground conductor in the dialogs below.

Figure 1-3 Terminals Dialogs

To model a longer length, just deembed the ports as shown below.

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

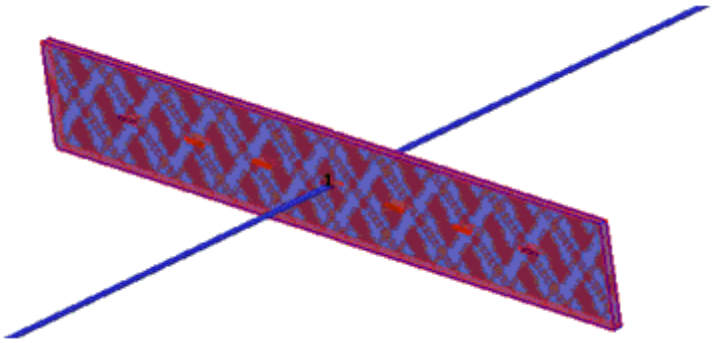


Figure 1-4 Ports with Deembedding

In the Post Processing tab of the Wave Port dialog box the deembedding distance = + (Simulation Length - Model Length)/2 where Simulation Length and Model Length are the design variables.

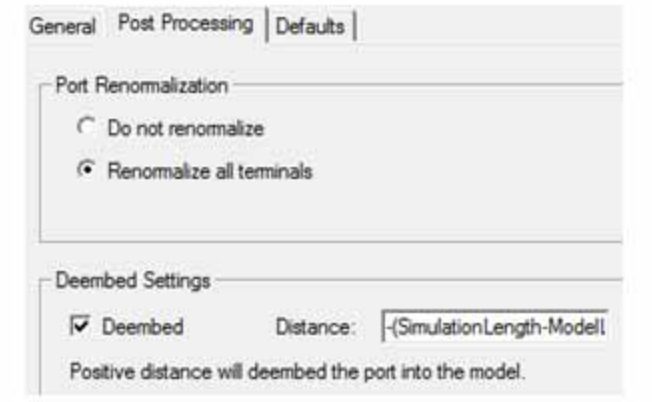


Figure 1-5 Post Processing Deembed Setting

The mesh plot is shown below.

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

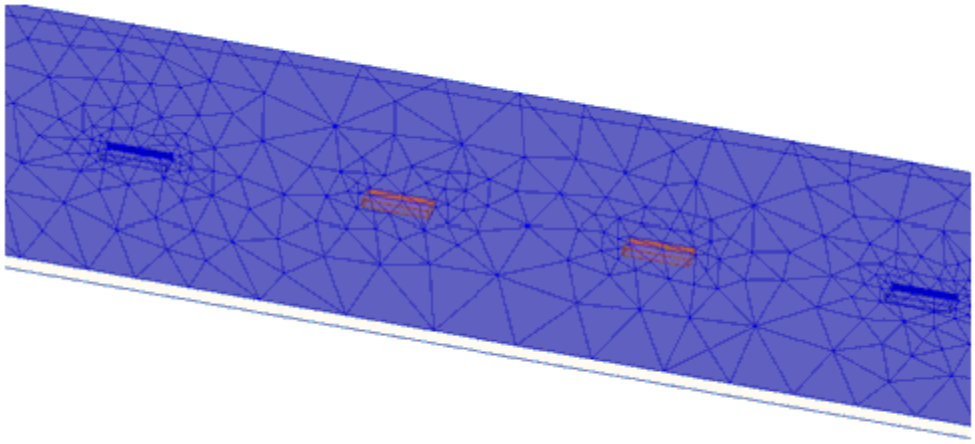
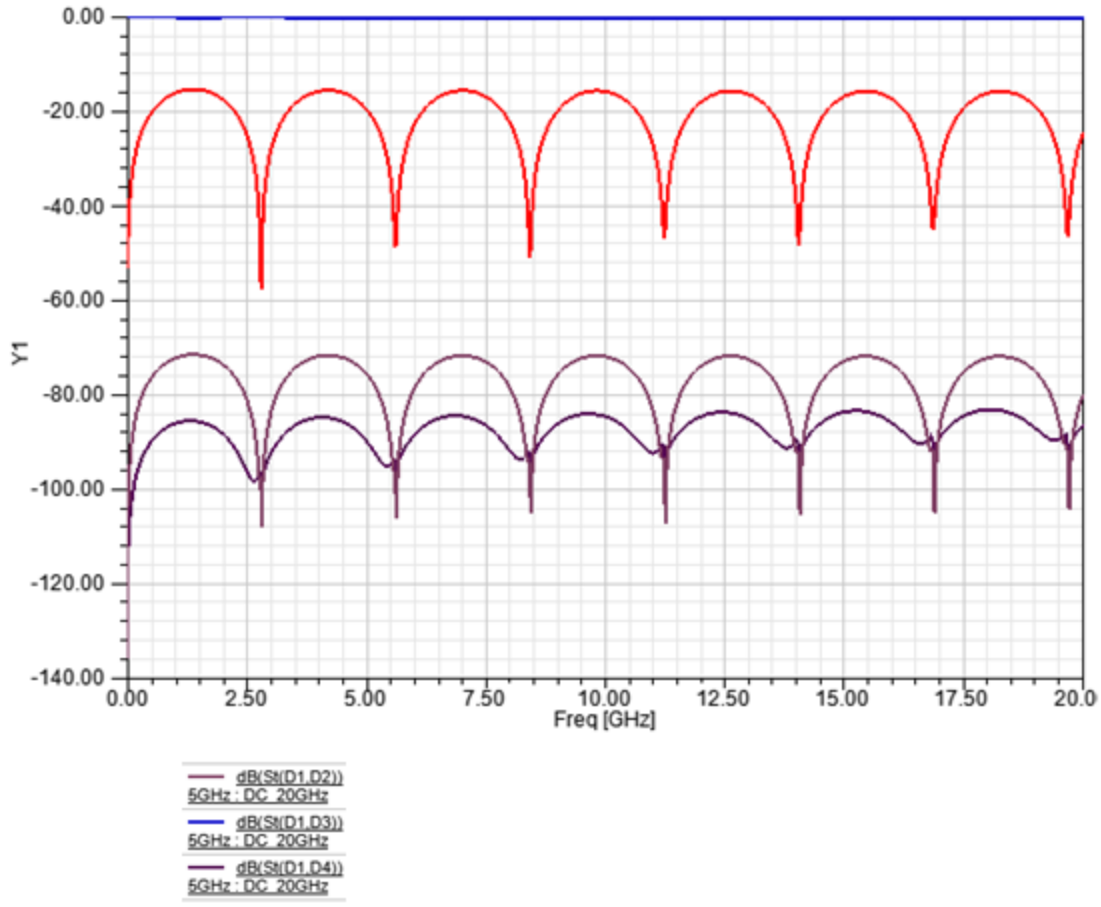


Figure 1-6 Stripline Mesh Plot

The results for the S-parameter plot are shown below.

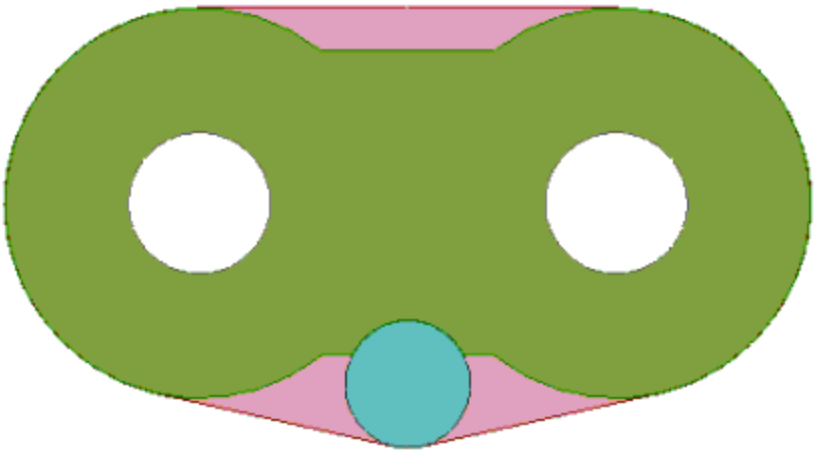


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1-7 Stripline S-Parameter Plot

Twinaxial Cable

Twinaxial cables are used for in-rack connections between supercomputers to carry the networking traffic. They are meant for the transmission of short-range signals. The following figure illustrates a twinaxial cable (28 wire gauge) design, which comprises two inner conductors (S1 and S2) made of silver, and a shield or drain made of aluminum enclosed in an air-box. The central conductor is insulated with a dielectric layer.

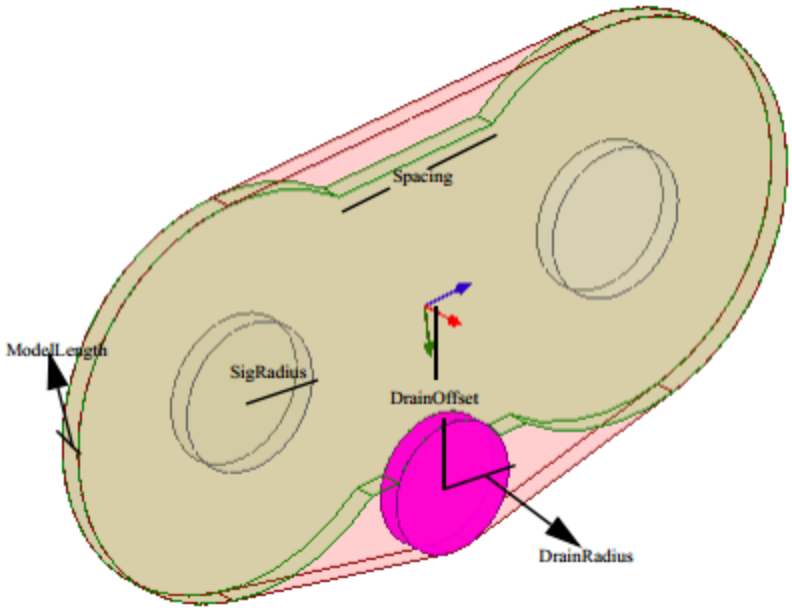


twinax cable design

The main intent is to draw a minimal length of the twinaxial cable and stretch it to any required length by using the post processing feature of *Deembedding*. Such a design saves simulation time and makes minimal use of computational resources and ensures efficient simulation, without explicitly modeling the actual length of the cable.

The design is parameterized as shown. The geometry is drawn using the parameters and boolean operations in such a way that all the individual objects that make up the geometry track with it. For example, when you change the values of the variables appearing in the **Properties** window, the model resizes accordingly and the objects with changed parameters are track with the geometry appropriately.

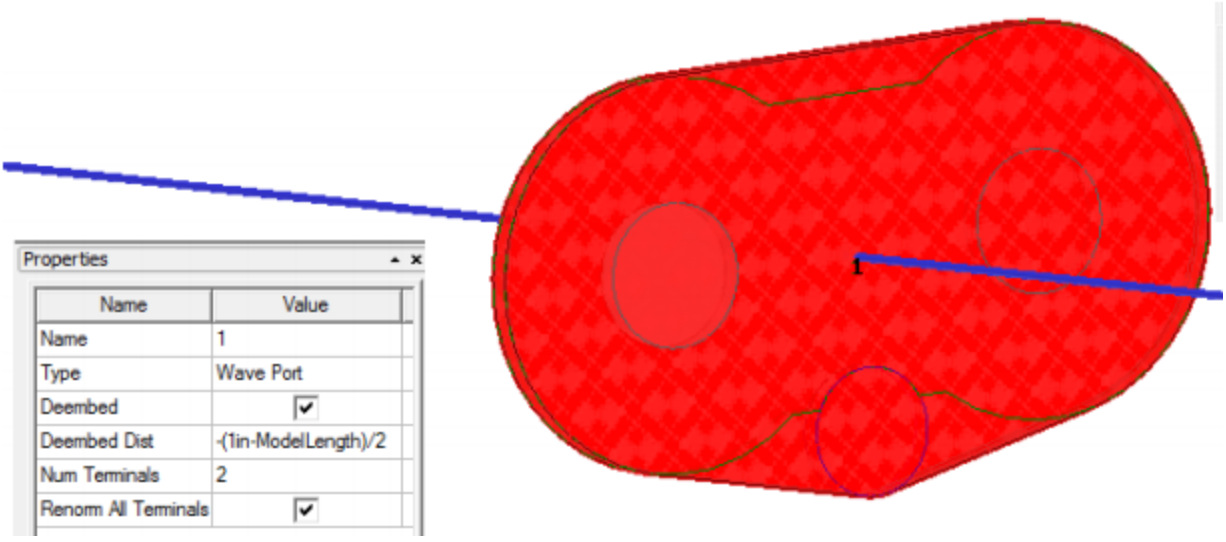
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



some of the parameters used in the design

Excitations

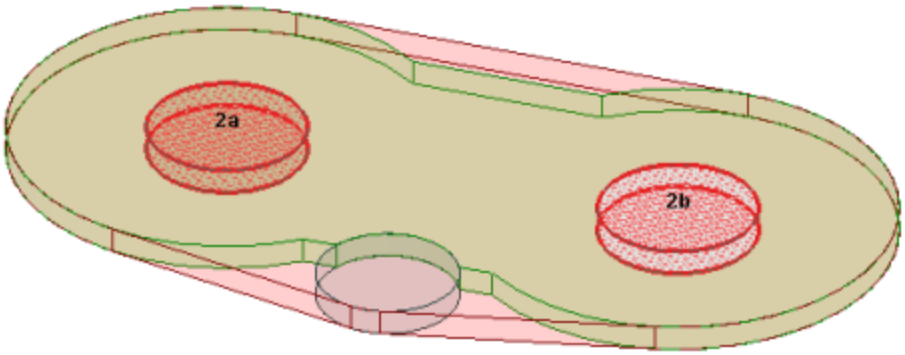
Right-click **Excitations** on the project tree and select **List** to open the **Design List** dialog box where you can select the terminals and wave ports that are assigned on this design. In the following figure, the two wave ports with their deembedding distances are highlighted.



wave ports and deembedding

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

Negative values of deembedding indicates that the wave ports are deembedded away from the structure to stretch them to the required lengths. The transmission line characteristics are calculated along the shifted reference plane due to the deembedding. Deembedding prevents explicit drawing of the entire cable lengths. The following figure shows the 4 terminals assigned in this design.

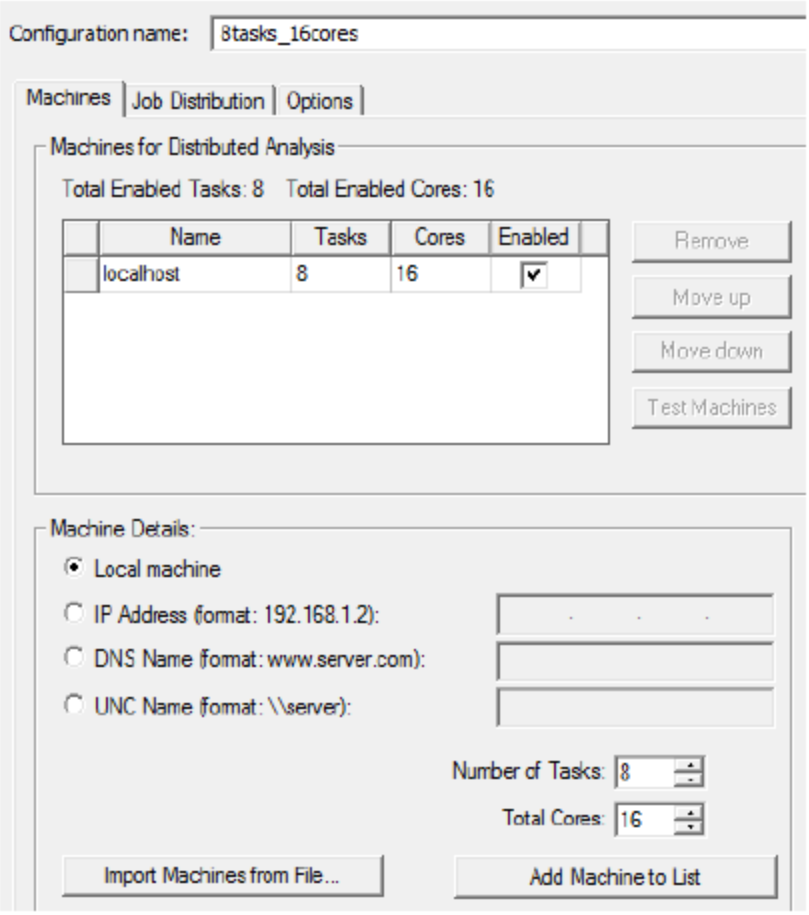


Twinaxial Cable terminals

HPC and Solution Setups

Run the design at an adapt frequency of 20 GHz. Since a parametric sweep is used (look under **Optimetrics**), this design is a good choice to employ HPC. You can access HPC settings from **Tools > Options > HPC and Analysis Options**. Click the **Add** button to open the **Analysis and Configuration** window where you can set the number of cores and tasks for the HPC simulation.

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



HPC analysis and configuration options

For example in figure above, 16 cores are available on the machine in which the design was simulated. In a set up of 8 tasks executed with these 16 cores, the sweep is run with 8 frequency points being solved in parallel using two cores of matrix multiprocessing for each frequency point. When such an analysis is executed on a single machine, the simulation is very efficient if the machine has enough shared memory to accommodate 8 simultaneous solves. Otherwise the analysis could be performed across multiple machines (that have HFSS installed in them) without requiring any additional HFSS license for each machine.

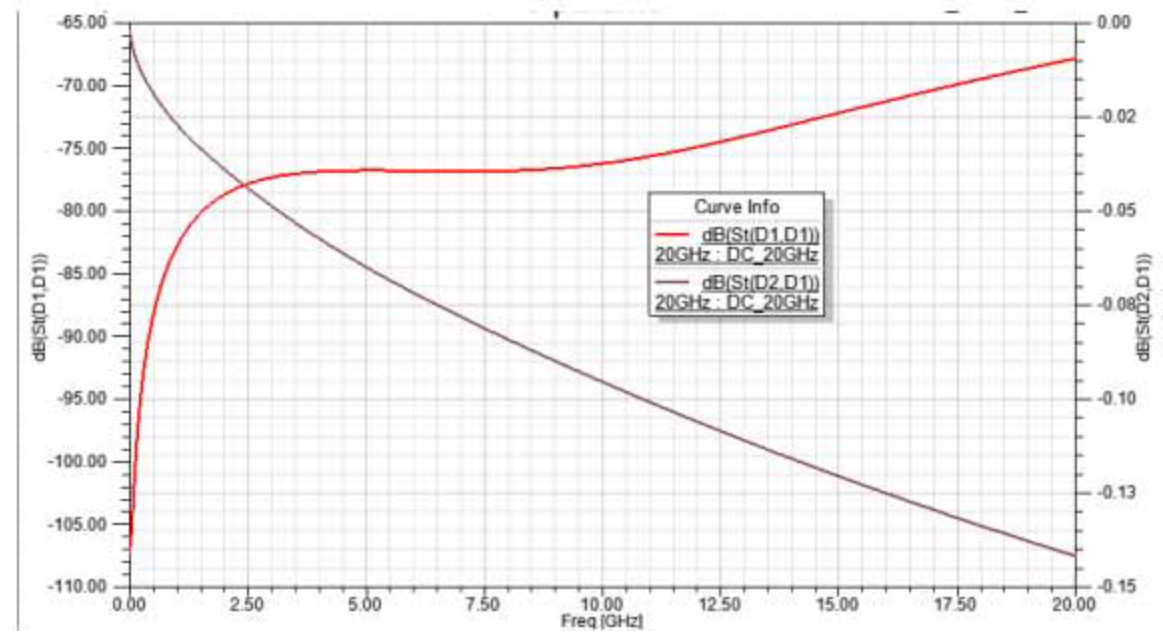
> **Note:**
>
> For more information about HPC, see the sections **Setting HPC and Analysis Options** and **Editing Distributed Machine Configurations** in the help.

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

The advantage of deembedding is that it saves the trouble of explicitly modeling the long cable lengths. Such a design is efficient and can be solved using minimum computational resources. By using HPC, the solution time is further reduced.

Results

The following figure shows the S-parameter plot.



s-parameter plot

Finding Information in the Online Help

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

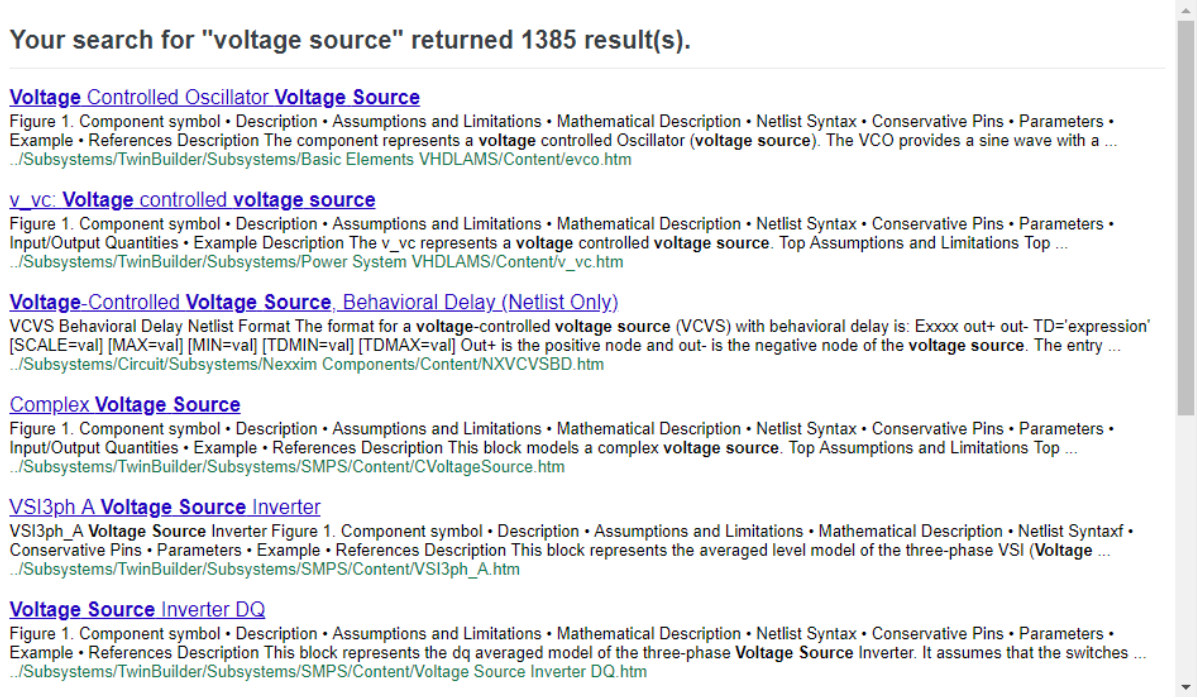
- Press **F1** on any open dialog box to open the relevant help topic.
- *A hierarchical table of contents* – Contents – Browse through the table of contents, expand entries, and close entries. Click an entry to see it in the content area.
- *A full text search* – To locate occurrences of a word or phrase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

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

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

Using the Search Function in the Help

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

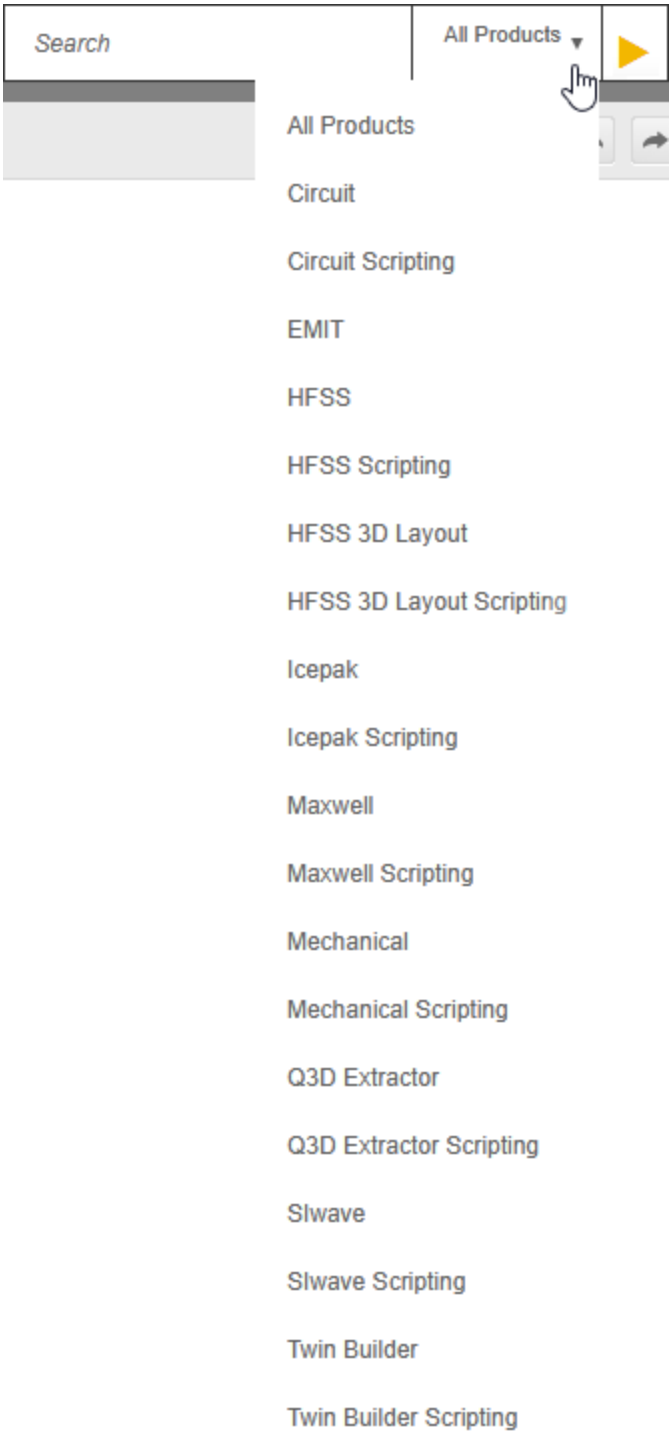


This method probably provides more hits than you want. The search function in the help provides several methods for making searches more specific.

Performing a Basic Search

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

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



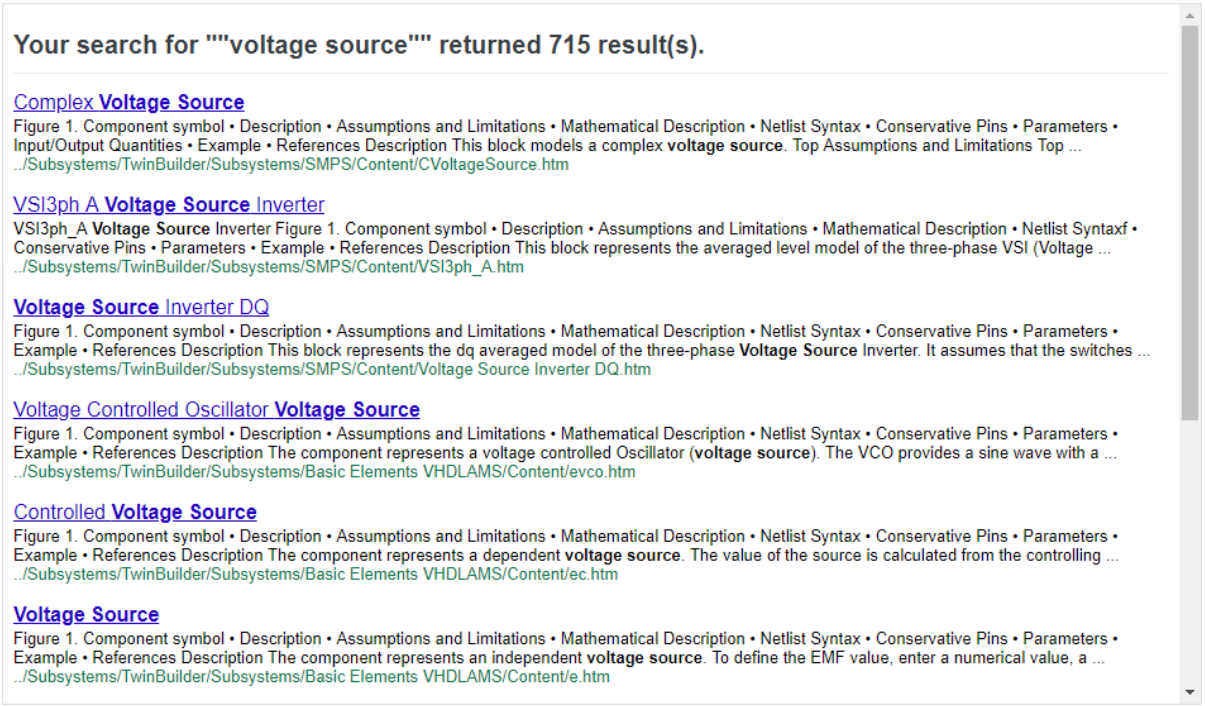
- If you accessed the help for a specific product by pressing **F1** in Electronics Desktop, the search box will have preselected that product. You can change the selection to a different product, a product's scripting guide, or all products.

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

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

Searching with Quotation Marks

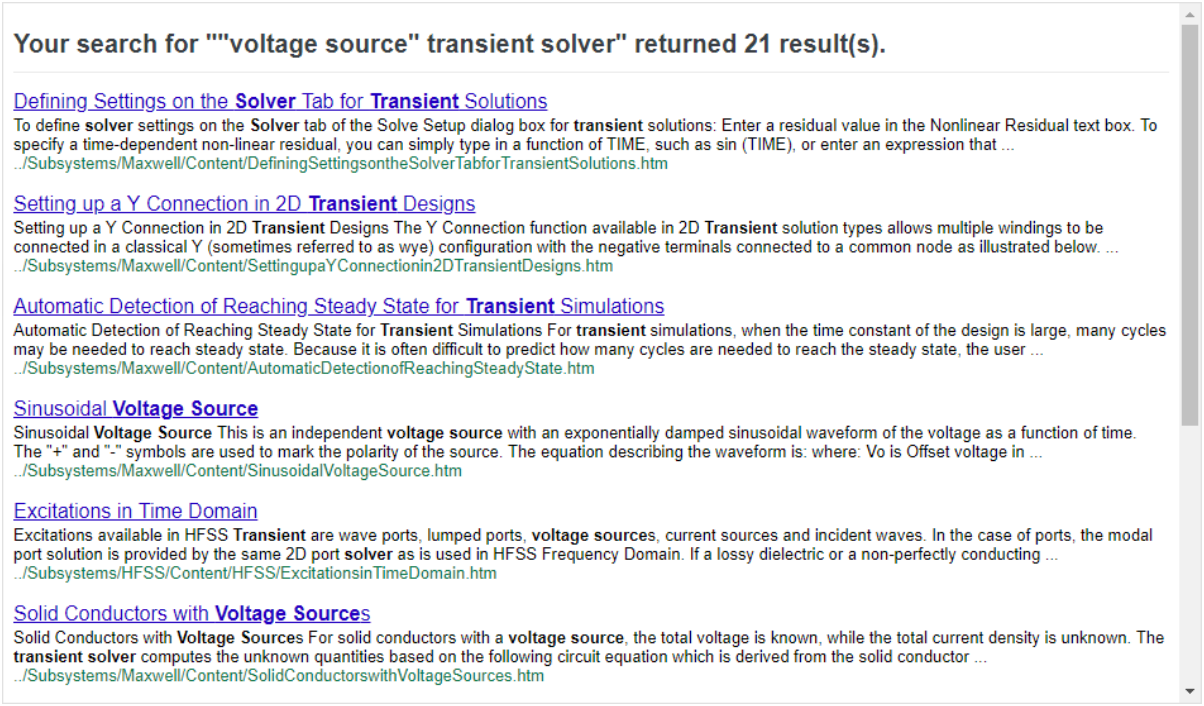
If you enter “voltage source” with quotation marks, the results show all topics that include the phrase.



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

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

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



Note:

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

Using Boolean Operators

Note:

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

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

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

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

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

Use parentheses to group terms and operators. For example:

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

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

Help Conventions

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

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

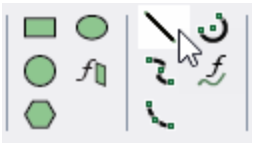
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

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

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

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

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

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

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

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

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

Getting Help: Ansys Technical Support

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

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

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

Help Menu

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

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

Context-Sensitive Help

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

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

Getting Help from Ansys Technical Support

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

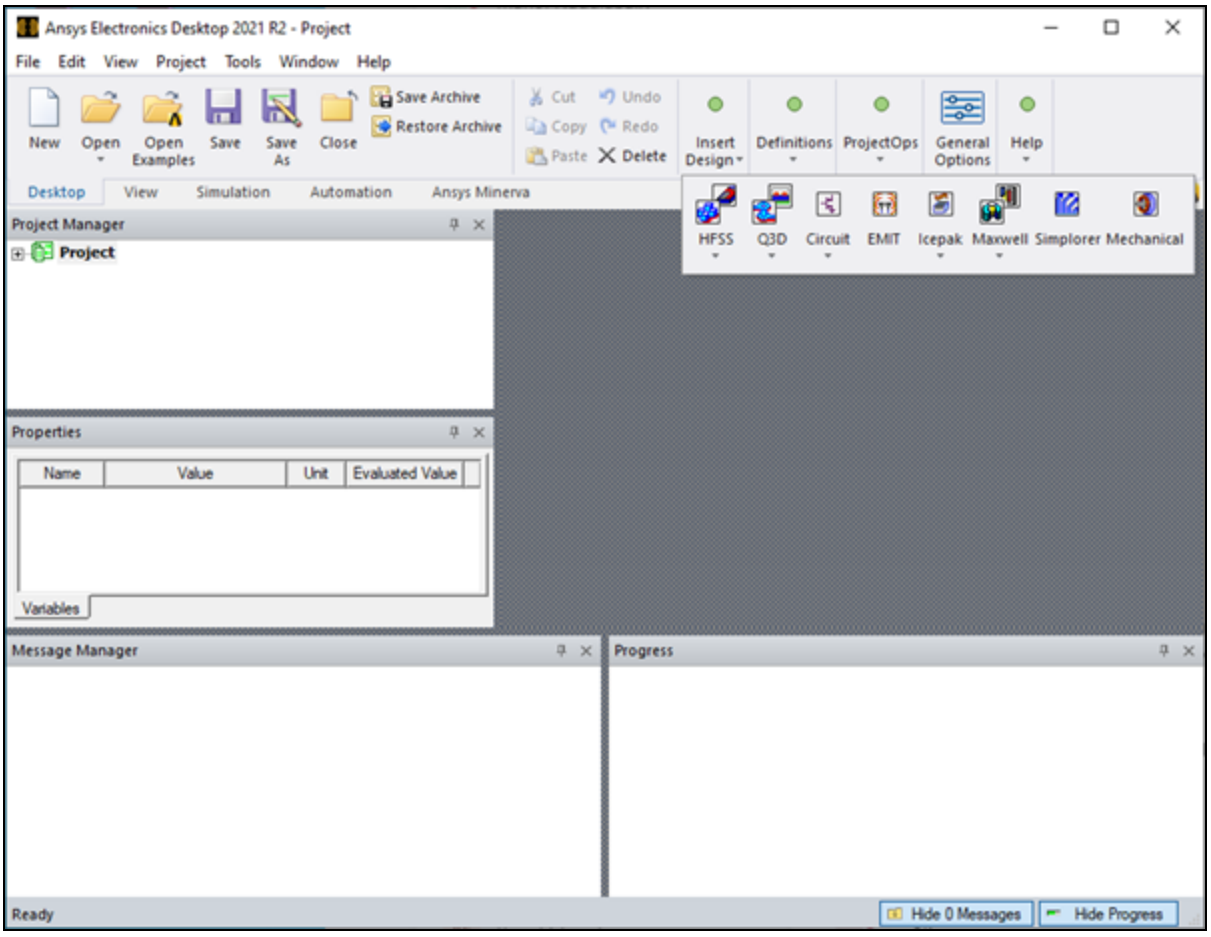
E-mail can work well for technical support. All Ansys EM software files are ASCII text and can be sent conveniently by email. When reporting difficulties, it is extremely helpful to include very specific information about what steps were taken or what stages the simulation reached. This allows more rapid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

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

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

2 - Getting Started with Ansys Electronics Desktop

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



The desktop includes the following design types:

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

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

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

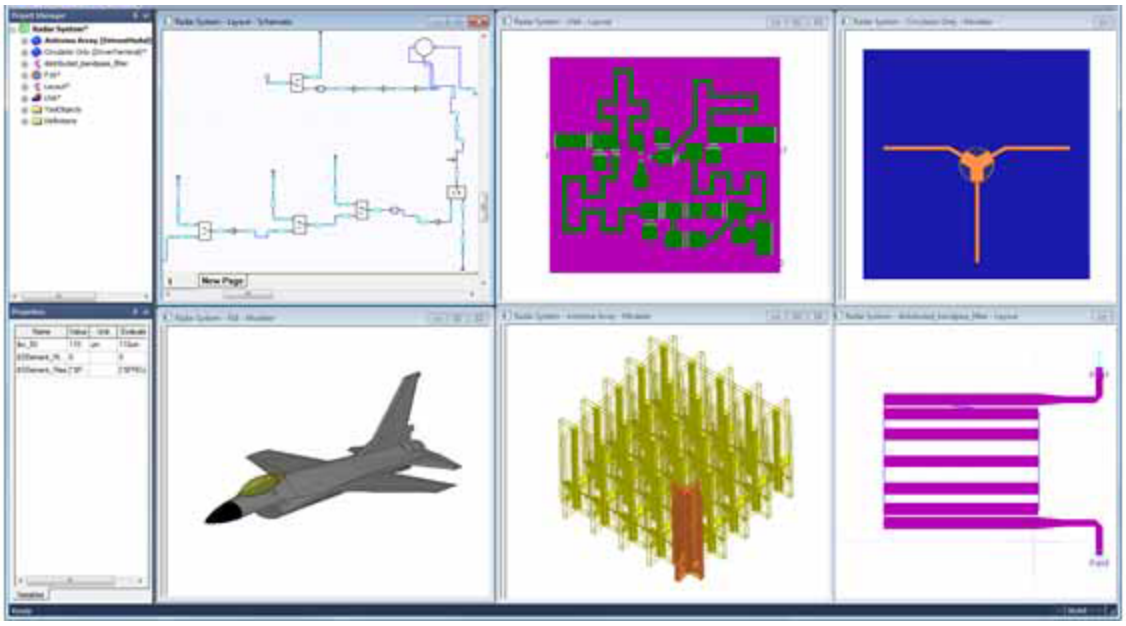
You can access all of these design types and features from the **Project** menu, and any combination of design types can be inserted into a single project file. The schematics can be used to wire up the different field solver models and create a model of a high-level system. Ansys Electronics Desktop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

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

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

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

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



Ansys Electronics Desktop Student

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

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

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

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

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

HFSS Student Limitations

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

Q3D Extractor and 2D Extractor Student Limitations

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

Circuit Limitations

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

Maxwell and RMxprt Student Limitations

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

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

Icepak Student Limitations

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

System Requirements

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

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

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

Limitations of Linux Installations:

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

Windows vs. Linux Installations

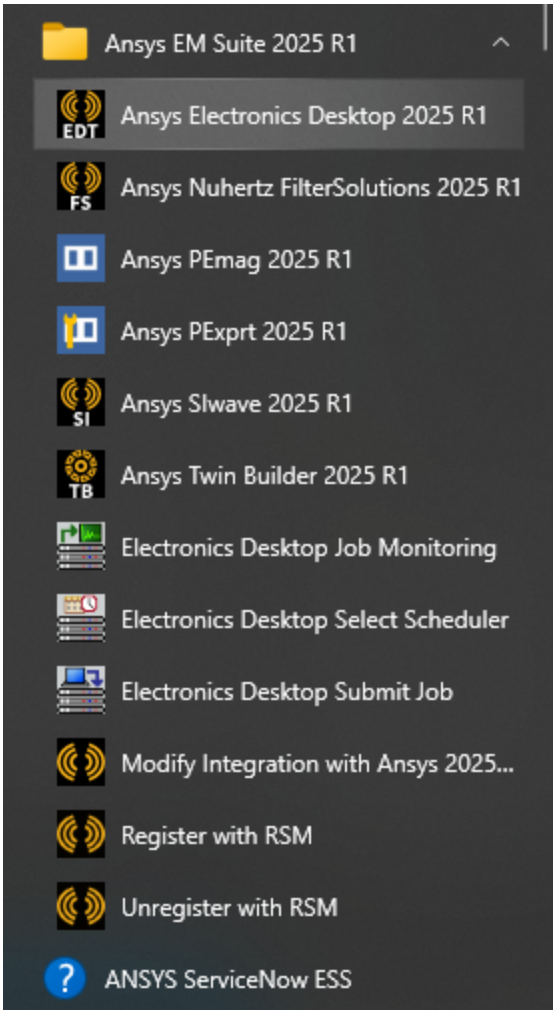
Launching Ansys Electronics Desktop

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

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

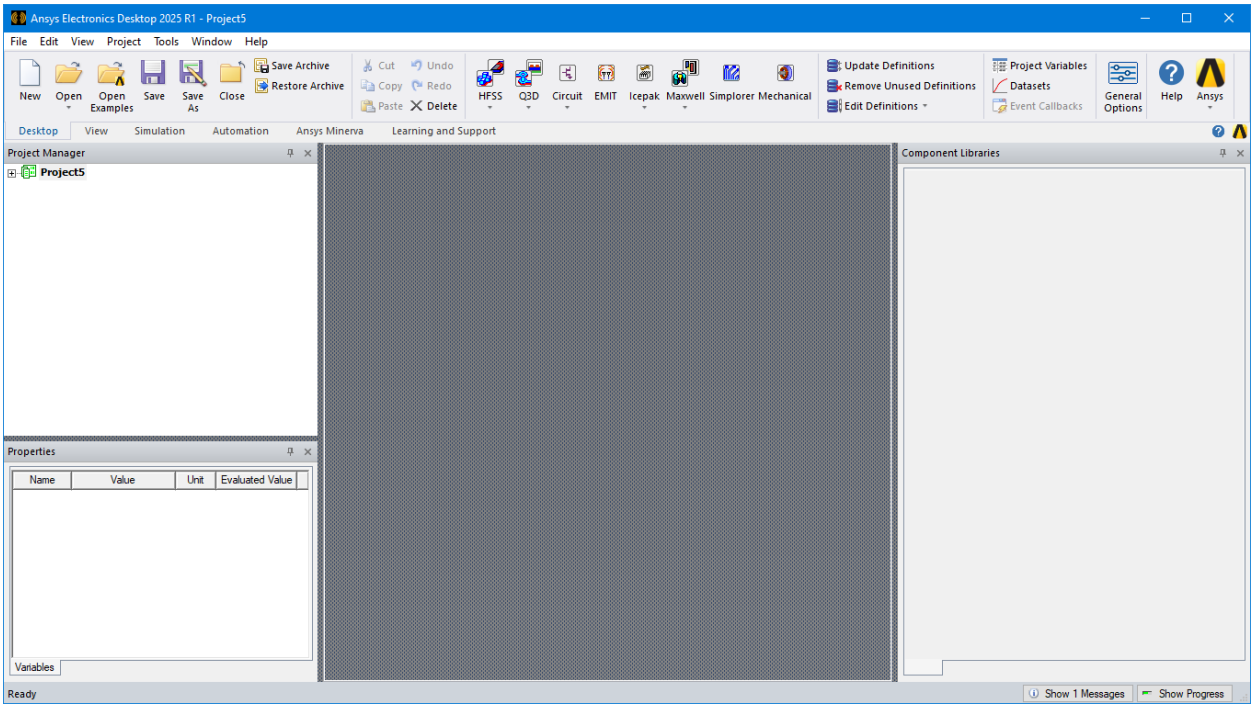
- From the Windows Start menu, select **Ansys EM Suite 2025 R1** > **Ansys Electronics Desktop 2025 R1**.

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



Ansys Electronics Desktop opens.

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



If the program fails to start, ensure that:

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

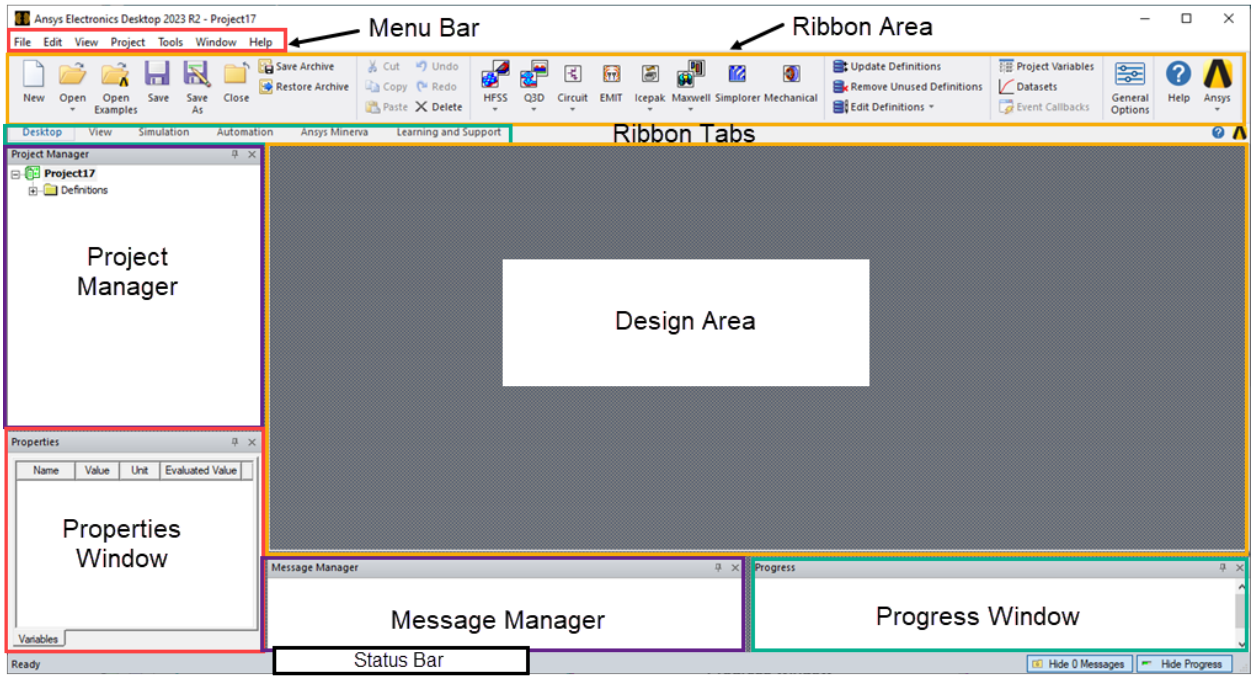
  This topic provides links to the Ansys installation guides (Windows and Linux) and product-specific platform support information.

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

Ansys Electronics Desktop Overview

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

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

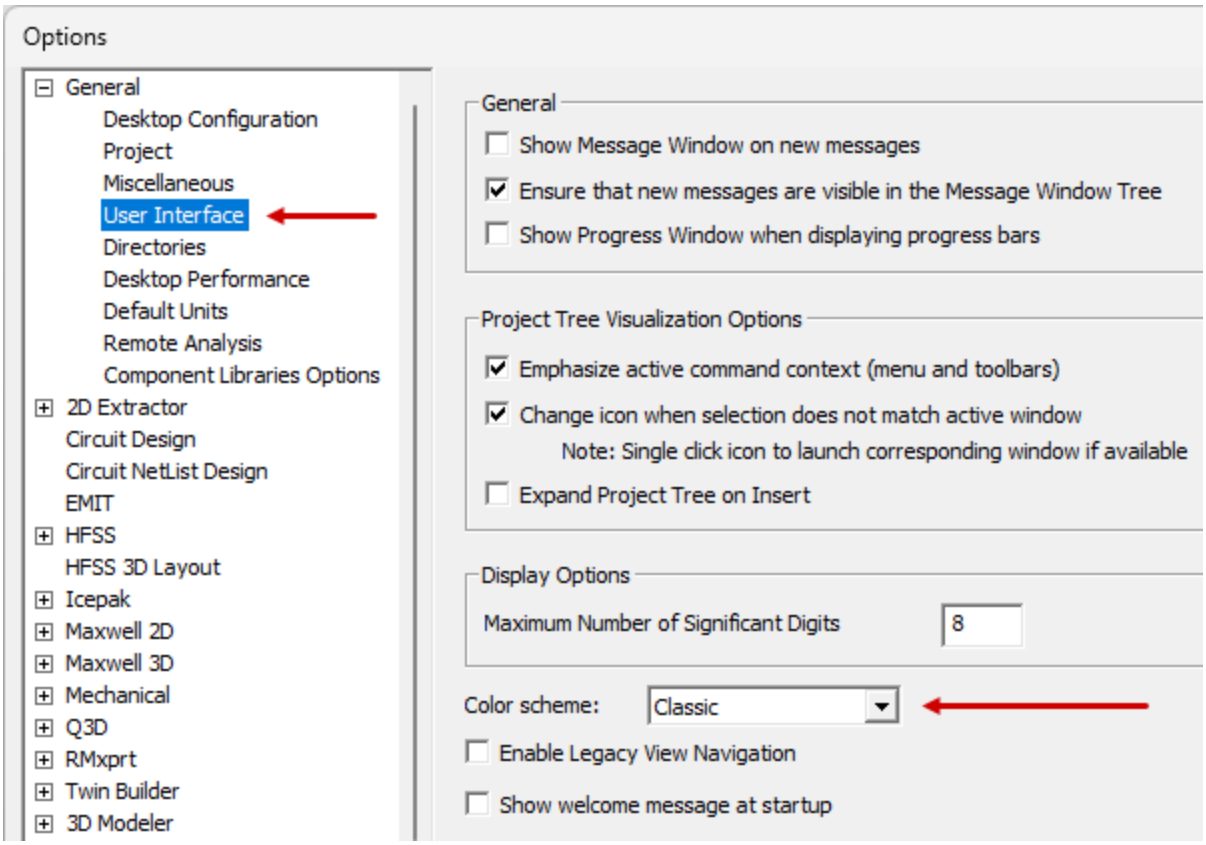


Choosing a Color Scheme [Beta]

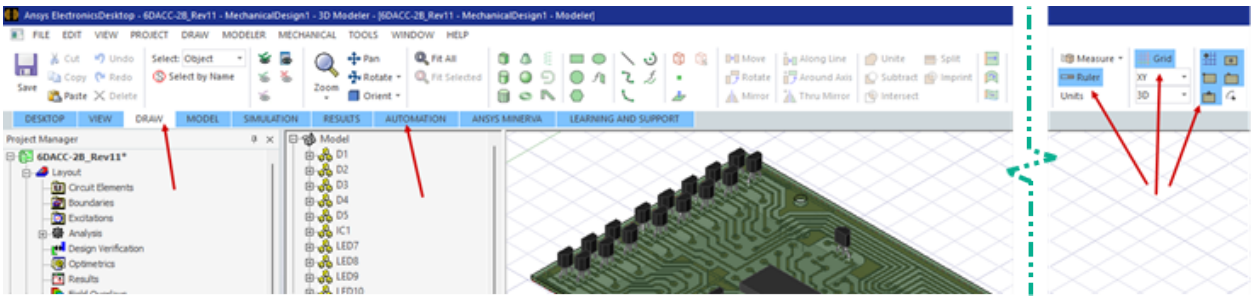
In addition to the classic Ansys Electronics Desktop color scheme, two additional color schemes are now available as beta features. Choose the desired color scheme as follows:

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

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

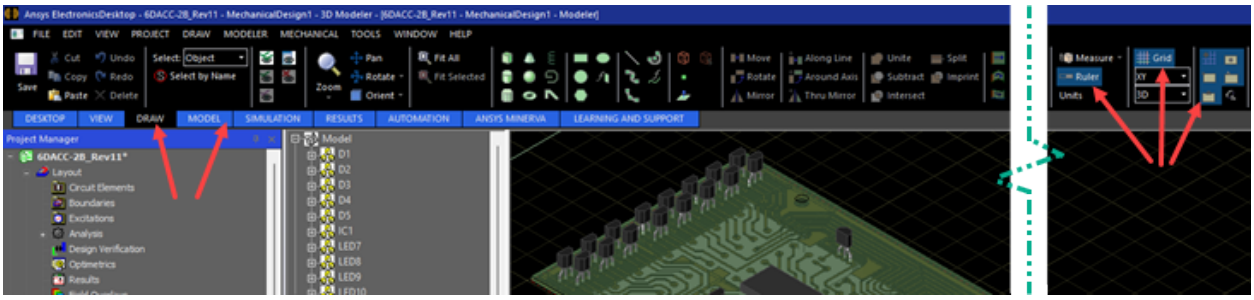


3. From the **Color scheme** drop-down menu, choose one of the following three options:
   - **Light (beta):** A light gray and white scheme with light blue highlighting for selected ribbon icons and for inactive ribbon tabs. The active ribbon has a white background (including the tab):

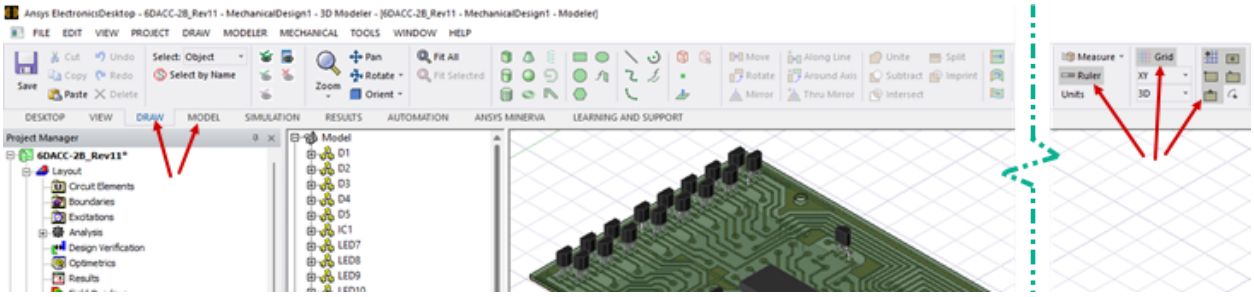


   - **Dark (beta):** A dark gray and black scheme with medium blue highlighting for selected ribbon icons and for inactive ribbon tabs. The active ribbon has a black background (including the tab):

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



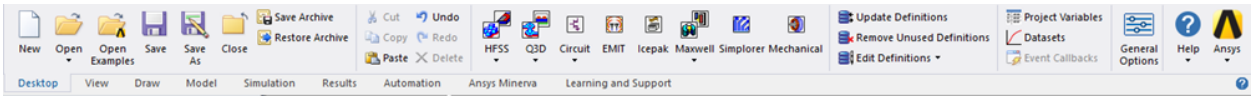
- **Classic:** A light gray and white scheme with medium gray highlighting for selected ribbon icons. The ribbon background is light gray. The name of the active ribbon tab has a blue font. A dark gray font is used for the inactive ribbon tab names:



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

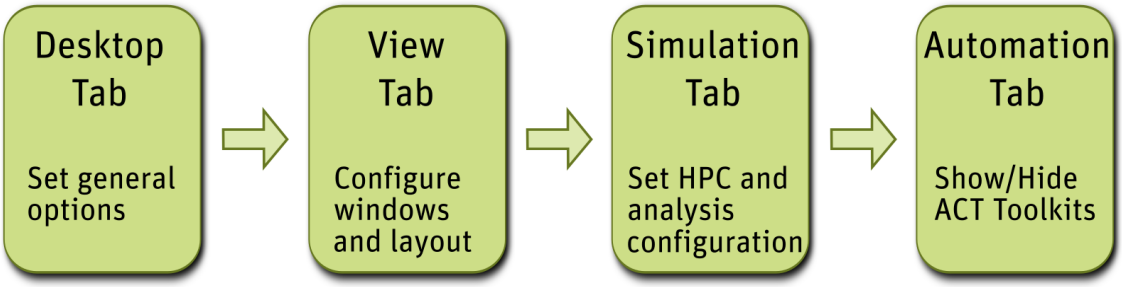
Working with Ribbons

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



The initial workflow for local configuration and personal customization is:

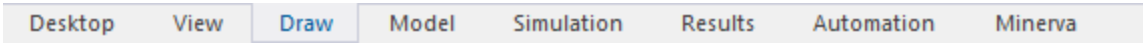
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



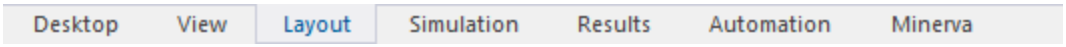
Available ribbon tabs depend on the design type. After you open or add a project and insert a design, you see additional ribbon tabs appear. The visible tabs and features are those that are appropriate for the design type and solver.

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

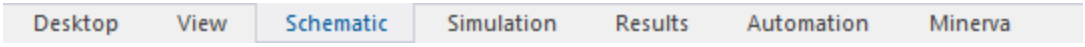
- For HFSS, Icepak, Maxwell, Mechanical, and Q3D designs, the **Draw**, **Model**, and **Results** tabs also appear:



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



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

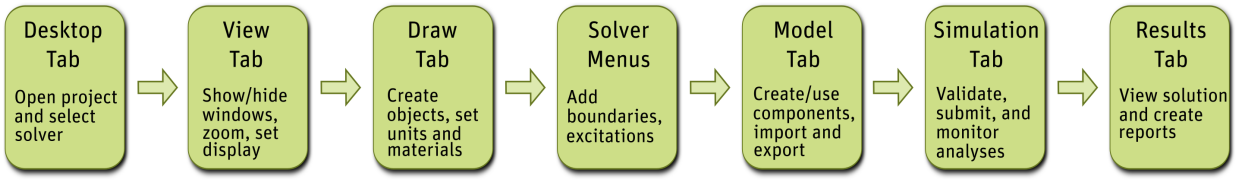


In addition to **Desktop**, **View**, **Minerva** and **Learning and Support** the ribbon tabs that are applicable to HFSS designs are described as follows:

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

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

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

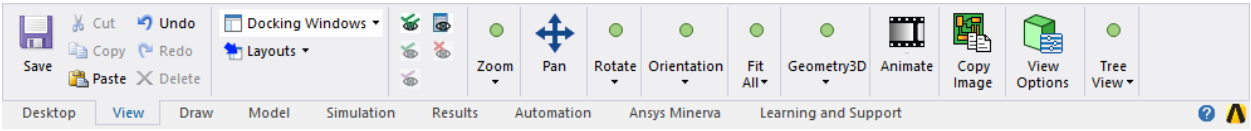


Note that workflow can vary depending on tasks at hand. For example, if you open a completed model in HFSS or use 3D components or the ACT Antenna Design toolkit (Windows only), you may not need the features of the **Draw** tab. If Validation on the **Simulation** tab identifies problems in a model,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

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

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

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



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

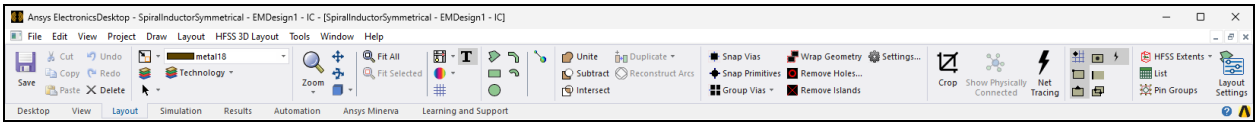
The **Learning and Support** tab provides easy access to Technical advice, instruction, and examples on Ansys websites.

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

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

IC Mode Layout Ribbon

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



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

Ansys Electronics Desktop Windows

Ansys Electronics Desktop contains the following windows:

- Project Manager
- Message Manager
- Progress
- Properties

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

- Component Libraries
- Layers
- Nets
- Components

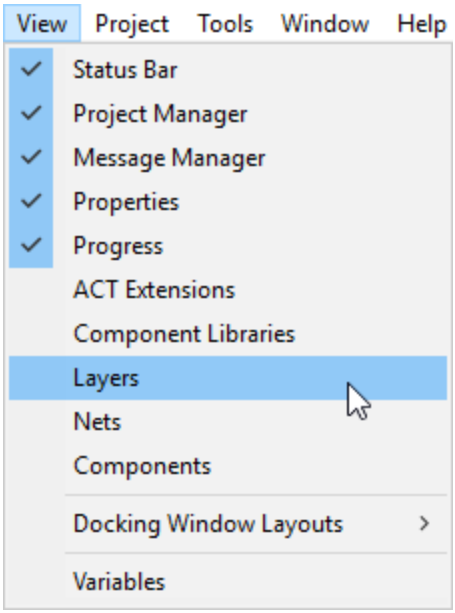
You can toggle the display of each window, auto-hide windows, or move and resize windows. Additionally, you can toggle the status bar or add ACT Extensions (Windows only).

Showing and Hiding Windows

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

From the **View** menu:

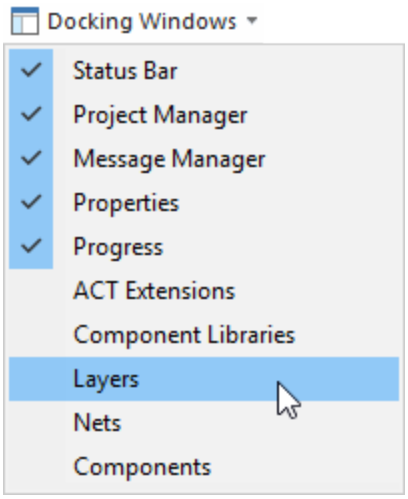
- Click a window to show or hide it.



From the **View** tab:

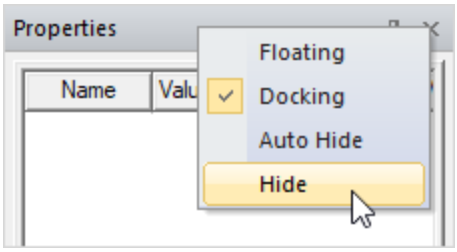
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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



You can also hide a window directly from the window:

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



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

Auto Hiding Windows

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

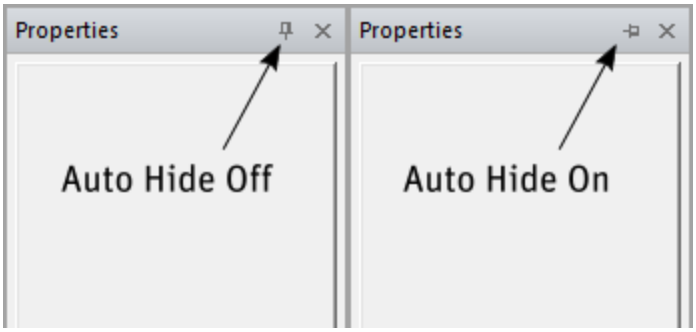
To automatically hide a window until the cursor is hovered over it:

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

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

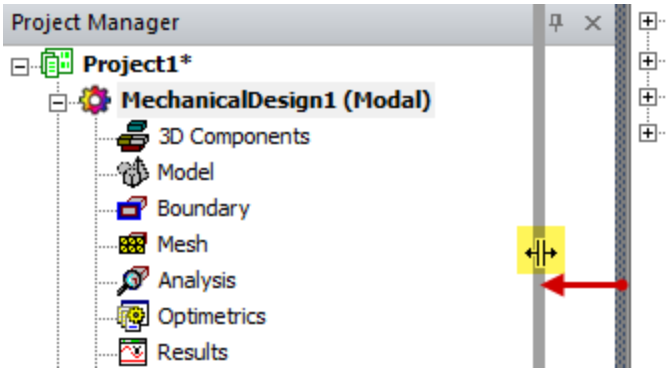
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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

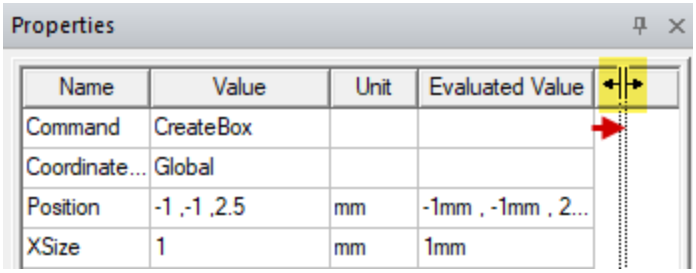


Moving and Resizing Windows

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



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



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

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

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

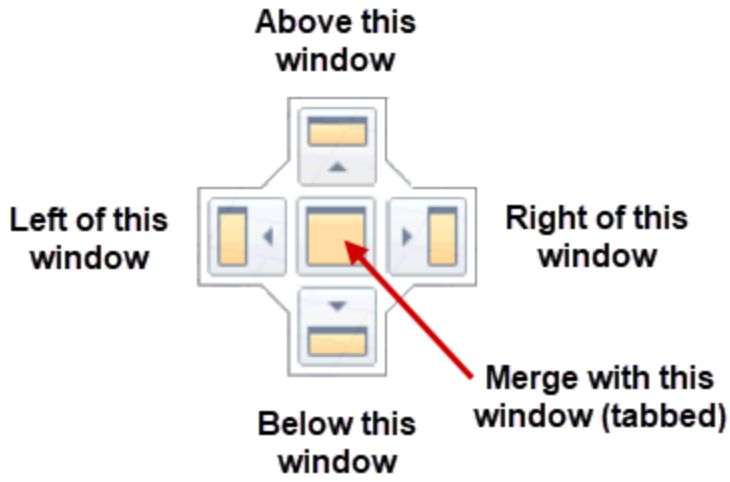
Left edge of user interface:

Right edge of user interface:

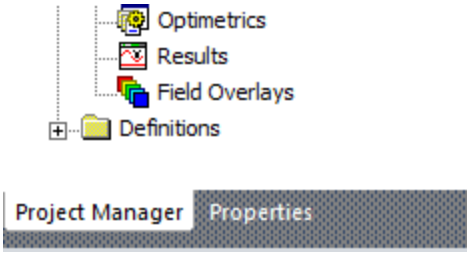
Top of user interface:

Bottom of user interface:

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



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



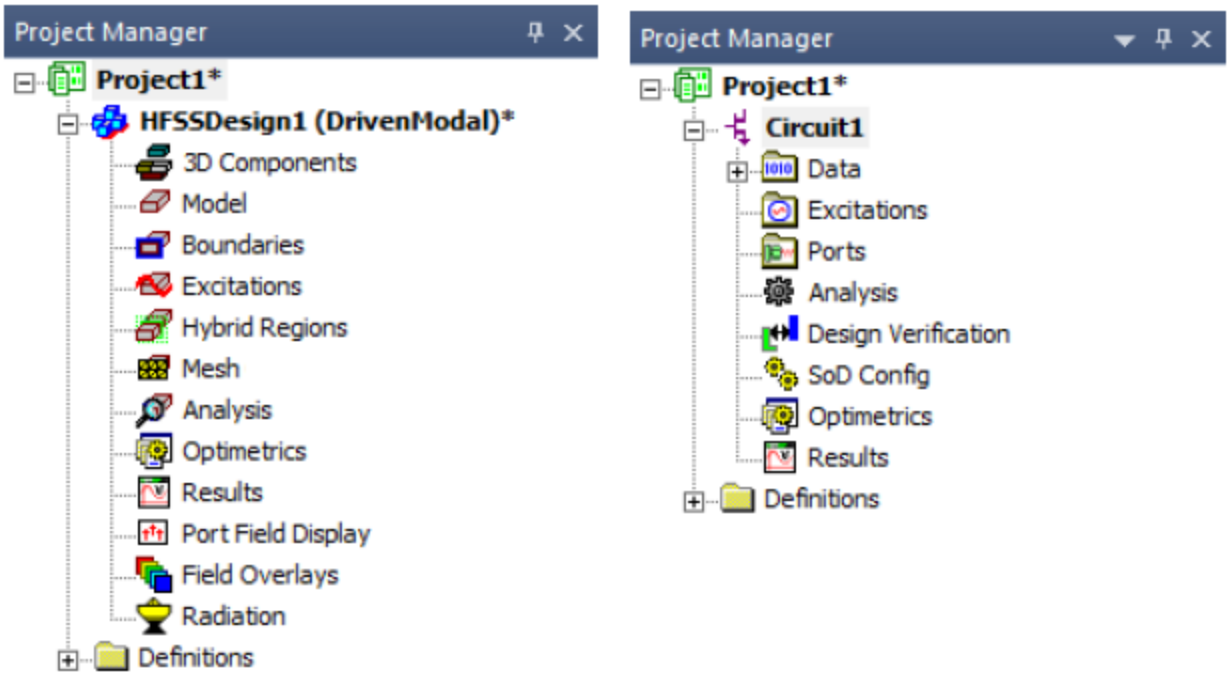
If the middle icon does not appear, it's because tabbed windows are not supported by one of the windows. For example, you cannot merge any window with the Modeler window.

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

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

Project Manager Window

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



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

- For any HFSS, Maxwell, or Q3D design, each project ultimately includes a geometric model, its boundary conditions and material assignments, and field solution and post-processing information. Any 3D Components appear under the 3D Components icon.
- For a Circuit design, the project includes data, excitations, ports, analysis setup, design verification, SoD configuration, Optimetrics, results, and relevant definitions.

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

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

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

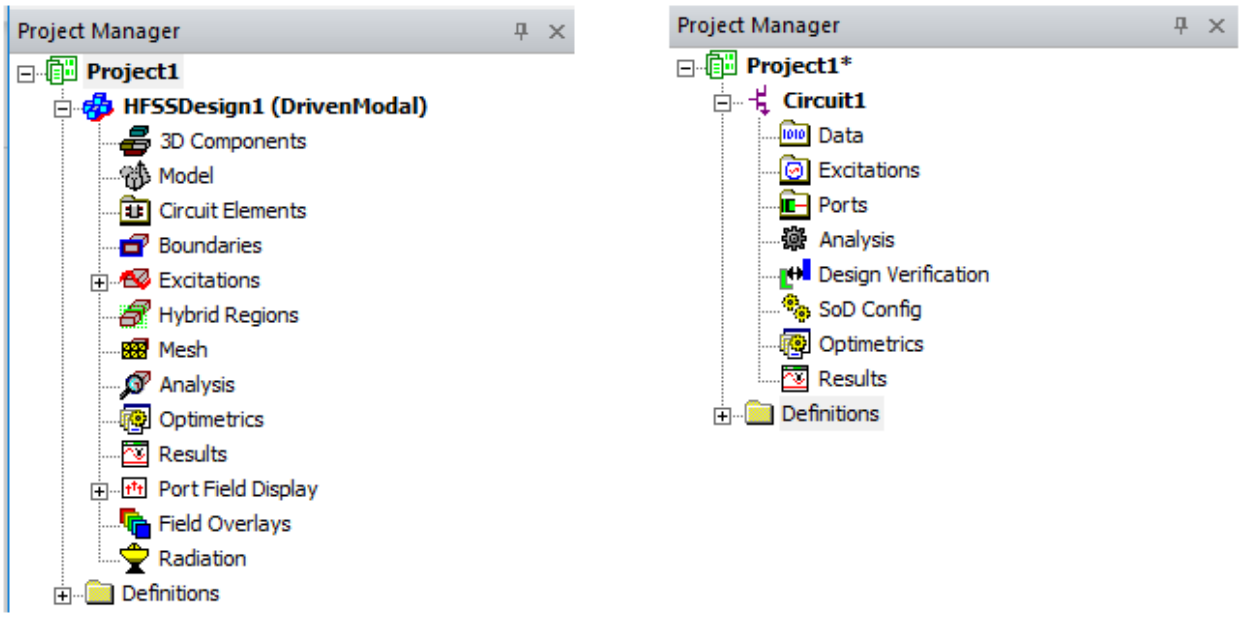
- Right-click in the toolbars area on the desktop, and then click **Project Manager** on the shortcut menu.

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

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

Working with the Project Tree

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



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

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

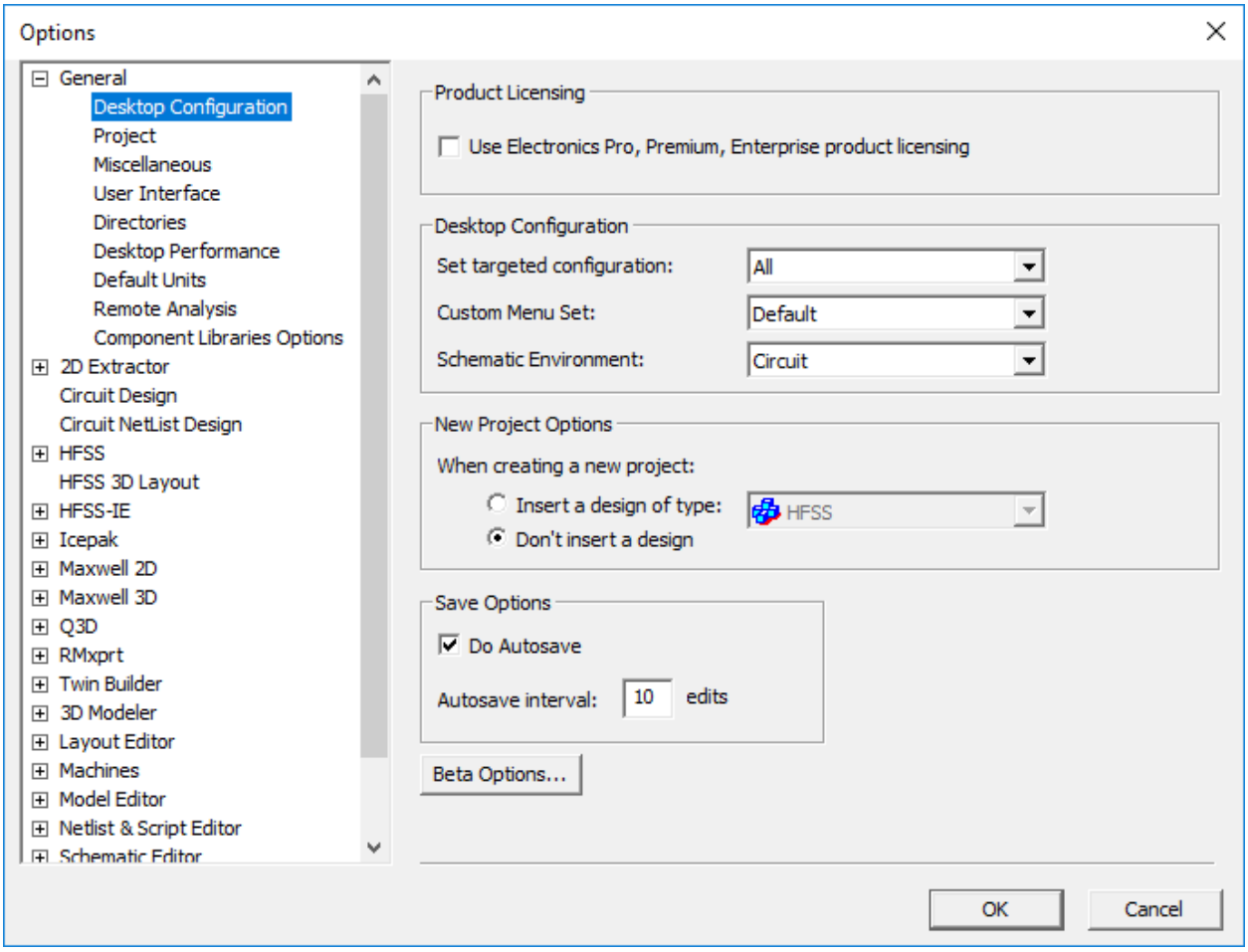
Setting the Project Tree to Expand Automatically

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

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

   The **Options** window appears.

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

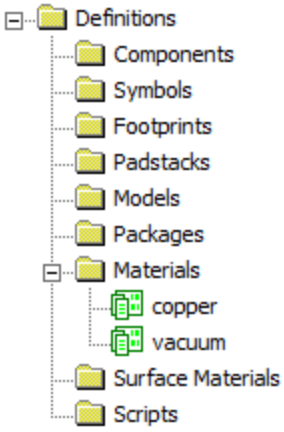


2. Click **User Interface**.
3. In the **Project Tree Visualization Options** area, check the **Expand Project Tree on Insert** check box.
4. Click **OK**.

Viewing Material Definitions

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

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



Viewing Ansys Electronics Desktop Design Details

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

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

3D Components	Displays any 3D Components added to the design.
Model	Displays the model geometries in the design.
Boundaries	Displays the boundary conditions assigned to an Ansys Electronics Desktop design, which specify the field behavior at the edges of the problem region and object interfaces.
Excitations	Displays the excitations assigned to an Ansys Electronics Desktop design, which are used to specify the sources of electromagnetic fields and charges, currents, or voltages on objects or surfaces in the design.
Hybrid Regions	Displays the hybrid regions defined for an HFSS design, which can be FE-BI, IE Region, PO Region, SBR+ Region, and Dielectric Cavity.
Mesh	Displays the mesh operations specified for objects or object faces. Mesh operations are optional mesh refinement settings that are specified before a mesh is generated.
Analysis	Displays the solution setups for an Ansys Electronics Desktop design. A solution setup specifies how Ansys Electronics Desktop will compute the solution.

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

Optimetrics	Displays any Optimetrics setups added to an Ansys Electronics Desktop design.
Results	Displays any post-processing reports that have been generated.
Port Field Display	Displays all port fields in the active model.
Field Overlays	Displays fields overlay plots, which are representations of basic or derived field quantities on surfaces or objects. Plot folders are listed under **Field Overlays.** These folders store the project's plots and can be customized. See Setting Field Plot Defaults for information on how to customize the plot folders.
Radiation	Displays far- and near-field setups added to an Ansys Electronics Desktop design.
Documentation	Displays files you have added as documentation.
Definitions	Displays definitions for HFSS, including Components, Materials, and more (as applicable).

Note:

To edit a project's design details:

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

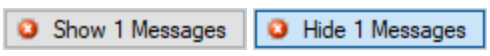
A window appears with that setup's parameters, which you can then edit.

Message Manager Window

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

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

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



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

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

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

Indicates an informative message.

Indicates a warning message that may require your attention.

Indicates an error message that may require your attention.

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

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

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

Setting the Message Manager to Open Automatically

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

Showing New Messages

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

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

   The **Options** window appears.

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

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

Automatically Expanding the Message Manager Tree

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

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

   The **Options** window appears.

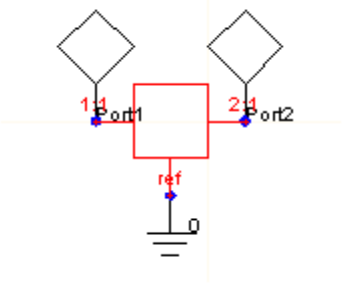
2. Expand **General** and click **User Interface**.
3. In the **General** area, check the **Ensure that new messages are visible in the Message Window Tree** check box.
4. Click **OK**.

Action Messages

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

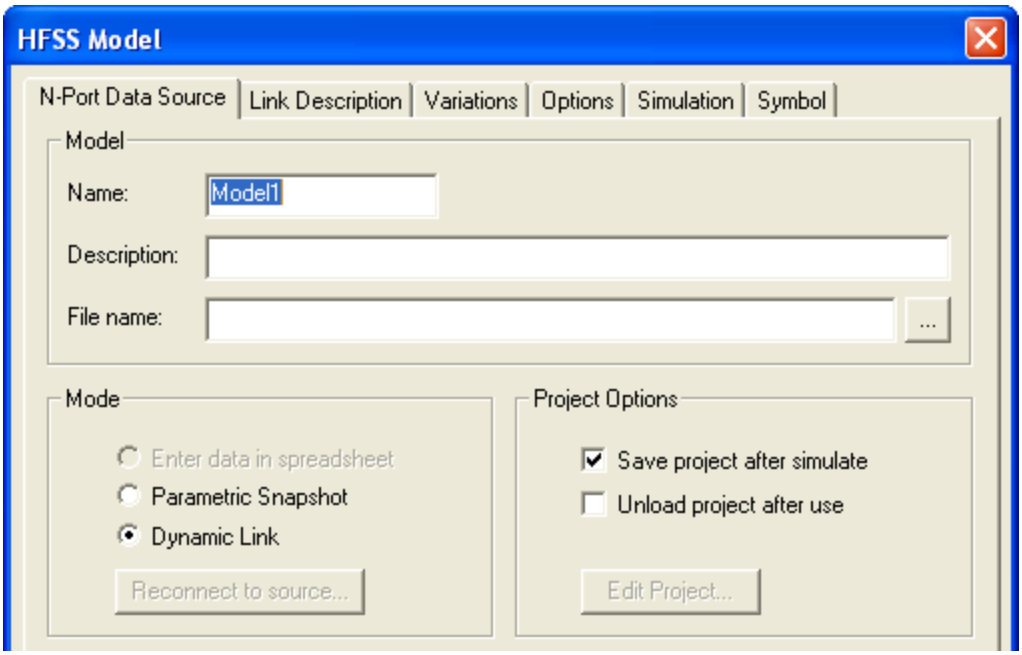
You can invoke a message action by clicking the magnifying glass icon, by double-clicking the message, or by right-clicking the message and selecting **Go To Reference** from the pop-up menu.

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



Another example is the opening of a related window:

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



Clearing Messages

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

Hiding the Message Manager Window until Messages Appear

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

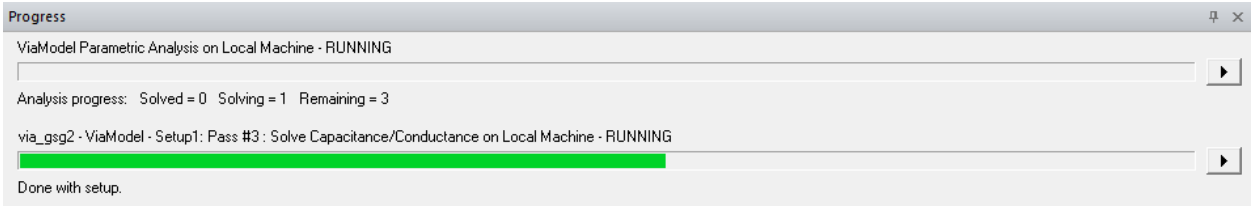
1. Select **View** > **Progress Window** to hide the window. For more information see the View drop-down menu.
2. Click **Tools** > **Options** > **General Options**.
3. Check the **Show message window on new messages** check box.

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

Progress Window

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

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



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

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

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

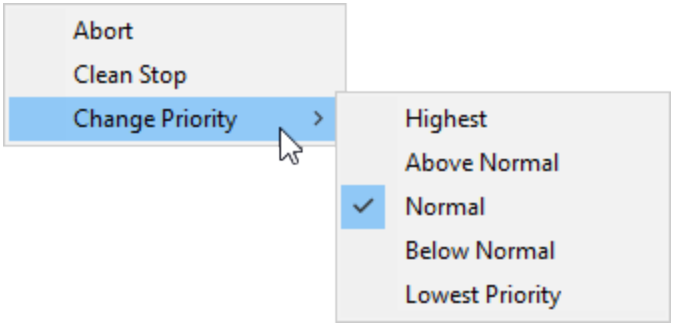
A check box appears next to this command if the **Progress** window is visible.

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

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

Stopping or Aborting Simulation Progress

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

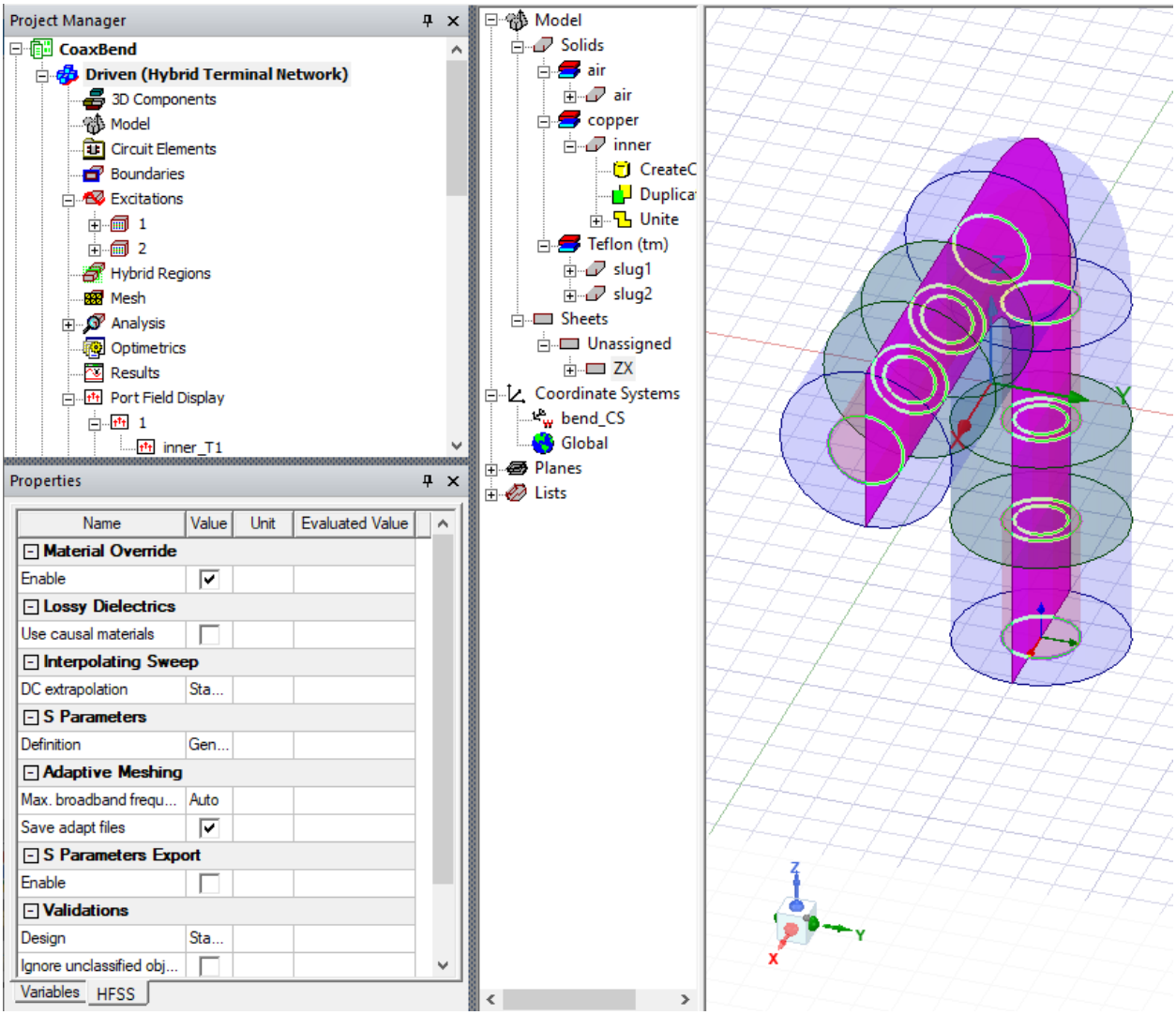


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties Window

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

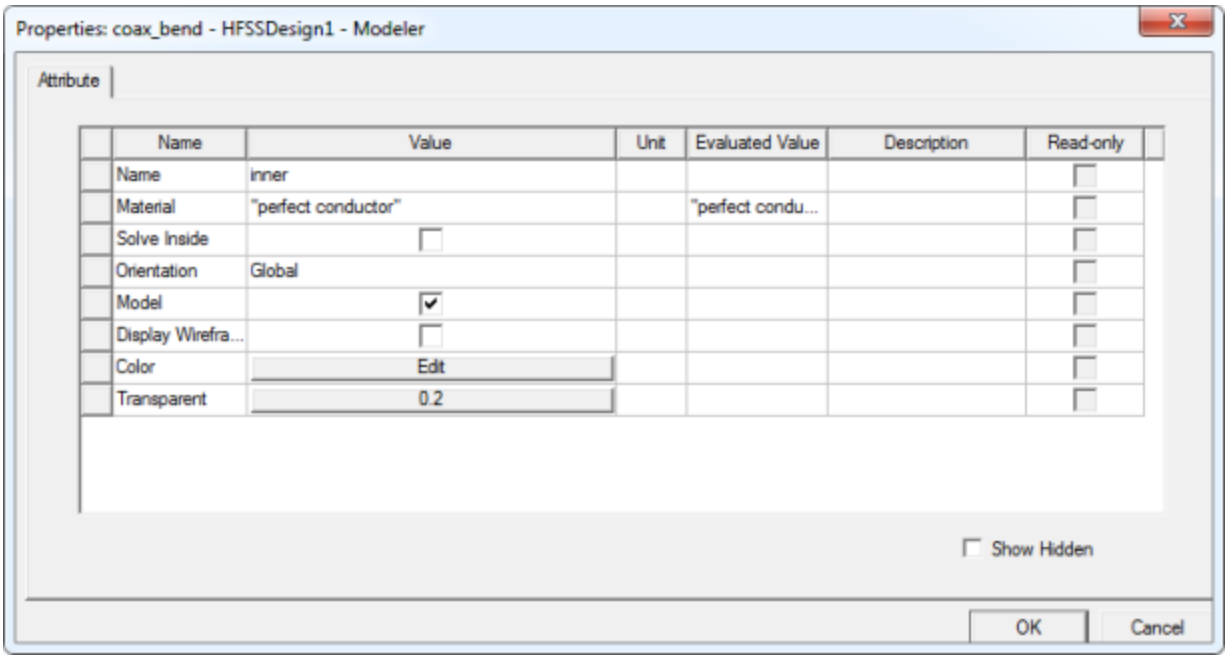
You can choose to show or hide a docked **Properties** window as part of the desktop.



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

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

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



Opening the Properties Window

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

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

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

Setting the Properties Window to Open Automatically

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

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

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

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

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

Showing and Hiding the Properties Window

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

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

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

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

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

Modifying Object Attributes Using the Properties Window

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

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

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

Modifying Object Command Properties Using the Properties Window

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

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

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

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

   Tip:

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

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

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

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

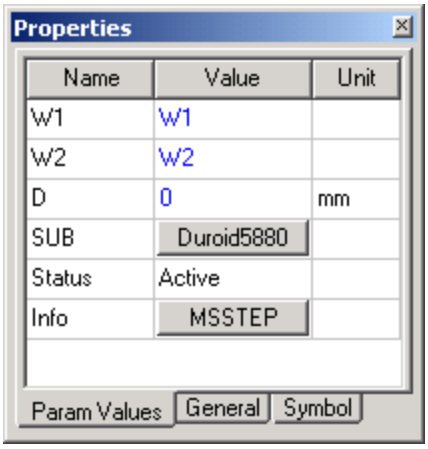
Property Window for Schematic and Layout Editors

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

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

Param Values Tab

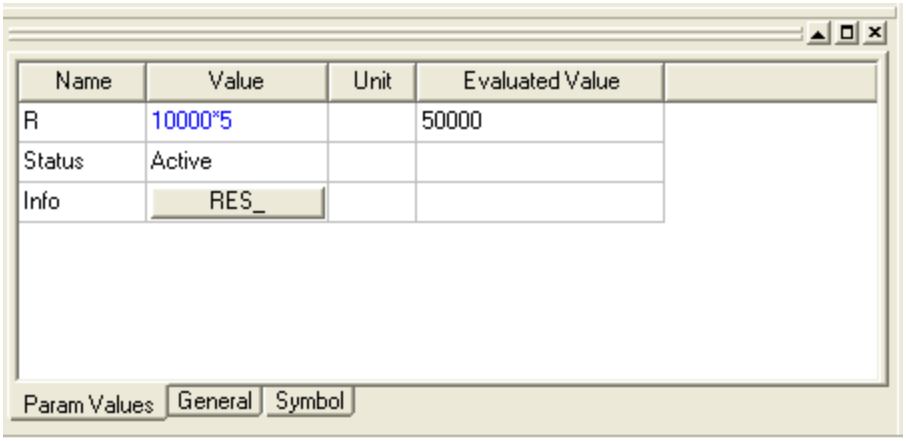
The **Param Values** tab in the **Property** window of the Ansys Electronics Desktop lists the simulation parameters of the component or components selected.



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

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

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



Launching Help from the Property Window

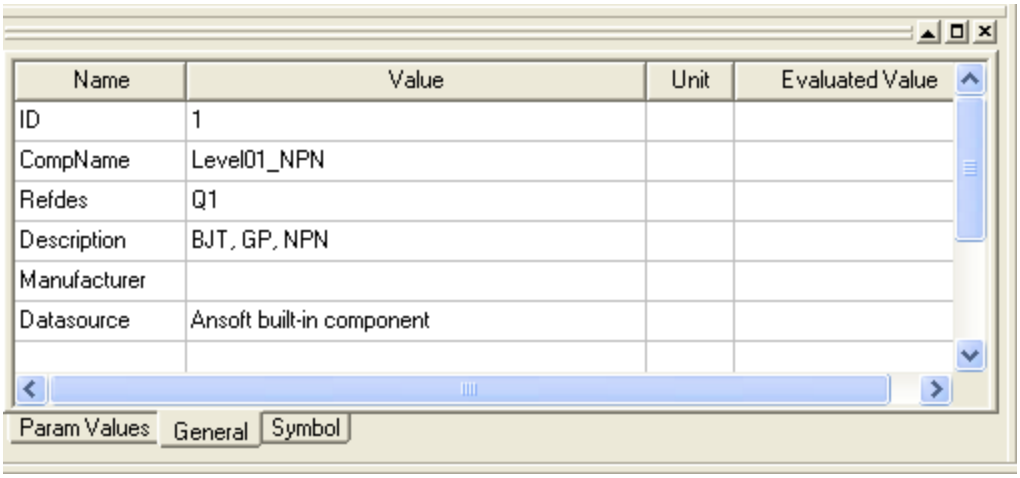
You can launch the online help for a component from the **Property** window. To do this:

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

General Tab

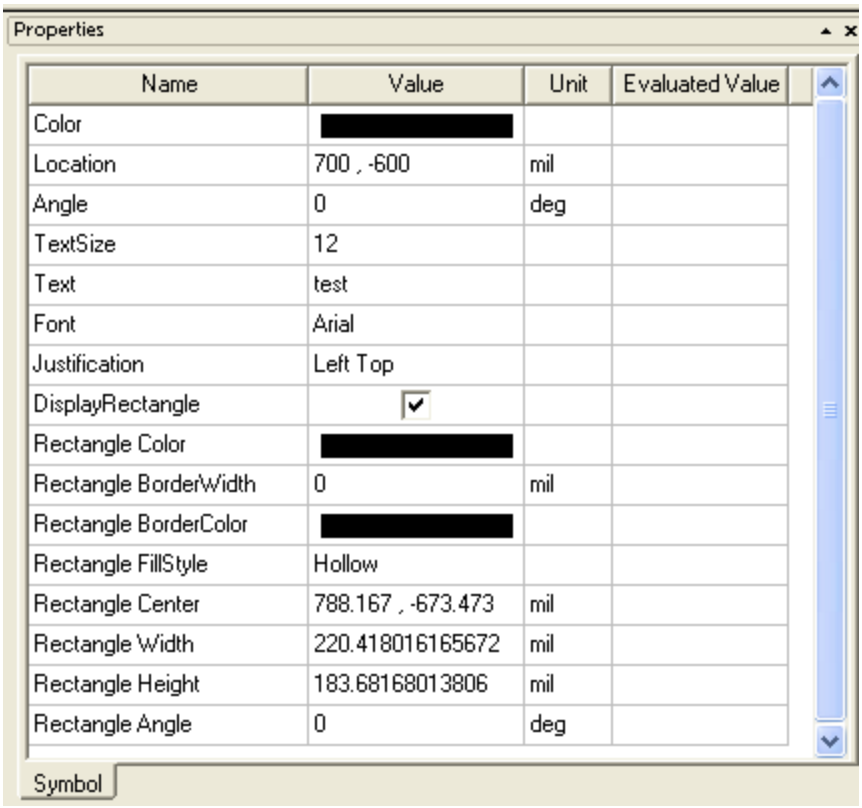
The **General** tab in the **Property** window of the Ansys Electronics Desktop lists the selection's name, symbol name, reference designator, and other data. These are generally not editable.

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



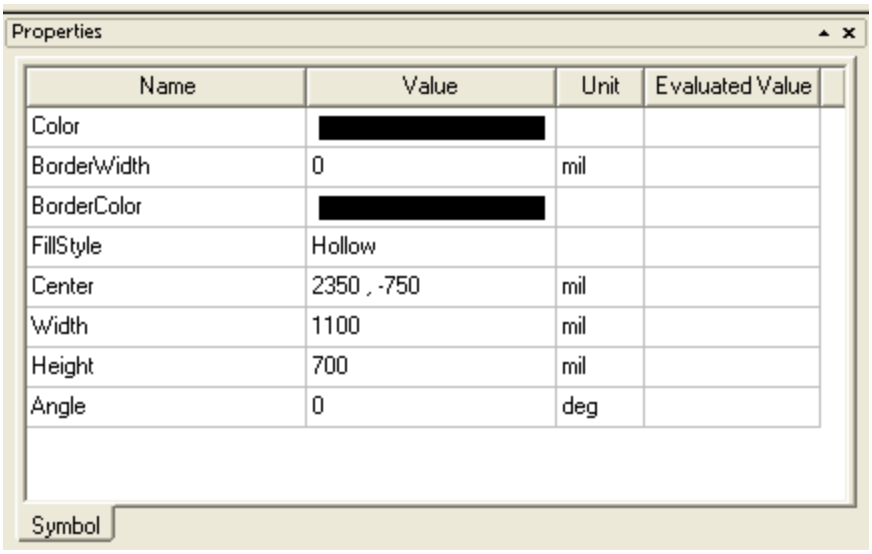
Symbol Tab

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

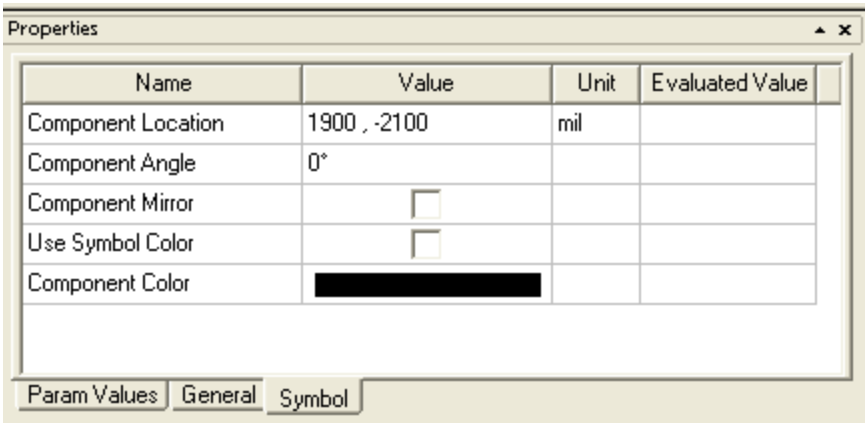


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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



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



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

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

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

- Click in the **Value** field to enter a new value for whatever symbol attribute you wish to change. Press **Return** to move the symbol to the new location.
- Click the colored bar displayed in the **Value** field to open a palette from which to select a new color for the symbol. The new color is applied when the symbol is unselected.

Closing the Property Window

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

For more information, see the View Pulldown Menu.

Properties Dialog Box

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

- Parameter Values
- General
- Symbol
- Property Displays

Opening the Properties Dialog Box

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

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

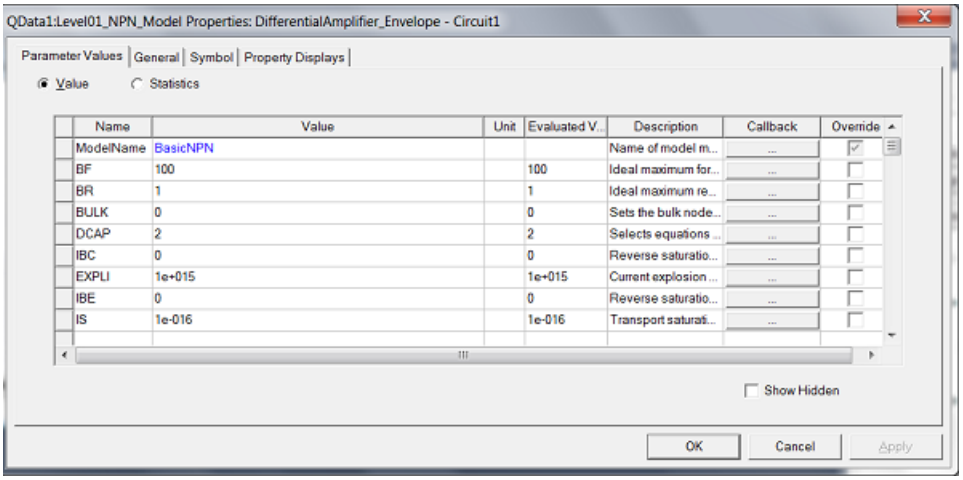
You can manipulate the **Properties** dialog box in the following ways:

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

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

Parameter Values Tab

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



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

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

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

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

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

General and Symbol Tabs

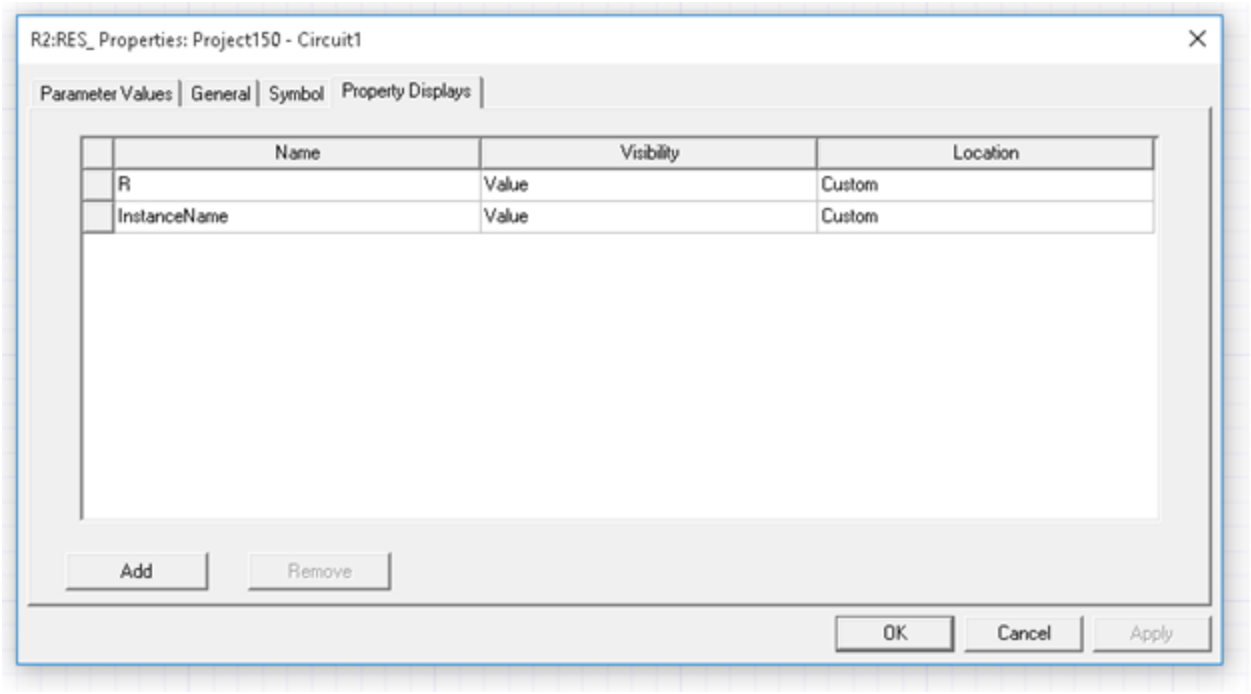
The **General** tab in the Properties Dialog Box lists the selection's name, symbol name, reference designator, and so on. These are generally not editable. The information is identical to that on the General tab of the Property Window.

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

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

Property Displays Tab

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



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

- **None** – results in no label being shown.
- **Name** – displays the parameter name (e.g., R).
- **Value** – displays the component value, which can be a single value (e.g., 10000) or an expression (e.g., 10000*5).

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

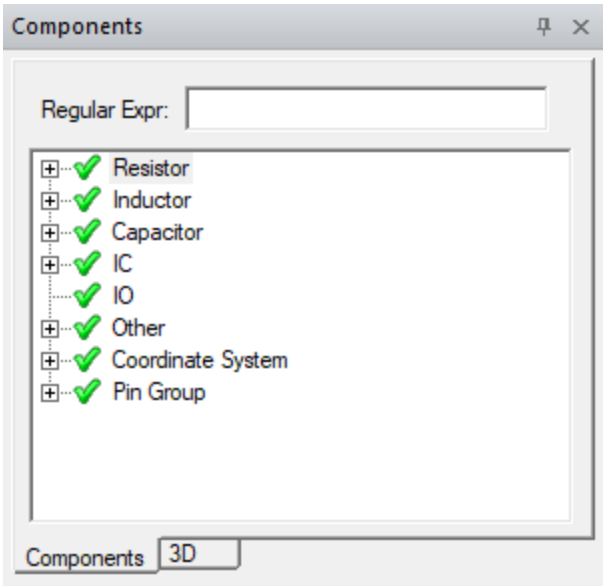
- **Both** – displays the parameter name and its value (e.g., R = 10000, R=10000*5).
- **Evaluated Value** – displays the evaluated value of an expression that has been used for the value of a parameter (e.g., 50000 for the expression 10000*5).
- **Evaluated Both** – displays the parameter name and the Evaluated Value (e.g., R = 50000 for the expression 10000*5).

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

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

Working with the Components Window

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



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

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

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

- **Type** – reclassifies the component type to **Resistor**, **Inductor**, **Capacitor**, **IC**, **IO**, or **Other**.
- **Enable**/**Disable** - enables or disables the component.
- **Model** – opens the **Component Model** window, where you can change the component model definition, including die, solder ball, and port properties. Changing the model scope will update every reference designator that belongs to the model definition.
- **Create Ports on Component** – adds ports for the component
- **Remove Ports on Component** – removes ports for the component.
- **Fit Selected** – fits the selected component in the active view window.

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

For more information, see Layout-based Component Encapsulation.

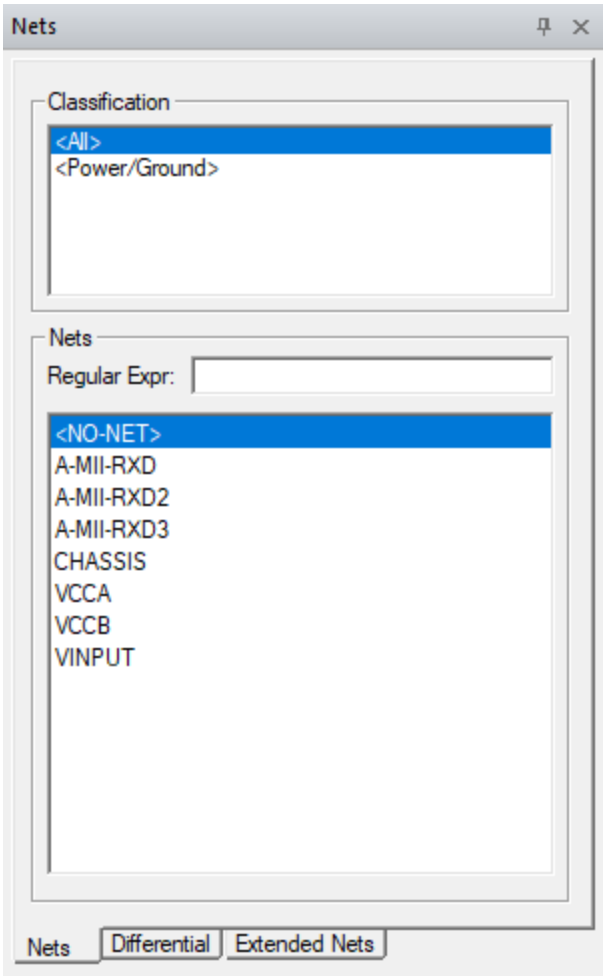
Working with the Nets Window

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

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

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

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



Nets Tab

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

Classification Pane

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

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

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

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

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

Nets Pane

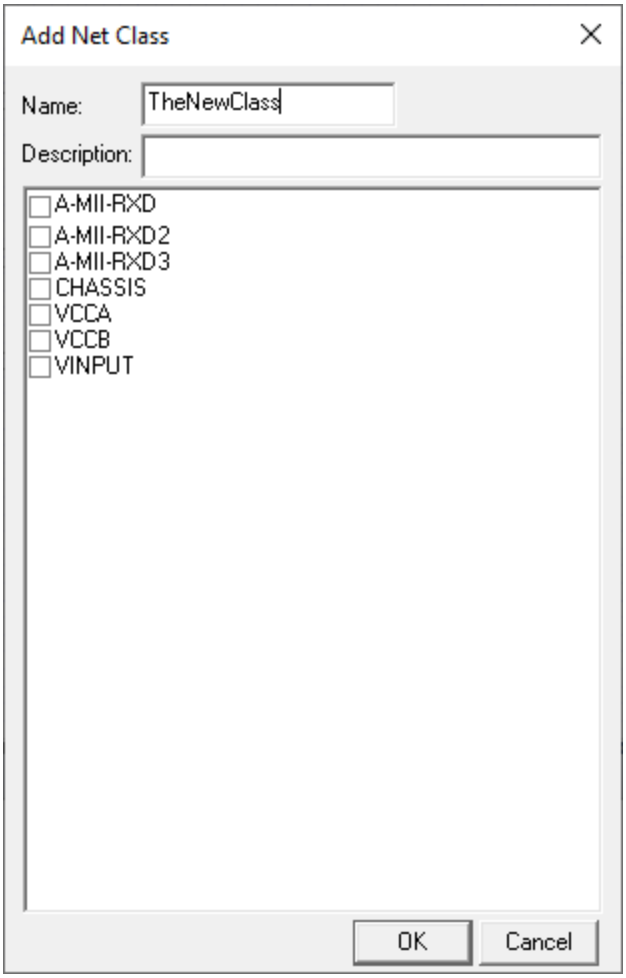
The displayed nets may be filtered using Perl Regular Expression syntax. In the **Nets** pane, the right-click menu contains additional options:

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

Add Net Class

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

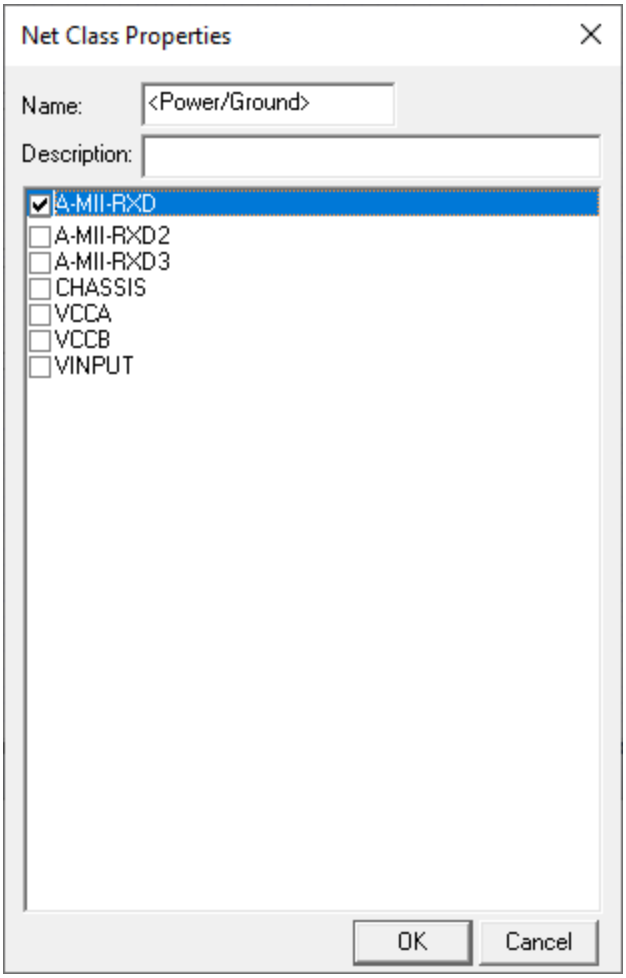
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Net Class Properties

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

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



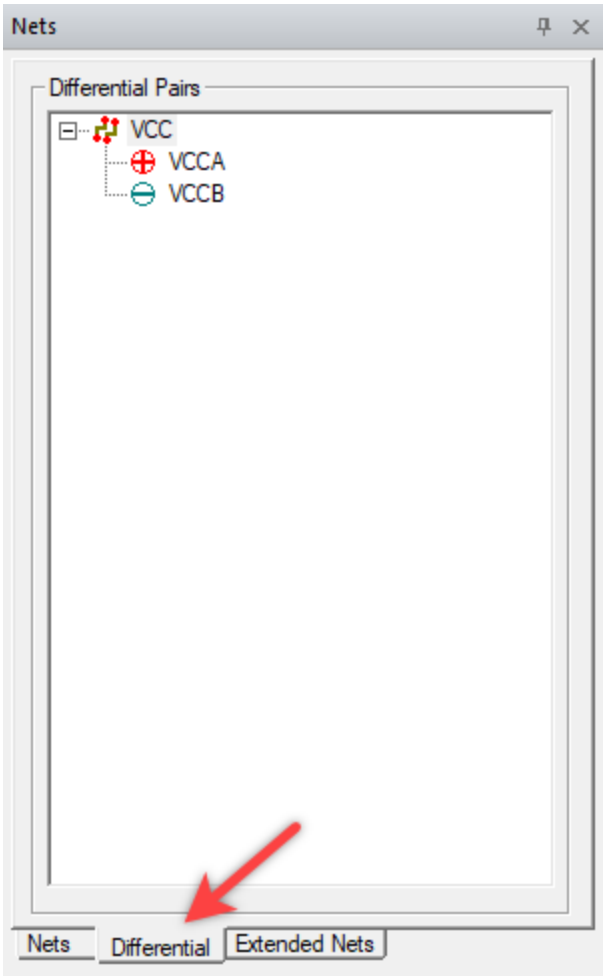
Differential Pairs

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

Viewing Differential Pairs

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

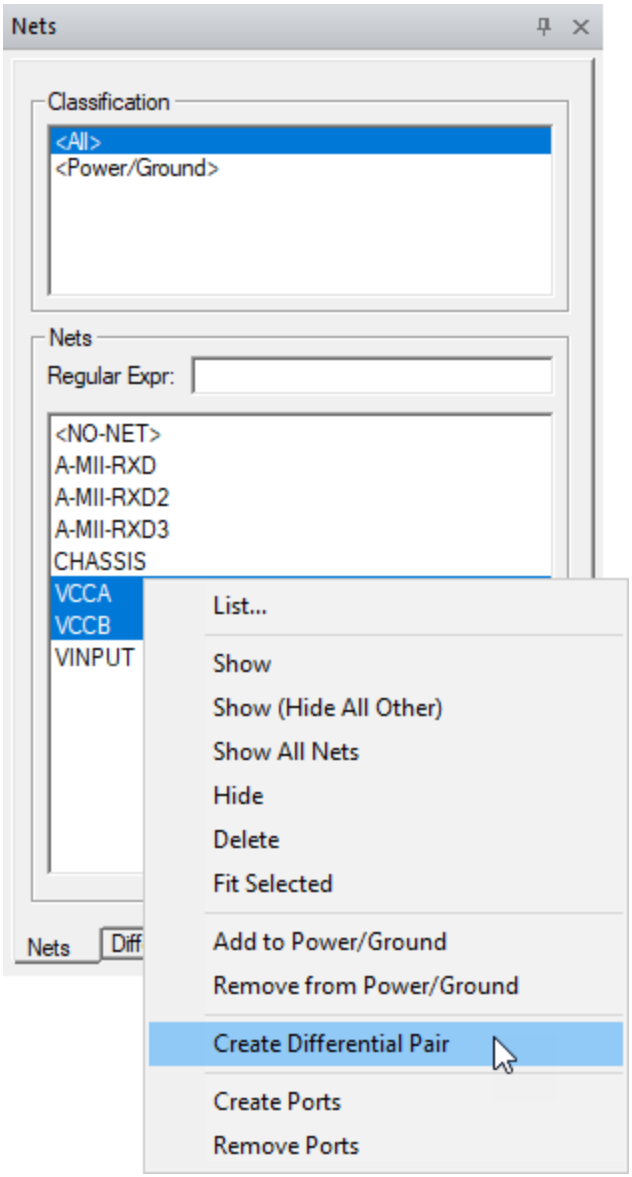
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Creating Differential Pairs

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

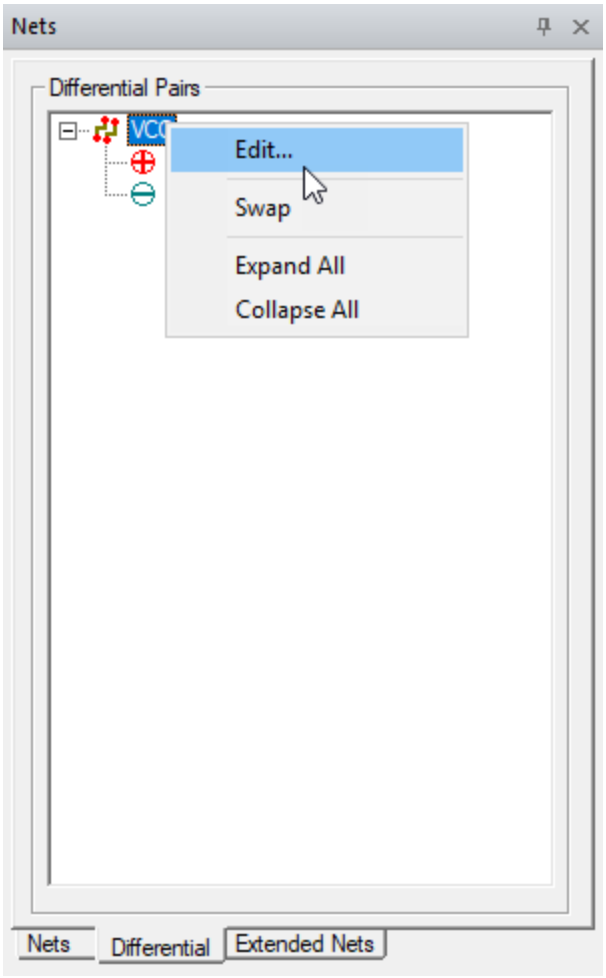
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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

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

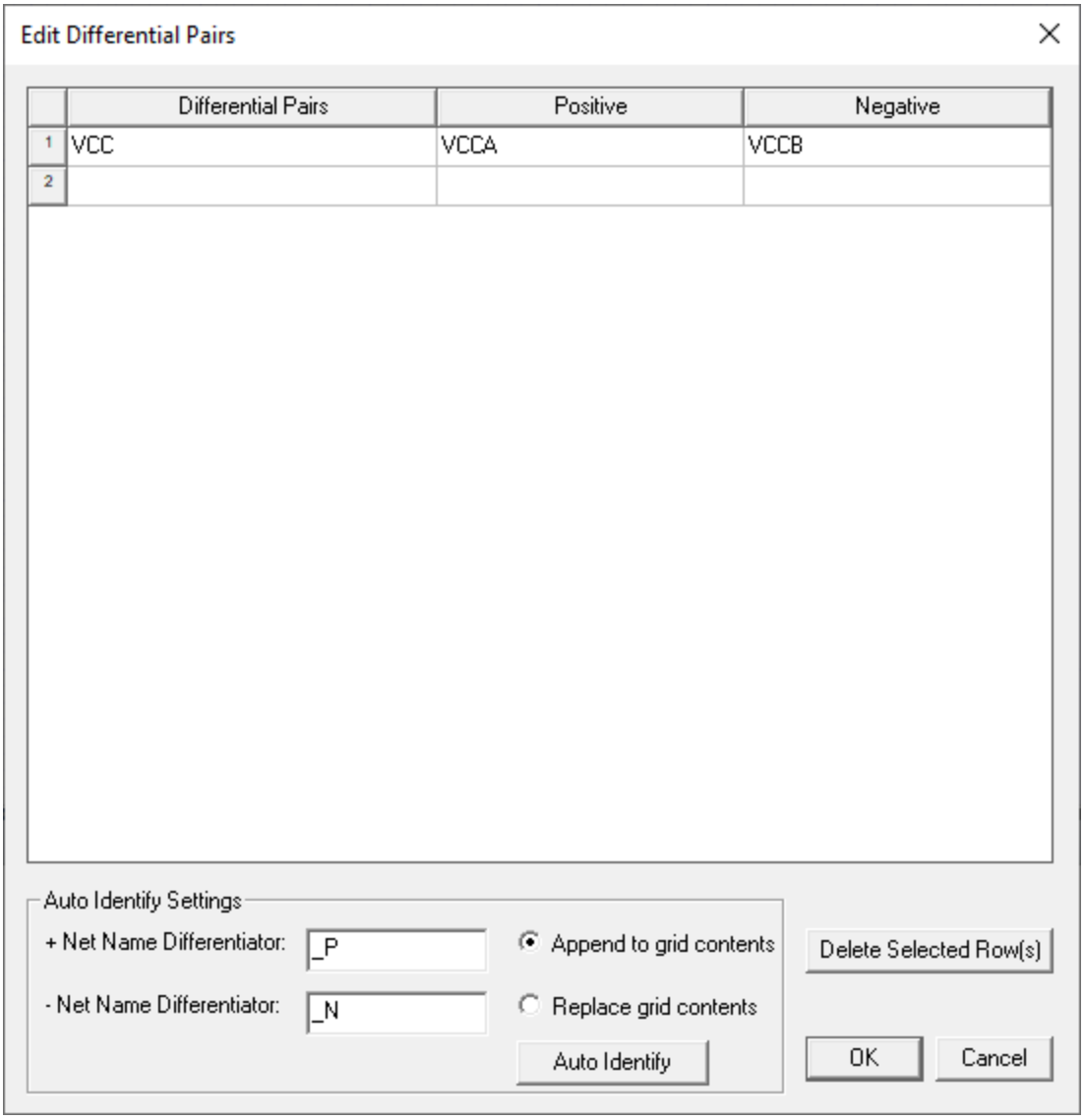
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Edit Differential Pairs Window

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

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



Extended Nets Tab

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

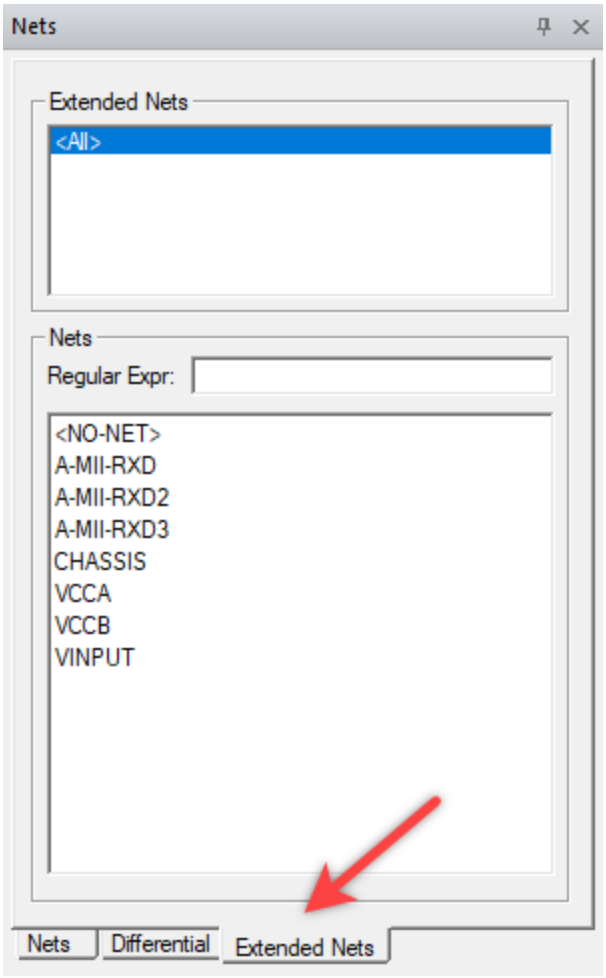
Extended nets are collections of nets that are similar to net classifications but with subtle differences. A net can be in only one extended net at a time, in contrast to being in multiple

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

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

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

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



Extended Nets Pane

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

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

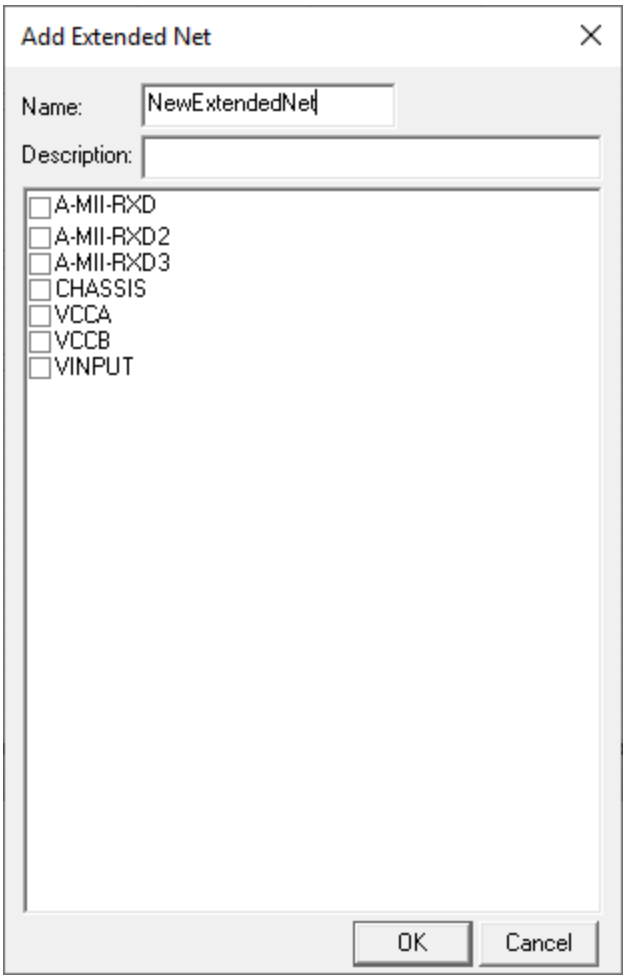
net. <All> is not an extended net. The **Extended Nets** pane's shortcut menu offers the following actions:

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

Create an Extended Net

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

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

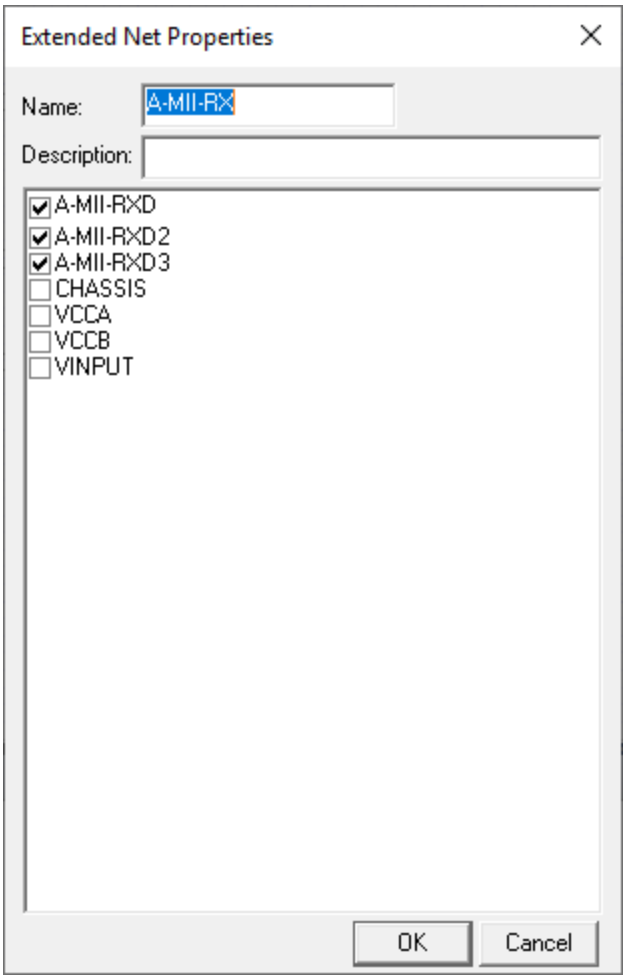


2. Enter a name and optional description.
3. Select the nets that will make up the extended net.
4. Click **OK**.

Editing an Extended Net

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

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

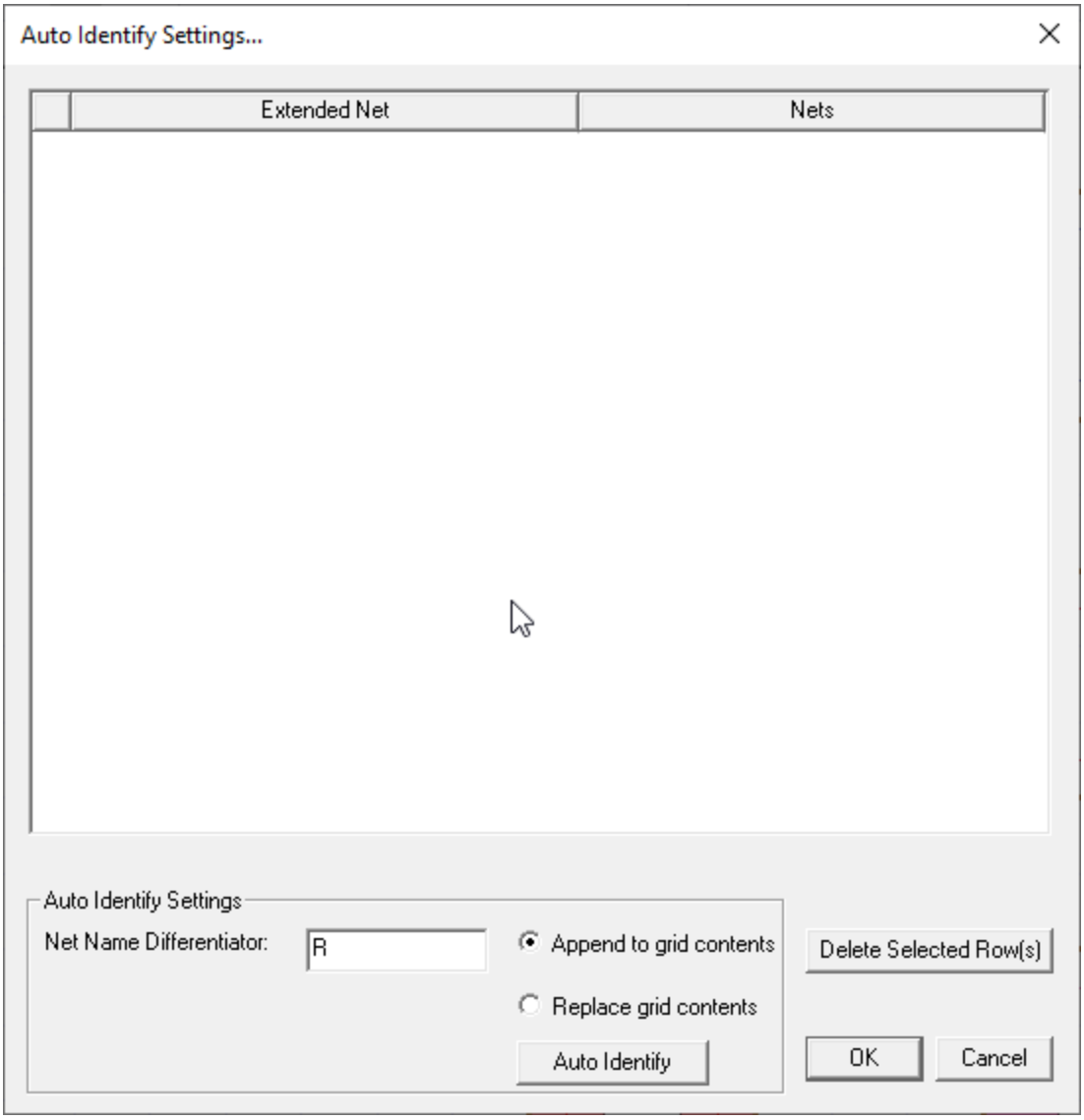


2. Change the name and optional description.
3. Select the nets that will make up the extended net.
4. Clear the nets that should not be in the extended net.
5. Click **OK**.

Auto Identify Settings window

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

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



Nets Pane

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

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

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

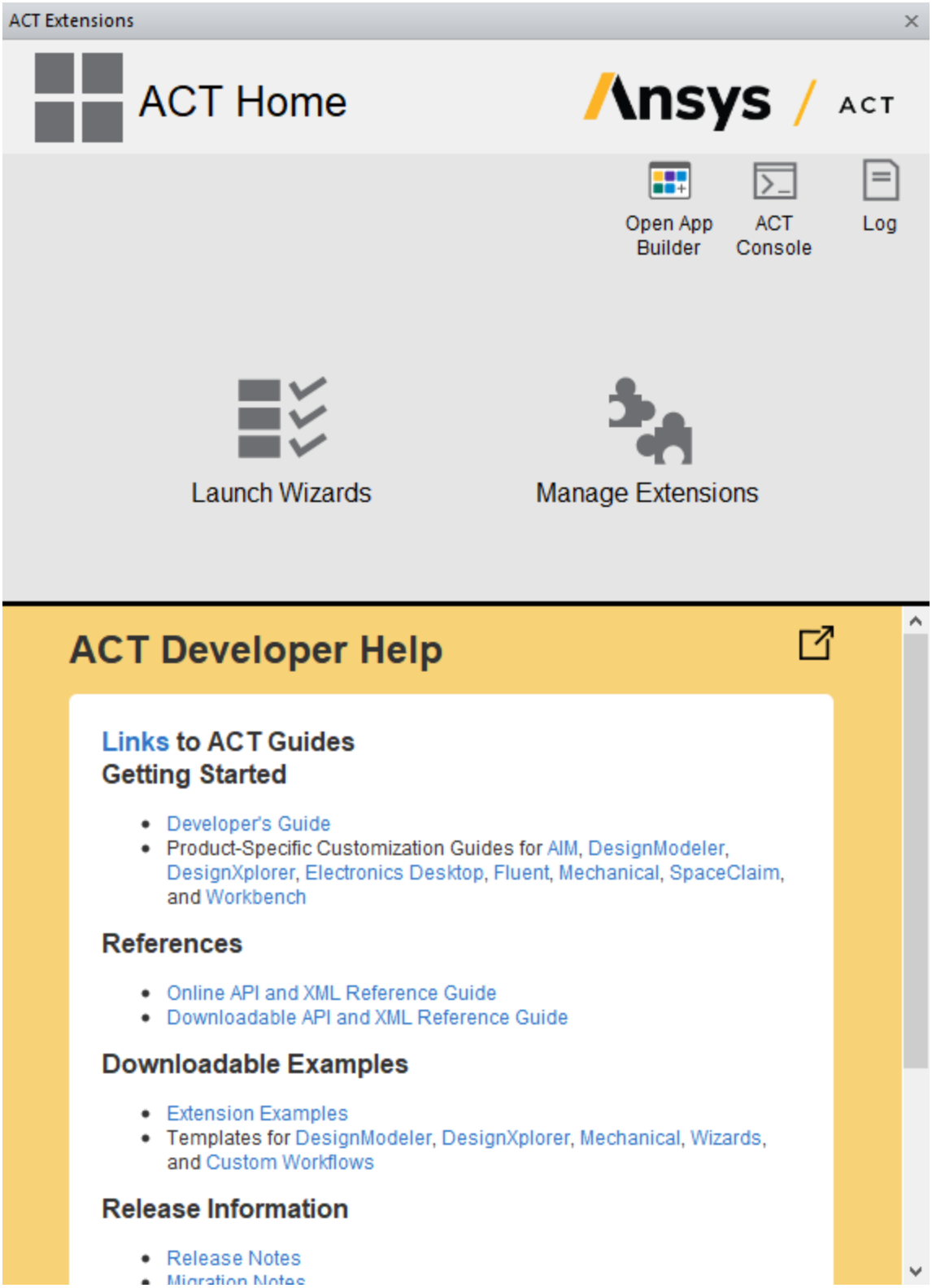
- **Delete** – deletes the selected nets from the project. Members of the net remain in the project.
- **Fit Selected** – zooms the Layout editor into show the selected nets.

ACT Extensions Window

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

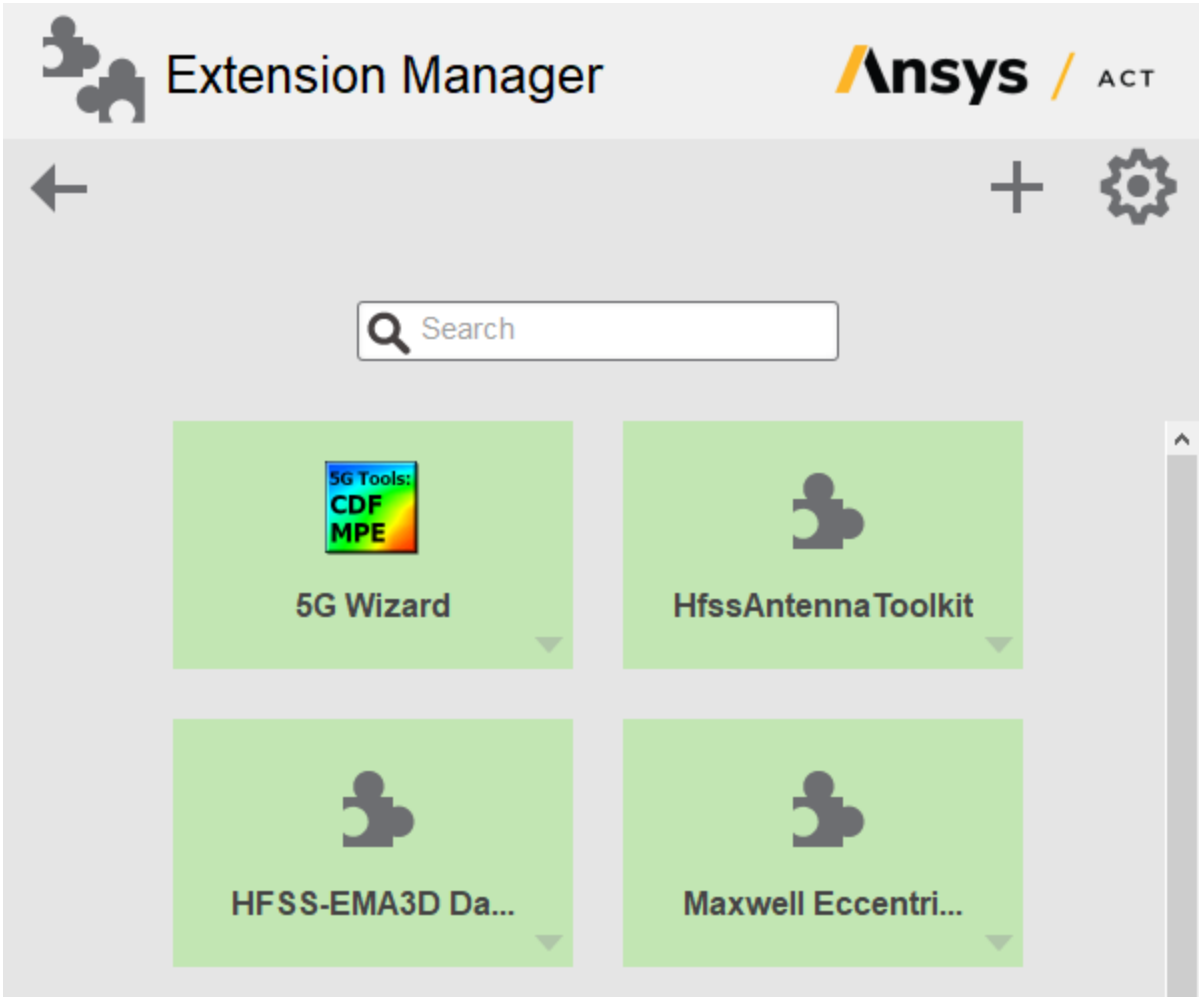
The **View** > **ACT Extensions** command opens the ACT Extensions window for toolkit extensions implemented via Ansys ACT. Depending on how you size and arrange the desktop, the appearance may vary. You may choose to detach and size the window to suit your needs.

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



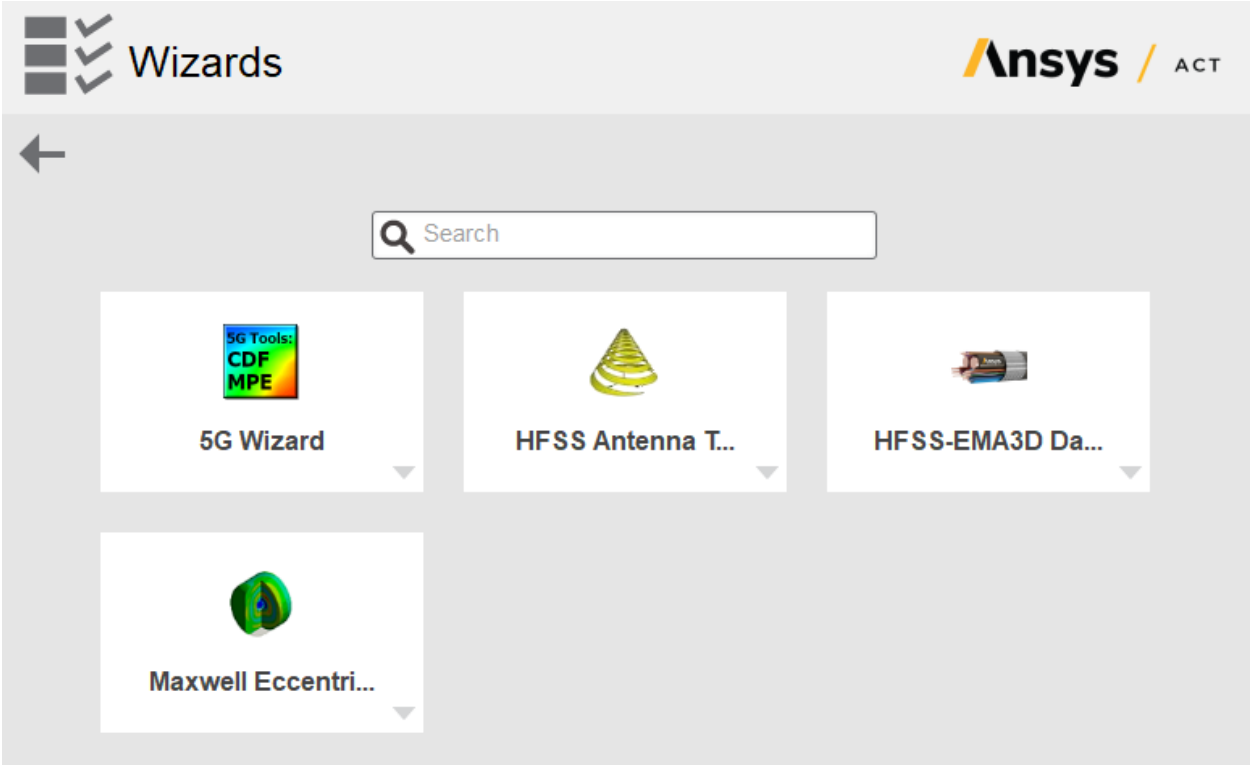
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

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



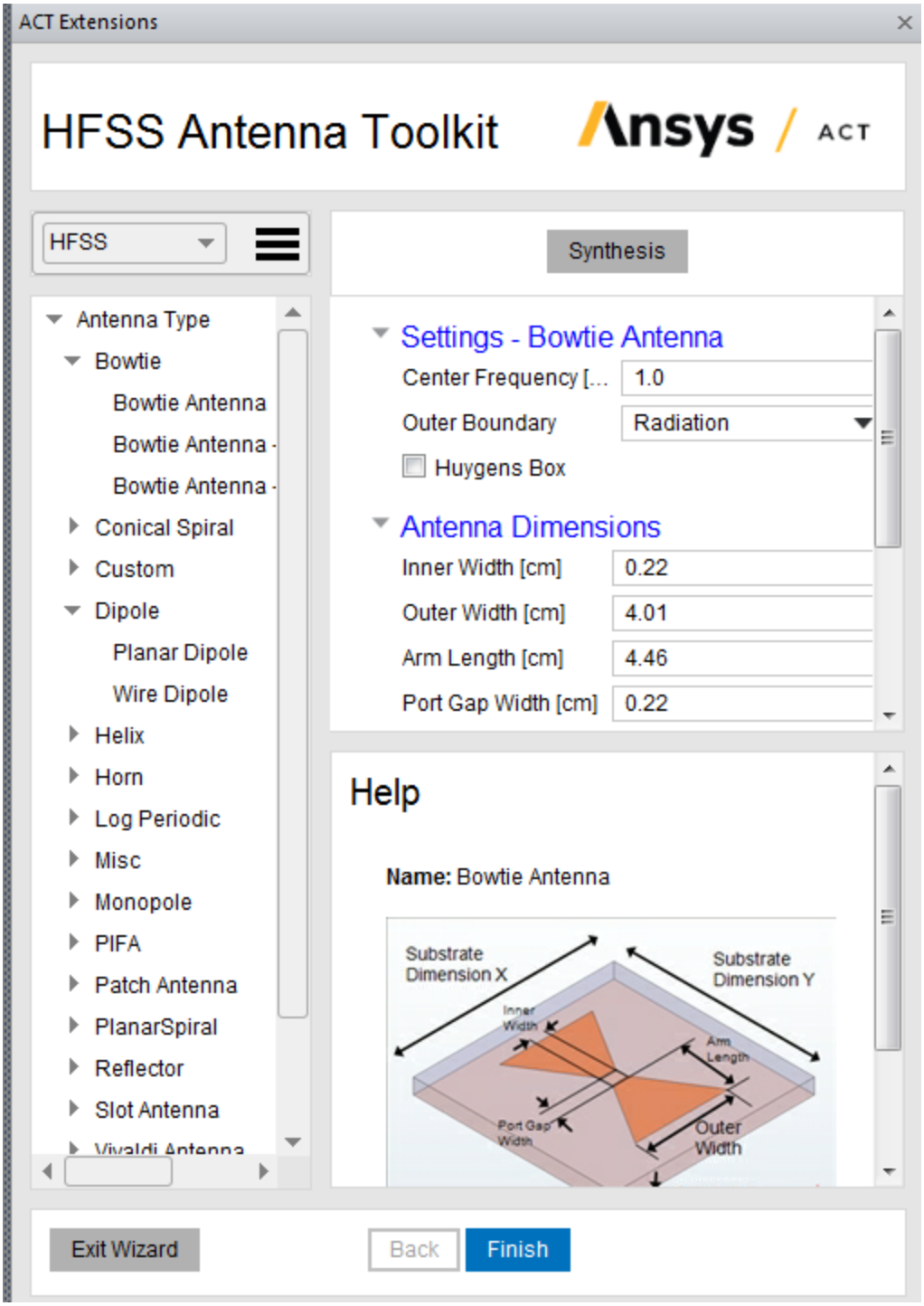
Click the **Launch Wizards** icon to load existing toolkits for activation. If there are many available extensions, you can use the **Search** bar. Search is not case sensitive.

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



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

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



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

Ansys Electronics Desktop Menus

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

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

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

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

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

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

Top Menu Bar

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

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

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

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

Shortcut Menu in the Toolbars Area

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

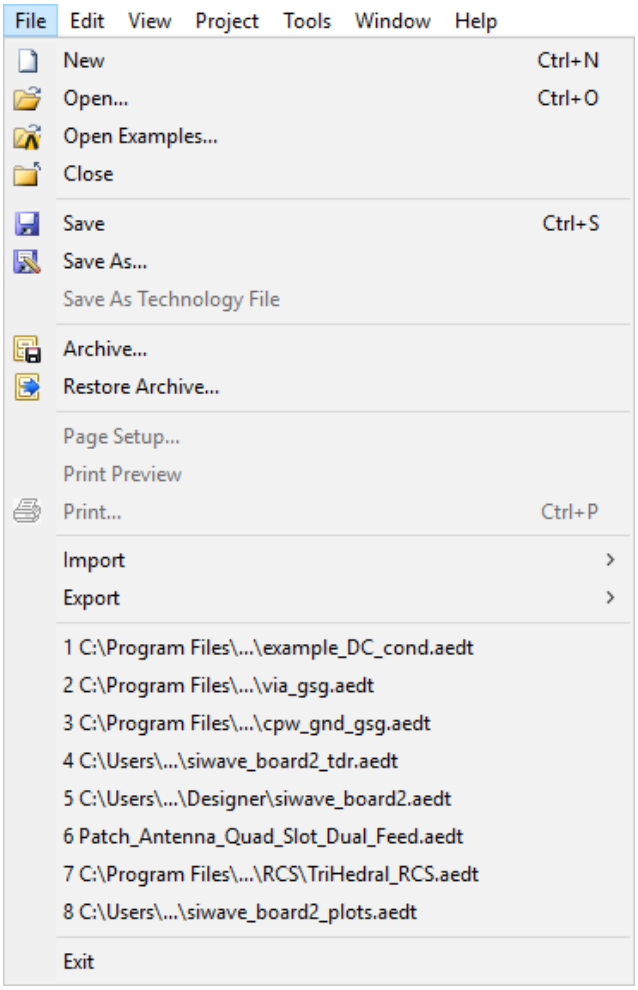
When a project is loaded, from left to right, the Ansys Electronics Desktop drop-down menus are:

File Edit View Project Draw Design-Editor Tools Window Help

These menus are described in the following sections.

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

File Menu



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

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

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

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

Edit Menu

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

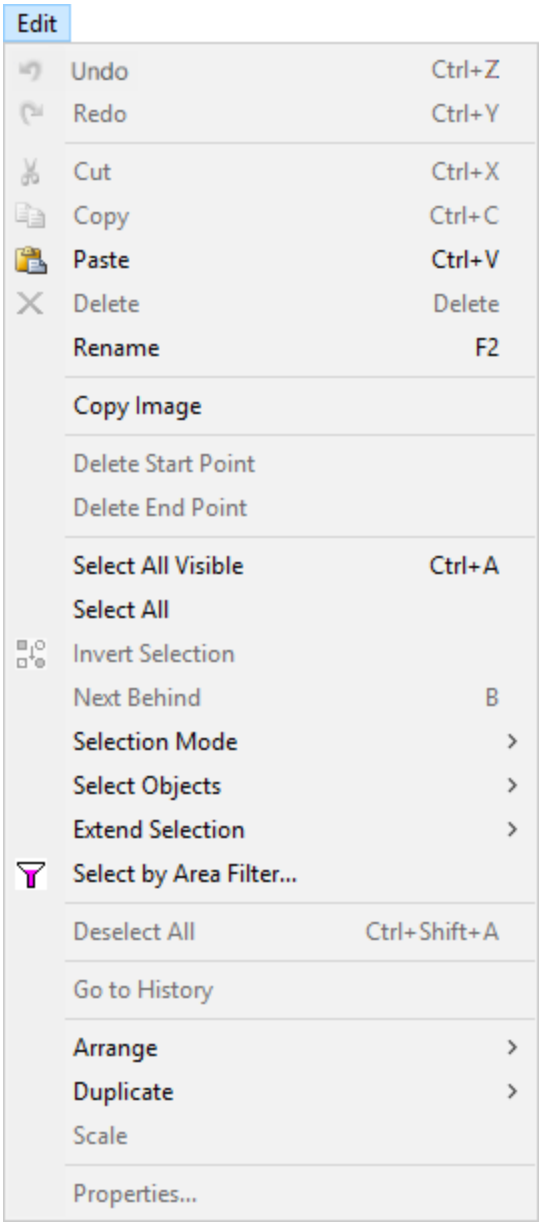
These menus include:

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

Edit Menu for 3D Modeler

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

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

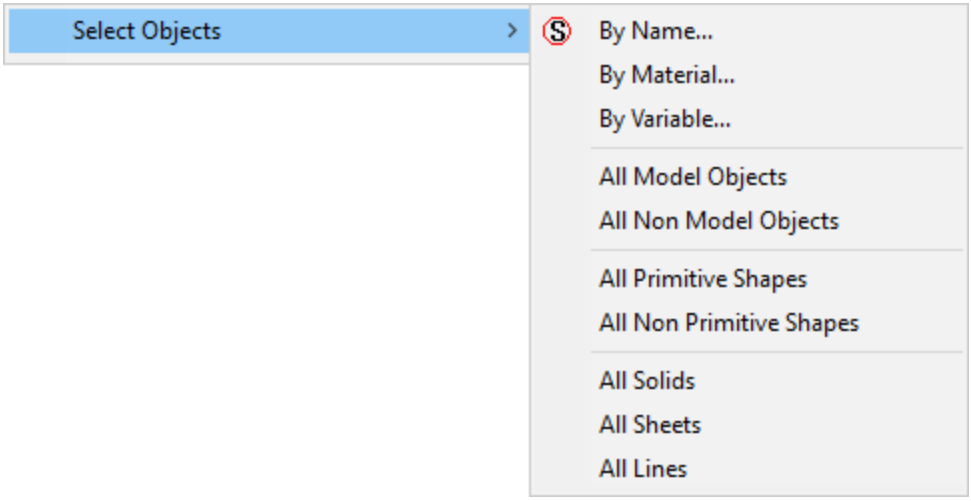


Comands will be available or not available for clicking depending on what is currently selected or on whether any operations have been performed, undone, or redone. The 3D Modeler's Edit menu contains the following options:

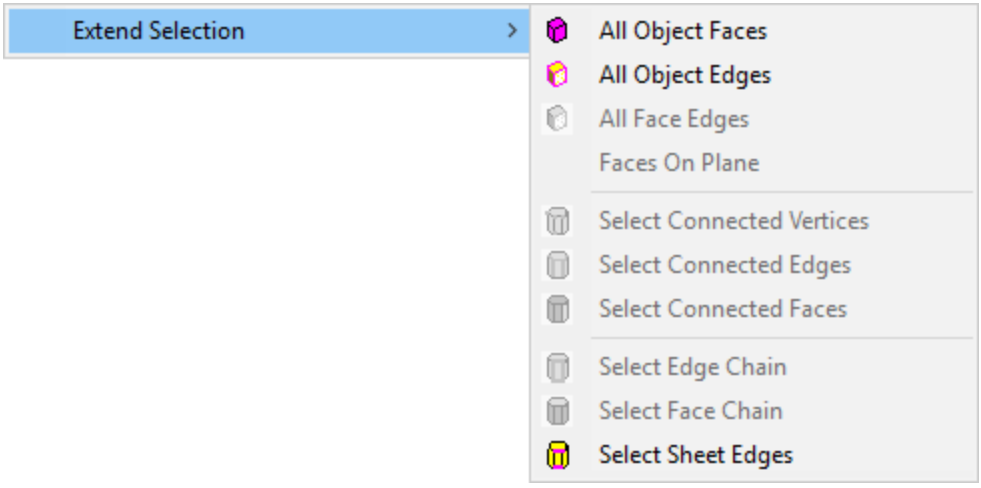
- **Undo** – allows you to undo the last action, which is specified after the word "Undo."
- **Redo** – allows you to redo an undone action, which is specified after the word "Redo."
- **Cut / Copy / Paste** – allows you to copy and paste elements.
- **Delete** – deletes a selected element.
- **Rename**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

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

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



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



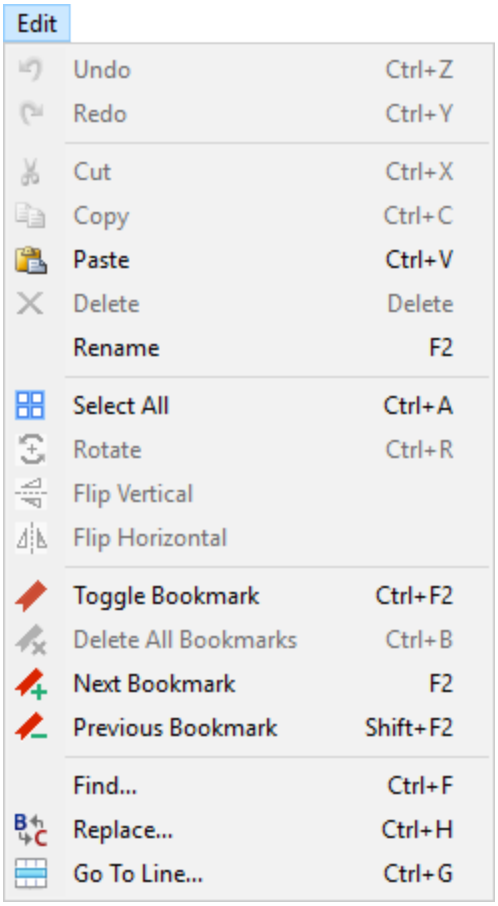
- **Select by Area Filter** – opens the **Select By Area Filters** window.
- **Deselect All** – deselects all selected objects.

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

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

Edit Menu for Netlist Editor

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



The Netlist Editor Edit menu contains the following options:

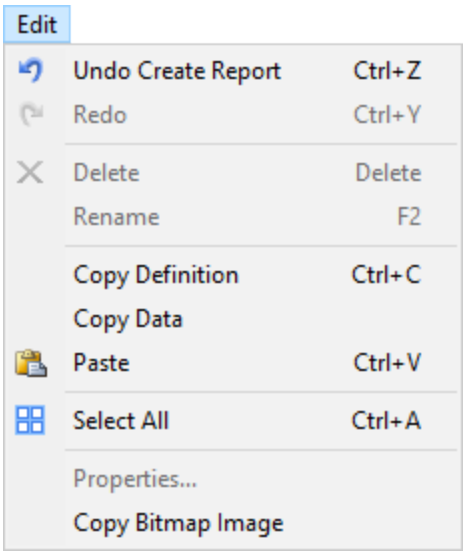
- **Undo** – allows you to undo the last action, which is specified after "Undo."
- **Redo** – allows you to redo an undone action, which is specified after "Redo."
- **Cut / Copy / Paste** – allows you to copy and paste netlist elements.
- **Delete** – deletes a selected element.
- **Rename** – renames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

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

- **Select All** – selects the entire netlist.
- **Rotate / Flip Vertical / Flip Horizontal** – deactivated options.
- **Toggle Bookmark** – toggles the display of bookmarks.
- **Delete All Bookmarks** – deletes all bookmarks.
- **Next Bookmark / Previous Bookmark** – allows you to navigate between bookmarked lines.
- **Find** – searches the netlist for a string of text.
- **Replace** – searches the netlist for a string of text and replaces it with different text that you specify.
- **Go To Line** – allows you to enter a line number and jump to that line.

Edit Menu for Report Window

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



The Report Window Edit menu contains the following options:

- **Undo** – allows you to undo the last action, which is specified after "Undo."
- **Redo** – allows you to redo an undone action, which is specified after "Redo."
- **Delete** – deletes a selected report element.
- **Rename** – renames a selected report element.
- **Copy Definition** – copies the report definition.
- **Copy Data** – copies report data.
- **Paste** – pastes the copied information at your selected location.
- **Select All** – selects everything in the report window.

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

- **Properties** – opens the relevant Properties window for the selected report element.
- **Copy Bitmap Image** – copies the entire report window as a bitmap image that can be pasted into another program or document.

View Menu

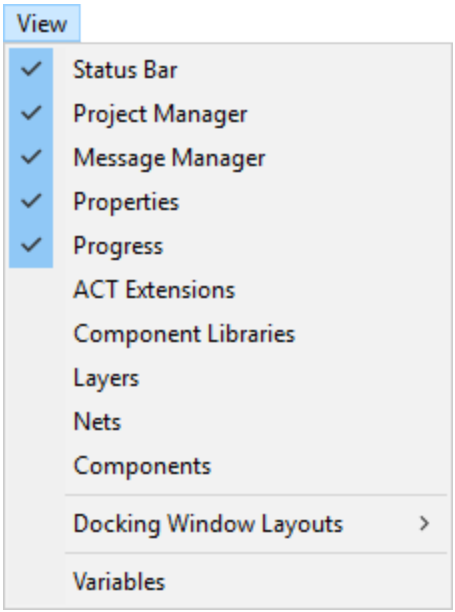
The **View** menu changes, depending on the active window in the Design Area.

These menus include:

- Basic View Menu
- Schematic Editor's View Menu
- Layout Editor's View Menu
- 3D Modeler's View Menu

Basic View Menu

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

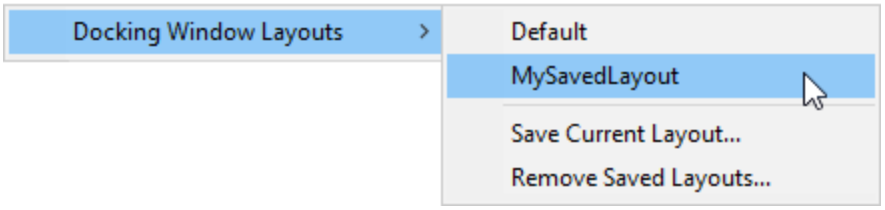


- The check boxes toggle the display of the **Status Bar**, **Message Manager Window**, **Project Manager**, **Properties Window**, and **Progress Window**. You can enable the display of **Component Libraries**, **Layers**, **Nets**, **Components** and **Variables** windows.
- The **ACT Extensions**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

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

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

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

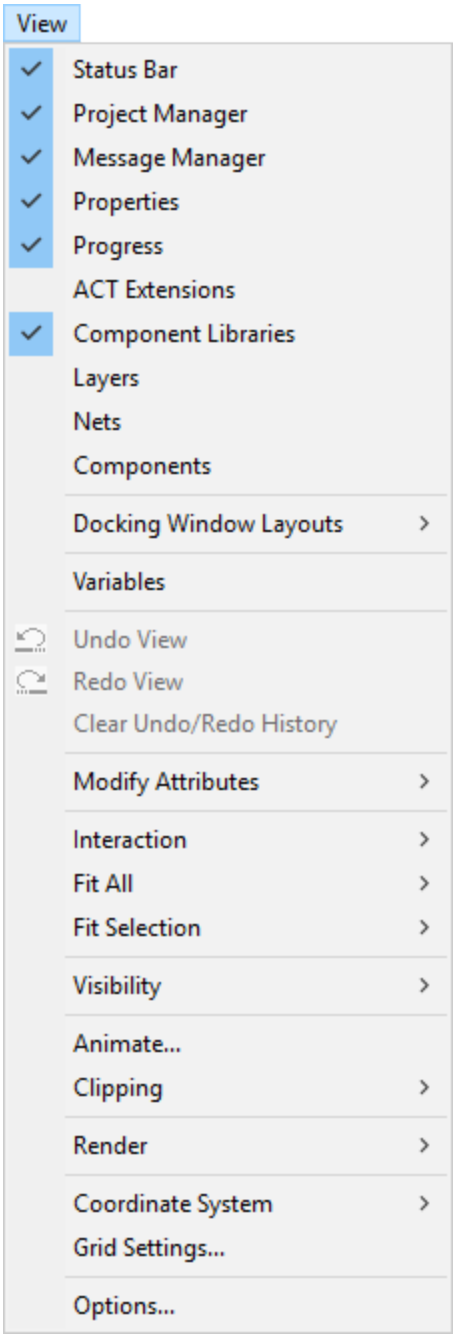


- The **Variables** option opens the **Project** and **Design** **Variables** window, allowing you to set variables.

3D Modeler View Menu

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

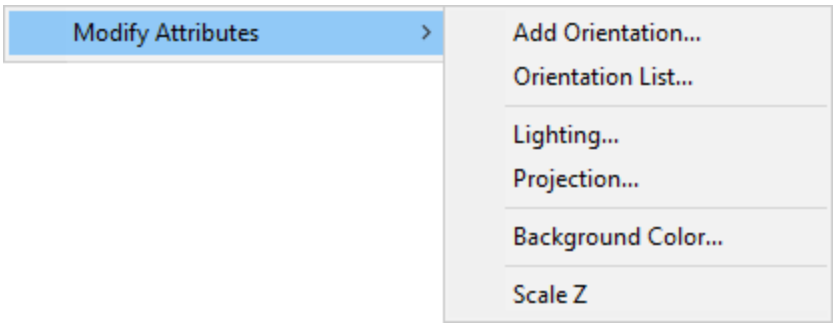
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



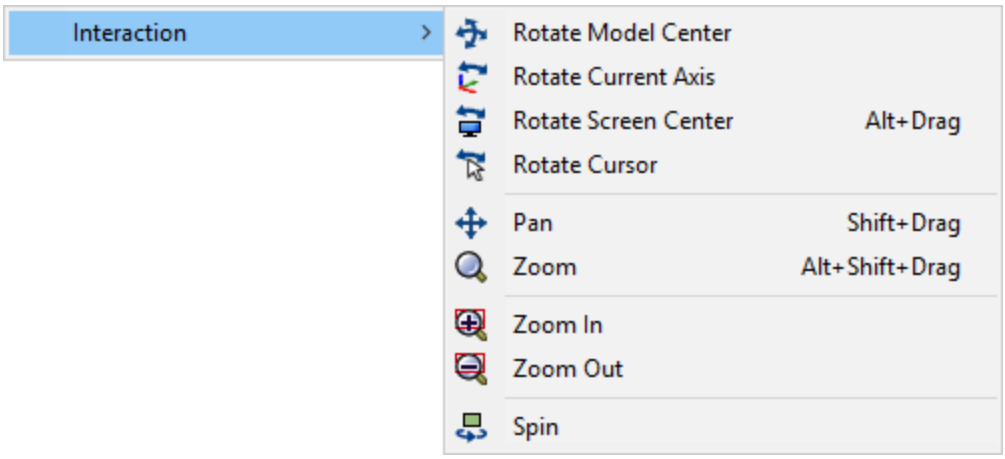
Thi View menu includes the options available on the Basic View Menu. The following additional options are available when a *Modeler* window is active. Note that a number of these options also appear in the View branch of the shortcut menu that appears when you right-click in the Modeler window.

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

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



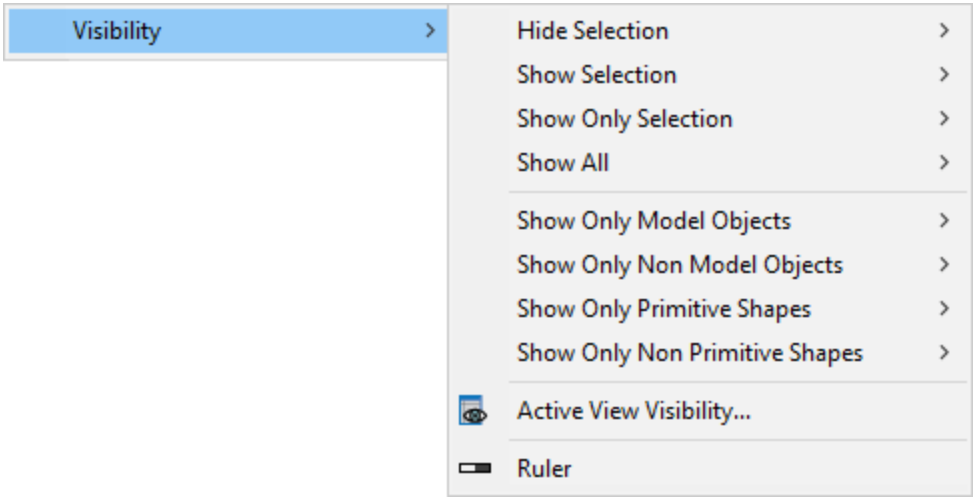
- **Interaction** – offers options to rotate, pan, zoom, and spin the drawing in the design area.



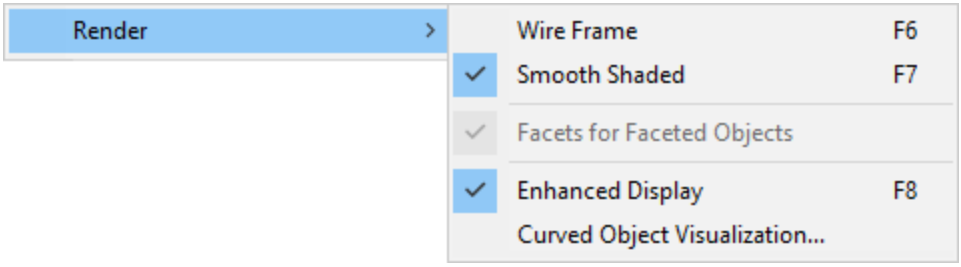
- **Fit All** – allows you to fit all objects to the view window.
- **Fit Selection** – allows you to fit selected objects to the view window.

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

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

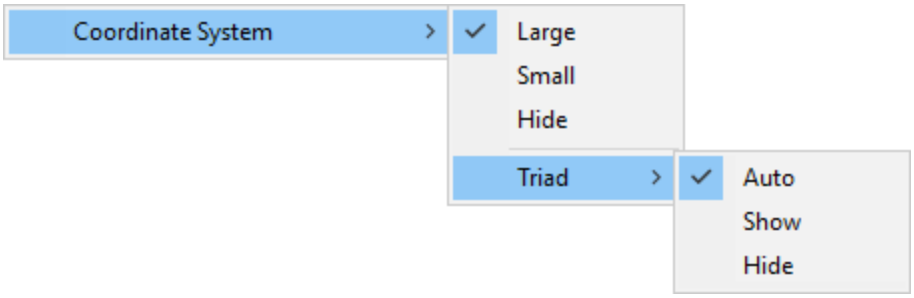


- **Animate** – opens the **Create Animation Setup** window, so that you can begin creating animations.
- **Clipping** – allows you to place a 3D coordinate system manipulator in the design area so that you can define clip planes.
- **Render** – offers options to switch between wire frame and smooth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

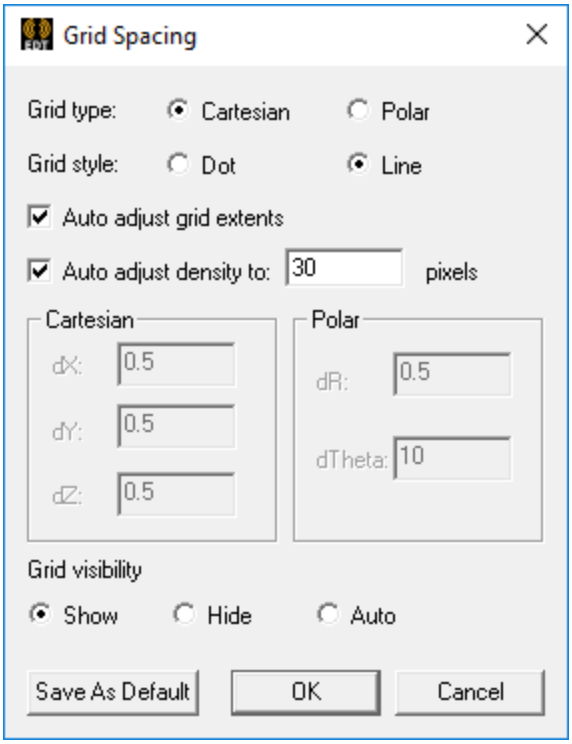


- **Coordinate System** – offers options to change how the coordinate system displays in the design area.

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

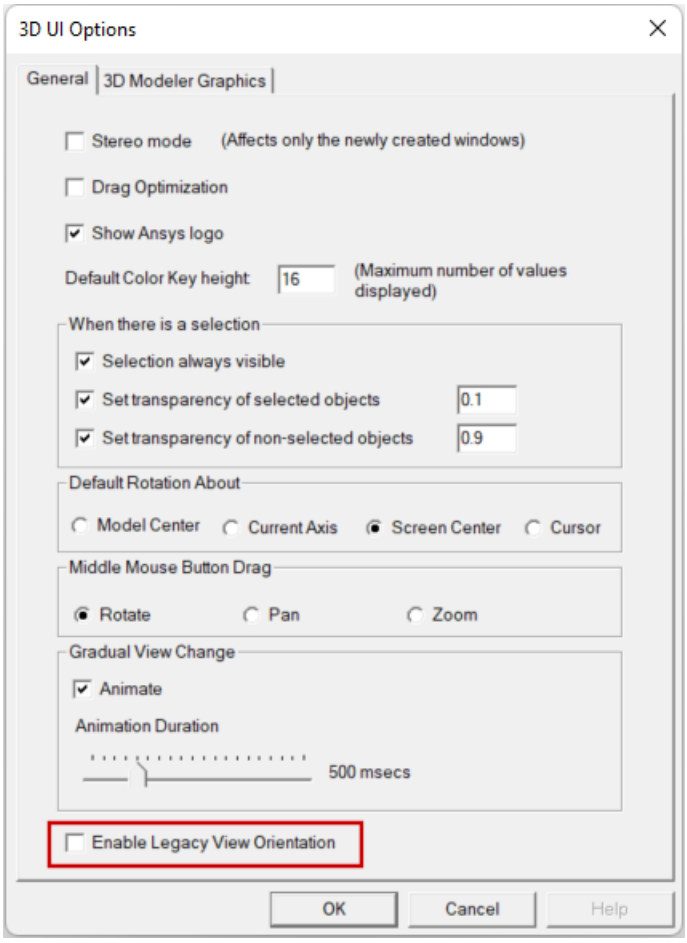


- **Grid Settings** – opens the **Grid Spacing** window, where you can adjust spacing.



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

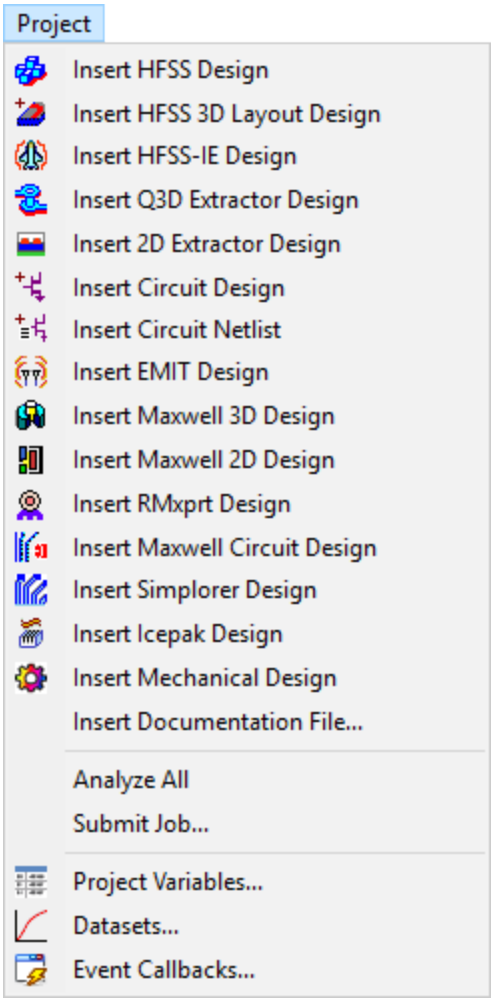
- **Options** – opens the **3D UI Options** window, where you can change additional settings.



Project Menu

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

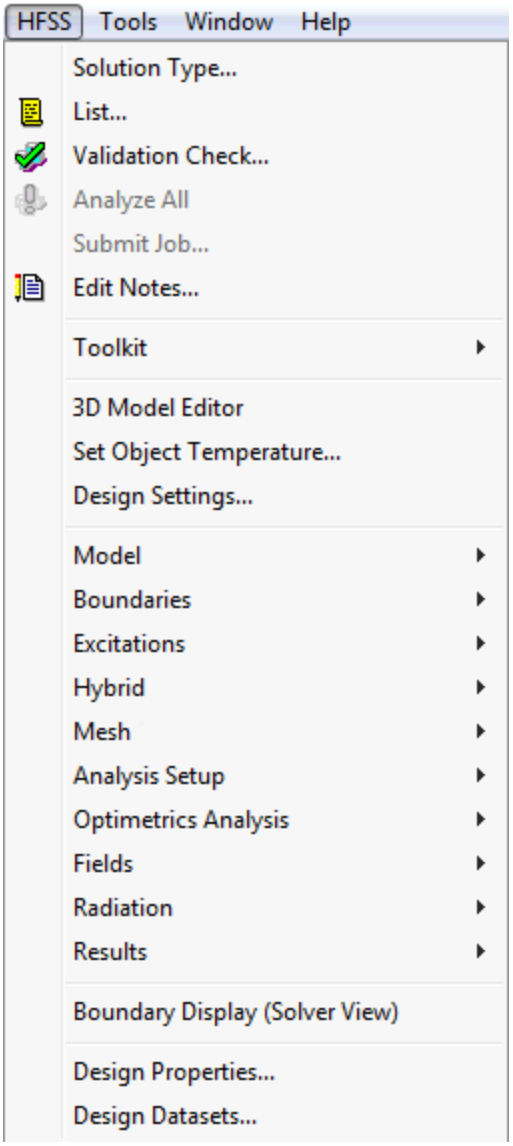
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Editor and Design Specific Menus

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

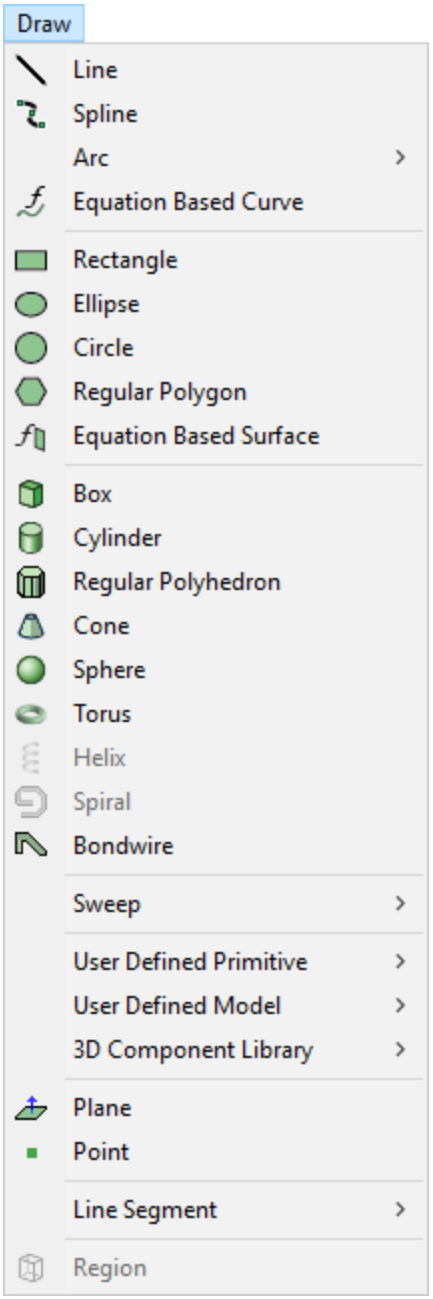
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Layout Editor Draw Menu

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

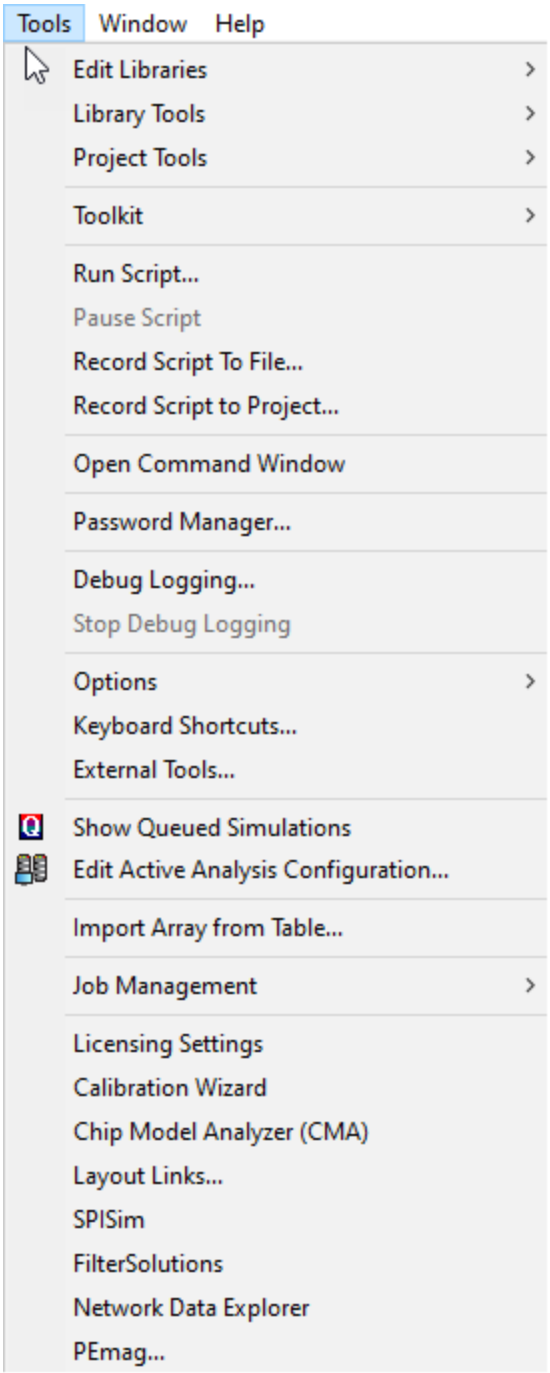
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tools Menu

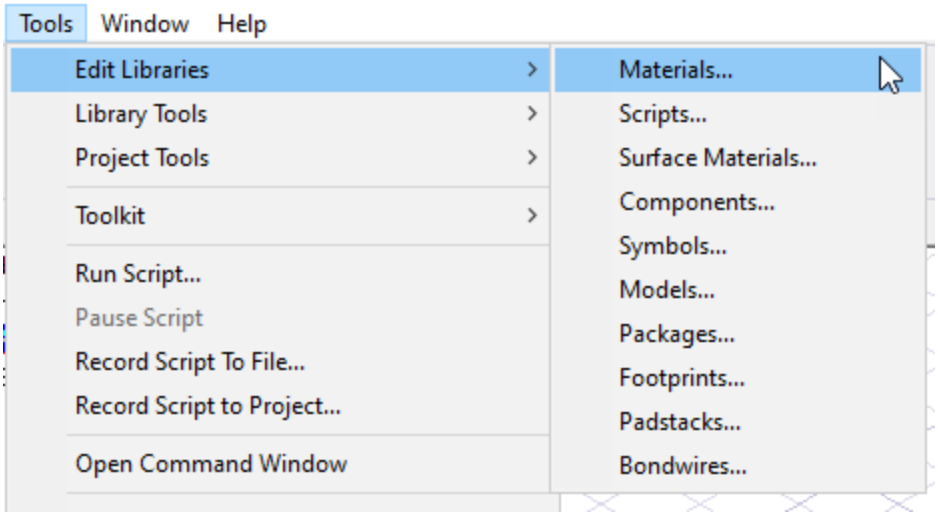
The **Tools** menu contains operations that are common to the analysis tools.

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

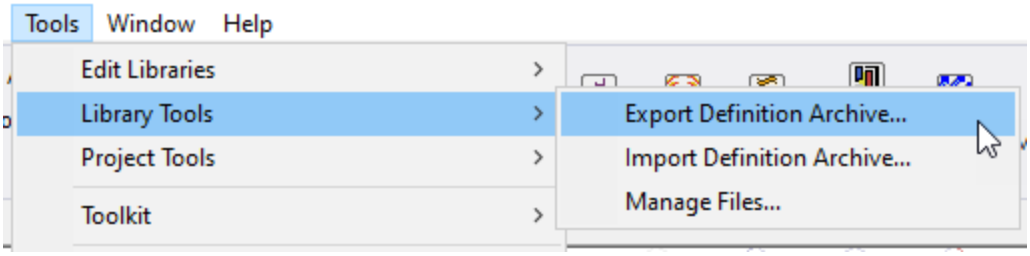


- Click **Tools>Edit Libraries** to view menus allowing access to the libraries available for the various Ansys Electronic Desktop solves, such as Materials, Scripts, and Components, Symbols and Models for Simplorer, Icepak, and Circuit.

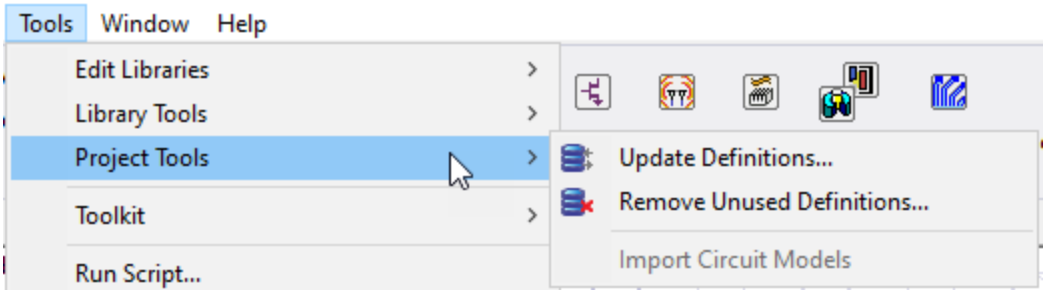
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



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



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



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

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

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

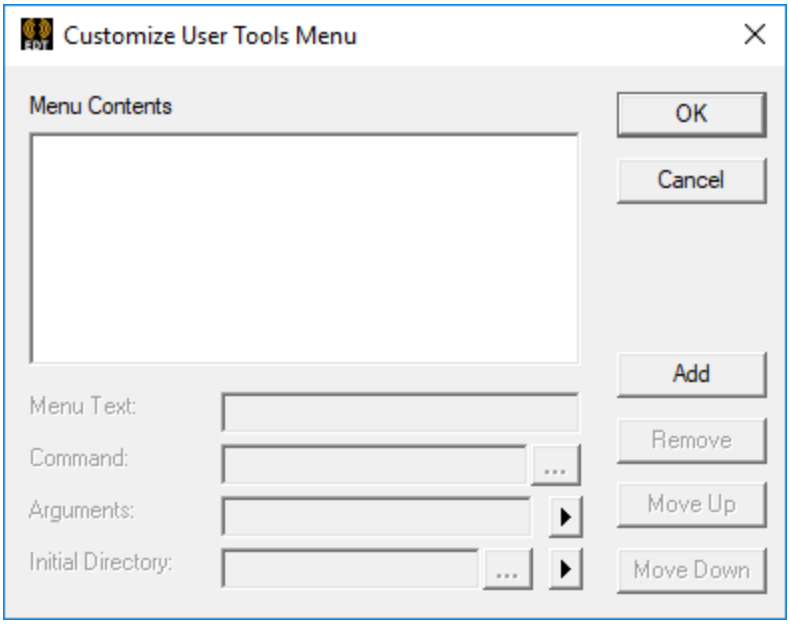
Adding External Tools to the Tools Menu

To add an executable to the Tools menu:

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

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

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



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

   This enables the following fields:

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

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

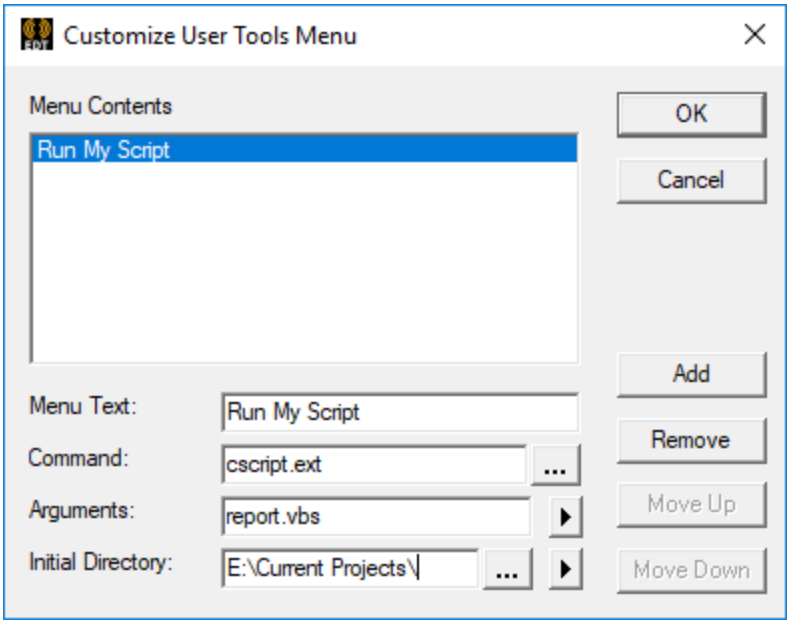
   Arguments field – accepts command arguments from the **>** button menu selections for File Path, File Directory, File Name, File Extension, Project Directory, or Temp Directory.

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

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

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

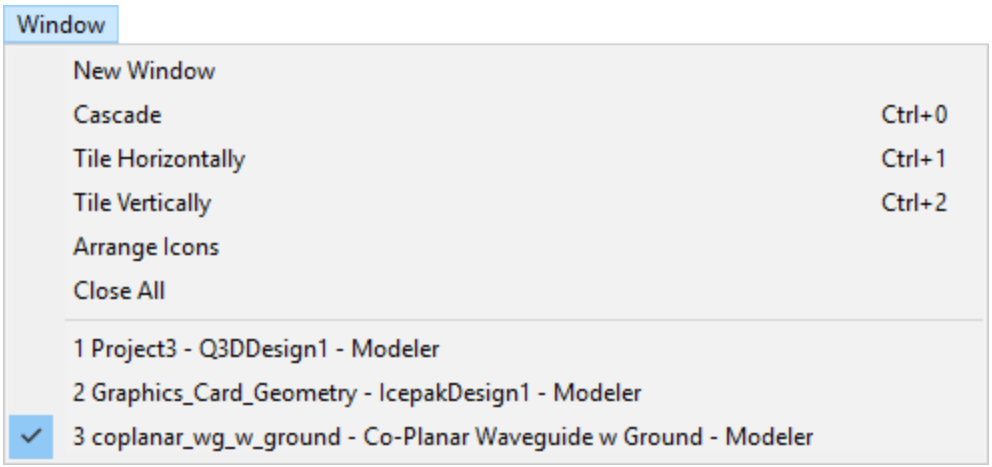
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This example shows the cscript.exe program added to the **Tools** menu as **Run My Script**. The command line argument to the csscript.exe program is report.vbs. You can also name the directory in which it will be run.

Window Menu

The **Window** menu contains common window control operations, as well as a list of projects that are currently open.



- Click **New Window** (or type **n**) to open a new window in the Design Area. The new window will show the active design.
- Click **Cascade** (or type **c**) to arrange the open design windows in overlapping sequence:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use the **Cascade** operation to restore all windows to their default sizes after one or more of them have been maximized (enlarged to fill the entire Design Area).

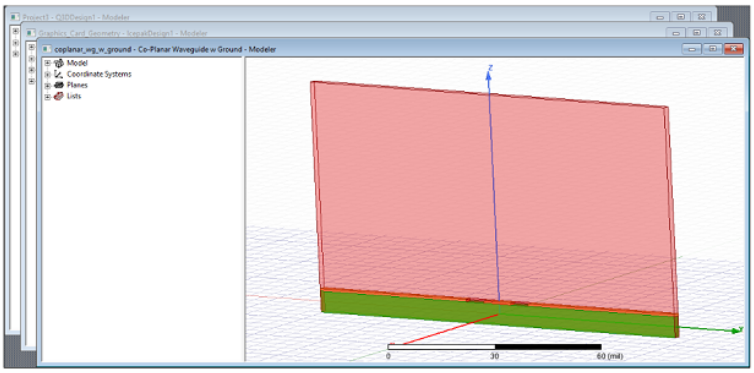
- Click **Tile Horizontally** (or type **h**) to arrange the open design windows in a top-to-bottom sequence:
- Click **Tile Vertically** (or type **v**) to arrange the open design windows in a side-to-side sequence:

> **Tip:**
>
> You can use the shortcut key sequences Ctrl+0, Ctrl+1, and Ctrl+2 to execute the Cascade, Tile Horizontal, and Tile Vertical operations, respectively. The shortcuts can be used at any time, bypassing the Window menu.

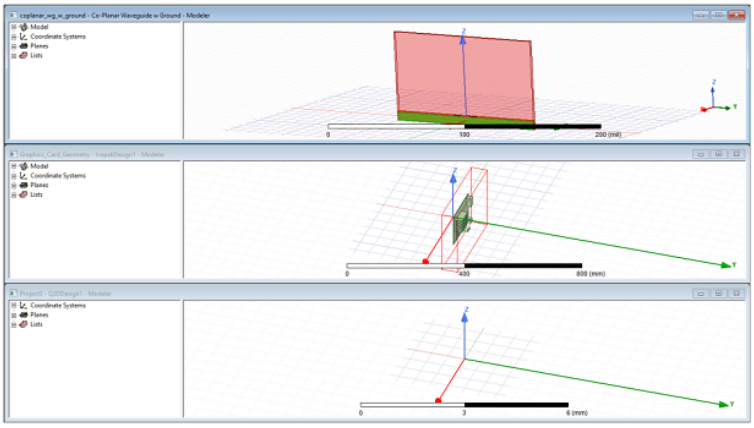
- Click **Arrange Icons** (or type **a**) to restore iconized windows to a neat row at the bottom of the Design Area, after the icons have been repositioned manually. See Working with Editor Windows for details on iconizing editor windows and on repositioning the iconized windows.
- Click **Close All** (or type **l**) to close all the editor windows in the Design Area.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

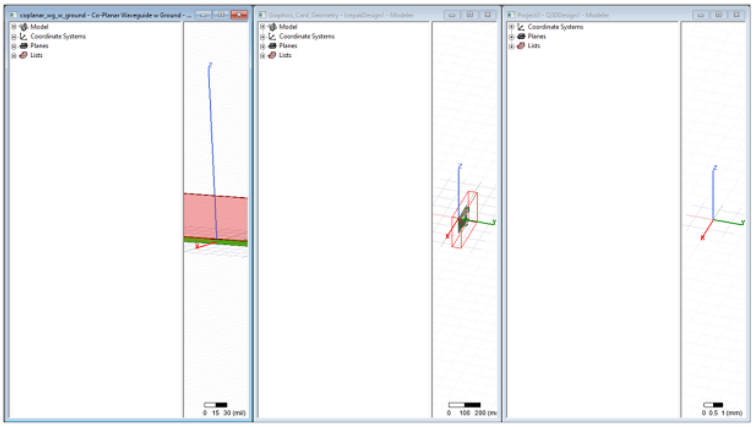
Cascade



Tile Horizontally



Tile Vertically



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Help Menu

Ansys Electronics Desktop features extensive help documentation, including text search and context-sensitive help items. To access the main help system, click **Help** on the top menu bar.

The Help Menu displays different selections depending upon the type of design inserted to the active project. The basic selections for HFSS are:

> **Note:** Ansys Electronics Desktop Student includes access to PDF documentation only.

- **HFSS Help** – opens the HFSS help within the Electronics help system. You can also access PDF versions from within the help system.
- **HFSS Scripting Help** – opens the HFSS scripting help.
- **HFSS Getting Started Guides** – opens a list of links to the HFSS Getting Started Guides, which walk you through projects that demonstrate product features.
- **HFSS PDFs** – provides access to PDFs for HFSS, including the main help, scripting guide, and Getting Started Guides.
- **Ansys Customer Support** – opens a browser page to the Ansys Customer Portal. At the website you can learn more about Ansys products and services and log on to contact Ansys technical support staff.
- **What's New in this Release** – opens a PDF that describes *What's New in Ansys Electronics Desktop* for 2025 R1.
- **Ansys Product Improvement Program** – opens a window describing the Product Improvement Program option.
- **License Settings** – Opens help for the License Settings tool.
- **About Ansys Electronics Desktop** – opens a dialog box that displays the Ansys Electromagnetics Suite release number and contains tabs that show information about the **Installed Components** and **Client License Settings**.
- **Ansys Innovation Courses** – opens a web page containing a wide range of Ansys Electronics Engineering courses using on-demand, self-paced video training and quizzes.
- **Ansys Learning Hub** – opens a web page with subscription based access to, virtual and self-paced learning across the Ansys Software portfolio.
- **Ansys Knowledge** – opens a web page to expert curated knowledge materials from FAQs to tutorials on simulation topics.
- **Ansys Learning Forum** – opens a web page to Ansys blog containing discussion and presentations from Ansys experts, partners and customers.
- **Customer Portal** – opens a web page to Ansys Product support.
- **About Electronics Desktop** - opens a dialog with version and release information, Ansys Electronics Desktop installed components, and licensing information.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Context-Sensitive Help

To access context-sensitive help from the Ansys Electronics Desktop user interface, press **F1** while your cursor is on an item. The help system specific to the product opens.

Obtaining Information about the Software and Release

To obtain information about the software and release:

1. Click **Help** > **About Ansys Electronics Desktop**
   The **About Ansys® Electromagnetics Suite [release number]** dialog box appears, listing information about the product.
2. Click the **Installed Components** tab to view a list of software installed.
3. Click the **Client License Settings** tab to view information about the following:
   - Provider Name
   - Ansys License Version
   - FlexNet Publisher Servers
   - Admin Directory
   - Customer Number
   - FLEXlm Version
   - Redirect Info
4. To export the software information:
   a. Click **Export**.
      The **Save As** dialog box appears.
   b. Browse to the location where you want to save the information as a text file.
   c. Type a name for the file in the **File name** text box. The **Save as type** drop-down menu is already specified as **Export (*.txt)**.
   d. Click **Save**.
5. Click **OK** to close the **About Ansys® Electromagnetics Suite [release number]** dialog box.

Shortcut Menus

A variety of shortcut menus — menus that appear when you right-click a selection or in a window — are available in the **3D Modeler** window, in the **Project Manager** window, in the **History Tree**, and in the **Progress** window.

Shortcut menu in the 3D Modeler window	Use the shortcut menu in the **3D Modeler** window to select, magnify, and move objects (zoom, rotate, etc.); change the view; perform boolean

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	operations; assign materials, boundaries, excitations, or mesh operations to objects; and work with field overlays.
Shortcut menus in the Project Manager window	Use the shortcut menus in the **Project Manager** window to manage Ansys Electronics Desktop project and design files and design properties; assign and edit boundaries, excitations, and mesh operations; add, analyze, and manage solution setups; add Optimetrics analyses; create post-processing reports; insert far- and near-field radiation setups; edit project definitions; and run Maxwell SPICE.
Shortcut menus in the History Tree	Use the shortcut menus in the **History** tree to expand or collapse groupings. If you select particular objects in the history tree, the shortcut menu lists the commands that you can apply to the selected object(s).
Shortcut menus in the Progress window	Use the shortcut menus in the **Progress** window during a simulation to **Abort** or **Clean Stop**.

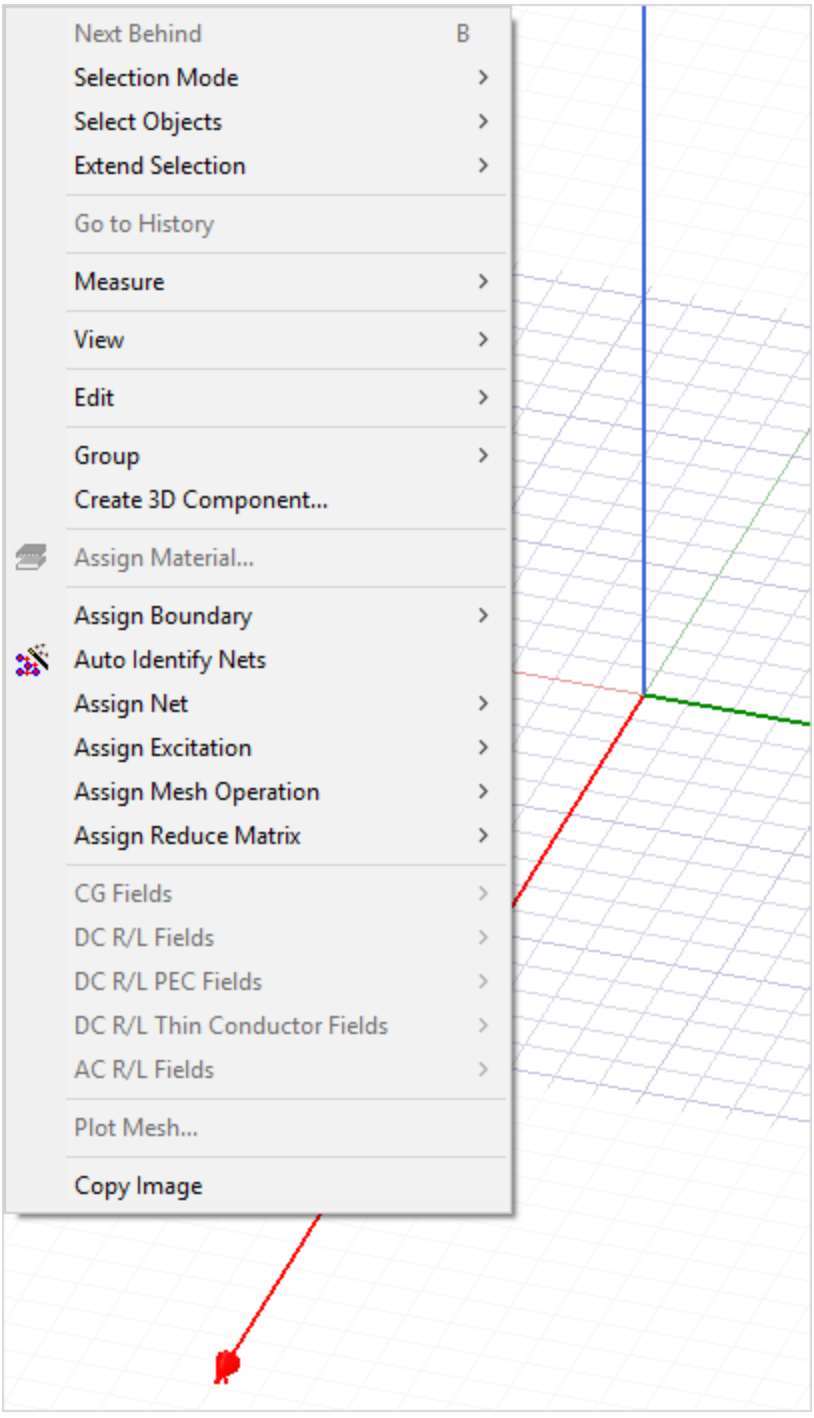
> **Note:**
>
> All commands available on shortcut menus are also available from the menu bar.

Shortcut Menu in the 3D Modeler Window

Use the shortcut menu in the **3D Modeler** window to select, magnify, and move objects (zoom, rotate, etc.); change the view; perform boolean operations; assign materials, boundaries, excitations, or mesh operations to objects; and work with field overlays.

To access the shortcut menu, right-click in the 3D Modeler window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

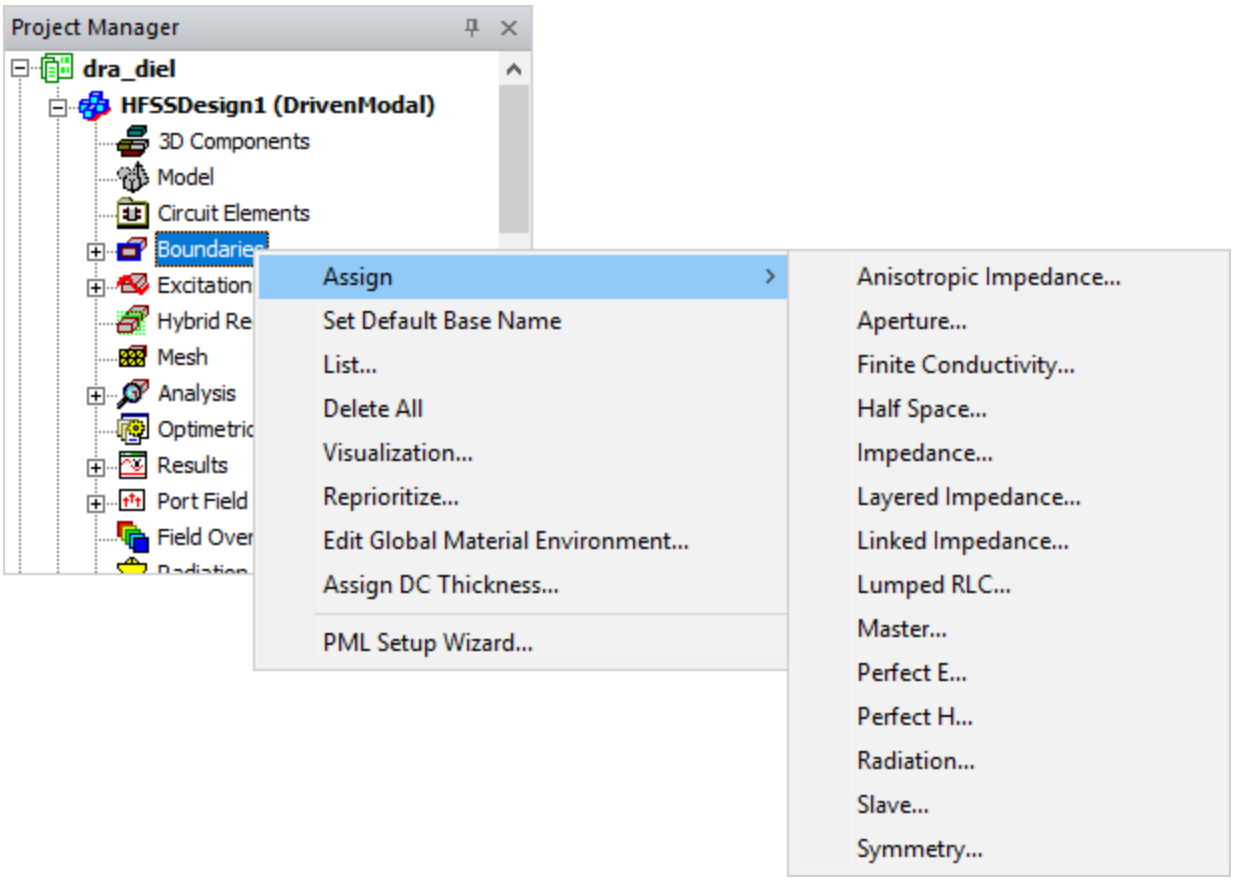


Shortcut Menus in the Project Manager Window

Each node, or item, in the project tree has a shortcut menu. For example, from the shortcut menu for the **Boundaries** icon, you can assign boundaries to selected objects; review

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

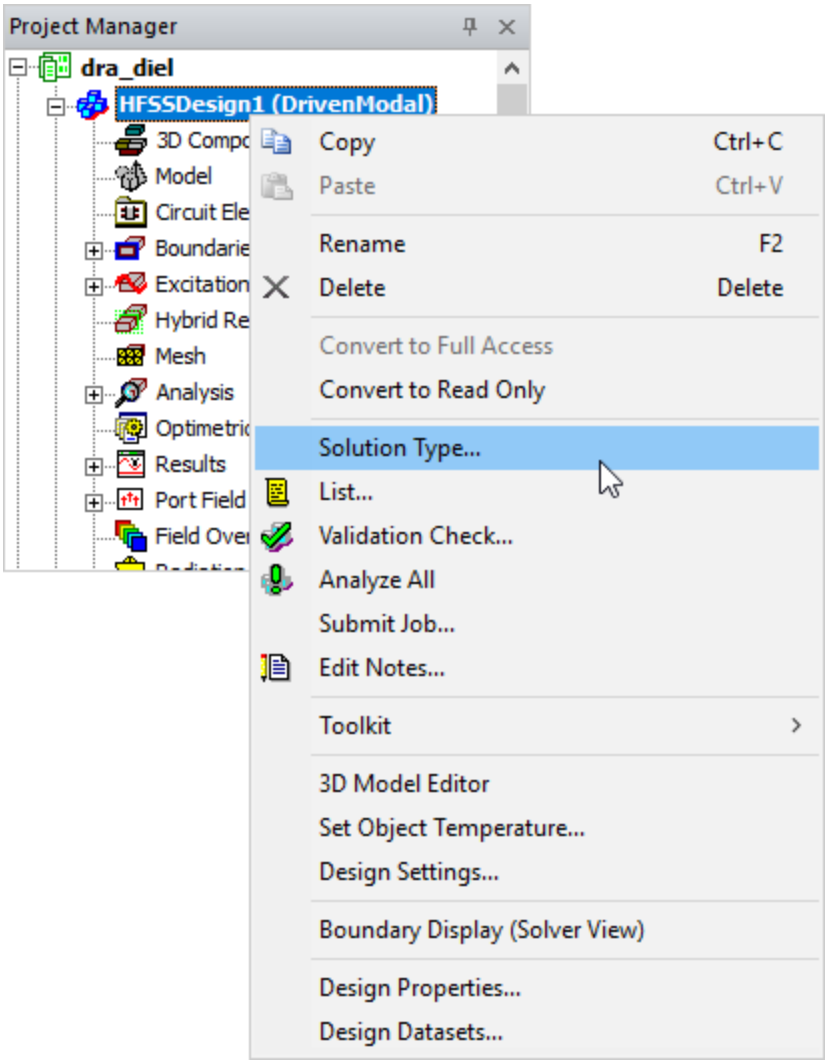
information for all the boundary assignments for the active design; remove all boundary assignments; show or hide a boundary's geometry, name, or vectors; change the priority of a previously assigned boundary; and use the PML Setup wizard to create a perfectly matched layer (PML) boundary.



Other nodes have shortcut menus appropriate for the context.

For example, the following figure shows the shortcut menu when you right-click the design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Customizing Ansys Electronics Desktop Menus

Ansys Electronics Desktop comes with a default Top Menu Bar. You can modify the top menu bar by using customized User Interface (UI) setups to add, remove, rename, and relocate commands. You can also add commands that execute external scripts or configure right-click menus for folders and items in the Project Tree, which allows for extensive customization of the Ansys Electronics Desktop Menus.

Customized setups are implemented using named subfolders that contain XML files used to configure customized menu settings. The subfolders included with Ansys Electronics Desktop are:

- config/UI/ElectronicsDesktop/EM
- config/UI/ElectronicsDesktop/RF

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- config/UI/ElectronicsDesktop/RF.0
- config/UI/ElectronicsDesktop/SI
- config/UI/ElectronicsDesktop/SI1.0
- config/UI/ElectronicsDesktop/SI2.0
- config/UI/ElectronicsDesktop/Twin Builder

Each customized UI type has its own subfolder. Within these subfolders, XML files hold the menu configurations as well as the right-click menus for that UI type.

Select Different UI Types

To change the menu display from the default UI to a different UI type:

- Click **Tools** > **Options** > **General Options** > **General** > **Desktop Configuration**.
- Choose the UI type that you wish to use from the **Custom Menu Set** drop-down menu and click **OK**.

To switch from a customized UI type to the default UI:

- Click **Tools** > **Revert To Default UI**.

Add a New Customized UI Type

A new folder needs to be added to the config/UI/ElectronicsDesktop folder for any new customized UI type. All XML files for this UI type must go in this folder. The new UI type will appear in the **Custom Menu Set** drop-down menu (**Tools** > **Options** > **General Options** > **General** > **Desktop Configuration**).

For any UI type, Ansys Electronics Desktop displays the default UI menus for any products/contexts that are not in the xml files. If there is error processing any XML file, the default UI menu is displayed for that product. Check the message window for the names of problematic XML files and suggestions on how to fix them.

XML files are only processed once when you first switch to that UI type. After you make changes to any XML file, in order for it to be reprocessed, navigate to **Tools** > **Options** > **General Options** > **General** > **Desktop Configuration** and reselect the UI type from the **Custom Menu Set** drop-down menu. Then click **OK**.

Names of the XML Files

Below is the list of XML files that can be placed in a folder for a new UI Type. If any of these XML files does not exist in this folder, the default menu setting displays for that product.

- 2D Extractor.xml – used for 2D Extractor projects.
- Circuit Design.xml – used for Circuit Design projects.
- Circuit Netlist.xml – used for Circuit Netlist projects.
- EMIT.xml – used for EMIT projects.
- HFSS 3D Layout Design.xml – used for HFSS 3D Layout Design projects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- HFSS.xml – used for HFSS projects.
- HFSS-IE.xml – used for HFSS-IE projects.
- Icepak.xml – used for Icepak projects.
- Maxwell 2D.xml – used for Maxwell 2D projects.
- Maxwell 3D.xml – used for Maxwell 3D projects.
- Maxwell Circuit.xml – used for Maxwell Circuit projects.
- NoDesignUI.xml – used for menu settings when no project is selected.
- Q3D Extractor.xml – used for Q3D projects.
- RightClickMenu.xml – used for all right-click menu settings.
- Twin Builder.xml – used for Twin Builder projects.

Valid XML Elements and Attributes

1. Root Element is **DesignerMenu**:
   - **xmlns** – required attribute which needs to be set to the following:

     <DesignerMenu xmlns="http://www.ansys.com/uiConfigMenu">

   - **UseProjectWindowSelectionContext** – child of DesignerMenu, appears zero or one time.

     > **Note:**
     >
     > This is only meaningful if it is used in NoDesignUI.xml. If this is set to true, then clicking in the Project Window on the Project icon or the Definitions icon (or a subitem) will show “Project” context in NoDesignUI.xml instead of active design context. No setting or setting this to false makes this UI type behaves the same as the default behavior of Ansys Electronics Desktop. The menus and toolbars are always shown for the active design unless there’s no design at all.

   - **Context** – child of DesignerMenu. It appears at least one time and has a required “name” attribute. For details on setting the context name, see the “Context Name” section below.
2. Child elements of **Context**:
   - **TopMenu** – For RightClickMenu.xml, do not use this element. You can specify child elements of TopMenu listed below under Context. For any other xml files, you need at least one TopMenu child element for any Context.

     Child elements of TopMenu:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **MenuName** – required, appears only one time.
- **popupMenu** – may appear one or multiple times; child elements are the same as those of Topmenu.
- **LeafMenu** – may appear one or multiple times.

  Child elements of **LeafMenu**:

  - **MenuName** – optional string.

    Pay attention to the character reference "&" in XML. In order for the name of the menu to be displayed as “Tools”, the XML syntax must be:

    <MenuName>&Tool</MenuName>

    “&“ is the character reference for ”&” in XML, while “&Tool” tells Ansys Electronics Desktop that Alt+T is the shortcut key for this menu.

  - **MenuID** – optional number.

    For a list of valid MenuIDs, please see ***Command IDs for Customizing AEDT Menus.xlsx*** under <*install_dir*>/v<*release_number*>/[Win64 or Linux64]/Help.

  - **ShowBitMap** – optional, “Yes” or “No” (default)
  - **Accelerator** – optional string (Example: Ctrl+N)

  > **Note:**
  >
  > To add a new LeafMenu, both “MenuName” and “MenuID” are required.

- **CustomMenu** – may appear one or multiple times; used to add a customized menu to run an external script (vbs or python).

  Child elements of **CustomMenu**:

  - **MenuName** – see LeafMenu for Usage
  - **ShowBitmap** – see LeafMenu for Usage
  - **Accelerator** – see LeafMenu for Usage
  - **ScriptPath** – required string; used to supply path to the script (use of $PROJECTDIR, $PERSONALLIB, $USERLIB, $SYSLIB variables is also allowed).

    Example:

    <ScriptPath>C:/Users/jwei/Python/HelloWorld.vbs</ScriptPath>

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or

<ScriptPath>$PERSONALLIB/HelloWorld.py</ScriptPath>

> **Note:**
>
> To add a new CustomMenu, both MenuName and ScriptPath are required.

- **Separator** – may appear one or multiple times; has no child element.

3. All elements have an optional attribute of **action**.

   Valid values are:

   - **add** – can be applied to all elements other than “DesignerMenu” or “Context” . This element and all its child elements will be added to its parent. An optional attribute of “position” can be followed here to specify the position of this newly added menu. Position starts from 1. If no position is specified, this menu will be appended to its parent menu.
   - **useDefault** – default menus will be used for this menu(matched by MenuName or MenuId). Any of it’s child menu will be processed according to its action setting.
   - **delete** – can be applied to all elements other than “DesignerMenu” or “Context” . This menu (matched by MenuName or MenuID) will be deleted from its parent.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

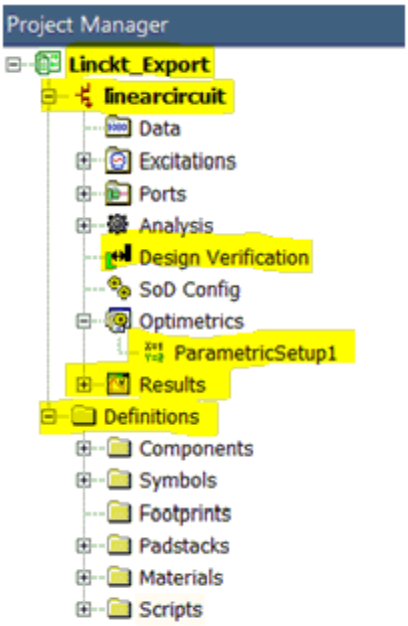
- For “DesignerMenu” and “Context”, only no action setting or “useDefault” is valid.
- The “DesignerMenu” action setting is significant because it is how you specify whether you want to construct your own menus or modify existing default menus. If this is set to “useDefault”, default menu settings are used for any menus not listed in your XML files. If no action is set, only the “File”, “Window” and “Help” menus display; no other default menus display. Menus listed in the XML file display after the “File” menu and before the “Window” menu. In this case, no action is needed for any other elements in the XML file. If any action is specified for any element, you will receive an error message and a default menu will be displayed for that context. A “Revert To Default” menu will be appended to the “Tools” menu. If no “Tools” menu is specified as a TopMenu, a “Tools” menu is created before the “Window” menu with one “Revert To Default” menu item.
- For any other elements, a missing action setting means that the parent action will be used.
- If any element’s action is set to “add”, then no action is needed for any of its child elements. If any action is specified, it will be ignored. This menu and all of its child menus will be added to the default menu.
- If any TopMenu or pop-upMenu’s action is set to “delete”, you don’t need to list any of its child menus. The menu will be deleted with all child menus.

Context Name

1. The “name” attribute specifies the name of the context for this product. You can have an "All" context if you set the action of DesignerMenu to UseDefault to specify a menu setting that you want to apply to all contexts for that product.
2. Valid context name for different xml files:
    - 2D Extractor.xml — “All”, “3d modeler”, “report2d”
    - CircuitDesign.xml — “All”, "SchematicEditor", “Layout”, “Netlist”, “report2d”
    - Circuit Netlist.xml —“All”, “Design”, “Netlist”
    - HFSS.xml — “All”, “3d modeler”, “report2d”
    - HFSS 3D Layout Design.xml — “All”, “Design”, “Layout”, “Layout3D Editor”, “report2d”
    - HFSS-IE.xml — “All”, “3d modeler”, “report2d”
    - NoDesignUI.xml — “All”, “No Context”, “project”, “FilterDesign”
    - Q3D Extractor.xml — “All”, “3d modeler”, “report2d”

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- RightClickMenu.xml — the following figure illustrates valid context names and how they display in Ansys Electronics Desktop.



  - **Project Folder** – "Project"; in the above figure, “Project” represents the “Linckt_Export” project folder.
  - **Design Instance Folder** – “Circuit Design”, “HFSS 3D Layout Design”, “HFSS”, “Q3D Extractor”, “Circuit Netlist”, “HFSS-IE”, “2D Extractor”
  - **Definitions Folder** – “Definitions”; in the above figure, the Definitions folder is highlighted.
  - For any Folder or Item under the Design Instance or Definitions folder, use the “/” notation to specify the path to the Folder or Item for context name. In the above figure: Use “Circuit Design/Results” for the Results folder. Use “Circuit Design/Design Verification” for the Design Verification folder.
    - For any folder or item under this, use the context name for the folder appended with “/Item”. In the above figure, “Circuit Design/Optimetrics/Item” represents ParametricSetup1.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> For the Report folder under Results, please use the context name for the Results folder appended with "Report"; for example: "Circuit Design/Results/Report".
>
> For the Trace folder under Report, please use the context name for the Report appended with "Trace"; for example: "Circuit Design/Results/Report/Trace"

Sample XML Files

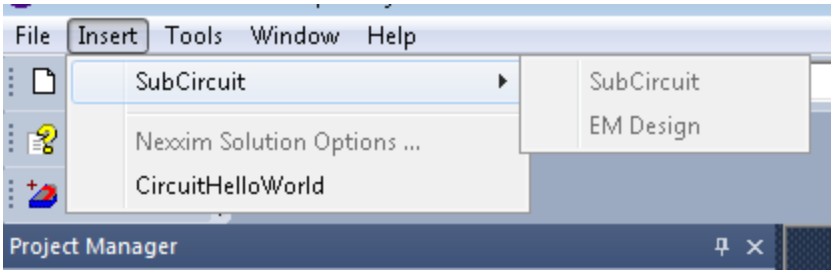
1. **Circuit Netlist.xml Sample**

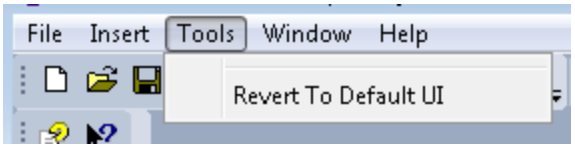
```
<DesignerMenu xmlns="http://www.ansys.com/uiConfigMenu">
   <Context name="Design">
      <TopMenu>
         <MenuName>Insert</MenuName>
         <pop-upMenu>
            <MenuName>S&amp;ubCircuit</MenuName>
            <LeafMenu>
               <MenuName>&amp;SubCircuit</MenuName>
               <MenuID>55500</MenuID>
               <ShowBitMap>No</ShowBitMap>
               <Accelerator></Accelerator>
            </LeafMenu>
            <LeafMenu>
               <MenuName>&amp;EM Design</MenuName>
               <MenuID>55502</MenuID>
               <ShowBitMap>No</ShowBitMap>
               <Accelerator></Accelerator>
            </LeafMenu>
         </pop-upMenu>
         <Separator></Separator>
         <LeafMenu>
            <MenuName>Nexxim Solution &amp;Options ...</MenuName>

            <MenuID>38460</MenuID>
            <ShowBitMap>No</ShowBitMap>
            <Accelerator></Accelerator>
         </LeafMenu>
         <CustomMenu>
            <MenuName>CircuitHelloWorld</MenuName>
            <ScriptPath>$USERLIB/HelloWorld.vbs</ScriptPath>
            <ShowBitMap>No</ShowBitMap>
         </CustomMenu>
      </TopMenu>
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    </Context>
</DesignerMenu>
```

Menus created by processing the above sample, Circuit Netlist.xml





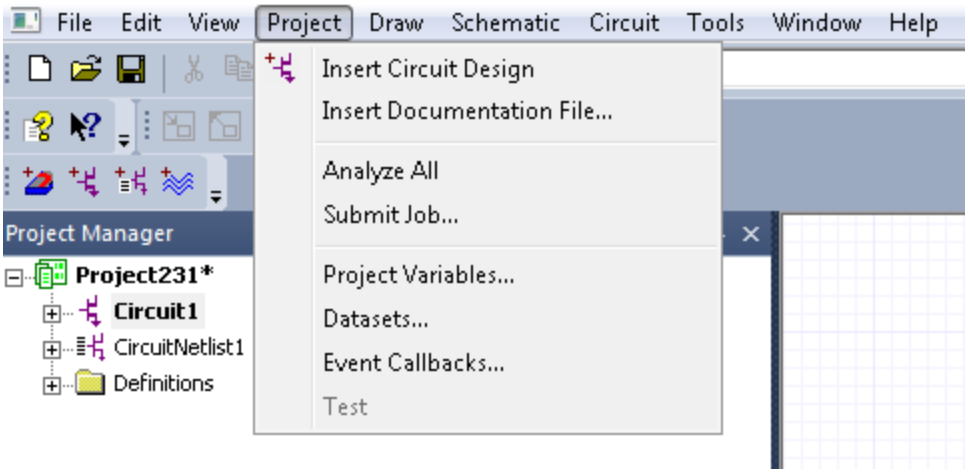
2. **Circuit Design.xml Sample**

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>
<DesignerMenu xmlns="http://www.ansys.com/uiConfigMenu"
action="useDefault">
  <UseProjectWindowSelectionContext>false</UseProjectWindowSelectionC
  ontext>
  <Context name="All">
    <TopMenu>
          <MenuName>&amp;Project</MenuName>
      <LeafMenu action="delete">
                <MenuName>Insert HFSS Design</MenuName>
      </LeafMenu>
      <LeafMenu action="delete">
                <MenuName>Insert HFSS 3D &amp;Layout
        Design</MenuName>
      </LeafMenu>
      <LeafMenu action="delete">
                <MenuName>Insert HFSS-IE Design</MenuName>
      </LeafMenu>
      <LeafMenu action="delete">
                <MenuName>Insert Q3D Extractor Design</MenuName>
      </LeafMenu>
      <LeafMenu action="delete">
```

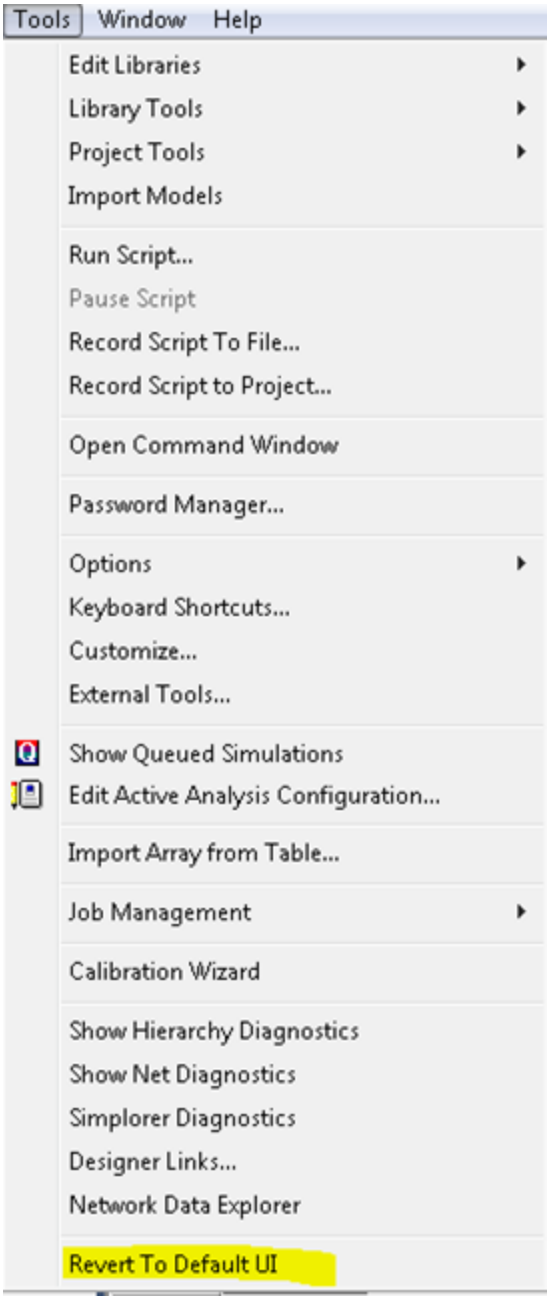

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
                <MenuName>Insert 2D Extractor Design</MenuName>
        </LeafMenu>
        <LeafMenu action="delete">
                <MenuName>Insert Circuit &amp;Netlist</MenuName>
        </LeafMenu>
        <LeafMenu action="add">
                <MenuName>Test</MenuName>
                <MenuID>3333</MenuID>
        </LeafMenu>
      </TopMenu>
   </Context>
</DesignerMenu>
```

Menus created by processing the above XML sample, Circuit Design.xml



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



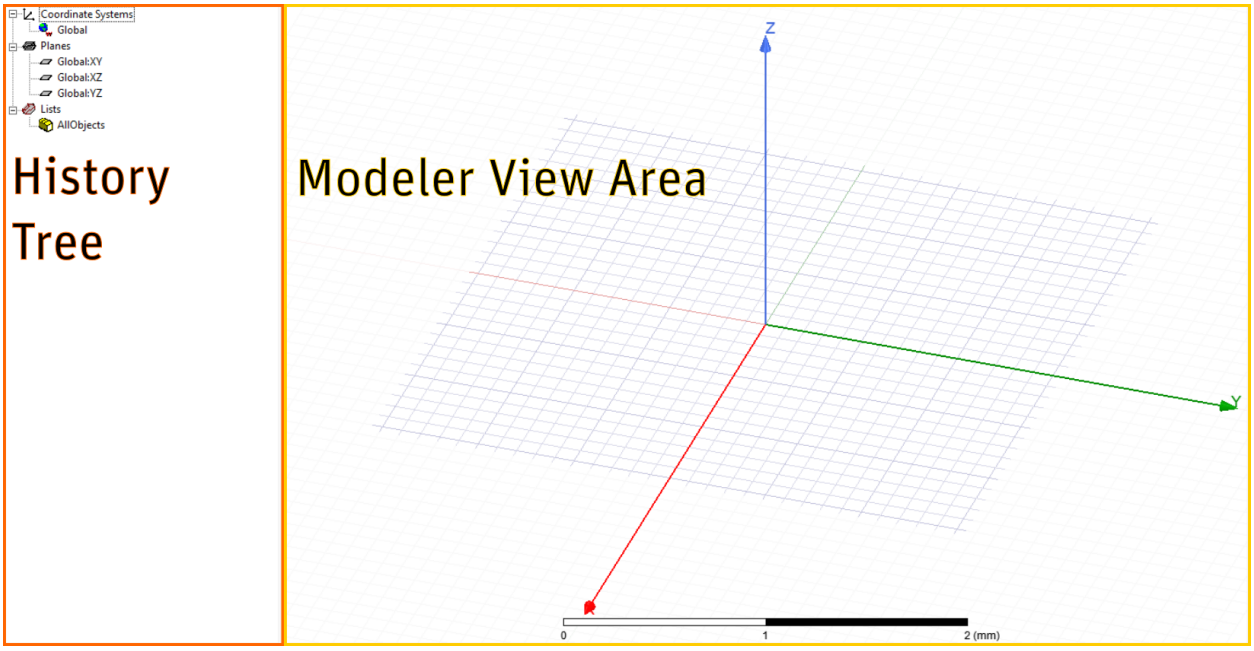
Ansys Electronics Desktop Design Area

The Design Area of the desktop can display one or more editor windows and report windows, depending on the type of designs you create or load. See the Desktop Windows topic for ways to manipulate the windows in the Design Area.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D Modeler Window

The **3D Modeler** window is the area where you create the model geometry. It appears to the right of the **Project Manager** window after you insert an Ansys Electronics Desktop design into a project. The **3D Modeler** window consists of the modeler view area, or grid, and the History Tree. The two areas are highlighted in the following figure:



You can open a new 3D Modeler window in one of two ways:

- Insert a new Ansys Electronics Desktop design into the current project, or
- Double-click an Ansys Electronics Desktop design in the Project Tree

The model you draw is saved with the current project when you click **File** > **Save**.

Objects are drawn in the **3D Modeler** window. You can create 3D objects using Ansys Electronics Desktop's Draw menu commands, or you can draw 1D and 2D objects and manipulate them to create 3D objects. See: Drawing a Model.

You can modify the view of objects in the **3D Modeler** window without changing their actual dimensions or positions. Besides menu and icon commands, you can also use **Alt+click**, mouse buttons, and keyboard shortcuts. See: Modifying the Model View.

Layout Window

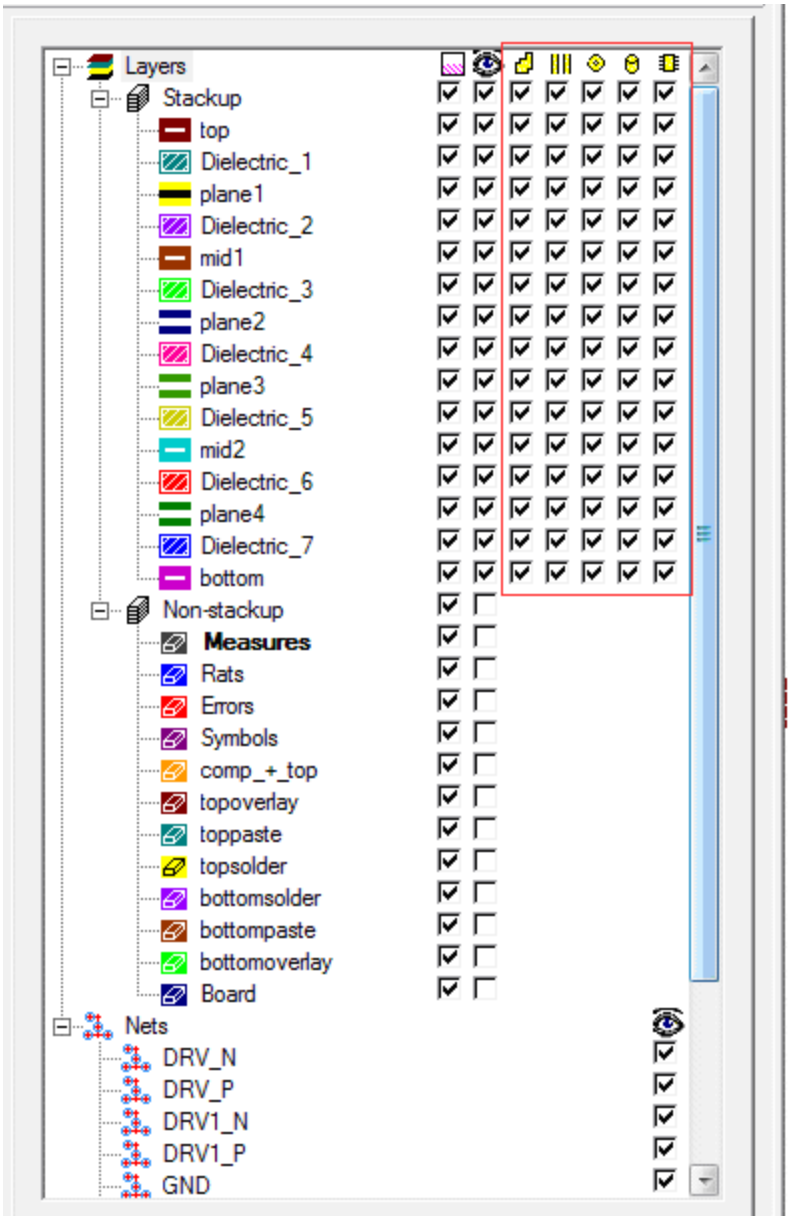
The Layout Window is a dockable Ansys Electronics Desktop window that can be resized and relocated, and can be used to view and configure various layout settings. Use the right-click

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

menu of the Layout Window to configure the following:

- **Set Active** – makes the current layer the active layer.
- **Show This Layer Only** – makes the current layer the only visible layer.
- **Show All Dielectrics** – makes all Dielectric layers visible.
- **Hide All Dielectrics** – makes all Dielectric layers invisible.
- **Show All Signals** – makes all Signal layers visible.
- **Hide All Signals** – makes all Signal layers invisible.

Use the **Layout** window to alter the following controls for setting visibility by layout-object type.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following controls are available:

Controls the visibility of shapes

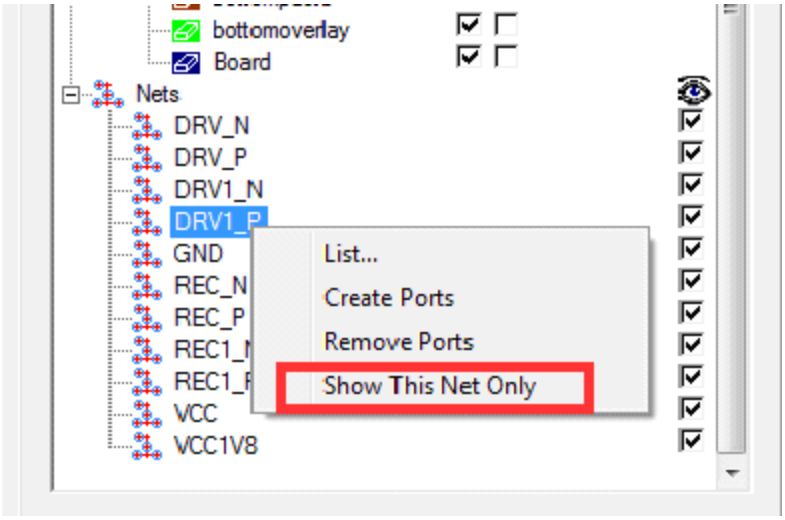
Controls the visibility of lines (paths)

Controls the visibility of pads

Controls the visibility of holes

Controls the visibility of components

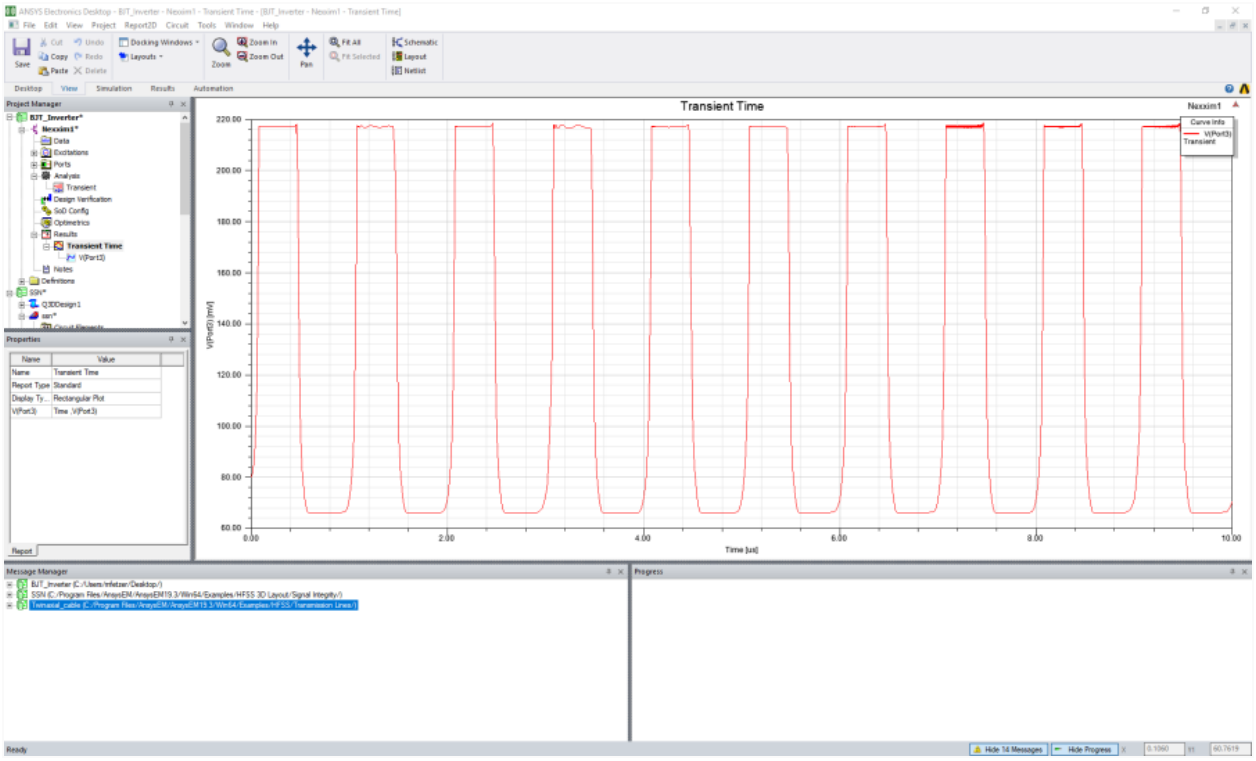
You can also turn visibility off for all nets but the selected net by right-clicking and selecting **Show This Net Only**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report Window

When a design has been successfully simulated, you can generate a report of results in a wide variety of forms, including XY graphs, polar graphs, 3D graphs, Smith charts, and data tables. Various attributes of each can be customized to your liking. The following shows a 2D report window in Electronics Desktop:

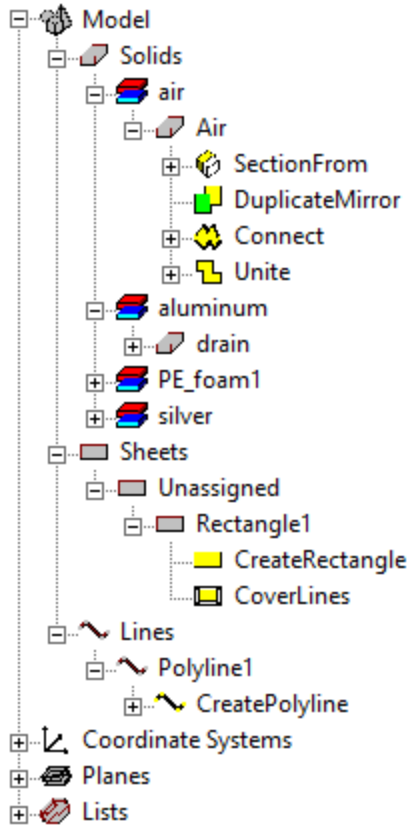


For more information, see Post Processing and Generating Reports.

Ansys Electronics Desktop History Tree

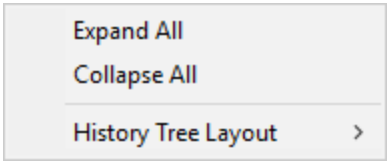
The history tree in the **3D Modeler** window lists the active model's structure and grid details.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Expand or collapse items in the tree using the **+** and **-** boxes.

Expand or collapse all items in a category by right-clicking to open a shortcut menu:



> **Note:**
>
> Some objects have more extensive shortcut menus than others, but all of them have these three options.

Selecting any item in the History Tree also selects it in the design area and populates the **Properties** window with its properties.

You can perform the following tasks with the History Tree:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Organize the History Tree
- Control the View of Objects in the History Tree
- Use Group Commands for Modeler Objects
- Select Objects in the History Tree
- View and Edit Commands on History Tree Objects

The History Tree can contain the following model details:

Invalid	Lists all invalid objects.
3D Component Names	Lists any 3D components added to the design. The History Tree lists their Solids, Coordinate Systems, and Planes separately from objects drawn in the modeler or imported to the modeler.
Solids	Lists the model's solid objects and a history of the commands carried out on each object.
Sheets	Lists the sheets in the model 3D design area. By default, HFSS groups sheet objects by boundary assignment.
Lines	Lists all line objects included in the active model. See Drawing a Straight Line for information on how to draw a line object.
Points	Lists all point objects included in the active model. See Drawing a Point for information on how to draw a point object.
Groups	Lists folders for Groups you have created or imported. See Group Commands for Modeler Objects.
Coordinate Systems	Lists all the coordinate systems for the active model. See Setting Coordinate Systems for more information on this model detail.
Planes	Lists the planes for all the coordinate systems. When you create a coordinate system, default planes are created on its xy, yz, and xz planes.
Lists	Lists the object or face lists for the active model. By default, a list called “AllObjects” appears. You can add a specific group of objects or surfaces for field plot or for calculation. See: Creating an Object List and Creating a Face List.

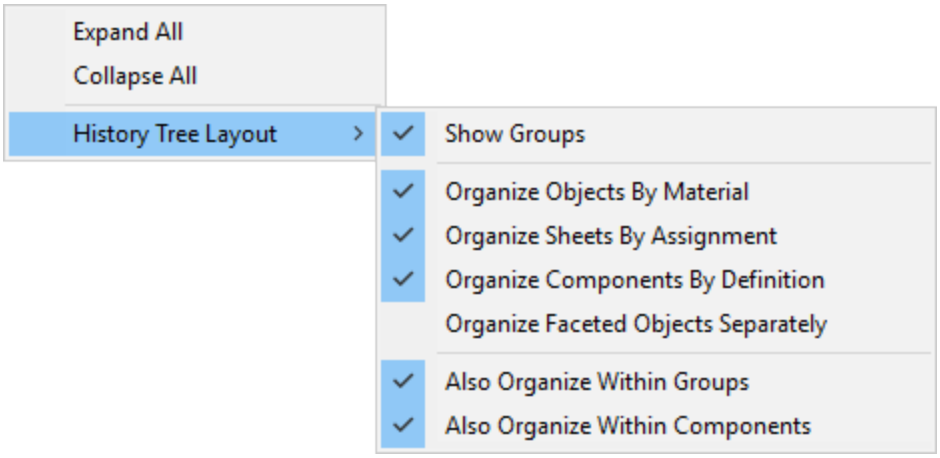
Note:

While objects created in HFSS can always be classed in the History Tree as a solid, sheet, or wire, some imported objects may have mixture of these. HFSS places such objects in an Unclassified folder.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Organizing the History Tree

To organize and sort objects in the History Tree, right-click any object in the history tree to display the shortcut menu and select **History Tree Layout**.



The shortcut menu contains the following commands:

- **Show Groups**
- **Organize Objects By Material:** This option is applicable to 3D objects and also to sheet objects that have the *Shell Element* attribute selected and a material selection specified (Mechanical - Thermal designs only). Imported solid objects that have no material specified yet are listed in a group named *Unassigned*. Non-model objects are listed in a *Non Model* group.
- **Organize Sheets By Assignment:** This option is applicable to sheet objects only (including Mechanical – Thermal shell objects with a boundary or excitation assigned). Sheet objects are grouped according to the boundary or excitation assigned. When a sheet is assigned both a boundary and an excitation, it is grouped according to the last assigned excitation (that is, the group name is the last assigned excitation). When multiple boundaries are assigned to a sheet, the sheet is grouped according to the boundary with the highest priority, and that boundary is used as the group name. The boundary priority depends on the design type, as follows:
  - **HFSS and Maxwell:** Priority is specified by the user.
  - **Icepak, Mechanical, and Q3D:** The last created boundary has the highest priority
- **Organize Components By Definition**
- **Organize Faceted Objects Separately**
- **Also Organize Within Groups**
- **Also Organize Within Components**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

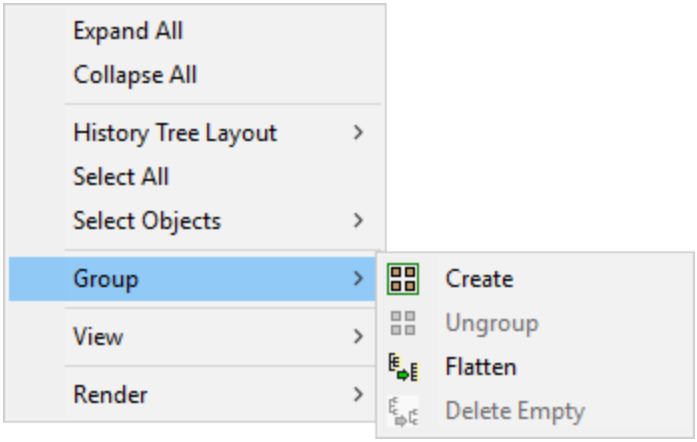
Controlling the View of Objects in the History Tree

To control the view and visibility of an object such as a box or PML, right-click an object in the history tree to display the shortcut menu and select **View**. The shortcut menu contains the following commands:

- **Fit in Active View**
- **Hide in Active View**
- **Show in Active View**
- **Fit in All Views**
- **Hide in All Views**
- **Show in All Views**

Group Commands for Modeler Objects

The 3D Modeler allows you to group objects in the History tree. Besides predefined folders under **History Tree Layout** for solids, sheets, and material assignments, a set of **Group** commands allows you to create groups, ungroup objects, flatten a selected group's hierarchy, and delete empty groups. Groups are not compatible with 3D components, although, you can organize 3D components by definition in the History tree.

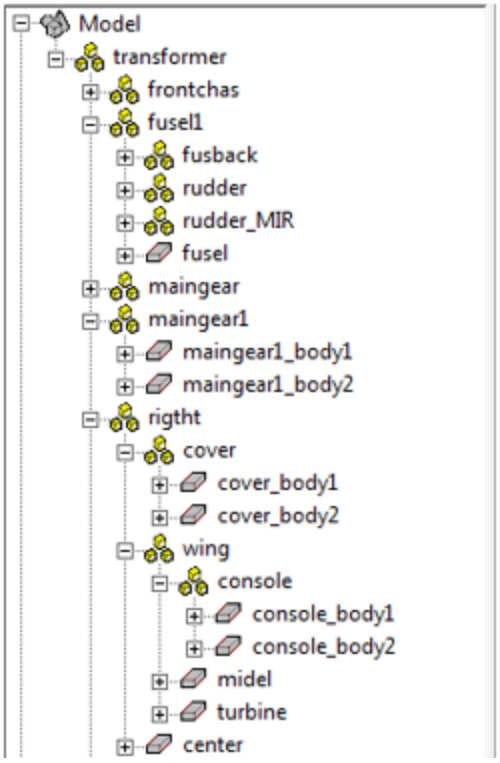


The **Group** commands support the following features:

- Group objects in the history tree. When objects are grouped, they show up under a sub-item in the history tree.
- Groups can contain sub-groups and sub-group containing further sub-groups.
- Groups permit moving objects from one group to any other group at any time.
- Groups are purely for organization of history tree. They do not affect solution in any way.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Groups permit you to bring in MCAD assemblies and sub-assemblies as groups. See: Importing 3D Model Files.



- Groups can contain objects, submodels (for example, UDM, 3D Component, CAD links) and groups. Coordinate systems, planes etc will not have any parent group.

Object groups have following important distinctions from 3D components:

- Groups do not encapsulate history of objects. They do not encapsulate parameters used by those objects.
- Groups do not have history tree operations. For example, Arrange operations apply to all objects of the group rather than the group.
- Groups are not independent. Delete of an object in one group could cause object in another group to be also deleted.

Accessing Group Commands

You can access Group commands in several ways: via **Modeler > Group**, via the right-click shortcut menu in the History tree, via the right-click shortcut menu in the Modeler window, and via the Group icons on the Model tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Group Command Descriptions

Group > **Create** works with or without a selection. If the History Tree Layout>Show Groups command is disabled, a dialog box reports that groups are currently hidden, and that if you OK to continue, groups will be shown. If there is no selection of objects for new group, an empty group is created under Model. If there are selections, a new group is created under same group as all the selections, provided all selections are under one group. Once a group is created, all selections move under the new group. If selections are under different groups, then a new group is created under Model. Selections could be objects, groups, 3D Components, or User Defined Model. The **Group** > **Create** command tries to find a more suitable group name where possible, based on names of all selections. If not, default group names are Group1, Group2, and so on. You can edit group names later.

Group > **Ungroup** is enabled only when you select one or more groups. Upon ungroup, all the contents of the selected group move under group's parent and the selected group is deleted. Note that **Group** > **Ungroup** is not recursive; it only affects the selected group is and all its children groups remain intact. Ungroup is also different from **Delete** (under **Edit** > **Delete**), which deletes a selected group and everything under it.

Group > **Flatten** is enabled when you select a group. You can select multiple groups at different levels. All of the selected group's contents (objects, sub-groups, and so forth) come directly under selected group. In a sense this is a recursive ungroup operation as all the children and grandchildren groups of selected group are ungrouped. If the History Tree Layout > Show Groups command is disabled, a dialog box reports that groups are currently hidden, and that if you OK to continue, groups will be shown.

Group > **Delete Empty** deletes all empty groups under the selected group. If a selected group is empty it is deleted. This command is helpful to clean up empty groups after you have moved objects from one group to another. If the History Tree Layout > Show Groups command is disabled, a dialog box reports that groups are currently hidden, and that if you click **OK** to continue, groups will be shown.

Operations on Groups

Edit/Delete

Delete operations delete group and all the children, including sub-groups, under the group.

Copy and Paste of Objects in Groups

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you copy and paste objects with groups to another design, group information is carried over, that is, new group in target design is created. If a group with the exact name exists, then the objects are moved under that group and no new group is created. When pasted in the same design, the pasted object moves under same parent group as original object.

Copy and Paste of Groups

When you copy and paste one or more groups in same design or to another design, a new group is created with unique name derived from original group name. Everything under that group, including any sub-groups, is copy/pasted as well.

Copy and Paste of Groups and Objects

If you multi-select a few objects and groups, all of them are pasted correctly. If you select a group and few objects under that group, copy and paste includes the complete group structure. This means that selection of objects under the selected group is ignored. If you want to only copy and paste few objects from group, you should only select those objects and NOT select the group.

Arrange Operations on Groups

You can select group to enable arrange operations like move, rotate and mirror. Note that arrange operations works directly on objects under selected group. So after arrange operation is done, the History tree shows the arrange operation under every object of that group.

Duplicate Operations on Groups

You can select groups to enable duplicate operations. Afterward, the History tree shows a duplicate operation under every object of that group. Newly created objects appear under a new group. There will be new group for every instance of duplicate.

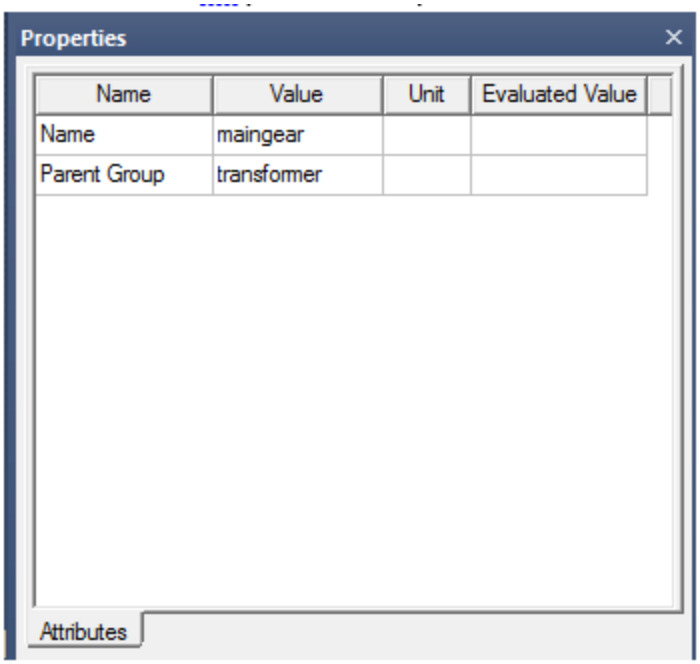
Other Operations on Groups and Objects in Groups

Modeling operations that create new objects, such as Create from Face or Edge, as well as Simplify and Separate. The newly created objects appear under a new group created under the original object's group, based on the tool option settings. See: 3D Modeler Options: Group Options.

Group Properties

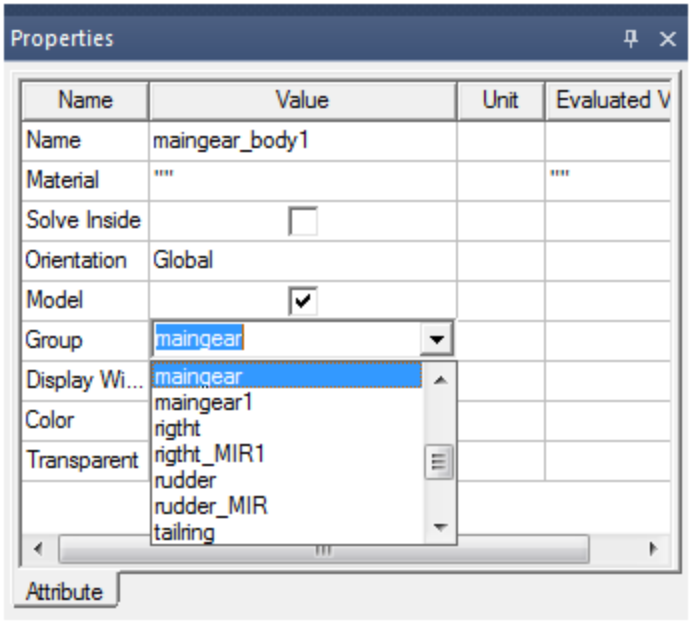
A Group’s Property tab is shown when you select a group in the history tree. The Group’s properties include Name and parent Group.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Group Property Window

Object, group and submodel properties have a Group property. You can edit the Group Property and this provides another way of setting an object's (or group's) group. You can select any of the existing groups listed or create a new group.



Group assignment could be changed in Object's property window

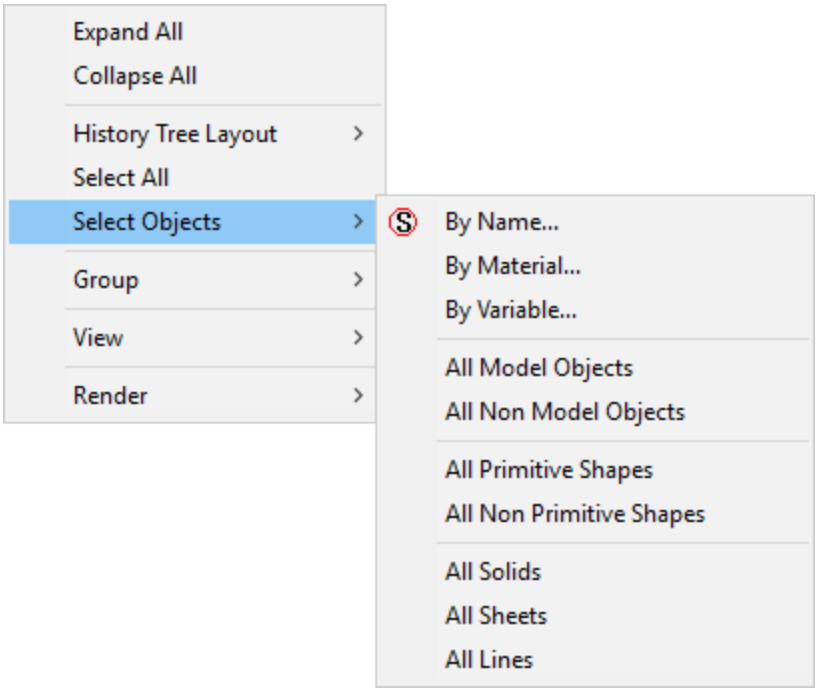
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting Objects in the History Tree

Selecting objects in the History Tree also selects them in the View window. This can be useful for complex objects, when it may be easier to find objects of interest by name or material because the object is inside or behind others.

Ctrl+click to make multiple selections in the History Tree. You can select a range of objects by clicking the first and using **Shift+click** to select all in the range. You can also click and drag the mouse to make rubber band selections. Only visible objects are selected. That is, if the hierarchy is closed under the selection, any operand parts are ignored and do not interfere with cut and paste operations.

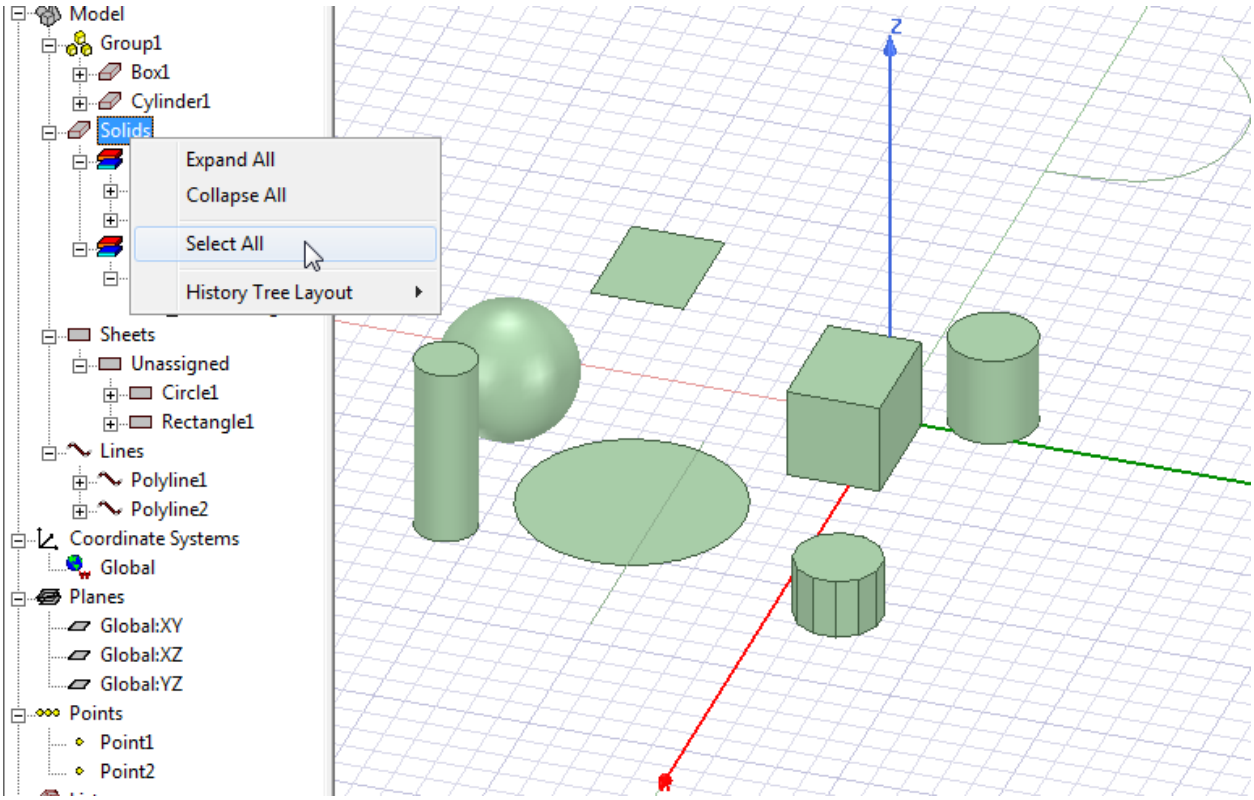
Right-click **Model** in the History Tree to see additional selection options:



You can right-click any History Tree object or grouping to display a shortcut menu with a **Select All** option. This lets you select all non-grouped solids, sheets, lines, non-model objects, or all unclassified objects, as well as all non-grouped objects assigned to specific materials.

For example, right-click Solids and then **Select All**:

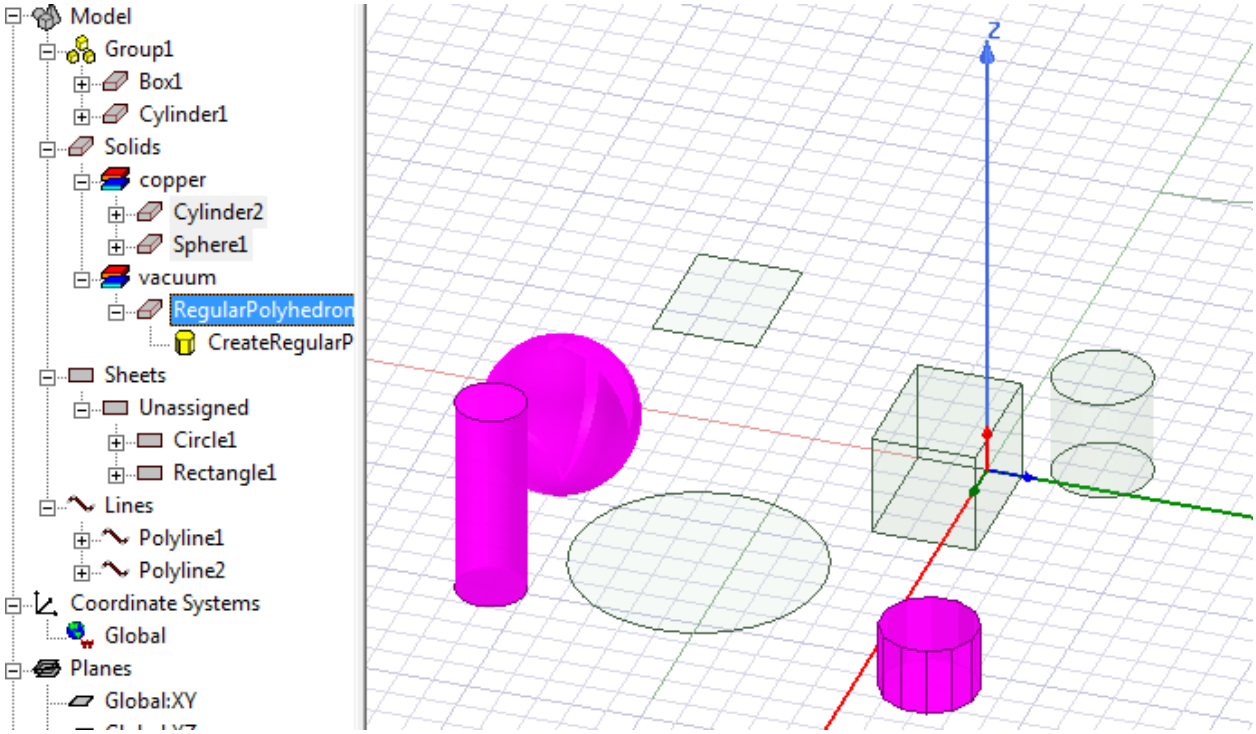
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After clicking **Select All**, the History tree and the **Model** window highlight the selected Solid Objects.

Notice that Box1 and Cylinder1 are not selected because they are in Group1.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



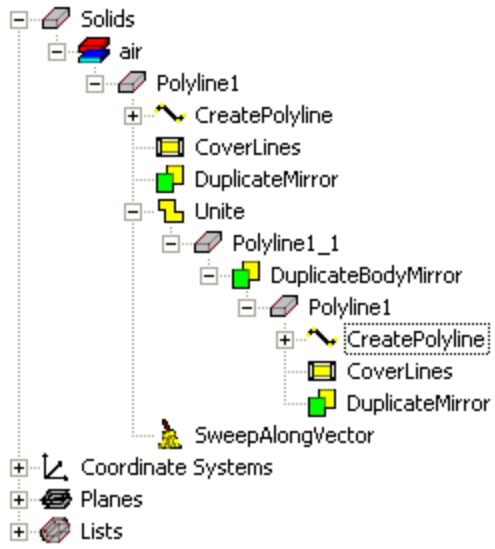
The right-click for Model and **Select All** does select the members of Group1 because they are part of the model.

If you select a geometric object, such as a Box or Cylinder, the shortcut menu **Select** command offers choices to select **All Faces**, **All Edges**, or **Sheet Edges** for that object.

Viewing and Editing Commands on History Tree Objects

The history tree also lists the history of all commands carried out on a model's objects (for example, “CreateBox” or “Subtract”). This history is displayed in the order in which it occurred. Here is a history tree from the waveguide combiner example project:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

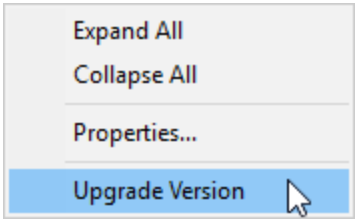


Selecting a command in the history highlights the object in the Modeler window and shows that object's properties (if available) in the docked **Properties** window. You can look at the fields in the **Properties** window to see any editable fields for that command, such as coordinate system, line type, coordinates, or units.

For some commands, such as **Edit** > **Arrange** > **Move** or boolean operations, selecting them in the history tree enables the delete icon (**X**) on the toolbar and the **Edit** > **Delete** menu. In these cases, you can delete those commands from the history tree as a way of undoing those operations. As an alternative to deletion, you can check **Suppress command** in the **Properties** window for that command. This undoes the effect of a command on an object without removing it from the History Tree.

Upgrading Version in the History Tree

Right-click an operation in the History Tree to see the **Upgrade Version** command.



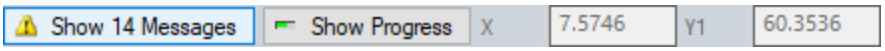
By default, the Modeler opens legacy projects using the modeling kernel version under which the project was saved, so that users don't see side effects from slight topology changes between versions. The **Upgrade Version** operation forces Electronics Desktop to use the latest modeling

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

kernel. Typically, this command would only be used when a needed fix is available for the newest version.

Ansys Electronics Desktop Status Bar

The status bar is located at the bottom of the application window. By default, it contains buttons to Show or Hide the Message window and Progress window.

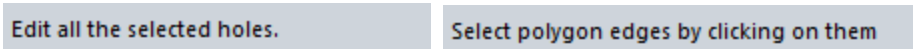


Note that the Show Messages button indicates the number of messages.

When more than one progress bar is active, the top progress bar is represented on the status bar with a progress indicator:

The status bar also displays helpful information about the current selection or command.

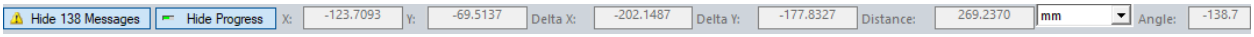
Directions for inputs appear on the left side of the status bar:



Depending on the command being performed, the status bar can display the following:

- **X**, **Y**, and **Z** coordinate boxes
- A drop-down menu for entering absolute, relative, cartesian, cylindrical, or spherical coordinates
- The model's units of measurement

These coordinate boxes and drop-down menus appear on the right side of the status bar:



To toggle status bar display:

- Click **View** > **Status Bar**.

A check mark next to this command indicates that the status bar is visible.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Keyboard Shortcuts

The following keyboard shortcuts apply to HFSS in general:

F1	Help (context-sensitive)
Ctrl + F4	Close window
Alt + F4	Close program
Ctrl + C	Copy
Ctrl + N	New Project
Ctrl + O	Open
Ctrl + P	Print
Ctrl + V	Paste
Ctrl + X	Cut
Ctrl + Y	Redo
Ctrl + Z	Undo
Ctrl + 0	Cascade windows
Ctrl + 1	Tile windows horizontally
Ctrl + 2	Tile windows vertically

See the subtopics in this Help branch for additional information concerning shortcuts and view navigation.

To customize the shortcut assignments, use **Tools** > **Keyboard Shortcuts**. Not all shortcuts are customizable.

Desktop Shortcuts

Modifier + key	Hold down the modifier, such as Shift or Ctrl, and press the key.

The following Ansys Electronics Desktop shortcut key combinations are available at any time:

Ctrl + N	New
Ctrl + O	Open
Ctrl + P	Print
Ctrl + S	Save
Ctrl + 0	Cascade windows
Ctrl + 1	Tile windows horizontally
Ctrl + 2	Tile windows vertically

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete	Delete
F1	Open help

3D Modeler Window Shortcuts

The following shortcut key combinations are available when the Modeler window is active. View navigation shortcuts are also applicable to 3D Plot windows. The table below is organized into four sections, with the shortcuts separated according to the following categories:

- Commands and options used for the selection of entities
- Choosing the entry mode to use when creating geometry
- View navigation (rotating, zooming, panning, choosing a view orientation)
- Choosing the geometry rendering mode (wireframe or shaded)

> **Note:**
>
> In the following table, **LMB** is an abbreviation for *Left Mouse Button*, and **MMB** means *Middle Mouse Button*.
>
> **Wheel** refers to rotation of the mouse wheel that is integrated with the middle button.

Entity Selection Commands and Options:	
B	Select face or object **behind** current selection
F	Select **faces** mode
O	Select **objects** mode
E	Select **edges** mode
V	Select **vertices** mode
M	**Multi-select** mode
Ctrl + A	Select **all** visible objects
Ctrl + Shift + A	Deselect all objects
Geometry Properties Entry Mode:	
F3	Switch to the **point** entry mode (using the mouse or coordinate text boxes to draw objects). Switching takes effect only after the drawing of an object has started.
F4	Switch to the **dialog box** entry mode (draw objects by specifying properties in the Command and Attribute tab of a dialog box). Switching takes effect only after the drawing of an object has started.
View Navigation:	
Ctrl + D	**Fit** view

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ctrl + '+'	**Zoom in** (screen center)
Ctrl + '-'	**Zoom out** (screen center)
Alt + double-click	Change the view orientation to one of nine predefined views depending on where you double-click within the Modeler window. See Changing the Model View with Alt+Double-Click Areas for more information.
Alt + *arrow keys*	**Rotate** the view about a vertical or horizontal axis. Each press of an arrow key rotates the view 2.5°.
Ctrl + *arrow keys*	**Pan** the view vertically or horizontally. Each press of an arrow key pans the view 10 screen pixels.
MMB + drag (no modifier)	Freely **rotate** the view orientation (except when clicking and dragging near the window border). When clicking near the window border, rotation is constrained about the screen-normal axis.
MMB + X + drag	Constrained **rotation** of the view orientation about the **X** axis
MMB + Y + drag	Constrained **rotation** of the view orientation about the **Y** axis
MMB + Z + drag	Constrained **rotation** of the view orientation about the **Z** axis
Wheel	Rotate the mouse wheel to **zoom** in and out. Rotate the top of the wheel forward (away from the mouse) to zoom **in** closer, and rotate it rearward to zoom **out**.
Shift + MMB + drag	**Zoom** in or out. Moving the cursor upward zooms in closer. Moving it downward zooms out. (* see note below)
Ctrl + MMB + drag	**Pan** the view (* see note below)
(*) **Note:** The listed actions for **Shift** and **Ctrl** keys in combination with the **MMB** are the view navigation assignments for 2025 R1. Simultaneously, legacy hot keys using **Shift** or **Alt + Shift** in combination with the **LMB** for these functions continue to be supported by default. Optionally, you can choose to disable the legacy shortcuts that were applicable to 2023 R2 and earlier versions. See Choosing the View Navigation Options for more information.	
Geometry Rendering Modes:	
F6	Render the geometry as wireframes (no shading of faces)
F7	Render the geometry with smooth, shaded faces

To customize shortcut assignments, use **Tools** > **Keyboard Shortcuts**. Not all shortcuts are customizable.

Schematic Shortcuts

Modifier + key	Hold down the modifier (such as Shift or Ctrl) and press the key.

The following shortcut key combinations are available when the **Schematic** window is active:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ctrl + Z	Undo
Ctrl + Y	Redo
Ctrl + X	Cut
Ctrl + C	Copy
Ctrl + V	Paste
Ctrl + A	Select all
Ctrl + "+"	Zoom in
Ctrl + "-"	Zoom out
Ctrl + Q	Zoom area
Ctrl + D	Fit drawing
Ctrl + W	Add wire
Ctrl + K	Cross-probe for selected components in layout editor
Ctrl + MMB + drag	Pan (see note below)
Shift + MMB + drag	Zoom (see note below)

Note:

For view navigation shortcuts (last two rows of the table above):

- **MMB** = Middle Mouse Button.
- These are the default view navigation assignments for version 2025 R1. Optionally, you can choose to use the legacy shortcuts that were appliable to 2023 R2 and earlier versions. See Choosing the View Navigation Options for more information.

Layout Shortcuts

Modifier + key	Hold down the modifier (such as Shift or Ctrl) and press the key.

The following shortcut key combinations are available when the **Layout** window is active:

Ctrl + A	Select all
Ctrl + Shift + A	Unselect all
Ctrl + C	Copy
Ctrl + D	Fit drawing
Ctrl + "+"	Zoom in
Ctrl + "-"	Zoom out

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ctrl + K	Cross-probe for selected components in schematic editor
Ctrl + M	Align microwave ports
Ctrl + Q	Zoom area
Ctrl + R	Rotate
Ctrl + V	Paste
Ctrl + W	Add connection
Ctrl + X	Cut
Ctrl + Y	Redo
Ctrl + Z	Undo
Delete	Delete
MMB + drag	Rotate (see note below)
Ctrl + MMB + drag	Pan (see note below)
Shift + MMB + drag	Zoom (see note below)

Note:

For view navigation shortcuts (last three rows of the table above):

- **MMB** = Middle Mouse Button.
- These are the default view navigation assignments for 2025 R1. Optionally, you can choose to use the legacy shortcuts that were appliable to 2023 R2 and earlier versions. See Choosing the View Navigation Options for more information.

Report Shortcuts

Modifier + key	Hold down the modifier (such as Shift or Ctrl) and press the key.

The following shortcut key combinations are available when a Report window is open:

Ctrl + A	Select all traces
Ctrl + C	Copy selected trace
Ctrl + V	Paste
Ctrl + X	Cut selected trace
Ctrl + Y	Redo insert report
Ctrl + Z	Undo insert report
Delete	Delete

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Netlist Editor Shortcuts

Modifier + key	Hold down the modifier (such as Shift or Ctrl) and press the key.

The following shortcut key combinations are available when a Netlist Editor window is open:

Ctrl + A	Select all traces
Ctrl + B	Delete all bookmarks
Ctrl + C	Copy
Ctrl + F	Find
Ctrl + G	Go to line number
Ctrl + R	Replace
Ctrl + V	Paste
Ctrl + X	Cut
Ctrl + Y	Redo
Ctrl + Z	Undo
Delete	Delete
F2	Next bookmark
Ctrl + F2	Toggle bookmark
Shift + F2	Previous bookmark

Choosing the View Navigation Options

New mouse button and modifier key assignments (hotkeys) have been added for navigating the model view (rotate, pan, and zoom functions). Simultaneously, legacy hotkeys for these functions continue to be supported by default. Optionally, you can choose to disable the legacy view navigation shortcuts that were applicable to version 2023 R2 and earlier, as follows:

1. Access the *Options* dialog box using one of the following two methods:
   - From the menu bar, click **Tools** > **Options** > **General Options**.
   - From the **Desktop** ribbon tab, click **General Options**.
2. In the tree at the left side of the dialog box, select **General** > **User Interface**. Then:
   - To use either the legacy or current view navigation buttons and hotkey combinations, select **Enable Legacy View Navigation**. This option is selected by default.
   - To use only the current view navigation behavior and disable legacy button/hotkey support, ensure that **Enable Legacy View Navigation** is cleared.
3. Click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The navigation option change becomes effective immediately (no program restart required).

Current vs. Legacy View Navigation

The following table summarizes changes in mouse-button and hotkey assignments between the current scheme (version 2024 R1 and newer) and the legacy scheme. "*Legacy*" assignments are applicable to version 2023 R2 or earlier and when the *Enable Legacy View Navigation* option is selected in newer versions):

> **Note:**
>
> The *"Current"* assignments are available whether or not the *Enable Legacy View Navigation* option is selected:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

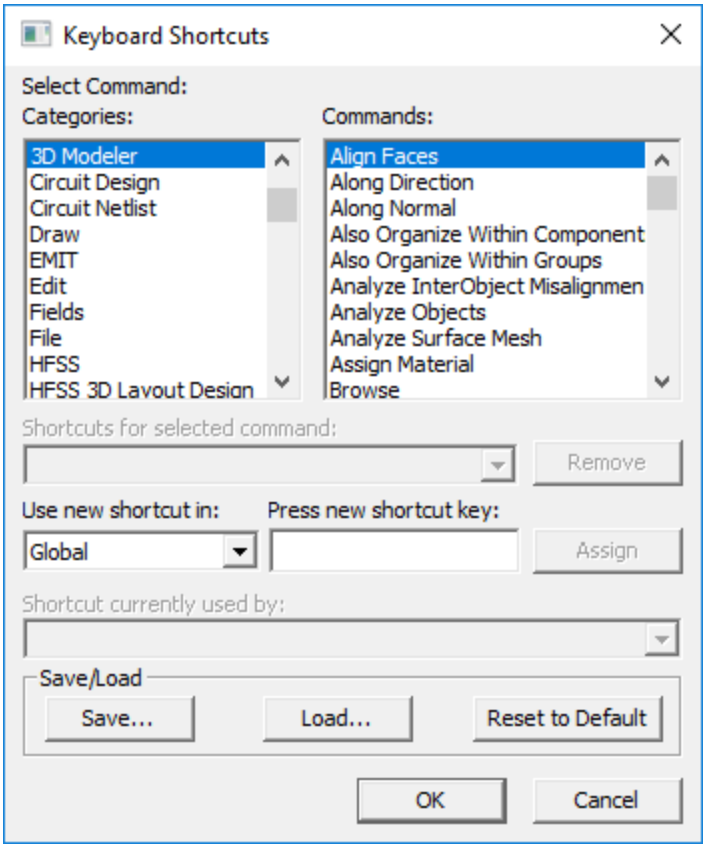
Function	Key / Mouse Button Assignment	
	Current	**Legacy**
Pan (all contexts)	Ctrl + MMB + drag or Ctrl + *arrow keys*	Shift + LMB + drag or Ctrl + *arrow keys*
Rotate (3D contexts)	MMB + drag or Alt + *arrow keys*	MMB + drag or Alt + LMB + drag or Alt + *arrow keys*
Zoom (all contexts)	Wheel or Shift + MMB + drag or Ctrl + "+", Ctrl + "-"	Wheel or Alt + Shift + LMB + drag or Ctrl + "+", Ctrl + "-"

- "**LMB**" = Left Mouse Button
- "**MMB**" = Middle Mouse Button
- "**Wheel**" refers to rotation of the mouse wheel.
- In some 2D contexts, where rotation is not applicable, you can use **MMB + drag** to pan the view for either scheme.
- For **Alt + *arrow keys***, rotation is about a vertical or horizontal axis, regardless of the model view orientation.
- For zooming, if using the "=\+" key, instead of the "+" key on the numeric keypad, do **not** press Shift.

Custom Keyboard Shortcuts

Click **Tools** > **Keyboard Shortcuts** to open the **Keyboard Shortcuts** window. Here, you can view existing assignments, create new shortcuts, and save or load assignment files.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selecting a **Command Category** populates the **Commands** list with the available commands for that category. If the command has an assigned shortcut, it is displayed in the **Shortcuts for the selected command** field. You can use the **Remove** button to disable the shortcut for the selected command. If the selected command does not have an assigned shortcut, the **Shortcuts for selected command** field and the **Remove** button are unavailable.

To create a new shortcut key:

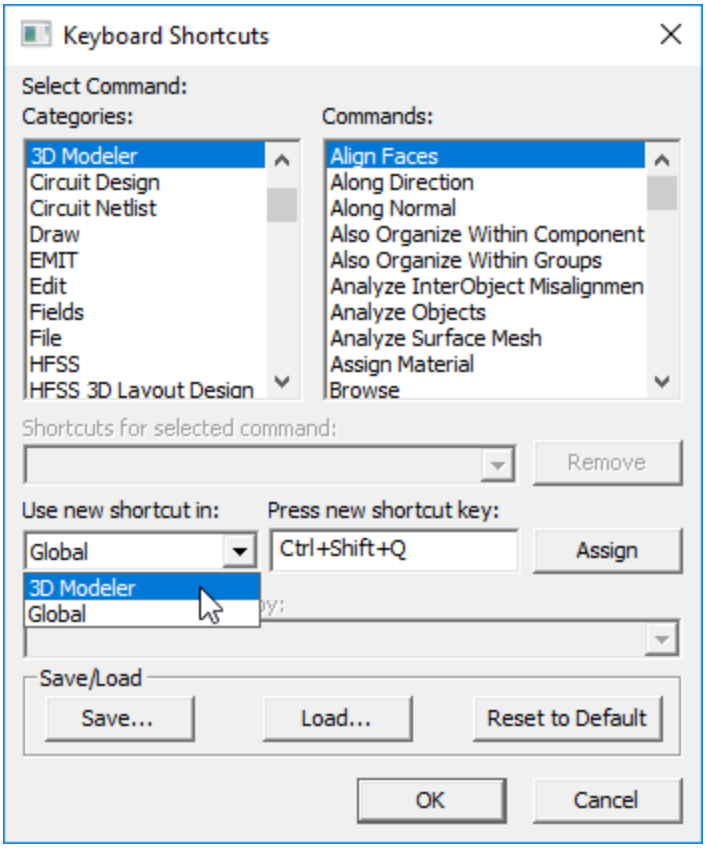
1. Select the **Category** and **Command**.
2. If you want to disable a current assignment for the selected command, click **Remove**.
3. To assign a keyboard shortcut, place the cursor in the **Press new shortcut key** field.

   The field displays the keystrokes you make. When you have made keystrokes, the **Assign** button becomes available. If you combine keystrokes, these are displayed with a plus sign (+) between them. For example, Ctrl+P or Alt+O.

   If the shortcut you select is currently in use, that information displays in the **Shortcut currently used by** field.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. The **Use new shortcut in** drop-down list displays **Global** by default, which means that the shortcut will apply to all applicable contexts. If a limited context exists, the menu offers a selection.



5. When you have made your desired assignments, you can save them to a named file.

   Click **Save...** to save the assignments to your desired location in Ansoft Keyboard Shortcut (*.aks) format.

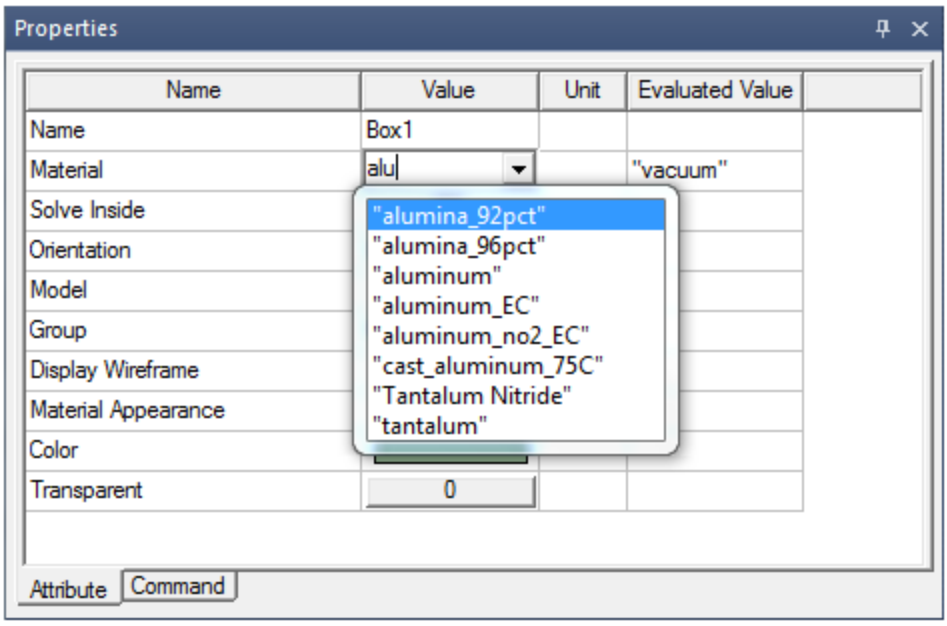
   If you have an existing *.aks file, use the **Load...** button to locate and select the file.

6. Click **OK** to save the current settings, or **Reset to Default**.

Auto-Complete for Variables and Properties in Electronics Desktop

When you edit a properties or variable text field, Electronics Desktop can display possible matches for what you type. This can help if a variable or material name is long. You can save time by selecting a pre-determined match rather than typing out the entire name. The following figure shows an example of auto-complete for material names:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

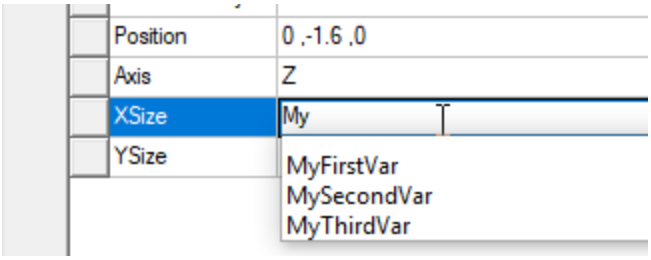


Using Auto-Complete

Certain commonly used text fields have auto-complete configured. When you start typing in these cases, matches display in a list below the text field. If there is no matching text, then no list displays. The list is automatically sized, but you can resize it. Electronics Desktop remembers the new size when doing additional auto-complete matching in the same text field. Switching to a different text field resets the sizing.

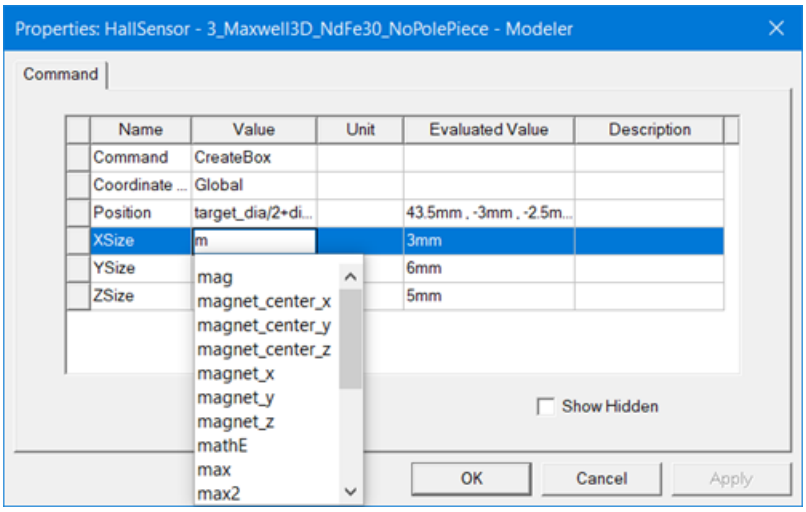
Hitting **Tab** or **Enter** accepts the current selection for auto-complete, which replaces the text typed with the full auto-complete match and hides the list. Otherwise, hitting the escape key **Esc** hides the auto-complete list. Typing more letters causes auto-complete matching to resume.

Where you select design properties, auto-complete works with expressions for the values, providing matches for the names of constants, intrinsic variables, functions, project variables (prefixed with “$”), and design properties. When project variables are displayed (that is, you select the project in the Project tree), auto-complete for value expressions also works, except without matching design properties. The variable auto-complete list does not include Separator Variables or Hidden variables since these could cause invalid formulas.



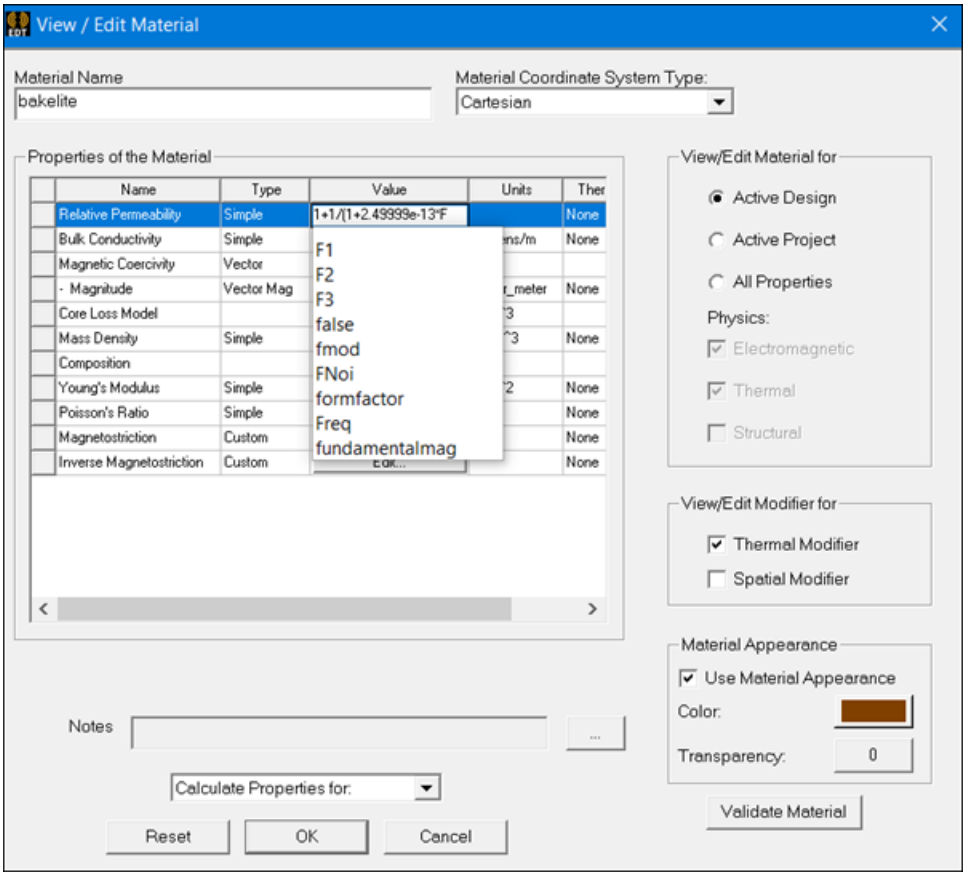
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties of other items may also have auto-complete configured to work with value expressions (for example, properties of a CreateBox command or of a circuit component). The following shows a design property with auto-complete matching:



Auto-complete also works with value expressions for material properties:

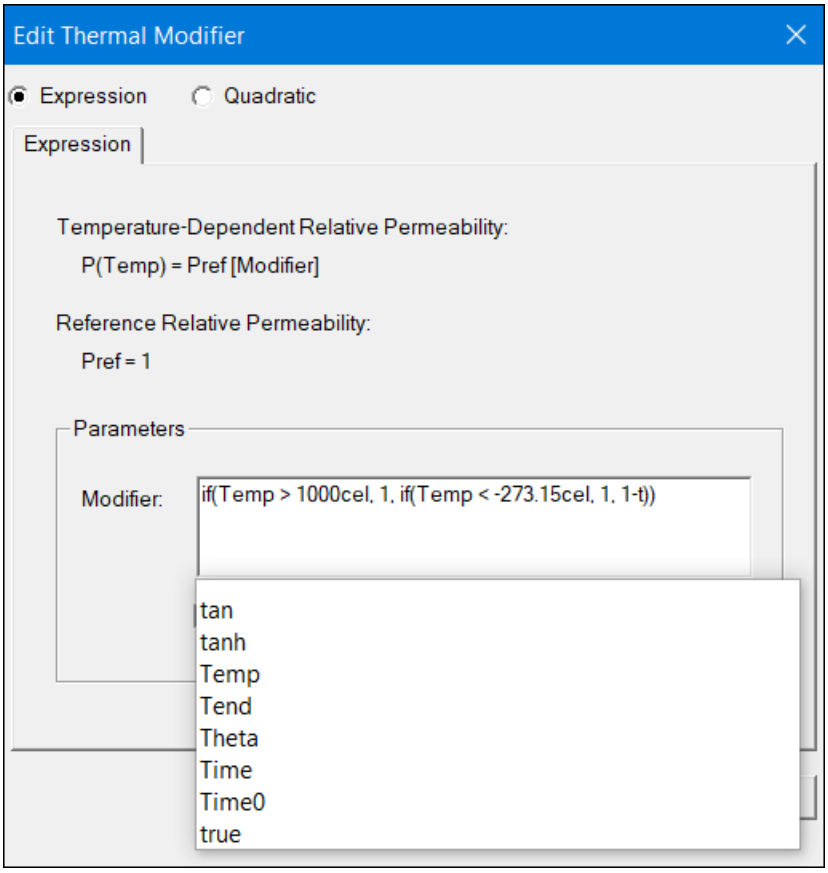
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note the scroll bar, indicating that there are more matches than those currently displayed.

Auto complete also works with thermal modifier expressions for Materials:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Using the Password Manager to Control Access to Resources

Ansys Electronics Desktop lets you specify library resources that require password access, and encryption of those resources. For convenience, the same password can apply to multiple resources. To access the **Password Manager,** click **Tools** > **Password Manager**. This displays the **Password Manager** window.

To Specify a New Password Protected Resource

1. Click **Tools** > **Password Manager**.
2. Click **New**.

   This opens the **New Encrypted Resource** dialog box.

3. Specify the name of the resource that you want to protect and click **OK**.

   This displays the **Enter Passwords** window, which contains radio buttons to let you:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enter Password and confirm for **Full Access** or for **Execute Only Access**.
- **Use Ansys Password** (for execute only). This option does not require you to enter a password, but still encrypts the library.

4. Once you have selected a radio button and specified any necessary passwords correctly, click **OK**.

   The **Password Manager** updates to show the resource.

To Encrypt a Resource

1. Click **Tools** > **Password Manager**.
2. The **Password Manager** window appears.
3. In the Password Manager, select an existing resource to highlight it.
4. Click **Encrypt File**.

   A file browser window appears.

5. Use the drop-down menu to select the appropriate **Files of Type** filter.

   For HFSS, choose Ansoft Library files. Any existing resources in the selected directory will appear.

6. When you have selected the appropriate resource, click **OK**.

   This encrypts the resource.

> **Note:**
>
> The **Expire** resource on option lets you select the date when the password expires for the relevant resource.

Running Ansys Electronics Desktop from a Command Line

Ansys Electronics Desktop includes line arguments that can be included when launching from a command line or terminal prompt. All command-line arguments are case-insensitive. The commands associated batch options can also be used with a Job Management Interface for submitting jobs to Ansys or RSM and other supported schedulers.

Command-line Syntax

```
ansysedt <options> <run command(s)> <project file/script file>
```

It is good practice to put quotation marks around the path to the solver executable, and around the full path to the project. This ensures that spaces in the path or project will not be an issue. The same is true of the design name, if there are spaces.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Run Commands

The following command line run commands are available for Ansys Electronics Desktop. Of the commands (BatchSave, BatchSolve, BatchExtract, RunScript, RunScriptandExit), one or none must be specified as arguments after the solver executable. When none is specified, you may specify a project or archive to open when Electronics Desktop launches, and can only use the **-Iconic** and **-Help** options. The commands are further described below:

- **-BatchSave** [options] **<***file or folder specifier***>**

  Saves a named project or folder containing one or more project files or folders to the current version. You can run this command with the **-Iconic**, **-Logfile**, and **-ng** options.

  If no migration is necessary (*e.g.* a project containing a circuit design): The original file will be overwritten with the new version, and no backup will be made.

  Special behavior when opening backed up files from the user interface: Files located in the AnsysEM_Backup directory would be the original non-migrated files that were backed up during an earlier –batchsave or open. Opening a file located in the AnsysEM_Backup directory via the open file dialog box will invoke the Parasolid migration, and the migrated file will be saved in the parent folder of the AnsysEM_Backup folder with a unique name so as not to overwrite any existing files.

  If a ***file*** is given, **-batchsave** will run on the specified file.

  If a ***folder*** is given, **-batchsave** will run on all files underneath the specified folder.

  .aedt, .aedtz, and .a3dcomp files will be processed.

  Allowed options:

  norecurse: only process files in top level of input folder

  forcearchive: project files are converted, then saved as archive files.

  previewonly: don't perform actual conversion, instead write preview of filenames to batchlog.

  outputfolder=<outputfolder>: save converted files to <outputfolder>, preserving subfolder structure.

- **-BatchSolve <***project file name***>**

  By default, this run command solves all adaptive setups, Optimetrics setups, and sweeps found in the project file. You can run this command with the **-Iconic**, **-Logfile**, **-ng**, and **-WaitForLicense** options. If parallel solve is possible, you can use the **-Distribute** option.

  If you wish to specify which setups **-BatchSolve** completes, you can use additional parameters:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **[*designName*]** – batch solve all setups for the specified design in the project file.
- **[*designName*]:Nominal** – batch solve all nominal setups for the specified design in the project file.
- **[*designName*]:Optimetrics** – batch solve all Optimetrics setups for the specified design in the project file.
- **[*designName*]:[*Nominal/Optimetrics*]:[*SetupName*]** – batch solve the specified Nominal or Optimetrics setup in the specified design.

If you wish to specify whether **-BatchSolve** setups are completed locally or remotely, you can use the following options:

- **-Local** – performs the **-batchsolve** on the local machine.
- **-Remote -machineList** – performs the **-batchsolve** on a remote machine. The <*machineList*> should provide a single hostname.
- **-Distributed -machineList** – performs a distributed **-batchsolve** using a specified machine list.

  The **-machineList** parameter for a **-Distributed** setup can be formatted three ways:

  - **-MachineList list= "<*machine1*>, <*machine2*>, ..."** – machine names (either by IP address or hostname) are separated by commas. If the list contains any whitespace, it must be enclosed in quotation marks. The number of distributed COM engines run on each host is equal to the number of times the hostname appears in the list. That is, if host1 appears in the list once, and host2 appears twice, then one COM engine will run on host1 and two COM engines will run on host2.

    list= accepts the following additional modifiers:

    <*MachineName***>:<***TasksOnMachine***>:<***CoresOnMachine***>:<***RAM Limit in %***>:<***GPUsOnMachine***>**

    Duplicate machine names are not permitted. The integer for <*CoresOnMachine*> must be greater than the integer for <*TasksOnMachine*>. RAM% must be of the form "<num>%" where num is an integer between 1 and 99. Please be sure to leave sufficient RAM for the operating system and applications. If -auto is specified with a machine list, the number of tasks for each machine must be -1.

    Example:

    ```
    list="Orion:4:8:90%:1, Aries:3:12, Pluto:6:12"
    ```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Duplicate machines are not allowed when specifying these additional modifiers. The number of cores must be greater than the number of tasks.

- **-MachineList file= "<***machineListFilepath***>"** – machine names (either by IP address or hostname) are listed in a file (one per line), and you specify the filepath. The number of distributed COM engines run on each host is equal to the number of times the hostname appears in the file. That is, if host1 appears in the file once, and host2 appears twice, then one COM engine will run on host1 and two COM engines will run on host2.

  file= accepts the following additional modifiers, in the file itself:

  <*MachineName***>:<***TasksOnMachine***>:<***CoresOnMachine***>:<***RAM Limit in %***>:<***GPUsOnMachine***>**

  Duplicate machine names are not permitted. The integer for <*CoresOnMachine*> must be greater than the integer for <*TasksOnMachine*>. RAM% must be of the form "<num>%" where num is an integer between 1 and 99. Please be sure to leave sufficient RAM for the operating system and applications. If -auto is specified with a machine list, the number of tasks for each machine must be -1.

  Example:

  ```
  "Orion:4:8:90%:1",
  "Aries:3:12",
  "Pluto:6:12",
  ```

  > **Note:**
  >
  > Duplicate machines are not allowed when specifying these additional modifiers. The number of cores must be greater than the number of tasks.

- **-MachineList num= "<***numberofDistributedEngines***>"** – This format is used when a scheduler (such as LSF, PBS, SGE or HPC) is used to manage the jobs sent to a cluster of hosts. In a scheduler environment, you can specify the number tasks for distributed processing. In this case, you do not specify the machine names after the flag because the names are provided by the scheduler. For example, in the Windows HPC environment, you can write the number of tasks as follows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
-MachineList num=4
```

Distributed setups can also take the following optional arguments. When these are not present, the behavior defaults to single-level distributed solutions with no change in order of precedence among possible distribution types.

The arguments are:

- **includeTypes= <***default***>|<***distribution type 1, distribution type 2***, ...>** – If included distribution types are specified, only the listed distribution types are enabled. If default is specified, the default set of enabled distribution types is used. To see valid distribution types for your design, click **Simulation** > **Analysis Config** to open the **Analysis Configuration** window and view the types on the **Job Distribution** tab.

  If the list contains any whitespace, it must be enclosed in quotation marks. For example:
  `"includeTypes=Frequencies,Mesh Assembly"`
- **excludeTypes= <***default***>|<***distribution type 1, distribution type 2***, ...>** – If excluded distributed types are specified, all distribution types except those listed will be enabled. If default is specified, the default set of enabled distribution types is used. To see a valid distribution types for your design, click **Simulation** > **Analysis Config** to open the **Analysis Configuration** window and view the types on the **Job Distribution** tab.

  If the list contains any whitespace, it must be enclosed in quotation marks. For example:
  `"excludeTypes=Frequencies,Mesh Assembly"`
- **maxLevels= <**1 | 2**>** – the maximum number of levels of job distribution (the current maximum is 2). See: Selecting Optimal Configurations for Distributed Analysis.
- **numLevel1=** – when two-level distribution is selected (**maxLevels=2**), this specifies the number of level 1 tasks.

-Auto [**NumDistributedVariations=<***num***>**]

This flag enables automatic HPC settings and must be used with one of the following options:

- **-machinelist list=**<*machine list*>, with tasks for each machine set to -1
- **-machinelist numcores=**<*num*>, under a scheduler

All design types being solved must support **-auto** or the solve will be aborted.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The NumDistributedVariations option can be used to specify the number of optimetrics variations to solve simultaneously. The default is to solve optimetrics variations sequentially.

Arguments with **-auto** in a scheduler environment:

- **numcores=**<*total number of cores*>

  Total number of cores, and can be used only with -auto and in a scheduler environment.

- **file=**"<*tmachine list file path*>"

  The specified file can contain line delimited machine specifiers as described above.

- **num=**<*tnum distributed tasks*>

  This is the total number of tasks and used only in a scheduler environment.

- **numgpus=**<*tnumber of GPUs to use*>

  This is the total number of GPUs and used only in a scheduler environment. numgpus must be combined with either **num=** or **numcores=**.

  You can also specify how a **-BatchSolve** distributes Optimetrics variations:

- **-auto** – Without additional parameters, the batch log file will specify that Optimetrics variations be solved sequentially. If -auto is specified with a machine list, the number of tasks for each machine must be -1.
- **-auto NumDistributedVariations=<***num***>** – You can specify an integer value greater than 1. This is the number of variations that will be solved in parallel.

- **-BatchExtract <***BatchExtract script file name***> <***project file name***>** – allows the following commands to be executed non-graphically via script and without checking out any GUI licenses:
  - ExportProfile
  - ExportConvergence
  - ExportMeshStats
  - ExportNetworkData
  - ExportNMFData
  - ExportEigenmodes
  - ExportTransientData
  - Update Reports
  - ExportToFile

  A project file *must* be specified when **-BatchExtract** is used. Commands in the script file will only be executed in the specified project.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Important:**
>
> - **-ng** must be used with **-BatchExtract**, or it will fail.
> - Only the scripts listed above are supported for **-BatchExtract**. Including unsupported script commands will terminate script execution.

- **-RunScript <***script file name***>** – runs the specified script. You can use the **-ScriptArgs** option to add one or more arguments to this command, and the **-Iconic** option.
- **-RunScriptAndExit <***script file name***>** – runs the specified script and exits Electronics Desktop. You can use the **-ScriptArgs** option to add one or more arguments to this command. You can also use **-Iconic**, **-Logfile**, and **-WaitForLicense**.

> **Note:**
>
> **-BatchSolve <***DesignName***>** is mutually exclusive with **-RunScriptAndExit <***ScriptName***>**.

- **-monitor** – during non-graphical analysis, you can monitor progress and messages. Progress, warning and info messages are logged to the standard output stream. Error and fatal messages are logged to the standard error stream. Schedulers intercept these streams and provide commands for display of this output. See individual scheduler documentation for specifics.
- **-grpcsrv** – For external cpython scripting support. If not specified with **ansysedt.exe**, the Start GRPV server listens on a port in the default range 50051:51051. If specified, Start GRPV server listening on the the specified port number. An error is issued if the port is already used. For example:

```
ansysedt.exe // Start GRPV server listening on a port in
the default range 50051:51051.
ansysedt.exe -grpcsrv portnumber // Start GRPV server
listening on the the port number. error if the port
already used.
ansysedt.exe -grpcsrv 50051:50150 // Start GRPV server
listening on a port range 50051:50150.
ansysedt.exe -grpcsrv 50051:100 // Start GRPV server
listening on a port range 50051:50151.
```

Job Management from the Command Line

- **-showmonitorjob** – Launch the monitor job dialog.
- **-showsubmitjob** – Launch the submit job dialog.
- **-showselectscheduler** – Launch the select scheduler dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Run Command Examples

A distributed **-BatchSolve** of a specified design's Optimetrics setups, with a specified machine list:

```
C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt -distributed -
machinelist list="255.255.1.1,255.255.1.2" -batchsolve
myDesign:Optimetrics "C:\myProject.aedt"
```

A **-BatchExtract** operation using paths to a script file and a project file:

> **Note:** The example is for an HFSS project. The path, project and design names will vary depending on what the user specifies.

```
ansysedt -ng -batchextract exportToFile.py "C:\Program Files\ANSYS
Inc\v251\AnsysEM\Examples\ElectronicsDesktop\HFSS\RF
Microwave\OptimTee.aedt"
```

Where exportToFile.py contains:

```
oDesktop.RestoreWindow()
oProject = oDesktop.SetActiveProject("OptimTee")
oDesign = oProject.SetActiveDesign("HFSSDesign1")
oModule = oDesign.GetModule("ReportSetup")
oModule.UpdateReports(["XY Plot 1"])
oModule.ExportToFile("XY Plot 1", "exportToFilePy.csv")
```

A **-BatchSolve** of a specified design's nominal setups, run in a minimized window, with a specified log file:

```
ansysedt -Iconic -LogFile "H:\Logs\mylog.log" -BatchSolve
myDesign:Nominal "H:\Projects\MyProject.aedt"
```

Specifying Project Files

Specifying a project file opens that project when Electronics Desktop launches. If **-BatchSolve** is set, the project will also be solved.

You can specify an archive file instead of a project file. If **-Batchsolve** is set, the project will be automatically restored and solved. Otherwise, you are prompted for a restore location, and the project will be restored and opened.

When a **-Batchsolve** is being performed on an archive file, you may also specify **-archiveoptions**:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **overwritefiles** – allows non-project/results-extracted files to overwrite existing files.
- **path= <***projectFilepath***>** – extracts the project file and associated files to the specified path. If not specified, the archive will be extracted into the same directory as the archive file.
- **repackageresults** – Add batchsolve results back to archive file.
- **winpath= <***windowsProjectFilepath***>** – specifies the Windows-specific path to the extracted project file. This is used when a batch job is to be run on a Linux system, but monitored on Windows.

Options

The following options can be associated with one or more of the run commands:

- **-autoextract** – exports profile (as text), convergence (as text), and report data (as CSV) for the requested project/design/setup in a batch job. Once the solve is complete, an export directory is created (for example, "Project1.aedtexport" for a project named "Project1.aedt") that contains a sub-directory for each design name. You can also specify `-autoextract "reports, fieldplots"` to also generate *.aedtplt files for each field plot and possible *.avz file (for import and display in Ensight) for all valid field plots. Export files reside within each design-name directory, and include setup name, design variation, job ID, and problem type, as applicable.

  > **Note:**
  >
  > - The -autoextract option is only valid when used with -BatchSolve
  > - The -autoextract option is automatically added for all Ansys Cloud Direct jobs submitted from Electronics Desktop. There is also an additional "reports" and/or "fieldplots" option that immediately follow "-autoextract". This causes all reports to be exported as CSV files at the end of the batch solve, after the profile and convergence have been exported.
  > - For example, you can specify `-autoextract "reports, fieldplots"` to also generate *.aedtplt files for each field plot and possible *.avz file for all valid field plots.

- **-batchoptions** – for batch jobs, specifies any of the options in **Tools** > **Options**. See additional information.
- **-batchoptionhelp** – opens a window showing -batchoptions help. The paths shown in this window can be used with batchoptions and the Update Registry Get and Set commands. See: Setting or Removing Option Values in Configuration Files: UpdateRegistry Command.
- **-distribute** – distributes a batch solve to multiple machines. This option must be combined with the **-BatchSolve** run command. See: Distributed Analysis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **-help** – opens a window displaying command line options. This can only be used without a run command.
- **-iconic** – runs Electronics Desktop with the window iconified (minimized).
- **-logfile <***filePath***>** – specifies a log file. If none is specified, <project_name>.log will be written to the <project_name>.batchinfo directory.
- **-monitor** – enables batch job output to standard output and standard error streams.
- **-ng** – runs Electronics Desktop in non-graphical mode. This *must* be used with the **-BatchExtract** command.
- **-validateonly** – If true, validation will be performed, but none of the setup will be solved. Allowed values: true, "= 1", false, "= 0".
- **-waitforlicense** – directs Electronics Desktop to wait for unavailable licenses.
- **-scriptargs** – used in conjunction with **-RunScript** or **-RunScriptAndExit**, adds arguments to the specified script. You can pass multiple arguments to **-scriptargs** by surrounding the arguments in quotation marks. For example:

  ```
  ansysedt.exe -scriptargs "Design1 Setup1" -RunScriptAndExit
  C:\temp\test.py
  ```

  In Python, the command line parameter following **-scriptargs** is passed without modification as a single string in the ScriptArgument python variable.

  In VBscript, the command line parameter following **-scriptargs** is split into multiple strings and converted to a VBscript collection which is accessible via the **AnsoftScript.Arguments** collection. To access these arguments, for example:

  ```
  msgbox AnsoftScript.Arguments(0) // Returns Design1
  msgbox AnsoftScript.Arguments(1) // Returns Setup1
  ```

  In either case, **Design1** is taken into Electronics Desktop as the first argument, and **Setup1** as the second argument. If you failed to use quotation marks, **Design1** would be taken as the first argument and **Setup1** would not be understood by Electronics Desktop.

-Batchoptions

All options that are specified through **Tools** > **Options** go into the user-level registry.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- Options are arranged as keys and values (in a structure similar to the *Windows Registry*). However, these options are not a part of the *Windows Registry* but are separately stored and maintained by the Ansys Electronics Desktop software.
- For access to options and functionality beyond what is directly accessible via the user interface or batch options, refer to the documentation of the **UpdateRegistry** tool. This tool is discussed in the following help topic and in the topics that follow it in the same branch of the product help:

  Setting or Removing Option Values in Configuration Files: UpdateRegistry Command

You can override the option registry entries via the **-batchoptions** command line. These overrides apply only to the current Desktop session. The registry setting overrides may be specified on the command line, or may be in a file with the file pathname specified on the command line. Batch jobs can be submitted from the command line or through Electronics Desktop's job submission window.

Large Scale DSO offers two new batchoptions related to the redistribution ability.

- LargeScaleDSO/VarRedistribution, where 0 disables redistribution (default), and 1 enables it.
- LargeScaleDSO/RedistributionLimit, is a positive integer specifying the minimum estimated remaining time (in minutes) for variations to redistribute to another task. The default is 3.

Note:

-batchoptions is only valid for batch jobs. It is ignored if you have not specified **-BatchSolve**, **-BatchSave**, or **-BatchExtract**.

-Batchoptions Examples

The batchoption **CreateStartingMesh** is available for the following products:

HFSS, HFSS-3DLayout, Icepak, Q3D, Q2D, Maxwell3D, Maxwell2D, and Mechanical.

When this option is set, only the initial mesh and manual mesh operations are completed for the batch solution.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> No adaptive meshing occurs for any Icepak or Mechanical solutions. Therefore, after solving, the final mesh will be identical to the starting mesh for these two design types.

This example enables CreateStartingMesh for HFSS, and runs a batch solution of the specified project:

```
ansysedt -batchoptions "'HFSS/CreateStartingMesh'=1" -batchsolve
"D:\projects\MyProject.aedt"
```

See: Additional Examples of -Batchoptions Use.

Export Options Files

The **Tools** > **Options** > **Export Options Files** command writes XML files containing the options settings at all levels to the specified directory. This feature is intended to make it easier for different users to use Ansys Electromagnetics Suite 2025 R1 installed on shared directories or network drives. See: Example Uses for Export Options Features.

Examples and Further Explanations of -batchoptions Use

This section provides further examples and explanations of -batchoptions.

- Example with registry settings specified on the command line
- Example with registry settings specified in a file
- When to Use the -batchoptions Desktop Command Line Option

The following examples use general Desktop and HFSS-specific settings. This feature is available for all desktop products.

- The registry path separator is the slash (/) character.
- Each complete registry key (that is, a registry path and option name) is enclosed in single quotes.
- Registry string values are enclosed in single quotes.
- After the -batchoptions switch, the set of registry keys and values that follows it must be enclosed in double quotes. However, if a batchoptions file is referenced (instead of listing the options directly on the command line), the double quotes are not used around the filename.
- Backslashes in registry key string values must be escaped with another backslash (\\ ), since a backslash by itself is an escape code within strings.

Example with registry settings specified on the command line

```
ansysedt.exe -batchsolve -batchoptions
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
"'Desktop/Settings/ProjectOptions/NumberOfProcessors'=4
'HFSS/NumCoresPerDistributedTask'=2   (* This option is not
currently available from the Add Batchoption dialog box.)
'Desktop/ProjectDirectory'='C:\\projects\test'" projectname.aedt
```

This command line overrides registry values of the NumberOfProcessors (Desktop/Settings/ProjectOptions key) and ProjectDirectory (Desktop key) options.

Note:

- Multiple registry settings may appear in a single -batchoptions value, separated by whitespace.
- The -batchoptions value must be enclosed in double quotes if it contains any whitespace.

Example with registry settings specified in a file

```
ansysedt.exe -batchsolve -batchoptions <filename>
projectname.aedt
```

where the referenced file, <*filename*>, contains:

```
$begin 'Config'
  'Desktop/Settings/ProjectOptions/NumberOfProcessors'=4
  'HFSS/NumCoresPerDistributedTask'=2
  'Desktop/ProjectDirectory'='C:/projects/test'
$end 'Config'
```

This command overrides the registry values of the NumberOfProcessors (Desktop/Settings/ProjectOptions key), NumCoresPerDistributedTask (HFSS key), and ProjectDirectory (Desktop key) options. These overrides apply only to the current Electronics Desktop session.

Note:

- The -batchoptions **<*filename*>** value must be enclosed in double quotes if it contains whitespace
- The **$begin 'Config'** and **$end 'Config'** lines are required

For additional options you can override from the command line with **-batchoptions**, see:

- For -batchoptions Use: Project Directory and Lib Paths
- For -batchoptions Use: TempDirectory
- For -batchoptions Use: Various Desktop Settings

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For -batchoptions Use: HFSS 3D Layout Options with Paths
- For -batchoptions Use: HFSS Options with Paths

When to Use the -batchoptions Desktop Command Line Option

You can set analysis parameters for batch mode jobs using the graphical user interface (GUI). For example, you can set all options for a batch job using the **Add Batch Option** dialog box, which is accessed through the **Submit Job To** dialog box. These parameter settings include the following solver options (several examples, not a complete list):

- HPCLicenseType
- tempdirectory
- Desktop/AutoExtractReports
- Desktop/Settings/ProjectOptions/MumberOfProcessors
- <*design_type*>/DefaultProcessPriority
- <*design_type*>/EnableGPU and/or EnableGPUForEye – Applicable to Circuit Design, EMIT, HFSS, HFSS 3DLayout, and Maxwell 3D (also EnableGPUForSBR for HFSS only)
- <*design_type*>/MPIVendor (either “Intel” or “Microsoft”) – Applicable to HFSS, HFSS 3DLayout, Icepak, Maxwell 2D, Maxwell 3D), and Q3D

For graphical analyses that do **not** use batch mode, you specify the analysis parameters using the GUI (**Tools** > **Options** > **General Options** or **HPC and Analysis Options**). These settings are written to the registry when you exit the Electronics Desktop program. The settings are read from the registry when the application is started. Therefore, when you start the Electronics Desktop application, all settings retain the values from the previous session of the same user on the same machine. If there was not a previous session of the same user on the same machine, then the values are obtained from other registry configuration files or from a default value.

When running a batch analysis, any setting that is not specified using the **-batchoptions** command line option is taken from the registry. This value is typically the setting from the last session of the same user on the same machine. However, the **-batchoptions** command line option allows you to override the parameter with values specified on the command line or in a batchoptions file. The values specified using the **-batchoptions** command line option only apply to the batch job, and do not affect the parameter values stored in the registry.

If important **-batchoptions** values are not specified when running a batch job, the paramaters could be affected by an interactive session running on the same host by the same user. Parameter changes can occur if the user sets an option in the GUI and then exits the program, or if another process that accesses the registry exits. To be sure of the desired batch job outcome, avoid changing options in concurrent interactive sessions or include the desired -batchoptions in the command line.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For -batchoptions Use: Project Directory and Lib Paths

The PersonalLib, syslib and userlib settings are a little different from other settings. If the final directory name is different from what is expected, then PersonalLib, syslib or userlib is appended as a final directory. In addition, these settings may come from a different registry value if the registry values shown above are not set.

Registry Key	Default Value	Units or Values	Description
Desktop/ProjectDirectory	subdirectory of user's HOME directory or “Documents” directory	Directory pathname	Directory where new projects are created
Desktop/PersonalLib	PersonalLib subdirectory of user's HOME directory or “Documents” directory	Directory pathname	Directory PersonalLib is appended if final directory is not PersonalLib
Desktop/syslib	syslib subdirectory of installation directory	Directory pathname	Directory syslib is appended if final directory is not syslib
Desktop/userlib	userlib subdirectory of installation directory	Directory pathname	Directory userlib is appended if final directory is not userlib

For -batchoptions Use: TempDirectory

Registry Key	Default Value	Units or Values	Description
TempDirectory	Set by installer	-	Directory for temporary files

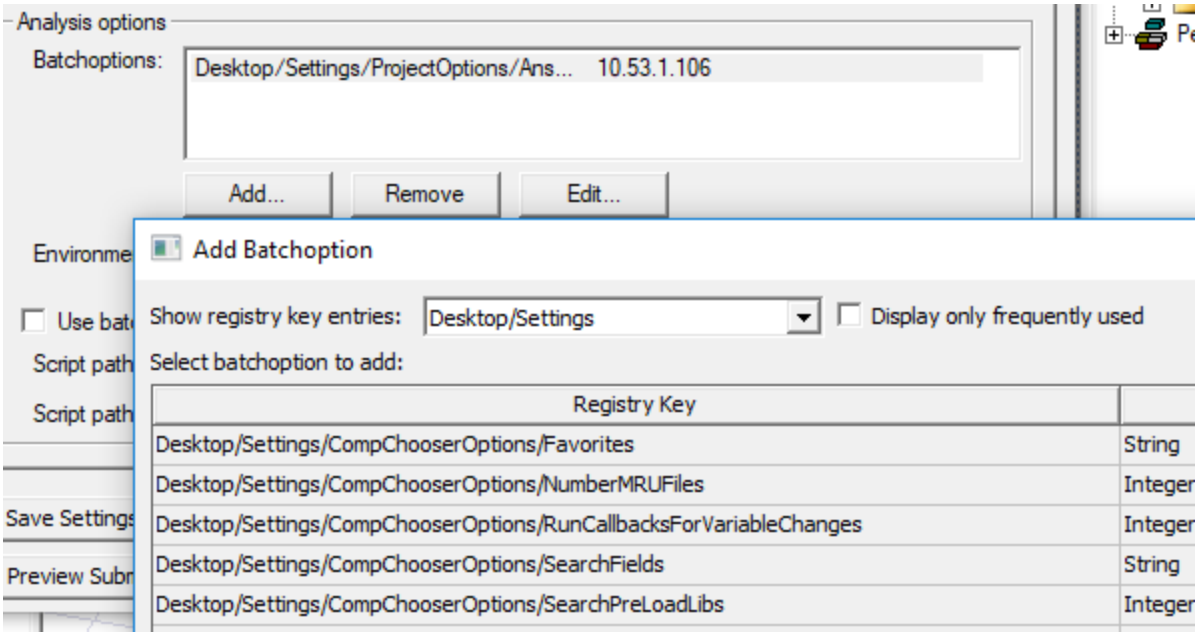
For -batchoptions Use: Various Desktop Settings

Note that most of these options only affect the GUI. To view these options from the *Submit Job To* window:

1. In the *Analysis options* area of the *Submit Job To* window, click **Add...**.
2. In the *Add Batchoption* window that appears, select **Desktop/Settings** from the **Show**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

registry key entries drop-down menu.



Registry Key	Default Value	Units or Values	Description
Desktop/Settings/ProjectOptions/AnimationMemory	200	Megabytes (MB)	Stop animations when available memory falls below this value
Desktop/Settings/ProjectOptions/AnsoftCOMPreferredIPAddress (see note below table)	“” (empty string)	IP address (as a string)	IP address used to connect from COM engines to ansysedt.exe

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Registry Key	Default Value	Units or Values	Description
Desktop/Settings/ProjectOptions/AnsysEMPreferredSubnetAddress (see note below table)	“” (empty string)	IP address (as a string)	Subnet used to connect COM engines to ansysedt.exe – allowed formats are: • IPv4 network prefix in CIDR notation Example: 123.123.123.0/24 • IPv4 network prefix with /subnet mask appended Example: 123.123.123.0/255.255.255.0
Desktop/Settings/ProjectOptions/AutoSaveInterval	10	edits	Number of edits to allow between autosaves
Desktop/Settings/ProjectOptions/AutoShowMessageWindow	1 (true)	0 (false) or 1 (true)	Show message window on new messages
Desktop/Settings/ProjectOptions/AutoShowProgressWindow	0 (false)	0 (false) or 1 (true)	Show progress window when starting a simulation
Desktop/Settings/ProjectOptions/DiskLimitForAbort	0	Megabytes (MB)	A warning is issued when available disk space falls below this value
Desktop/Settings/ProjectOptions/DoAutoSave	1 (true)	0 (false) or 1 (true)	Enables autosaves if true
Desktop/Settings/ProjectOptions/DrawStateIconsInProjectTree3	1 (true)	0 (false) or 1 (true)	Change icon when selection does not match active window

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Registry Key	Default Value	Units or Values	Description
Desktop/Settings/ProjectOptions/ExpandMessageTreeOnInsert	1 (true)	0 (false) or 1 (true)	Ensure that new messages are visible in the message window tree
Desktop/Settings/ProjectOptions/ExpandOnInsert	0 (false)	0 (false) or 1 (true)	Expand project tree on insert
Desktop/Settings/ProjectOptions/HighlightActiveContextInProjectTree2	1 (true)	0 (false) or 1 (true)	Emphasize active command context (menu and toolbars)
Desktop/Settings/ProjectOptions/SavePreviewImagesInProject File	1 (true)	0 (false) or 1 (true)	Save preview images in project file
Desktop/Settings/ProjectOptions/UpdateReportOnFileOpen	0 (false)	0 (false) or 1 (true)	Update reports on file open

Note:

The preferredIP address and preferred subnet address settings are mutually exclusive. If both are specified to be non-empty strings, then the preferred IP address takes precedence, and the preferred subnet address is ignored. This feature is typically used for cluster environments using batch solves. The setting can be made via batchoptions but can also be done via UpdateRegistry.

For -batchoptions Use: HFSS Options

Here are the HFSS batch ptions with the complete key path, option names, and values. You can find most of the available batch options in a help dialog box, if you use a Command Window to open the -batchoptions help:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Registry Key	Default Value	Values	Description
HPCLicenseType	“pack”	“pool”, “pack”, or "none"	For HFSS Designs: “pool” = multiprocessing enabled by HPC licensing; “pack” = multiprocessing enabled by HPC Pack licensing
tempdirectory	string		Temp directory path
HFSS/AllowOffCore	1	0 (false), 1 (true)	Specifies if the solver will be able to use off core memory.
HFSS/CreateStartingMesh	1	0 (false), 1 (true)	If true, the starting mesh (with mesh operations if applicable) will be created, but none of the setup will be solved. Allowed values: 0 (false), 1 (true)
HFSS/DefaultProcessPriority	Normal	"Critical", "Above Normal", "Normal", "Below Normal", "Idle"	Specifies the default priority of distributed tasks.
HFSS/EnableGPU	0	0 (false), 1 (true)	Allow GPU to be used for transient and frequency domain solutions. GPU acceleration will

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Registry Key	Default Value	Values	Description
			be disabled when automatic HPC setting is selected. Allowed values: 0 (false), 1 (true)
HFSS/EnableGPUforSBR	1	0 (false), 1 (true)	Allow GPU to be used for SBR+ solutions. Allowed values: 0 (false), 1 (true)
HFSS/HPCLicenseType	“pack”	“pool” or “pack” or "none"	For HFSS Designs: “pool” = multiprocessing enabled by HPC licensing; “pack” = multiprocessing enabled by HPC Pack licensing
HFSS/MPIVendor	"Intel"	“Microsoft”, “Intel”.	Which MPI libraries to use for remote communication. Type: String
HFSS/MPIVersion	"Default"	"Default", "2018", "2021"	This setting specifies the MPI version. If not specified, then the default version for the MPI vendor will be used. The valid values depend on the MPI vendor. Allowed values: "Default", "2018", "2021"
HFSS/RemoteSpawnCommand	SSH	"RSH", "SSH", "Scheduler"	This setting

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Registry Key	Default Value	Values	Description
			specifies the command that will be used to launch distributed solver jobs. The value Scheduler is only valid for jobs running under an LSF or SGE scheduler, and only if the MPI Vendor is Intel. Type: String.
HFSS/ValidateOnly	1	0 (False), 1 (True).	If true, validation will be performed, but none of the setup will be solved.
HFSS/SolveAdaptiveOnly		0 (False), 1 (True).	If true, only the adaptive portion of the setup will be solved. Applies only if a single setup is selected.

Batchoptions: Override Registry Entry

Any registry entry can be overridden via a command-line option.

Example: batchoptions specified on a command line

```
ansysedt -batchsolve -batchoptions
"'Desktop/Preferences/ProjectOptions/NumberOfProcessors'=4 'HFSS 3D
Layout Design/NumCoresPerDistributedTask'=2" projectname.aedt
```

Example: batchoptions specified in a file

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
ansysedt -batchsolve -batchoptions numprocfile projectname.aedt
```

where numprocfile contains

```
$begin 'Config'
   'Desktop/Preferences/ProjectOptions/NumberOfProcessors'=4
   'HFSS 3D Layout/NumCoresPerDistributedTask'=2
$end 'Config'
```

Running Ansys Electronics Desktop from a Windows Remote Terminal

When running HFSS from a remote terminal, there are some performance and behavior issues to consider. These issues are due to the interaction of bandwidth/opengl drivers/remote-terminal-protocol

- Showing axes when interactively drawing objects will slow the performance.
- Remote OpenGL performance will be slower in general. Graphics card and driver quality helps.
- All 3D windows will be closed when you switch from remote PC to a console or from a console to remote. This is to avoid display/opengl instability during the switch.
- Grid will not be turned off while viewing a plot from a remote desktop. The mouse over highlights on 2D plots may appear as not totally overlapping the line color or as thin dotted lines.

Using Windows HPC Commands

HPC Integration allows you to submit jobs directly using Ansys Electromagnetics command line arguments for batch solves. Ansys Electromagnetics products must be accessible from the same directory on all machines. The Ansys Electromagnetics command line syntax is documented here. You must pass in a -distributed flag as part of the Ansys Electromagnetics command line arguments if you want to run a distributed simulation.

For information concerning the supported HPC software, go the the Platform Support webpage and click the links to the following topics:

- *Ansys 2025 R1 - Job Schedulers and Queuing Systems Support*
- *Ansys 2025 R1 - Message Passing Interface Support for Parallel Computing*

Before running a job you must click **Tools** > **Job Management** > **Select Scheduler** and use the dialog box to designate the head node of a cluster. You can then click **Tools** > **Job Management** > **Submit Job** to submit the batch commands for the job.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Customizing Ansys Electronics Desktop with Ansys ACT

With Ansys ACT, you can create custom applications or extensions to customize supported Ansys products, including Electronics Desktop.

> **Note:**
>
> The **ACT Extensions** window and the design wizards it contains (*5G Wizard, HFSS Antenna Design Toolkit, HFSS-EMA3D Link,* and *Maxwell Eccentricities*) are only available for the Windows version of the Ansys Electronics Desktop software. These items are not available when the software is installed on a Linux platform.

An ACT guided process extension enables you to leverage both the functionality of Electronics Desktop and the scripting capabilities of the Workbench/AIM framework API. You can manipulate existing features and simulation components, organizing them as needed to produce a custom automated process. A guided process extension is exposed in Electronics Desktop as a wizard that provides step-by-step simulation guidance within the application workflow.

To access ACT functionality in Electronics Desktop, open the ACT Home page by clicking **View** > **ACT Extensions**.

For more information, see ACT Simulation Wizards and Electronics Desktop Wizard in the *Ansys ACT Developer's Guide*.

Installing PyAEDT (Beta)

PyAEDT is a Python library that interacts with the AEDT API to make scripting simpler for the end user. It supports all AEDT 3D products (HFSS, Icepak, Maxwell 3D, and Q3D Extractor), 2D tools, Ansys Mechanical, EMIT, Circuit tools like Nexxim, system simulation tools like Twin Builder, and layout tools like HFSS 3D Layout and EDB. Additionally, it enables the end user to have a CPython interface with AEDT. Its class and method structures simplify operation for the end user, enabling more Pythonic code while reusing information as much as possible across the various APIs.

Documentation for PyAEDT can be found online at: https://aedt.docs.pyansys.com/version/stable/

Installing PyAEDT adds three items to the **Tools** > **Toolkit** > **PersonalLib** menu and to the **Automation** tab:

- **Console** – launches the PyAEDT console.
- **Jupyter Notebook** – launches Jupyter Notebook (a computational notebook) in an internet browser.
- **Run PyAEDT Script** – launches a file browser allowing you to select a Python script to run via PyAEDT.

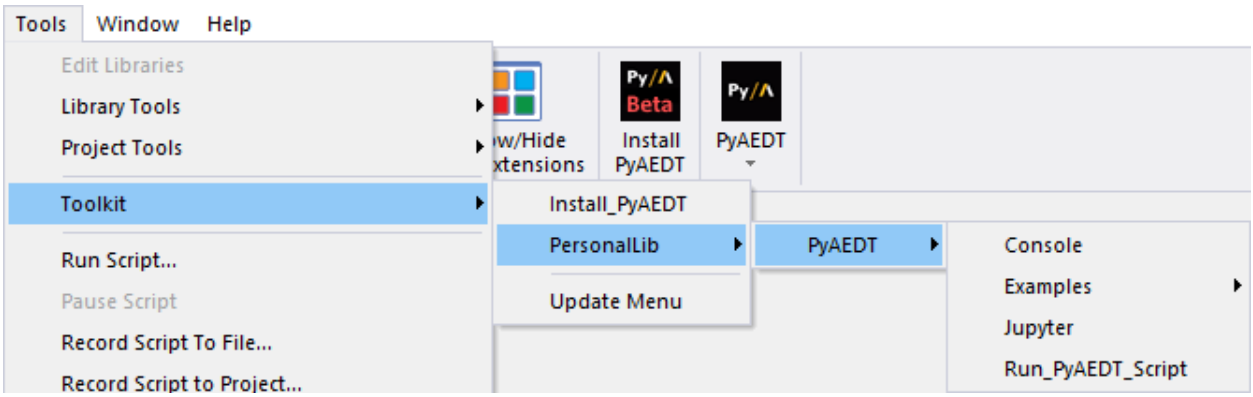
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

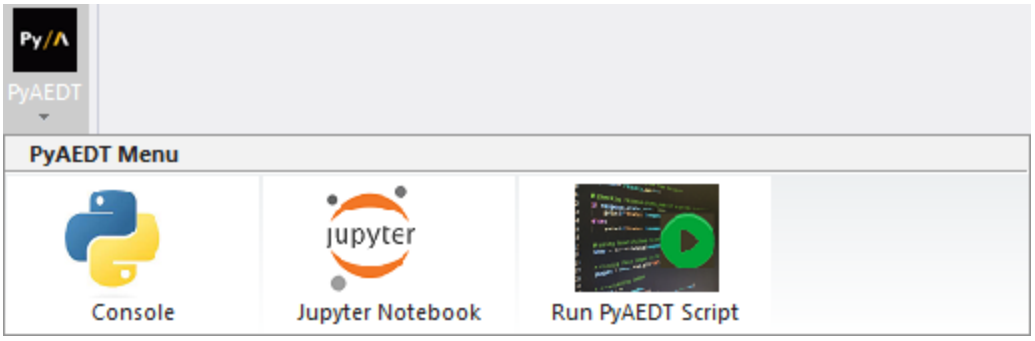
To install PyAEDT:

- From the **Automation** tab, click **Install PyAEDT**.

A web browser launches, and takes you to detailed installation instructions.

When installation is complete, the **Tools** > **Toolkit** > **PersonalLib** menu and the **Automation** tab update to display PyAEDT menu options:





Debug Logging

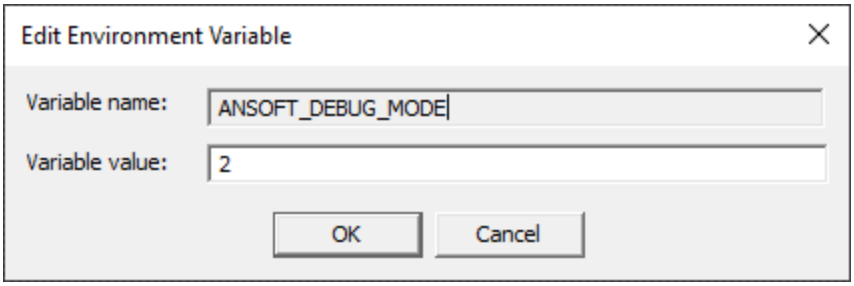
Electronics Desktop provides error logging capabilities.

To begin logging errors:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Tools** > **Debug Logging**.

   The **Edit Environment Variable** window appears, showing the ANSOFT_DEBUG_ MODE variable.

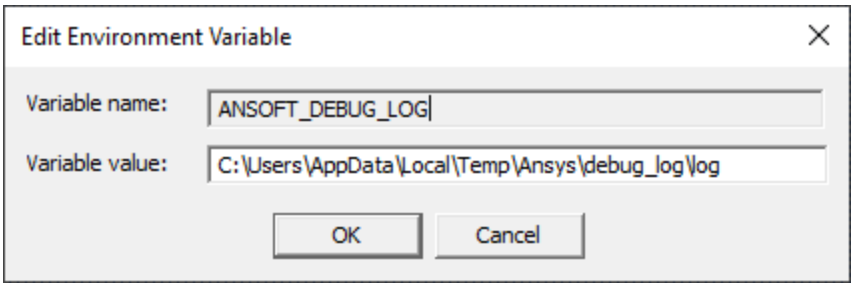


2. Enter a **Variable value**.

   The default value is 2, and should not be changed unless directed by technical support.

3. Click **OK**.

   The **Edit Environment Variable** window updates to display the ANSOFT_DEBUG_LOG variable.



4. Enter a folder path where log files (*.log) will be stored.
5. Click **OK**.

   Errors are logged to the specified folder until you disable logging.

To stop logging errors:

- Click **Tools** > **Stop Debug Logging**.

Antennas: Application Specific Modeling Guide

You can access the antennamdgd.pdf under the installation directory: "...\ANSYS Inc\v251\AnsysEM\Help\HFSS".

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This Application Specific *Modeling Guide* is written for Ansys Electronics Desktop beginners as well as experienced users who are using HFSS for the first time. This brief manual provides guidelines for modeling antennas in HFSS.

Spiral Inductors on Silicon Substrate: Application Specific Modeling Guide

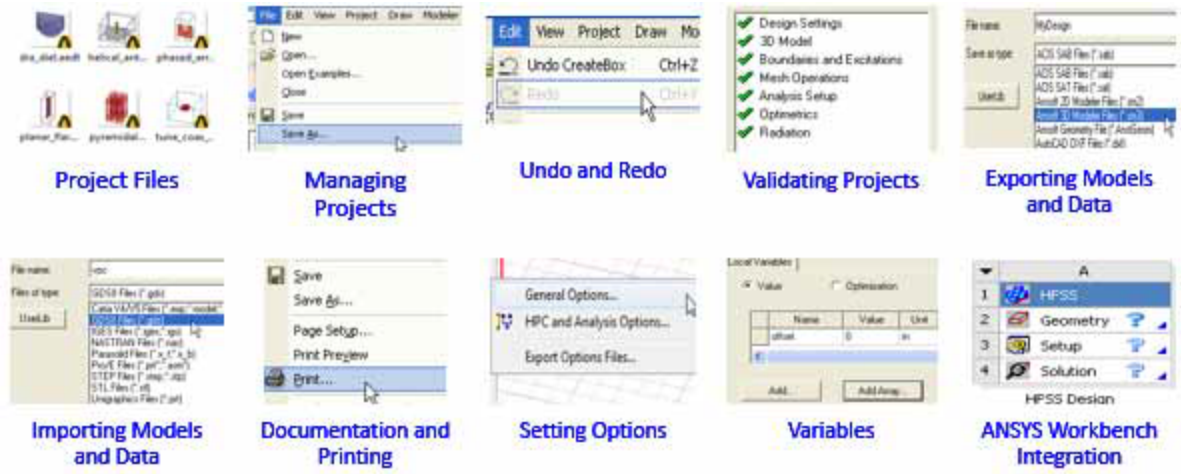
You can access Spiralmodelingguide.pdf under "<*install_dir*>\ANSYS Inc\v251\AnsysEM\Help\HFSS".

This Application Specific *Modeling Guide* is written for Ansys Electronics Desktop beginners as well as experienced users who are using HFSS for the first time. This brief manual provides guidelines for modeling spiral inductors in HFSS.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 - Working with Ansys Electronics Desktop Projects

An Ansys Electronics Desktop project is a folder that includes one or more models, or *designs*. Each design ultimately includes a geometric model, its boundary conditions and material assignments, and field solution and post-processing information, or a schematic or netlist.



A new project called *Projectn* is automatically created when the software is launched. By option, a design named Design*n* is automatically created for a new project. You can also open a new project by clicking **File** > **New**. In general, use the **File** menu commands to manage projects. If you move or change the names of files without using these commands, the software may not be able to find information necessary to solve the model.

> **Note:**
>
> Not all options and capabilities documented in this section apply to all design types.

Ansys Electronics Desktop Files

When you create any project in Ansys Electronics Desktop, it is given an .aedt file extension and stored in the directory you specify. For Legacy projects, the transition is handled as follows:

- When opening a legacy project in interactive mode, a dialog box appears, informing you that the legacy file extension is no longer supported and that if you continue, the project will be converted to the new .aedt extension. If you agree, the project and results directory are renamed/moved to the new file extensions immediately. The read continues with the standard code for reading previous version projects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The Ansys Electronics Desktop 2025R1 version cannot open projects created with 2022R2 or earlier that were using ACIS modeler.
- Attempts to run batch solve or non-graphical with a legacy project return an error. There is no automatic/hidden conversion of file extensions. Note that the existing BatchSave command can be used to convert many projects to the new extension and version.
- Workbench integration has not been modified. It continues to open legacy projects without a warning and copies results from legacy to the new extension.

HFSS Files

When you create an HFSS project, it is given an *.aedt* file extension and stored in the directory you specify. Any files related to that project are also stored in that directory.

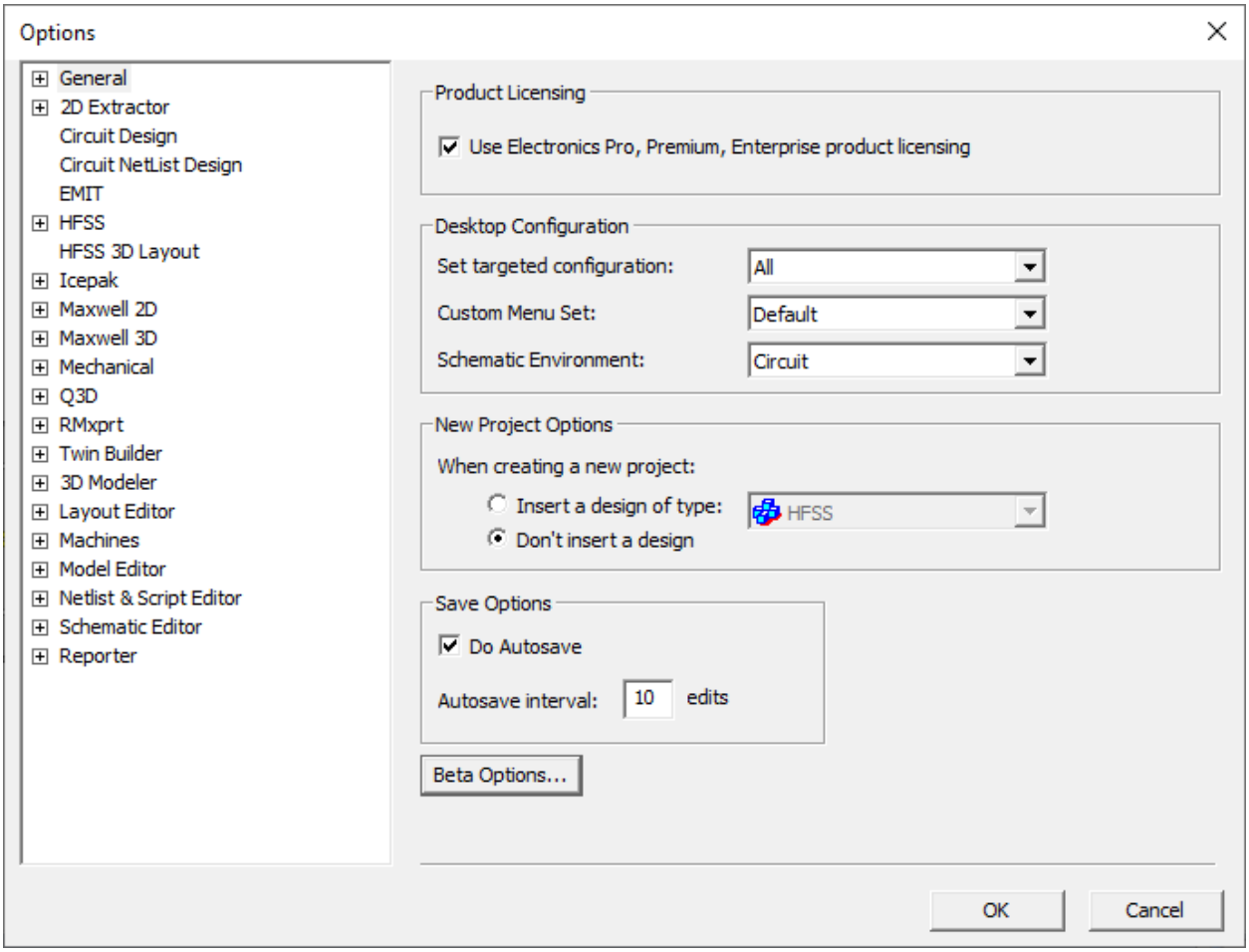
Some common HFSS file and folder types are listed below:

.aedt	HFSS or other Ansys Electronics Desktop project.
.aedti	Ansys Electronics Desktop Import file (*.aedti) format is for importing exported projects from ANSYS Discovery, an ANSYS Workbench tool. This format allows import of side by side *.aedti and the correponding geometry definition *.x_t into the Ansys Electronics Desktop, and creates a ready-to-solve HFSS project.
design_name. ***aedt*****results**	HFSS folder containing results data for a design. It resides in the project.aedtresults folder.
project_name. **aedtresults**	HFSS folder containing results data for a project.
*project_name.***asol**	The .asol file contains the database of all solved variations and where the resulting data is stored in the *design*.aedtresults folder. This file is stored in the *project_name*.**aedtresults** folder.
.hfss	HFSS file extension before Ansys Electronics Desktop.
.pjt	Ansys Electronics Desktop version 8.5 and earlier project.
.anfp	Ansys PCB neutral file

Setting Project Options

Tools > **Options** > **General Options** opens the **Options** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The left pane provides access to the following sets of options:

- General
- 2D Extractor
- Circuit Design
- Circuit Netlist Design
- HFSS
- HFSS 3D Layout
- Icepak
- Maxwell 2D
- Maxwell 3D
- Mechanical
- Q3D
- RMxprt

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Twin Builder
- 3D Modeler
- Layout Editor
- Machines
- Model Editor
- Netlist and Script Editor
- Schematic Editor
- Reporter

Use **Tools** > **HPC and Analysis Options** to specify active configuration per design type, queuing, distributed memory vendor, HPC licensing, select a MPI vendor, and whether to enable GPU for transient solves.

The **Tools** > **Options** > **Export Options Files** command writes xml files containing the Options settings at all levels to the specified directory. The **Tools** > **Options** > **Export Options** feature is intended to make it easier for different users to use Ansys Electromagnetics tools installed on shared directories or network drives. The Example Uses for Export Options Features section outlines some use cases enabled by this feature.

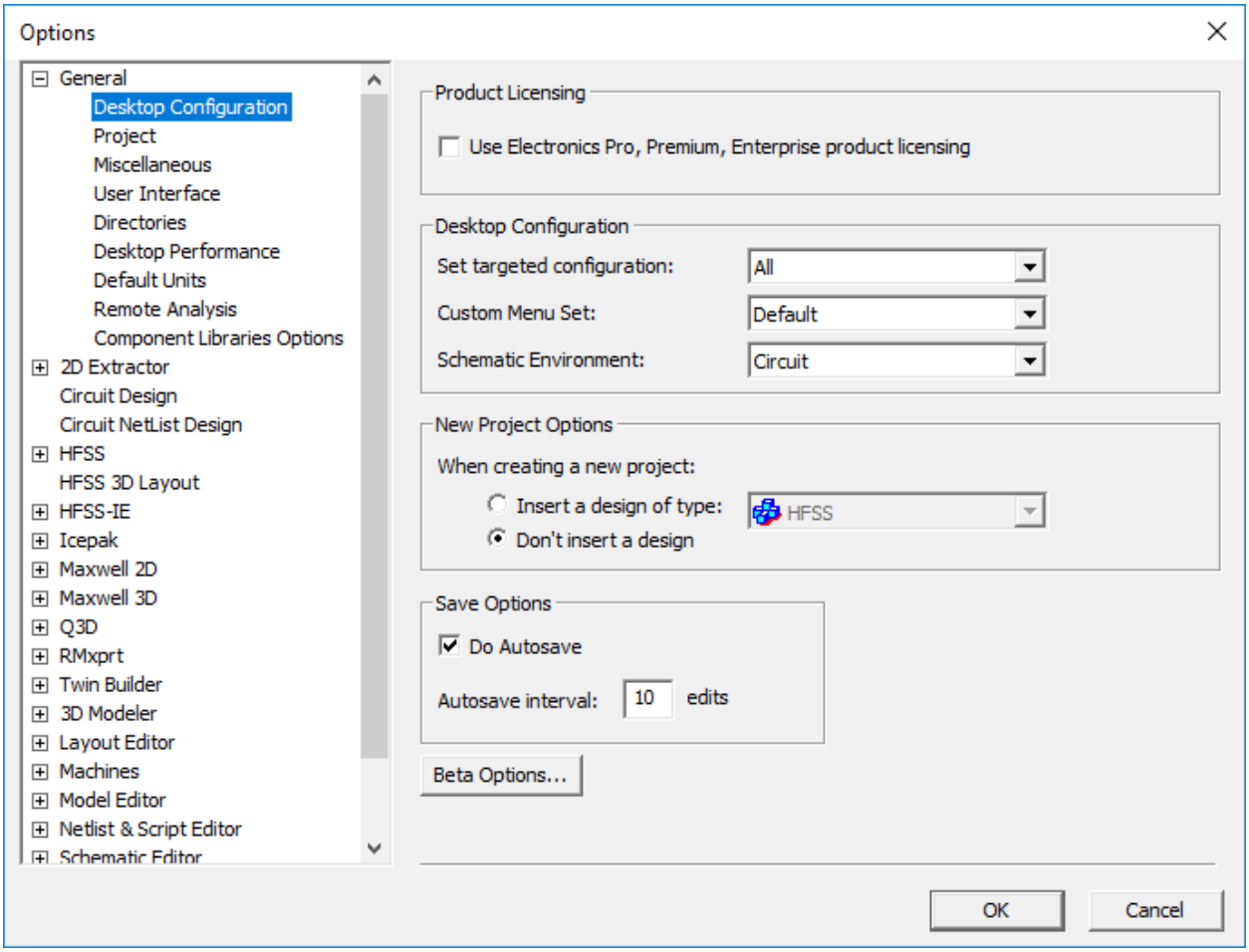
Setting General Options

To set general options in Ansys Electronics Desktop:

1. Click **Tools** > **Options** > **General Options**.

   The **Options** window opens with the Desktop Configuration options selected by default.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the left pane, select the entries below **General** to display the associated options:

- Desktop Configuration
- Project
- Miscellaneous
- User Interface
- Directories
- Desktop Performance
- Default Units
- Remote Analysis
- Component Libraries Options

2. Click each entry and make the desired selections.

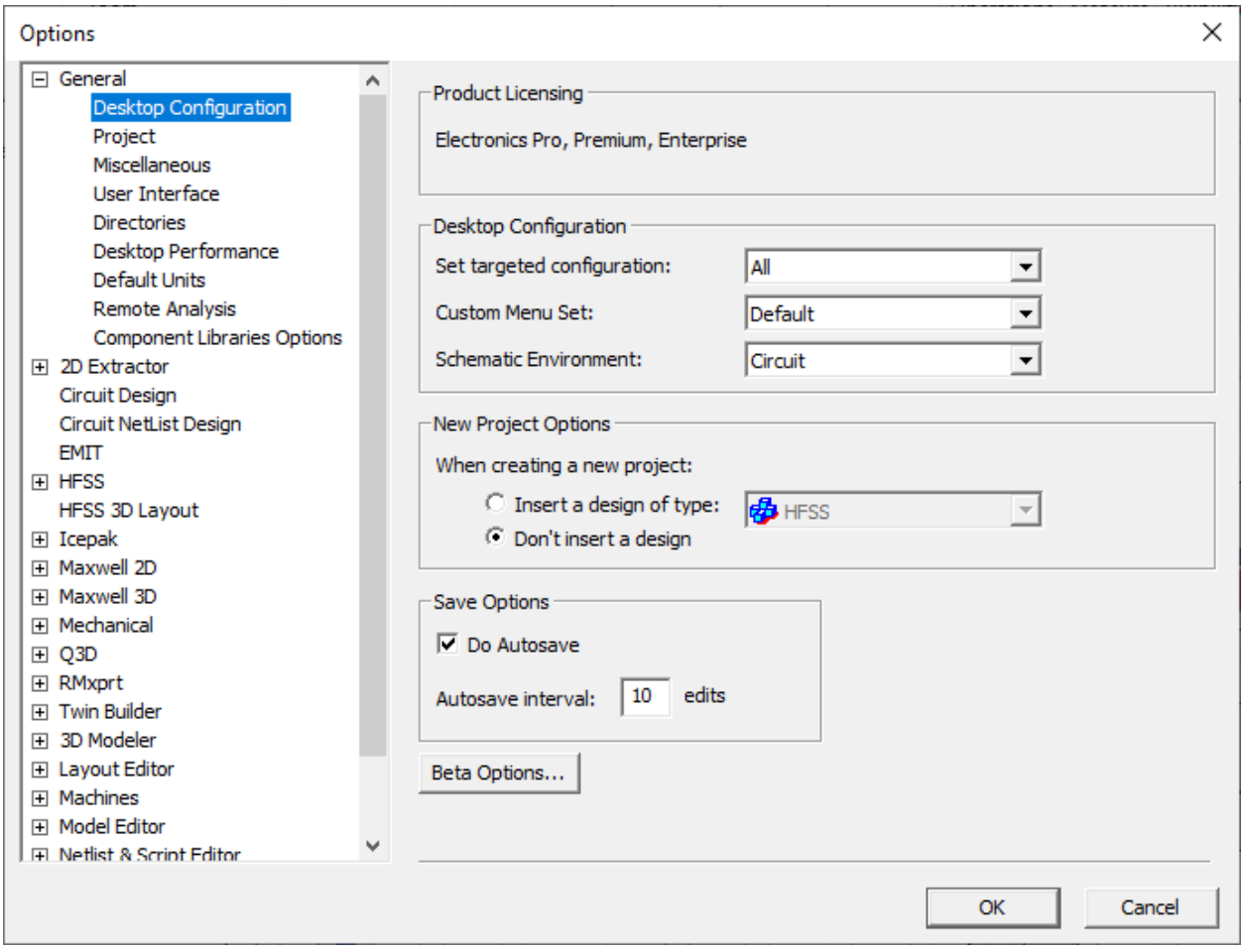
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK**.

> **Note:**
>
> Not all option settings are applied immediately in the current design. Some settings take effect in newly created designs.

General Options: Desktop Configuration

Under General Options, **Desktop Configuration** options allow you to customize Electronics Desktop in a way that suits your work priorities. These include options for menus, new projects, save intervals, and beta features.



Ansys Electronics Desktop now uses Electronics Pro, Premium, Enterprise (PPE) product licensing. Legacy product licensing and DSO are no longer supported.

With PPE, HPC licensing is used to enable all cores, GPUs, and distributed tasks.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more details, see Setting HPC and Analysis Options.

In the **Desktop Configuration** area, options include:

- **Set Targeted Configuration** – Choosing a Targeted Configuration changes the Custom Menu Set and Schematic Environment for your area of focus.

  Choices are:

  - **All** – Default option. Sets the **Custom Menu Set** to **Default**, the **Schematic Environment** to **Circuit**, and the default design type to **HFSS**.
  - **EM** – Electromagnetic focus. Sets the **Custom Menu Set** to **EM**, the **Schematic Environment** to **Twin Builder**, and the default design type to **Maxwell 3D**.
  - **RF** – Radio Frequency focus. Sets the **Custom Menu Set to** RF, the **Schematic Environment** to **Circuit**, and the default design type to **HFSS**.
  - **SI** – Signal Integrity focus. Sets the **Custom Menu Set** to **SI**, the **Schematic Environment** to **Circuit**, and the default design type to **HFSS 3D Layout**.
  - **Twin Builder** – Sets everything to default to **Twin Builder**.
- **Custom Menu Set** – Changes which menu options are available in Electronics Desktop.

  Choices are:

  - **Default** – All solvers appear on the **Project** menu.
  - **EM** – Only electromagnetics solvers appear on the **Project** menu.
  - **RF** – Only radio frequency solvers appear on the **Project** menu.
  - **RF.0** – All solvers appear on the **Project** menu. The **HFSS RF Setup** menu appears.
  - **SI** – Only signal integrity solvers appear on the **Project** menu.
  - **SI1.0** – Only signal integrity solvers appear on the **Project** menu. The **Import**, **Solution Setup**, **Automation**, and **Definitions** menus appear.
  - **SI2.0** – All solvers appear on the **Project** menu. The **HFSS SI Setup** menu appears.
  - **Twin Builder** – Only Twin Builder appears on the **Project** menu.
- **Schematic Environment** – Select **Circuit**, **Twin Builder**, or **Maxwell**.

In the **New Project Options** area, options include:

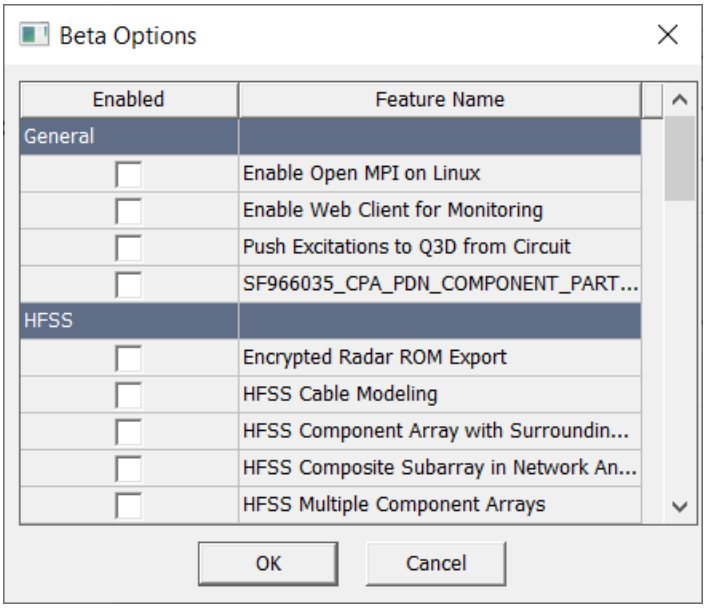
- **When Creating a New Project** – Select whether to insert a design when creating a new project. If you choose to insert a design by default, use the drop-down menu to select the default design type.

In the **Save Options** area, options include:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Do Autosave** – Electronics Desktop has autosave enabled by default. Deselect this option to disable it.
- **Autosave Interval** – When **Do Autosave** is enabled, select the number of edits at which Electronics Desktop autosaves.

Click the **Beta Options** button to open a window listing beta options. You may need to scroll or size the window to view all options.

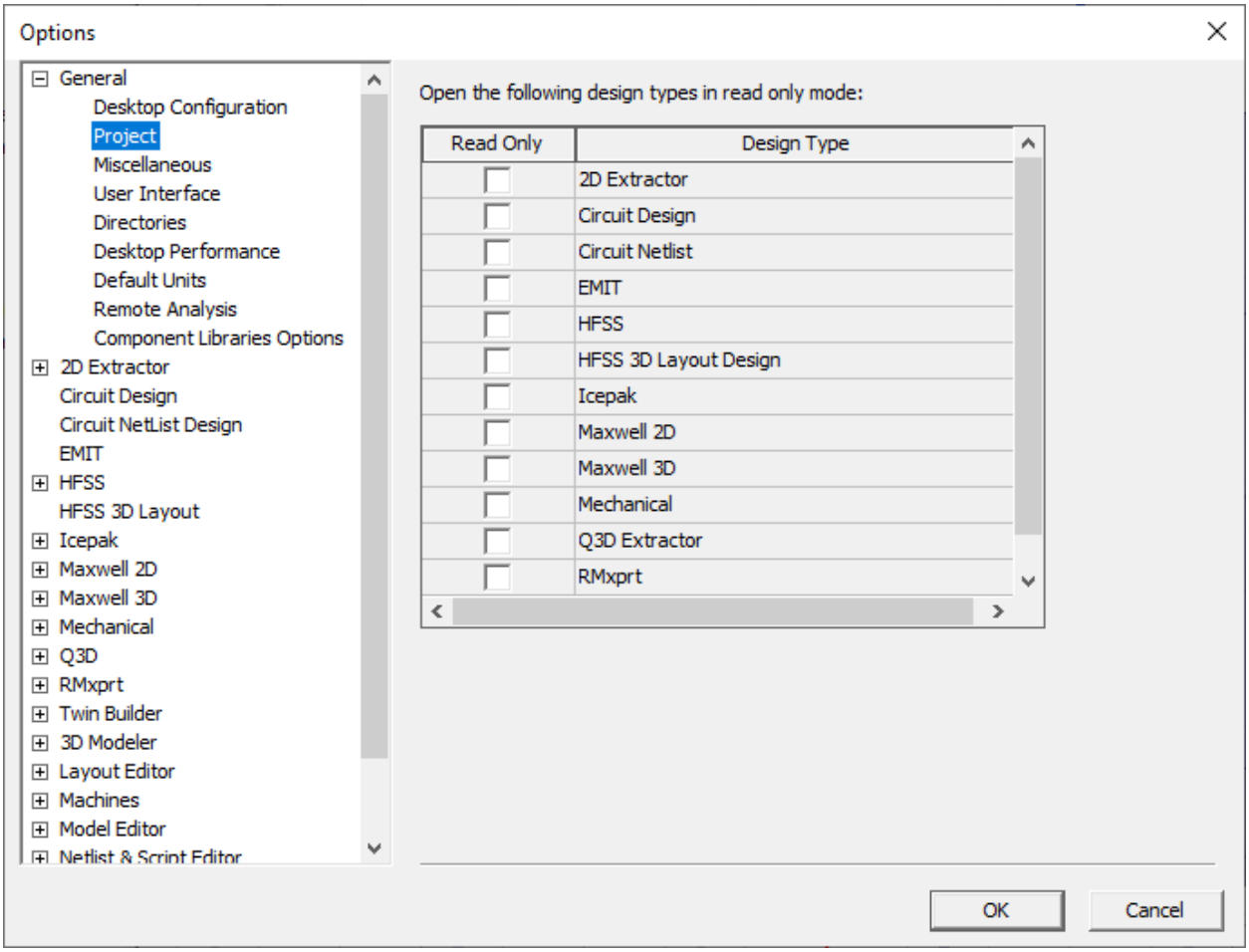


From there, enable or disable options, and click **OK**. You will need to restart to enable your selections.

General Options: Project Options

Under General Options, **Project** options allow you to select design types to open in read-only mode. This can help prevent accidental changes.

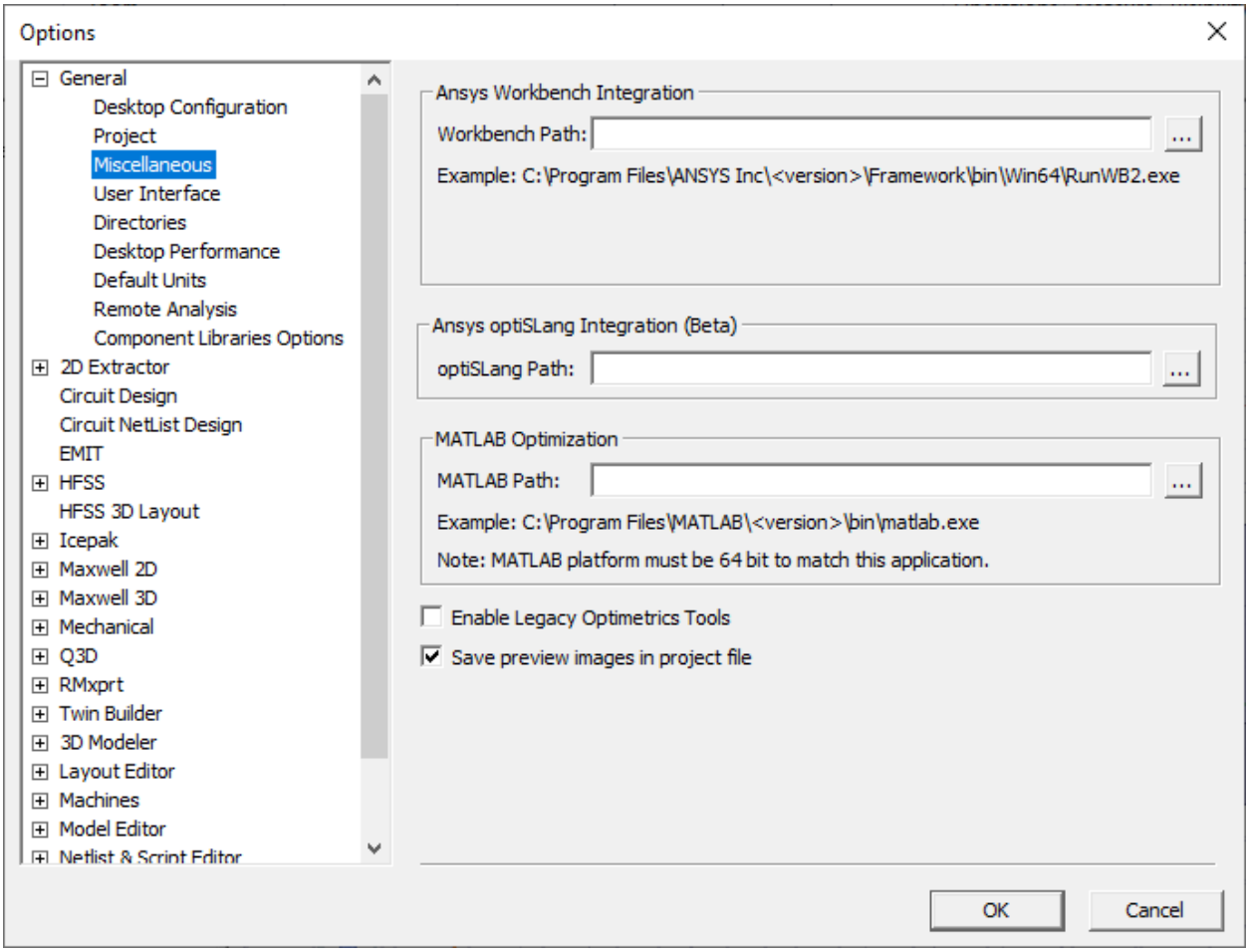
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



General Options: Miscellaneous Options

Under General Options, **Miscellaneous** options allow you to specify paths for Ansys Workbench Integration, optiSLang path, MATLAB Optimization and to enable Legacy Optimetrics Tools.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Workbench Integration allows you to specify the path to your Ansys Workbench installation. This path can be used in Optimetrics for connecting to the Design Xplorer.

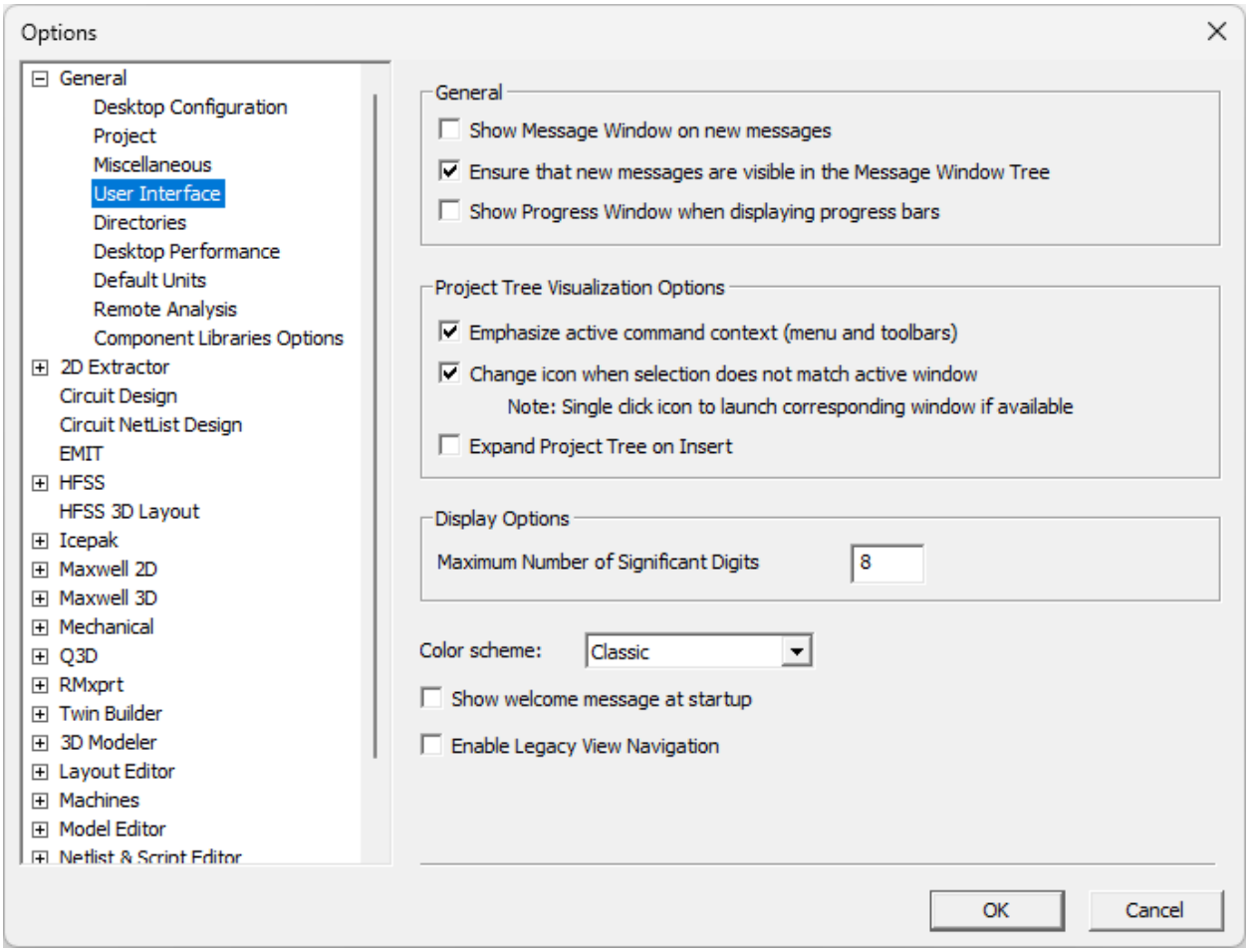
MATLAB Optimization allows you to specify the path to your installation of MATLAB. MATLAB can be used as an Optimizer.

> **Note:**
>
> The platform for MATLAB and Ansys Electronics Desktop must match (for example, a 64 bit version of MATLAB).

General Options: User Interface Options

Under General Options, **User Interface** options allow you to change how Electronics Desktop displays messages, command text, the project tree, welcome messages, and more.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **General** area, options include:

- **Show Message Window on new messages** – when selected, the message window automatically opens if a message arrives.
- **Ensure that new messages are visible in the Message Window Tree** – when selected, the message window expands as needed to display messages.
- **Show Progress Window when displaying progress bars** – when selected, the progress window automatically opens while simulations are in progress.

In the **Project Tree Visualization Options** area, options include:

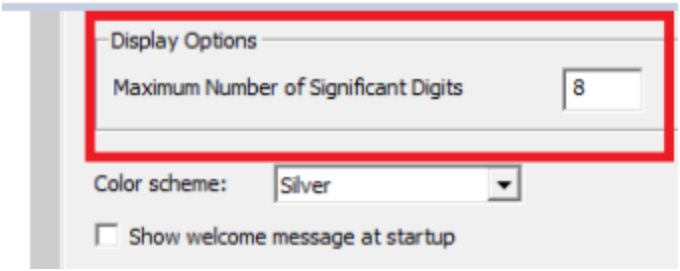
- **Emphasize active command context** – when selected, active elements in the Project Tree display in bold text.
- **Change icon when selection does not match active window** – when selected, a small, window-shaped overlay icon displays in the corner of the selected Project Tree element. This icon changes when the data in the active window is unrelated to the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

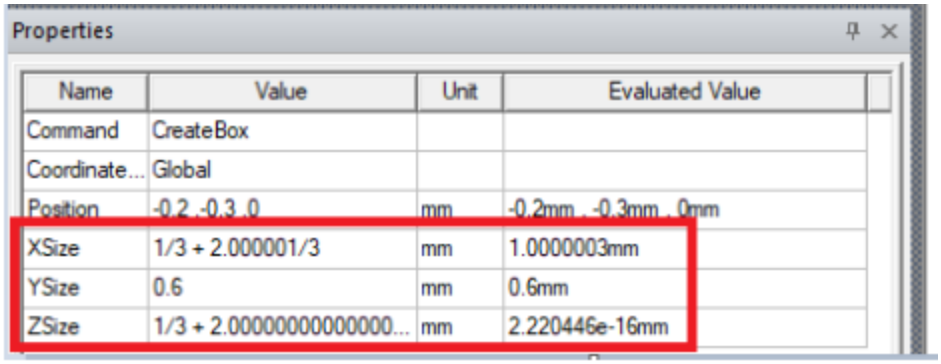
selected project item (data affecting the same model is considered to be related). Clicking the icon opens the window and brings it into focus.

- **Expand Project Tree on Insert** – when selected, the Project Tree automatically expands when you insert a new design.

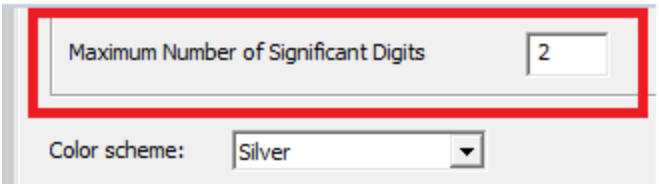
In the **Display Options** area, specify the **Maximum Number of Significant Digits** to display. The default is 8 and the maximum is 20. This affects the digits displayed in the **Solutions** dialog box, evaluated variable values, Animation dialog boxes, Optimetrics, Reports, and so forth. For example, here is the default.



So some evaluated variable values with significant digits appear as follows.

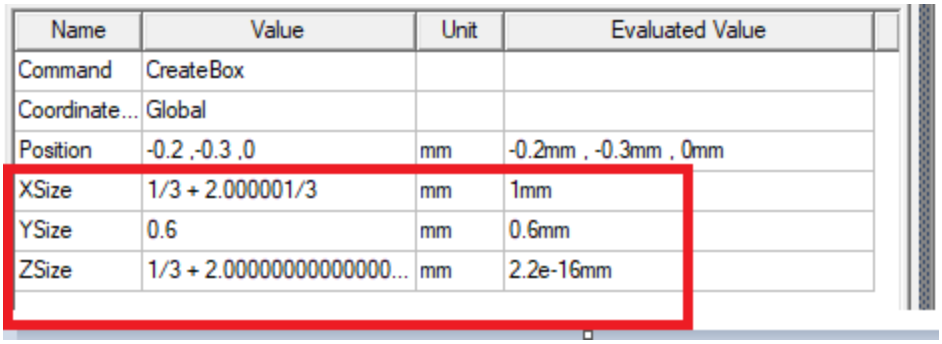


Change Precision = 2. This lets you display near-zero values in the Properties window as 0.



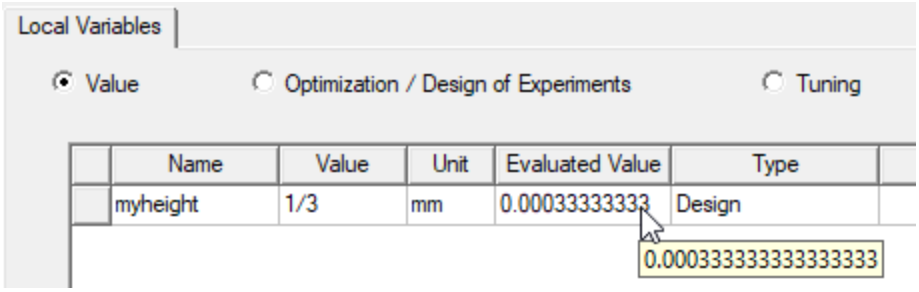
You can see that XSize is now rounded off to cleanly display 1mm. The decimal part of ZSize also shows cleaner value, but it is not rounded off to zero.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

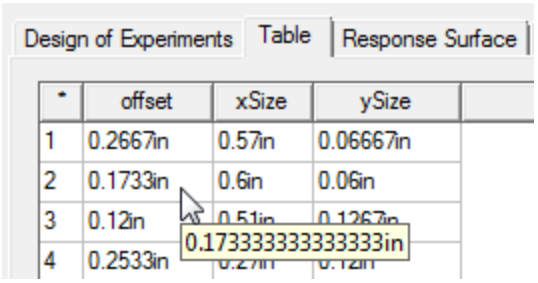


Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate...	Global		
Position	-0.2 ,-0.3 ,0	mm	-0.2mm , -0.3mm , 0mm
XSize	1/3 + 2.000001/3	mm	1mm
YSize	0.6	mm	0.6mm
ZSize	1/3 + 2.00000000000000...	mm	2.2e-16mm

You can still see the full precision values in tooltips by holding a cursor over the displayed value.



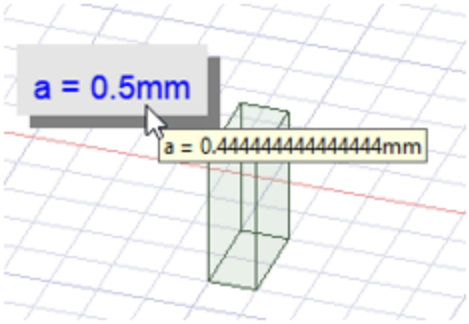
In the case of variable values, if you have assigned more significant digits, you will see these when editing the variable value. In the case of table displays of values, the tooltip display shows all available digits when the mouse pointer is over a result:

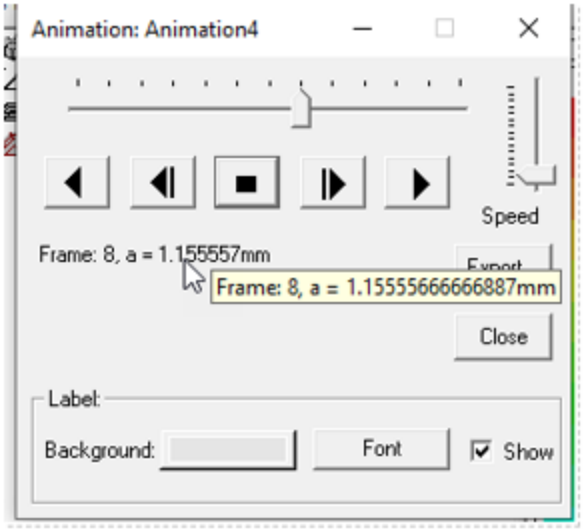


If you set the **Maximum Number of Significant Digits** to a lower value, Change Precision = 2. You can see that XSize is now rounded off to cleanly display 1mm. The decimal part of ZSize also shows cleaner value, but it is not rounded off to zero.

The tooltip functions to show internal digits in throughout the Ansys Electronics Desktop interface.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Select a **Color Scheme**. The choices are **Light (Beta)**, **Dark (Beta)**, or **Classic**: See Choosing a Color Scheme for more information and samples of each scheme.

Select **Show welcome message at startup** if you want to see a welcome message when Electronics Desktop opens.

Enable legacy view navigation enables you to choose between two schemes for view navigation keyboard shortcuts and mouse button assignments, as follows:

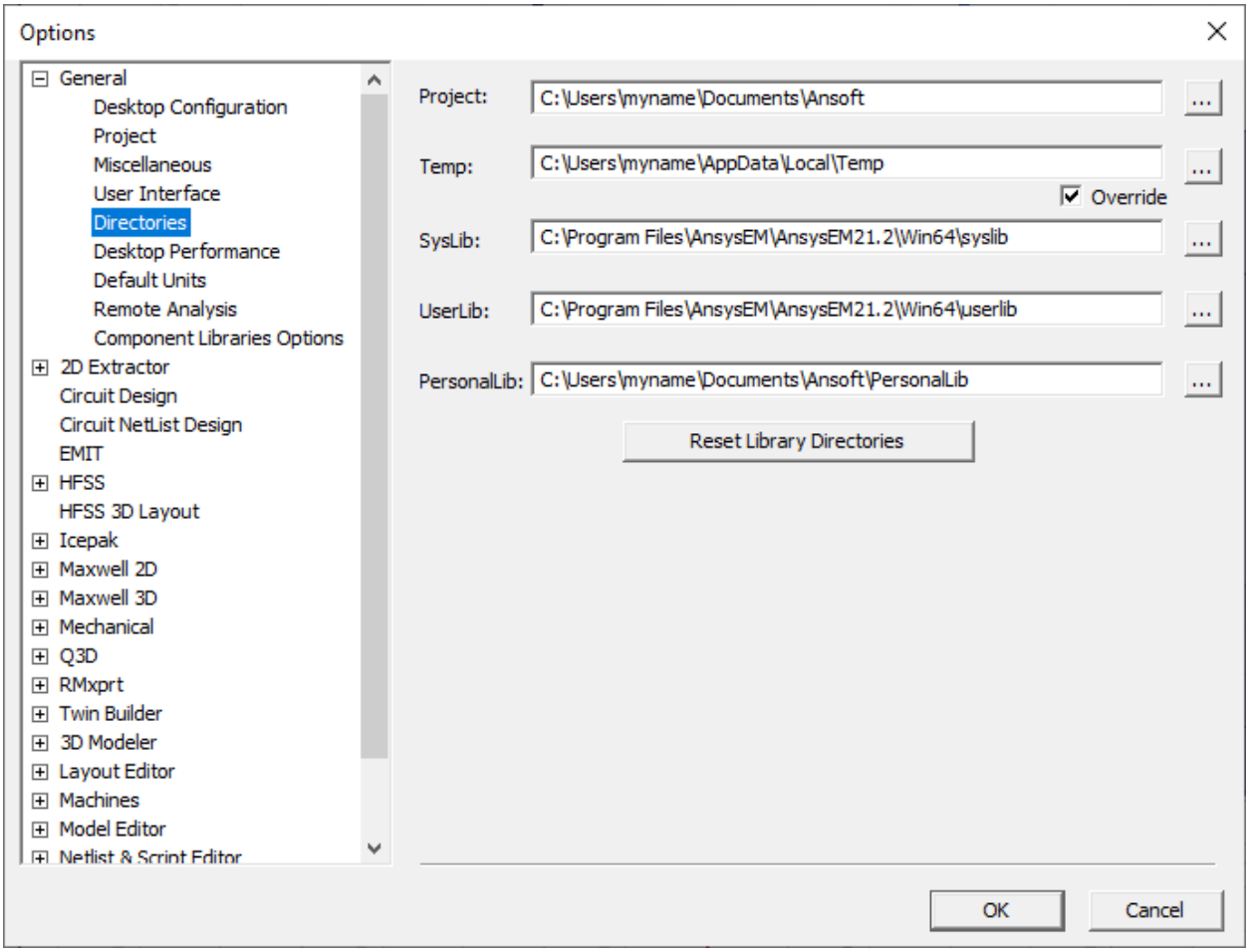
- **Cleared:** Use only the view navigation mouse-button and hotkey assignments introduced in Ansys Electronics Desktop version 2024 R1 and applicable to subsequent versions too. Legacy (2023 R2 and earlier) mouse-button and hotkey assignments will *not* be recognized when this option is cleared.
- **Selected:** Both the legacy (2023 R2 and earlier) and current (2024 R1 and newer) mouse-button and hotkey assignments are supported, and either can be used for view navigation.

See Choosing the View Navigation Options for more information, including a detailed comparison of the two schemes.

General Options: Directories Options

Under General Options, **Directories** options allow you to get the paths for Project, Temp, SysLib, UserLib, and PersonalLib directories.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Use the **[...]** buttons to browse to paths and click **OK**. You will need to select **Override** to change the Temp directory.

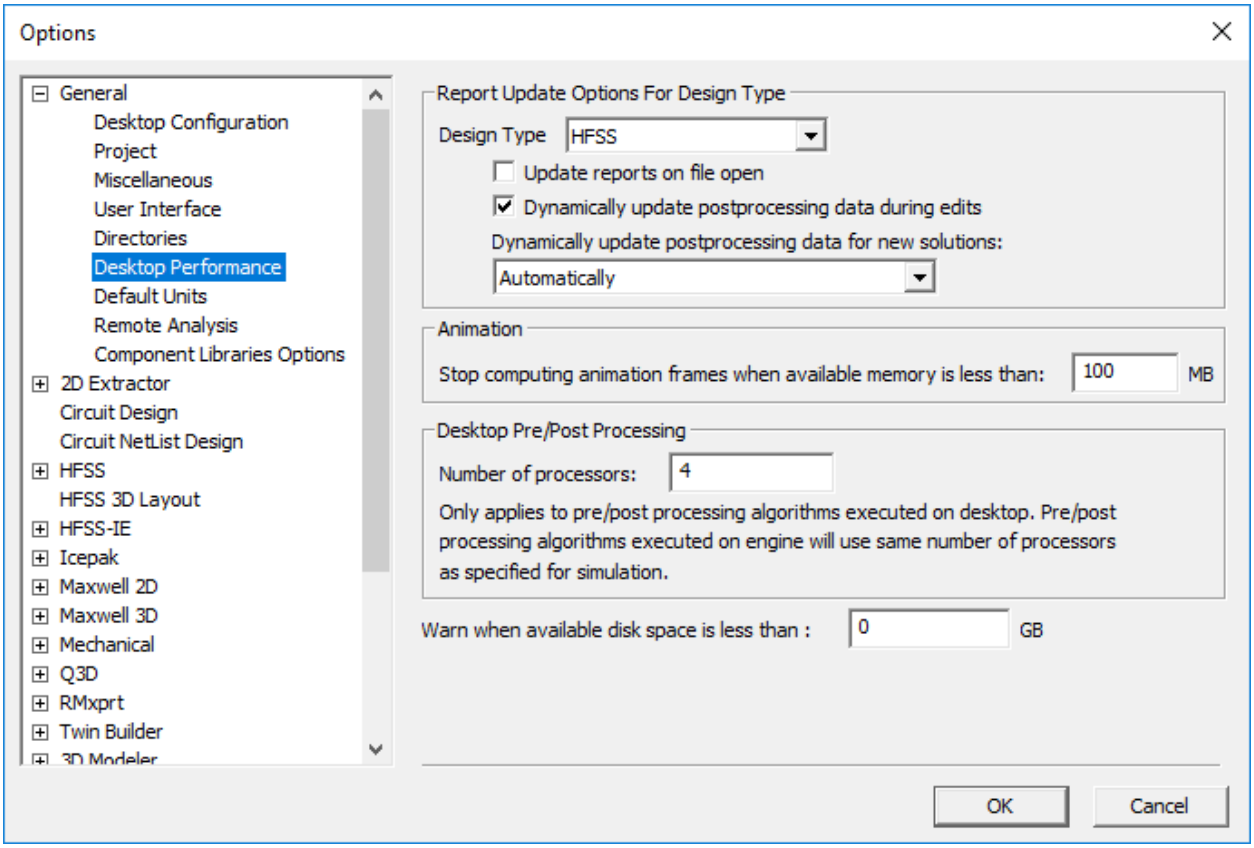
If you need to restore defaults, click **Reset Library Directories**.

> **Note:**
>
> Your changes will be reflected in the User Defined Primitives menu on next startup.

General Options: Desktop Performance

Under General Options, **Desktop Performance** options allow you to change settings for animation, pre/post-processing, disk space warnings, and updating reports.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



These options are set on the **Desktop Performance** panel under **General** in the **Options** window.

In the **Report Update Options for Design Type** area, select a **Design Type**. For each type, you can set the following options:

- **Update Reports on File Open** – when selected, reports are automatically updated whenever an existing file with solution data is opened for viewing/editing.
- **Dynamically Update Postprocessing Data During Edits** – when selected, report plots and overlays update as you edit their parameters.
- **Dynamically Update Postprocessing Data for New Solutions** – Because updating reports during analysis can impact solution time, you can specify how often your reports are updated:
    - **After Each Variation** – updates reports after analysis of each variation has been completed. Used for an Optimetric or parametric analysis.
    - **Automatically** – balances report and field plot updating with solution time.

      For Adaptive Passes, plots update at the end of each solution pass. For Last Adaptive or Transient, plots update at the end of the transient or adaptive solution.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, reports may be updated after each adaptive pass but field plots will not be updated until the solution is complete.

- **Immediately** – updates reports and plots as soon as data comes from the solver.

  This option will have the greatest impact on overall solution time, but affords the fastest updates to reports and field plots. Caution should be used in selecting this option. Some types of reports and field plots may take a long time to update, especially as mesh size increases.

- **Never** – updates reports only upon manual intervention. This prevents updates from impacting solution time.
- **On Completion** – updates reports once, when the solution completes.

> **Note:**
>
> Updates done on completion are done after the solve has been completed, and the time for that update is not included in the simulation profile.

In the **Animation** area, you can elect to **Stop computing animation frames when available memory is less than** a specified value, in MB (the default is 100MB). This setting is used to prevent problems related to low memory should an animation require large memory allocation.

In the **Desktop Pre/Post Processing** area, you can specify the **Number of Processors**. This option only affects pre- and post-processing (not solve or simulation). Pre-processing algorithms can take advantage of multiple processors for visualization and faceting of 3D models, model validation for 3D products, auto net identification for Q3D, and more. The default value is determined by the number of logical processors on the machine running Electronics Desktop:

- The default core usage per desktop session (UI + solve) is set to 2/3 of the logical processors on the machine.
- The cores for default local config is max{4, 1/3 of the logical processors on the machine}
- The default number of processors for pre/post is min{1, 2/3 of the logical processors - default_cores_for_local_config}
- When the core usage per desktop session cannot be evenly distributed between solve and pre/post, more cores will be assigned to solve.

Example 1:

- logical processors = 24
- cores usage per desktop session = 16
- local hpc set to 4 cores
- processors for pre/post set to 12

Example 2:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- logical processors = 20
- cores usage per desktop session = 13
- local hpc set to 4 cores
- processors for pre/post set to 9

Example 3:

- logical processors = 5
- cores usage per desktop session = 3
- local hpc set to 2
- processors for pre/post set to 1

You can also elect to **Warn when available disk space is less than** a specified value, in GB.

General Options: Default Units

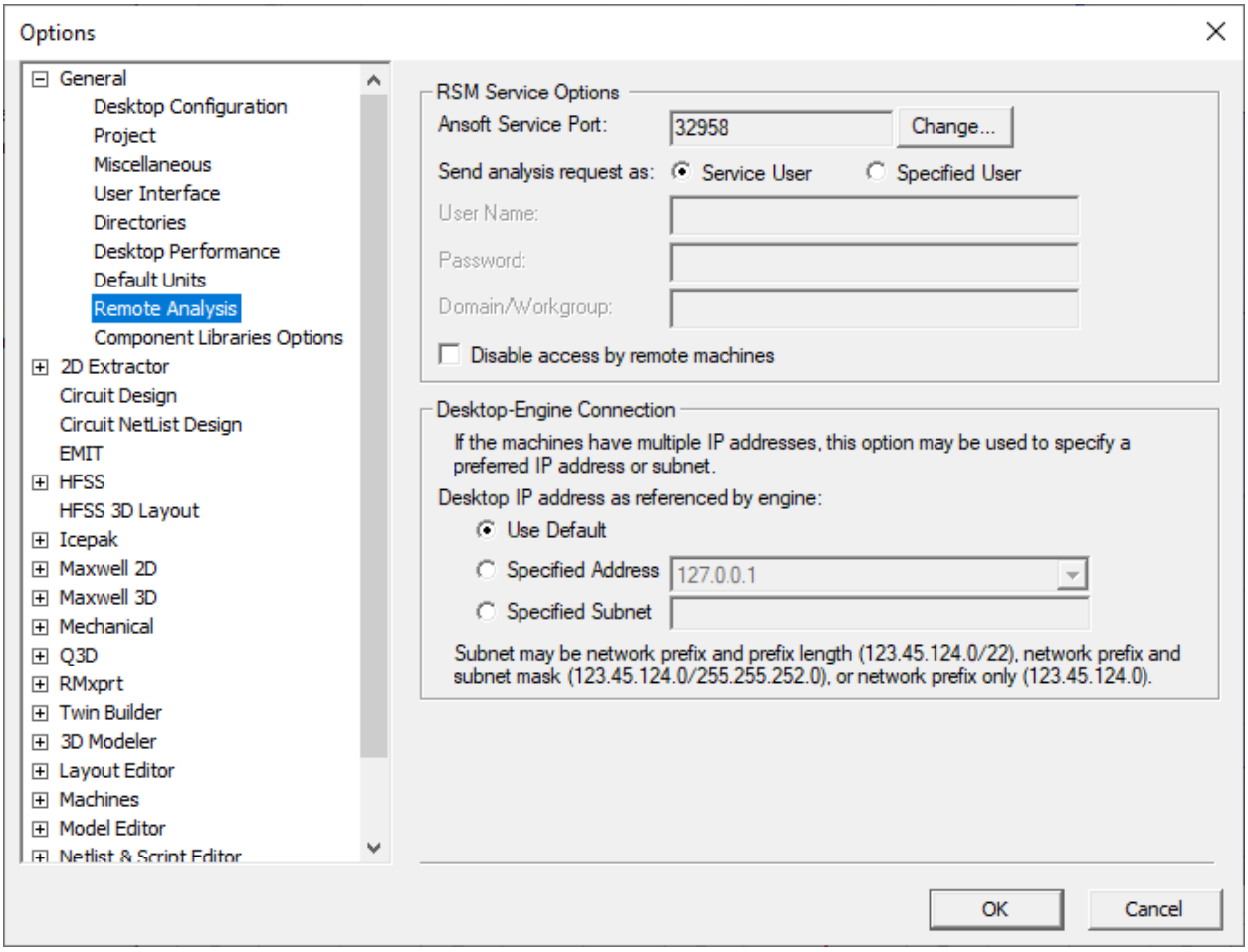
Under General Options, **Default Units** options allow you to set default units for the following:

- Length
- Angle
- Time
- Temperature
- Torque
- Magnetic Induction
- Pressure
- Frequency
- Power
- Voltage
- Current
- Speed
- Mass
- Conductance
- Resistance
- Inductance
- Capacitance
- Force
- Angular Speed
- Magnetic Field Strength

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Options: Remote Analysis Options

Under General Options, **Remote Analysis** options allow you to launch all analyses as a service or specified user rather than as the current user.



In the **RSM Service Options** area, options include:

- **Ansoft Service Port** – Click **Change** to update the port number. Ansys Electromagnetics RSM Service should be running on this port for all distributed machines.
- **Send Analysis Request As** – Select either **Service User** or **Specified User**. Selecting **Specified User** enables the **User Name**, **Password**, and **Domain/Workgroup** fields.

> **Note:**
>
> If any of the remote machines is Linux-based, you must specify the current user.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Disable Access By Remote Machines** – If desired, select to disable access for remote machines.

When multiple IP addresses are available, the **Desktop-Engine Connection** area allows you to specify the preferred IP address for communication:

- **Use Default** – your system's default IP address.
- **Specified Address** – an IP address you specify.
- **Specified Subnet** – a subnet you specify. Subnet may be network prefix and prefix length (123.12.123.0/22), network prefix and subnet mask (123.12.123.0/255.255.252.0), or network prefix only (123.23.123.0).

Changing the Listening Port used by Ansys RSM Service

To change the listening port used by the RSM Service, you must change the ansoftrsmservice.cfg file, as follows:

Specify the ListenPort within a 'CommDetails' block, which must be within a 'Default:CommDetails' block, which must be within the top level block of the file (the 'AnsoftCOMDaemon' block). The following example changes the listen port from 32958 to 32957, with these blocks at the beginning of the file:

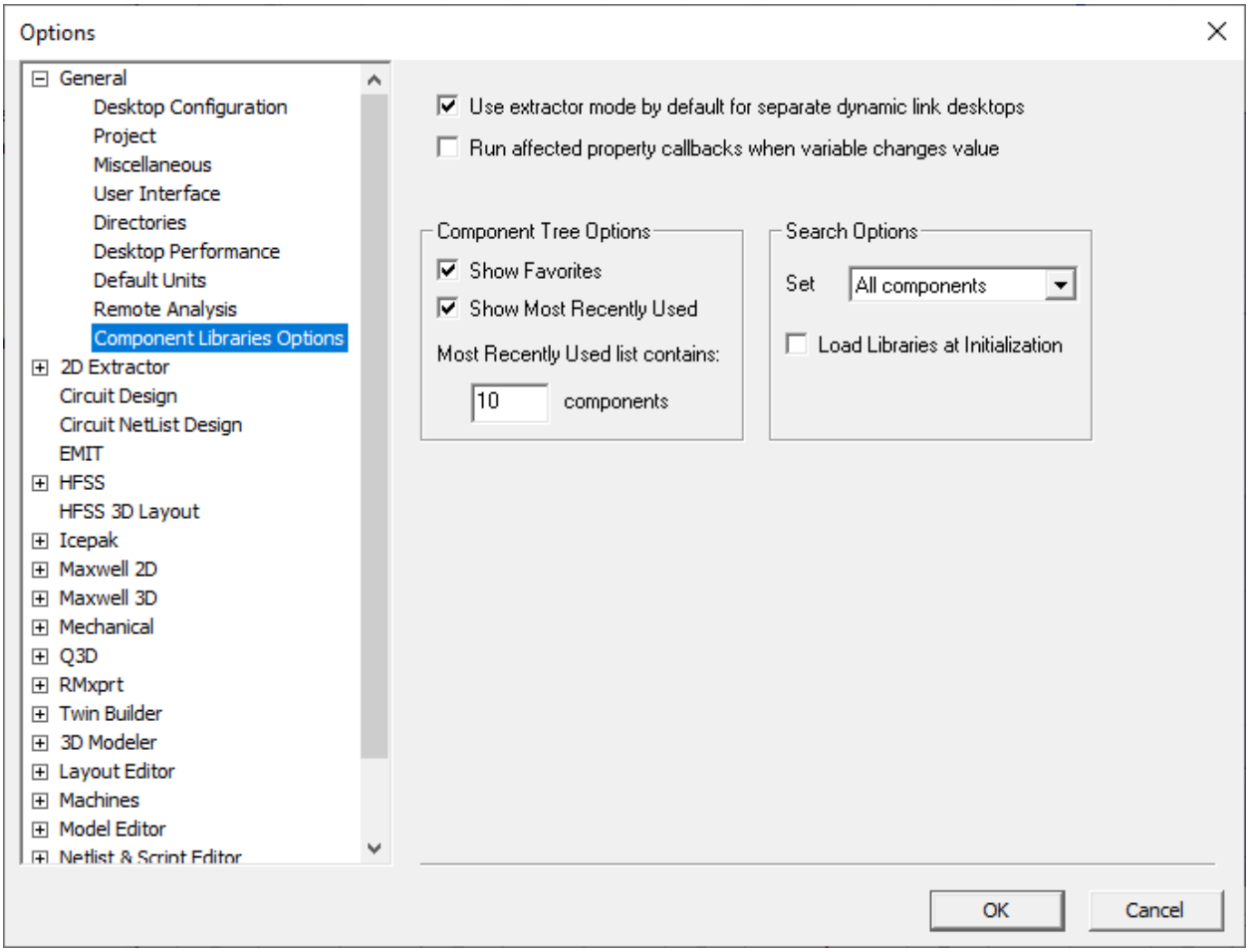
```
$begin 'AnsoftCOMDaemon'
  $begin 'Default:CommDetails'
    $begin 'CommDetails'
      ListenPort='32957'
    $end 'CommDetails'
  $end 'Default:CommDetails'
  . . . .
$end 'AnsoftCOMDaemon'
```

For the second level block, ensure that there is a single colon character and no spaces or tabs separating the two parts of the block name 'Default:CommDetails'. The third level block, with name 'CommDetails' is also required. Use caution when editing this file by hand, because any typos in the block or value names may cause the data to be ignored.

General Options: Component Libraries Options

Under General Options, **Component Libraries** options allow you to change how Electronics Desktop handles components.

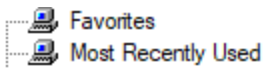
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can elect to **Use extractor mode by default for separate dynamic link desktops**, or to **Run affected property callbacks when variable changes value**.

In the **Component Tree Options** area, options include:

- **Show Favorites** – enables the **Favorites** folder in the **Component Libraries** tree.
- **Show Most Recently Used** – enables the **Most Recently Used** folder in the **Component Libraries** tree. Use the field to specify the number of components shown in the **Most Recently Used** folder. The default is 10.



In the **Search Options** area, options include:

- **Set** – specifies the search set when searching the Component Libraries. Choose to search **All components**, **Current list only**, or **Append to current list**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Load Libraries at Initialization** – this option can slow initialization.

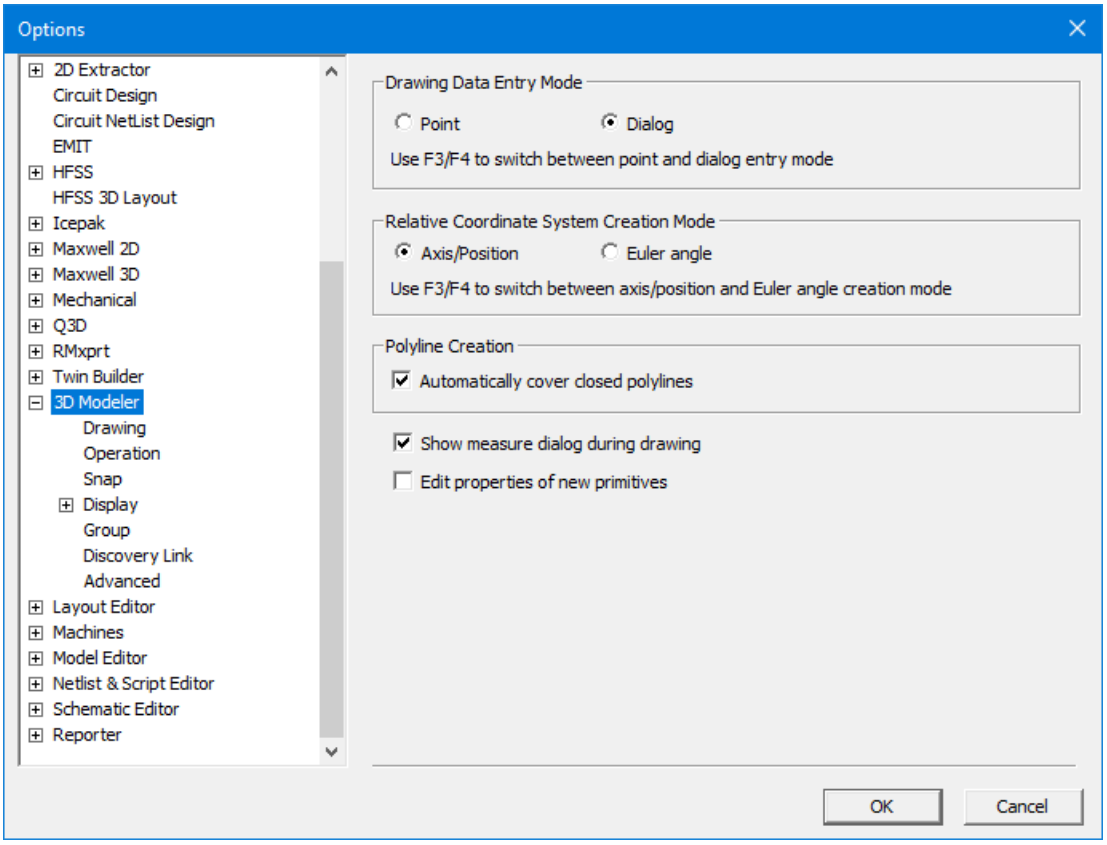
Setting 3D Modeler Options

To set 3D Modeler options in Ansys Electronics Desktop:

1. Click **Tools** > **Options** > **General Options**.

   The **Options** window opens with the Desktop Configuration options selected by default.

   In the left pane, select 3D Modeler to view modeler options:



   In the left pane, select the entries below **General** to display the associated options:

   - Drawing
   - Operation
   - Snap
   - Display
   - Group

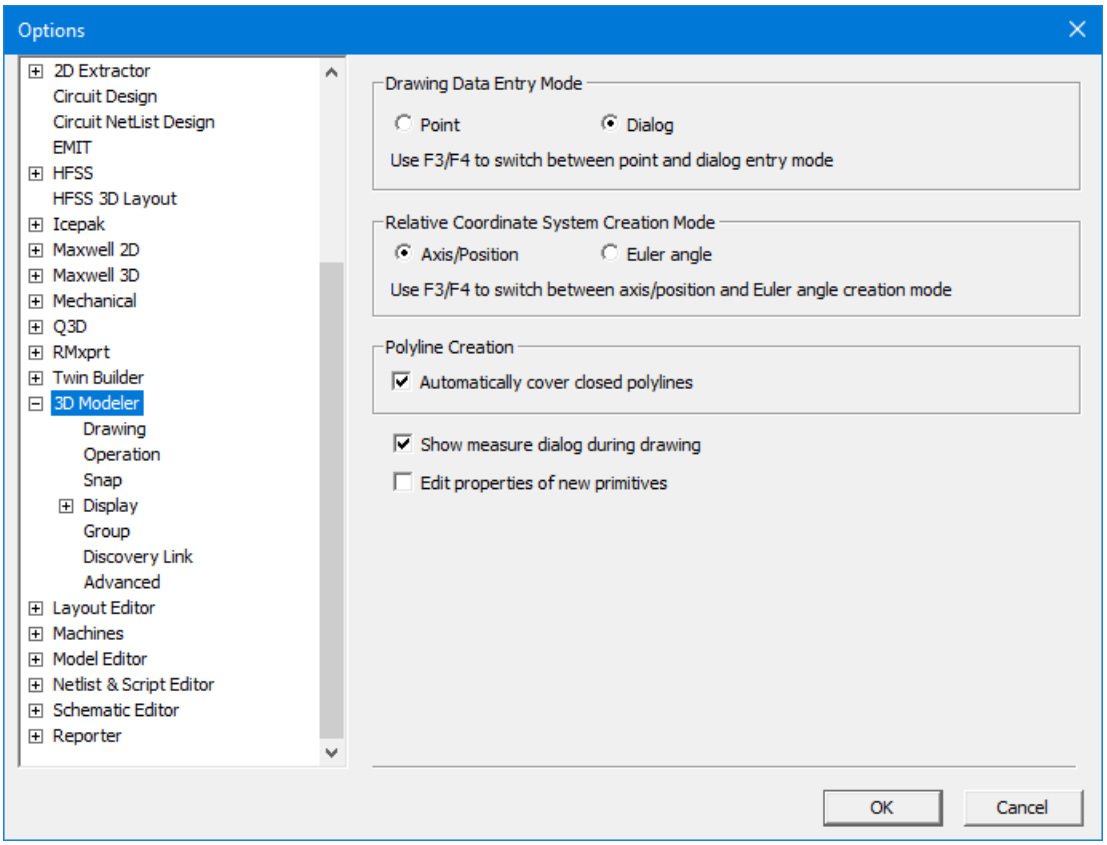
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Discovery Link
- Advanced

2. Click each entry and make the desired selections.
3. Click **OK** to apply your preferences.

3D Modeler Options: Drawing Options

Under 3D Modeler options, **Drawing** options allow you to decide how Electronics Desktop draws objects.



In the **Drawing Data Entry Mode** area, select **Point** to draw new objects directly via the mouse, or select **Dialog** to prompt a **Properties** dialog box that allows you to enter object dimensions. Dialog mode works with the equation-based line and all two- and three-dimensional objects.

In the **Relative Coordinate System Creation Mode** area, select **Axis/Position** or **Euler angle**. This determines how the relative coordinate system is created from the **Modeler** > **Coordinate System** > **Create** > **Relative CS** menu. When **Axis/Position** is selected, relative coordinate systems are created in Draw mode. When **Euler angle** is selected, they are created from values entered in a dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

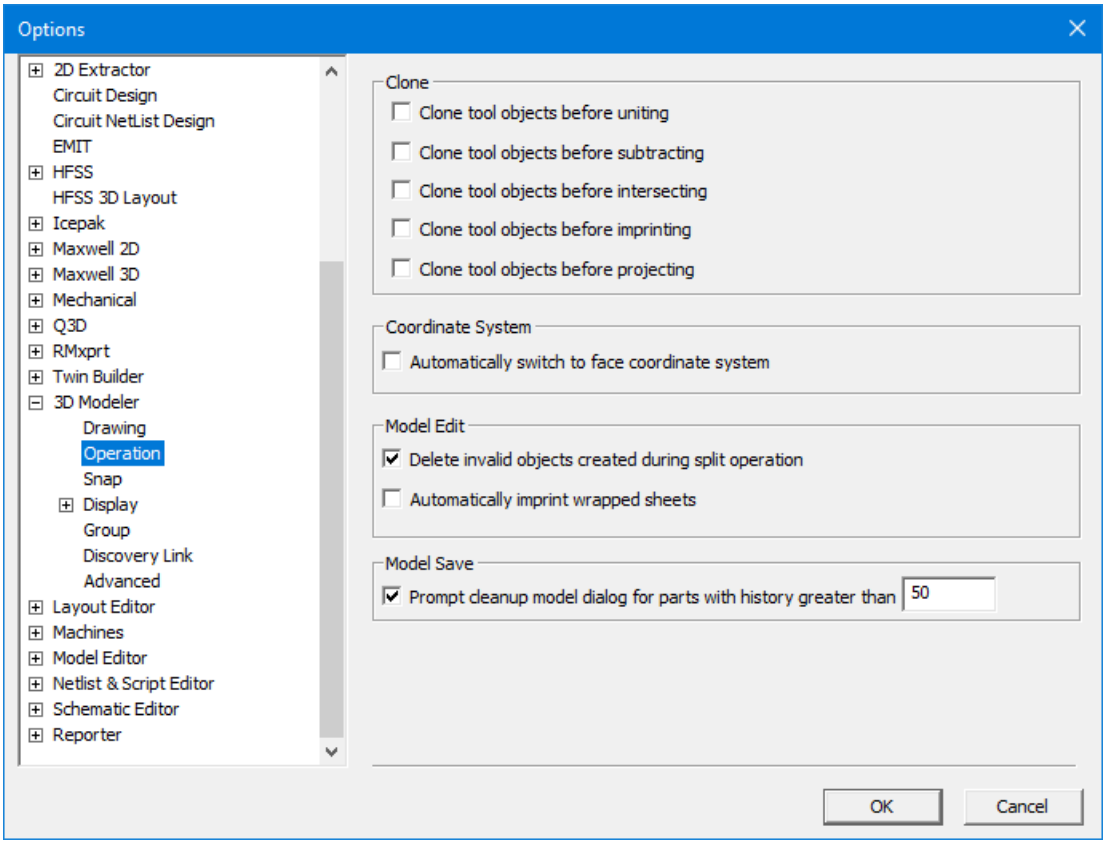
In the **Polyline Creation** area, enable or disable **Automatically cover closed polylines**. When enabled, polylines become sheet objects and are listed as such in the History tree. When deselected, closed polylines are left as uncovered objects and listed as lines in the History tree.

Other options are:

- **Show measure dialog during drawing** – specifies whether a **Measure** dialog box appears upon creation of a new primitive. The dialog box shows the coordinates of the current cursor position.
- **Edit properties of new primitives** – specifies whether a **Properties** dialog box display when you create a new object in the modeling workspace.

3D Modeler Options: Operation Options

Under 3D Modeler options, **Operation** options allow you to select options for clone, coordinate system, model editing, and model save operations.



In the **Clone** area, enable or disable options. By default, the modeler deletes tool objects when performing operations on objects. You may specify that the modeler make a clone (copy) of the tool object before these operations:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Clone tool objects before uniting**
- **Clone tool objects before subtracting**
- **Clone tool objects before intersecting**
- **Clone tool objects before imprinting**
- **Clone tool objects before projecting**

In the **Coordinate System** area, decide whether or not to **Automatically switch to face coordinate system**.

By default, the modeler operates within the user-selected coordinate system. If this option is enabled, when you select a face and create a new object, the modeler first creates a face coordinate system consistent with the selected face and the new object is created within the face coordinate system. When this option is disabled, you must manually create a Face Coordinate System before creating an object related to it.

In the **Model Edit** area, enable or disable the following options:

- **Delete invalid objects created during split operation** – When selected, the modeler deletes invalid objects created during split operation. When deselected, invalid objects can be freely created.
- **Automatically imprint wrapped sheets** – Imprinting is a boolean operation, whereas wrapping is not.

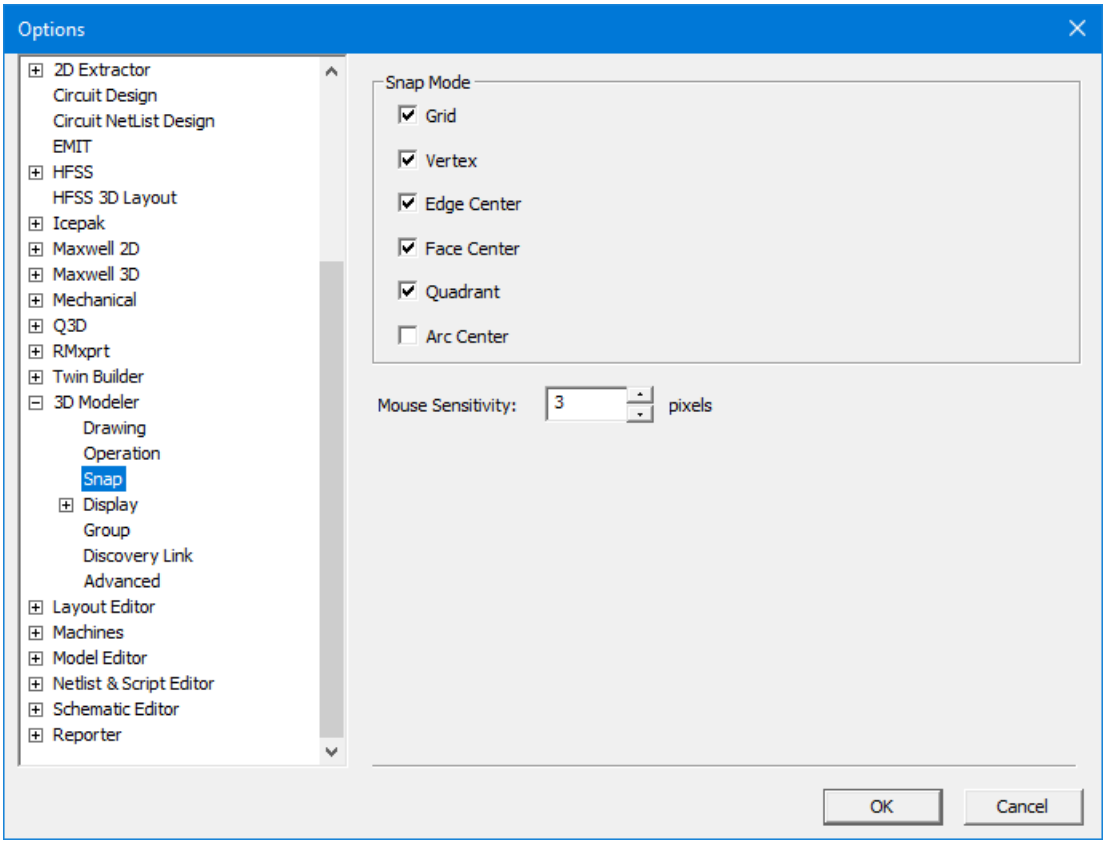
  See: Imprinting an Object and Wrap Sheet Command.

In the **Model Save** area, you can enable/disable being prompted for model history cleanup. You can also specify the number of actions on a part that will cause a prompt. The default value is 50.

3D Modeler Options: Snap Options

Under 3D Modeler options, **Snap** options allow you to decide how Electronics Desktop snaps objects. By default, most objects are set to snap (adhere) to the nearest point on the grid. The coordinates of this point are used, rather than the exact location of the mouse.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **Snap Mode** area, select the points you would like the cursor to snap to:

- Grid
- Vertex
- Edge Center
- Face Center
- Quadrant
- Arc Center

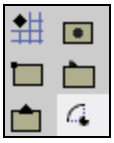
Use the **Mouse Sensitivity** option to select how many pixels you must move the mouse before a different snap point is selected. Values between 1 and 20 are accepted, with 1 being the most sensitive and 20 being the least sensitive. The default is 3.

You can also access snap settings:

- **From the Modeling Workspace** – While in draw mode, right-click and select **Snap Mode**.
- **From the Toolbar** – Select **Modeler** > **Snap Mode**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **From the Draw Ribbon** – Select and deselect the snap icons.



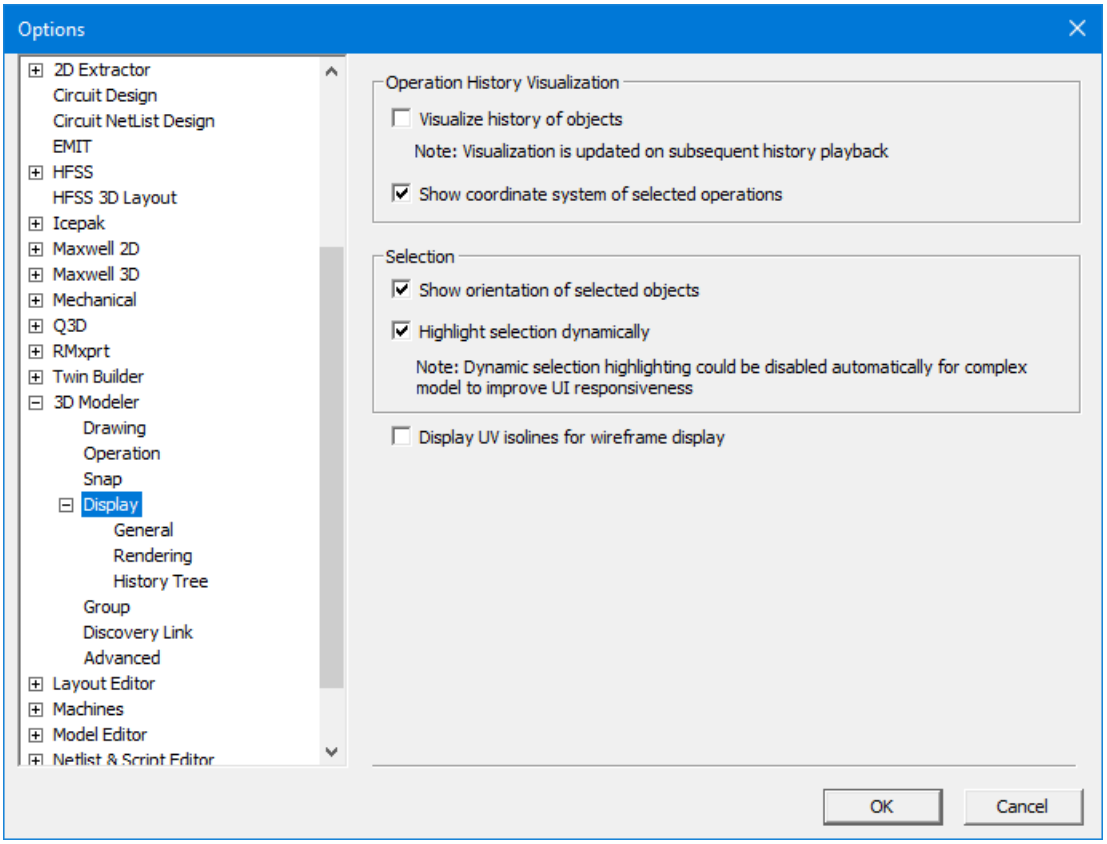
Hover over each icon for a tooltip indicating its equivalent menu option.

> **Important:**
>
> In general, you should select at least one of the snap options . If none is selected, the software is in "free mode" and selects whatever point you click, regardless of its coordinates. This can cause problems creating closed objects because the point you select may appear to be the vertex point of an open object, but may not be the exact coordinates.

3D Modeler Options: Display Options

Under 3D Modeler options, **Display** options allow you to change how Electronics Desktop displays your project tree, history, and objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



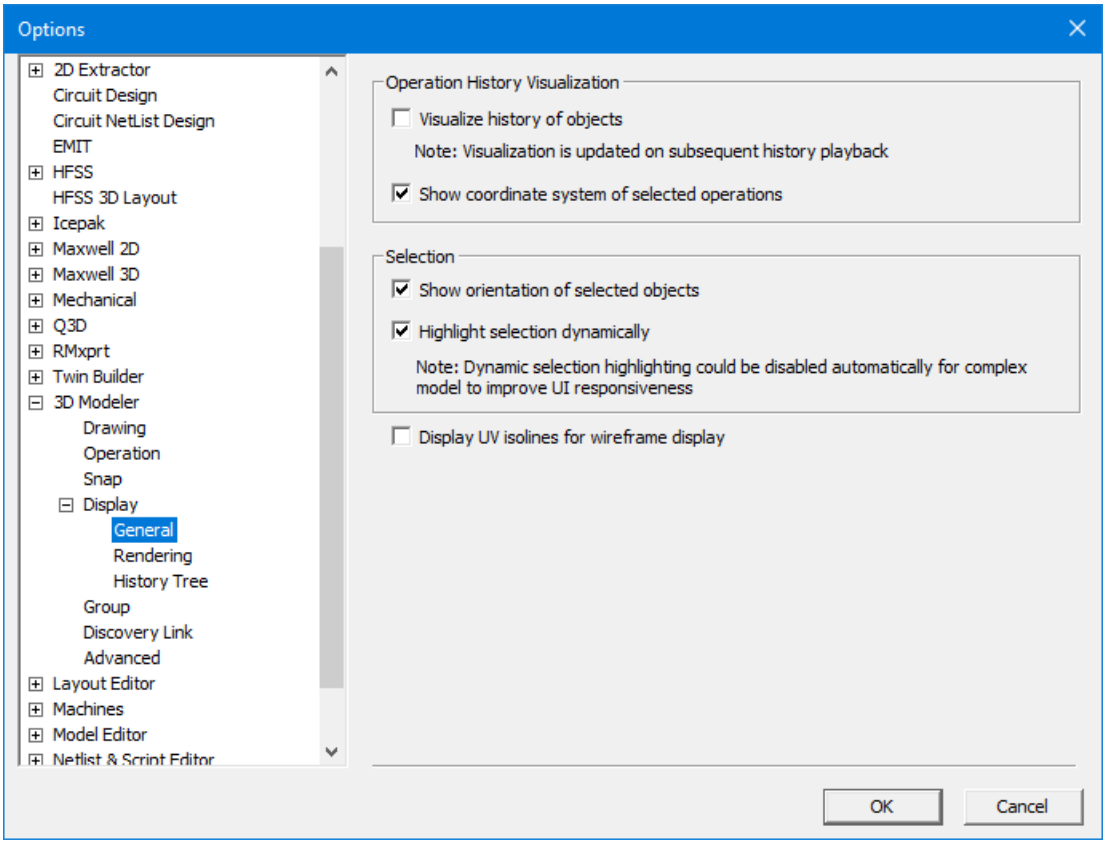
In the left pane, select the entries below **Display** to display the associated options:

- General
- Rendering
- History Tree

Display Options: General Options

Under Display Options, **General** options allow you to change how objects are selected and how their history is visualized.

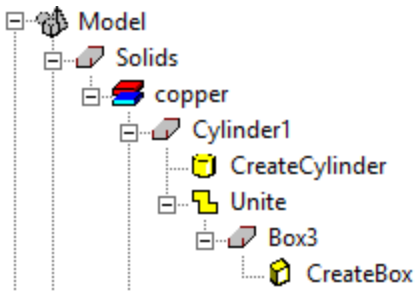
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **Operation History Visualization** area, set the following options:

- **Visualize history of objects** – When selecting an object in the history tree, this option lets you view an outline of each part that comprises the object. This can help you visualize an object that has been merged with another object. Clearing this selection removes visualization of objects that are part of the model history. For large models, this is faster and uses less memory.

  Visualized history:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> You must restart Electronics Desktop for this option to take effect.

- **Show coordinate system of selected operations** – Select or clear.

In the **Selection** area, set the following options:

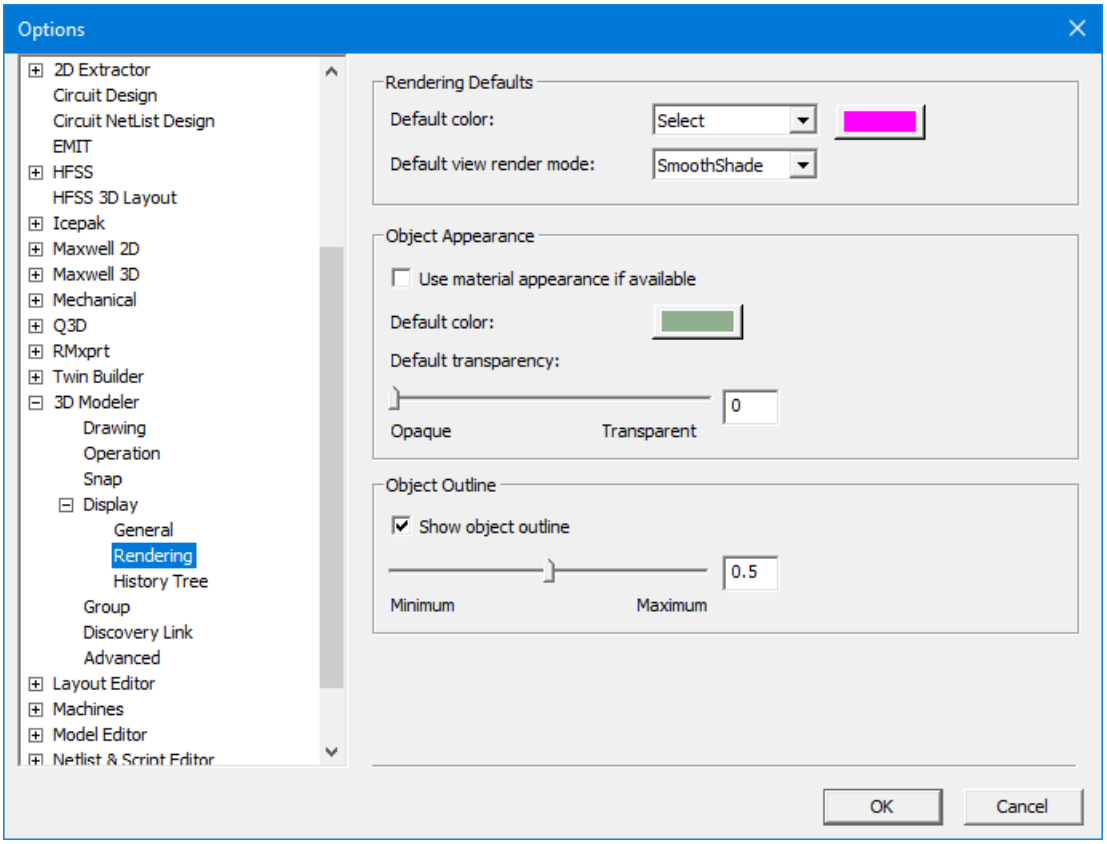
- **Show orientation of selected objects** – Select or clear.
- **Highlight selection dynamically** – This option causes objects or faces (depending on the selection mode) to be highlighted when you pass the mouse pointer over them. You may want to turn this off for complicated models as it can reduce responsiveness. If Electronics Desktop identifies the model to be large enough, dynamic highlighting is automatically ignored. In this case you will see a message about this.

You can also **Display UV isolines for wireframe display**. For models with curved faces, you may prefer to clear this option to simplify the wire-frame display and speed up rendering.

Display Options: Rendering Options

Under Display Options, **Rendering** options allow you to change rendering and object appearance defaults.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **Rendering Defaults** area, set the following options:

- **Default color** – Use the drop-down menu to choose the **Select**, **Highlight**, or **Rubberband** action. Click the color button to select a color for each action.
- **Default view render mode** – Select either **WireFrame** or **SmoothShade**. When dealing with complicated geometries, choose **WireFrame** rendering for speed.

In the **Object Appearance** area, set the following options:

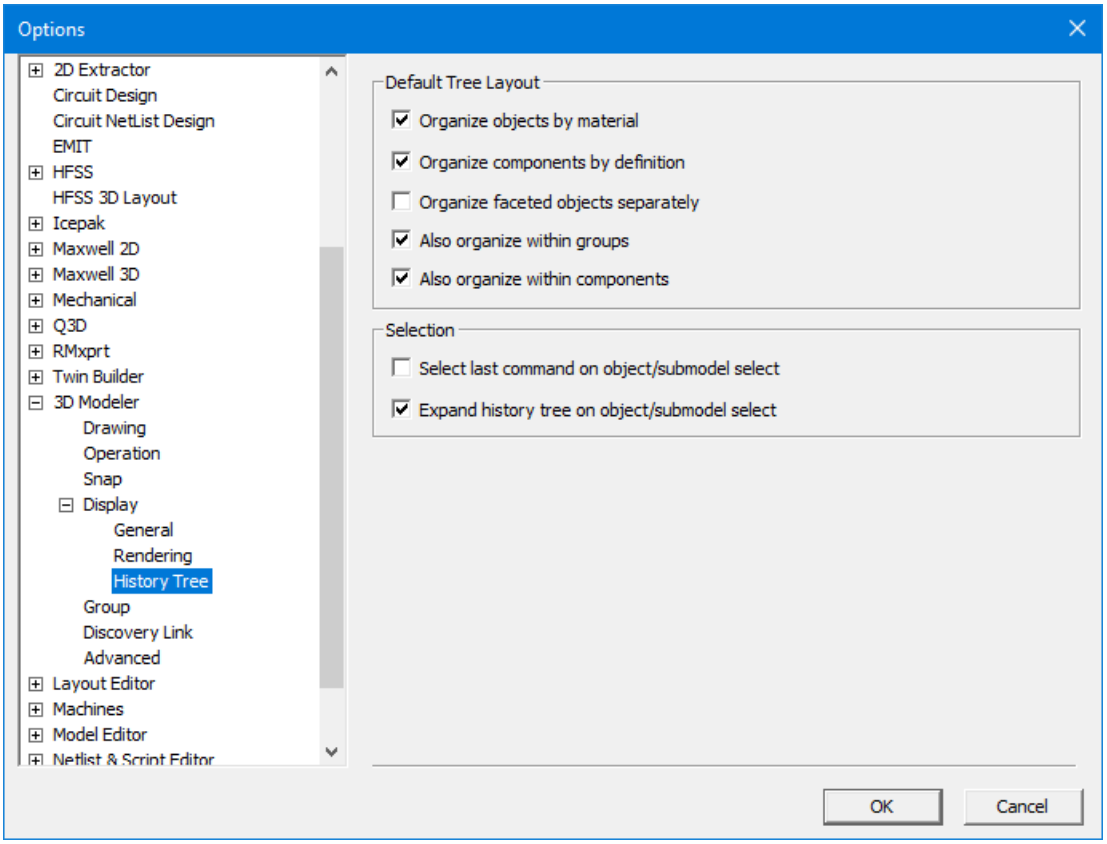
- **Use material appearance if available** – When this option is selected, default color and transparency used for newly created objects come from material settings. Some materials may not have default appearance specifications; in those cases, default color and transparency specified under **Rendering Defaults** are used.
- **Default color** – Click to select the default color.
- **Default transparency** – Use the slider to select the default transparency.

In the **Object Outline** area, select or clear **Show object outline**. If you elect to show the object outline, use the slider to determine the outline thickness. Value must be between 0 and 1.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Display Options: History Tree Options

Under Display Options, **History Tree** options allow you to change the history tree display and selection response.



In the **History Tree Layout** area, change the organization of the history tree as desired:

- **Organize objects by material**
- **Organize components by definition**
- **Organize faceted objects separately**
- **Also organize within groups**
- **Also organize within components**

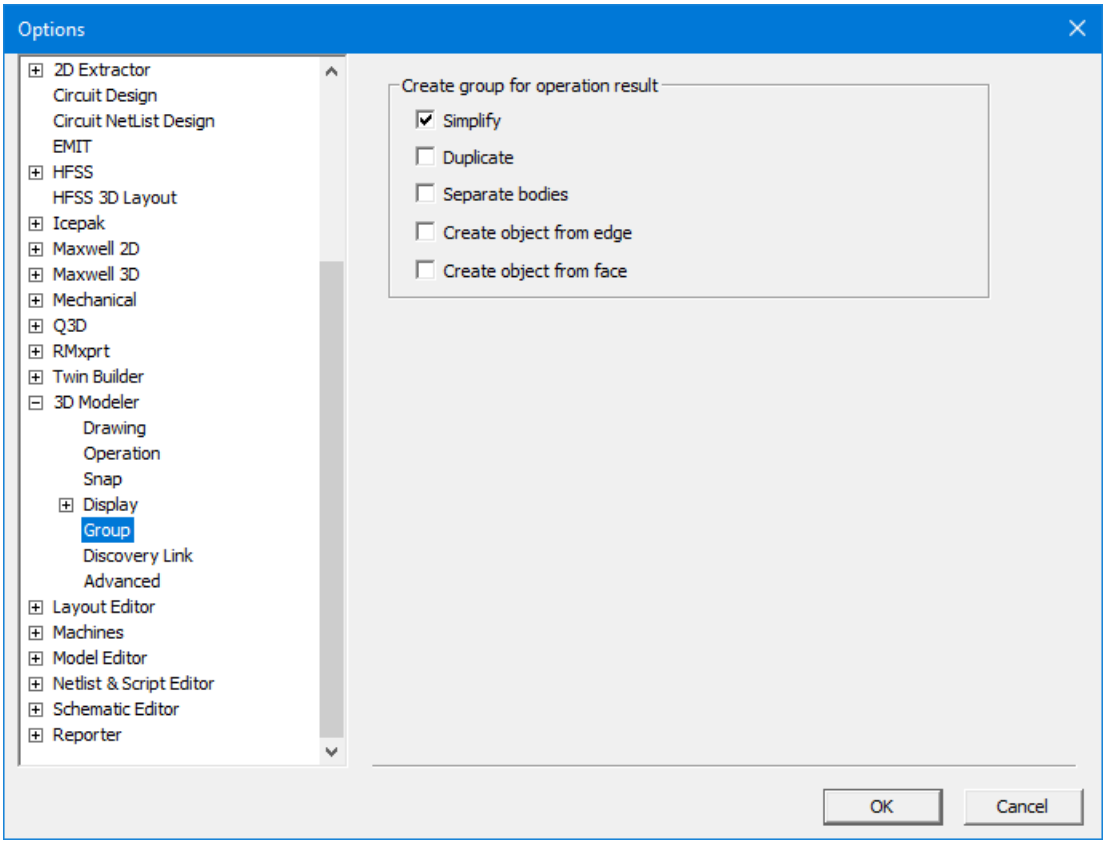
In the **Selection** area, set the following options:

- **Select last command on object/submodel select** – When selected, the history tree is expanded after operations on object properties, even if the tree is collapsed for the item. The command is selected.
- **Expand history tree on object/submodel select** – When selected, selecting an object automatically opens the history tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D Modeler Options: Group Options

Under 3D Modeler options, **Group** options determine how groups are created based on operations.



In the **Create group for operation result** area, select or clear the following actions:

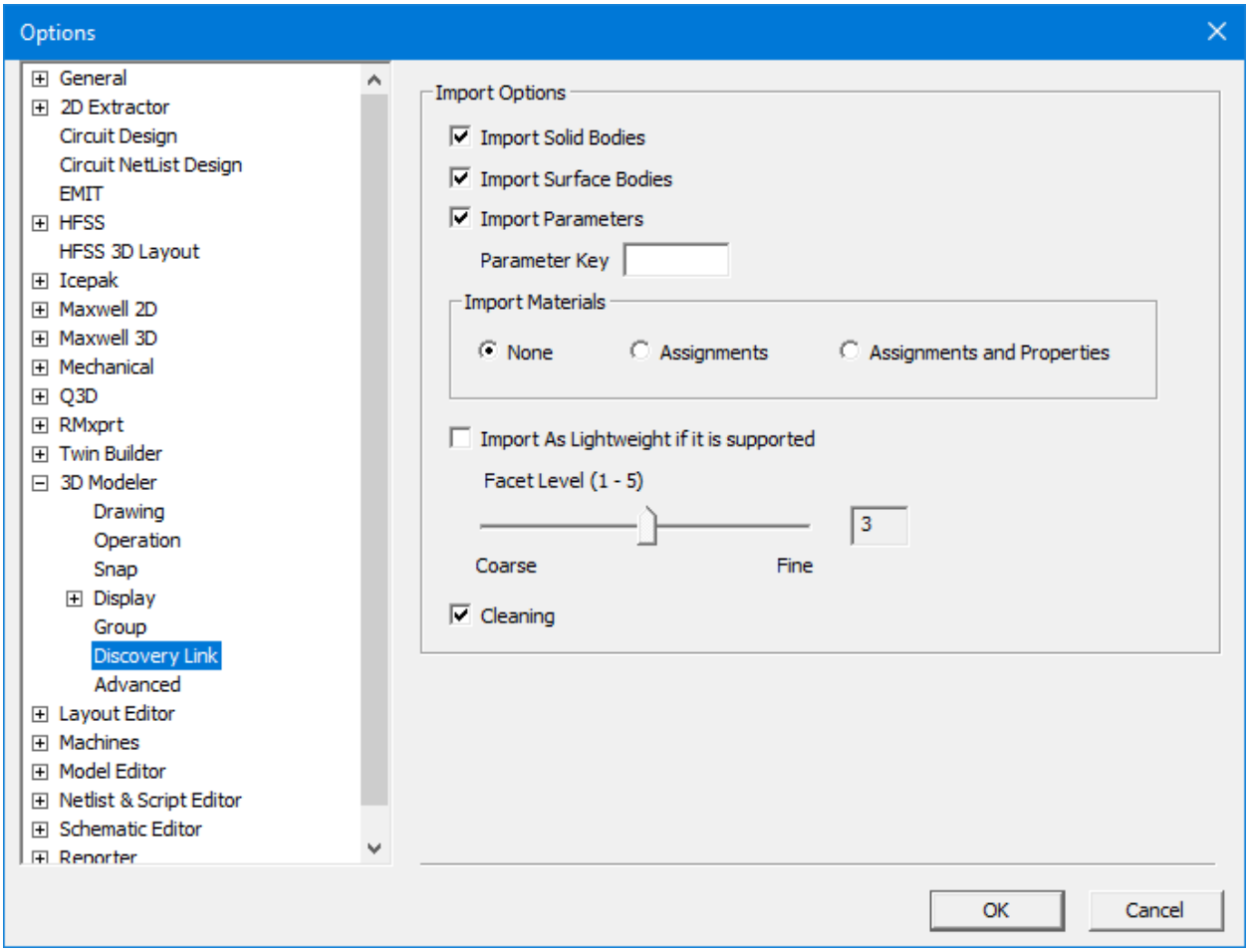
- **Simplify**
- **Duplicate**
- **Separate bodies**
- **Create object from edge**
- **Create object from face**

When any selected action is performed, a group will be created in the History Tree.

3D Modeler Options: Discovery Link Options

Under 3D Modeler options, **Discovery Link** options determine how linked Discovery projects are imported.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To specify Discovery link settings, select or clear the following check boxes in the **Discovery Link** section:

- **Import Solid Bodies** – Enabled by default.
- **Import Surface Bodies** – Enabled by default.
- **Import Parameters** – Enabled by default.
- **Parameter Key** – A string used to filter out parameters. By default no string is entered and all Discovery parameters are imported.
- **Import Material Assignments** – **None** by default.
    - **Assignments** – Imports the material names without the material properties. The material properties will be obtained from a material with the same name that exists in the user's project. With this option, the system will first search for the material within the project's materials. If no matching material is found, the search will extend to the sys libraries. (See Working with Material Libraries for details on libraries.) If a material with the same name exists in a sys library, it will be imported into the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

project from the library. If a material with the same name cannot be found in the project, a violation will be issued after the Discovery link is created.

- **Assignments and properties** – Imports material with its material properties. A project material will be created in AEDT with imported properties. Note that if a project material with same name already exists, then it will be used instead of a new material being created.

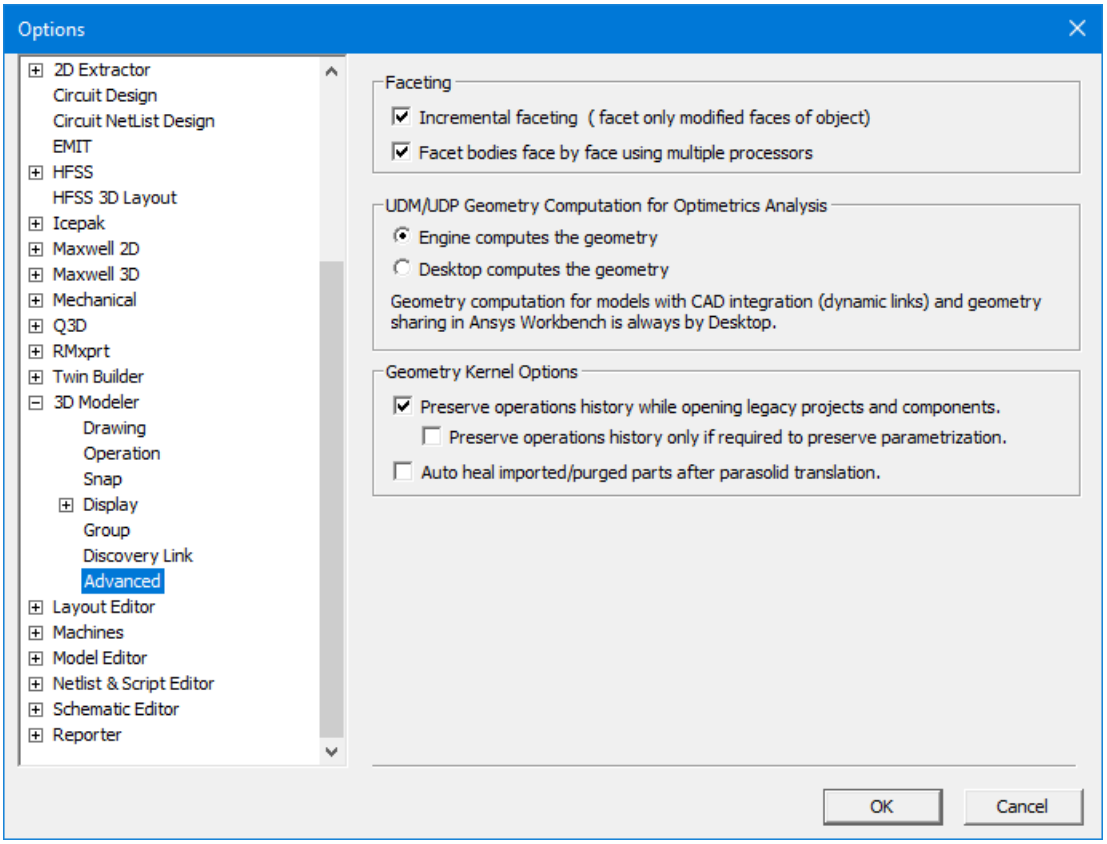
> **Note:** Discovery currently only supports exporting mechanical and thermal properties, so the **Assignment with properties** option is only supported in Mechanical and Icepak designs. For HFSS, Maxwell, and Q3D designs, this option will be ignored.

- **Import As Lightweight if it is supported** – This feature allows you to import a geometry from Discovery as a lightweight geometry into a design with SBR+ solution type. The Facet Level can be set on a 1-5 range, from coarse to fine, if the geometry in Discovery is not faceted. If it is faceted in Discovery, the facets will be imported directly and the Facet Level setting will be ignored.

3D Modeler Options: Advanced Options

Under 3D Modeler options, **Advanced** options help configure faceting and UDM/UDP.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **Faceting** area, select **Incremental faceting** or **Facet bodies face by face using multiple processors**. Face by face will be slower.

In the **UDM/UDP Geometry Computation for Optimetrics Analysis** area, select whether **Engine** or **Desktop** computes the geometry.

In the **Geometry Kernel Options**, enable **Preserve operations history while opening legacy projects** to preserve operations history while opening legacy projects that used the ACIS kernel. When opening legacy projects, the conversion is a one-time event. A renamed copy of the original file is preserved. The default is to preserve operations history. If the option is disabled, the operations history is purged. The option to purge history is available because:

- Some operations cannot be replayed in Parasolid.
- Purging provides backup to translate geometry.
- Parameterization is lost.
- Expect a better conversion success rate with purged models.

Setting Reporter Options

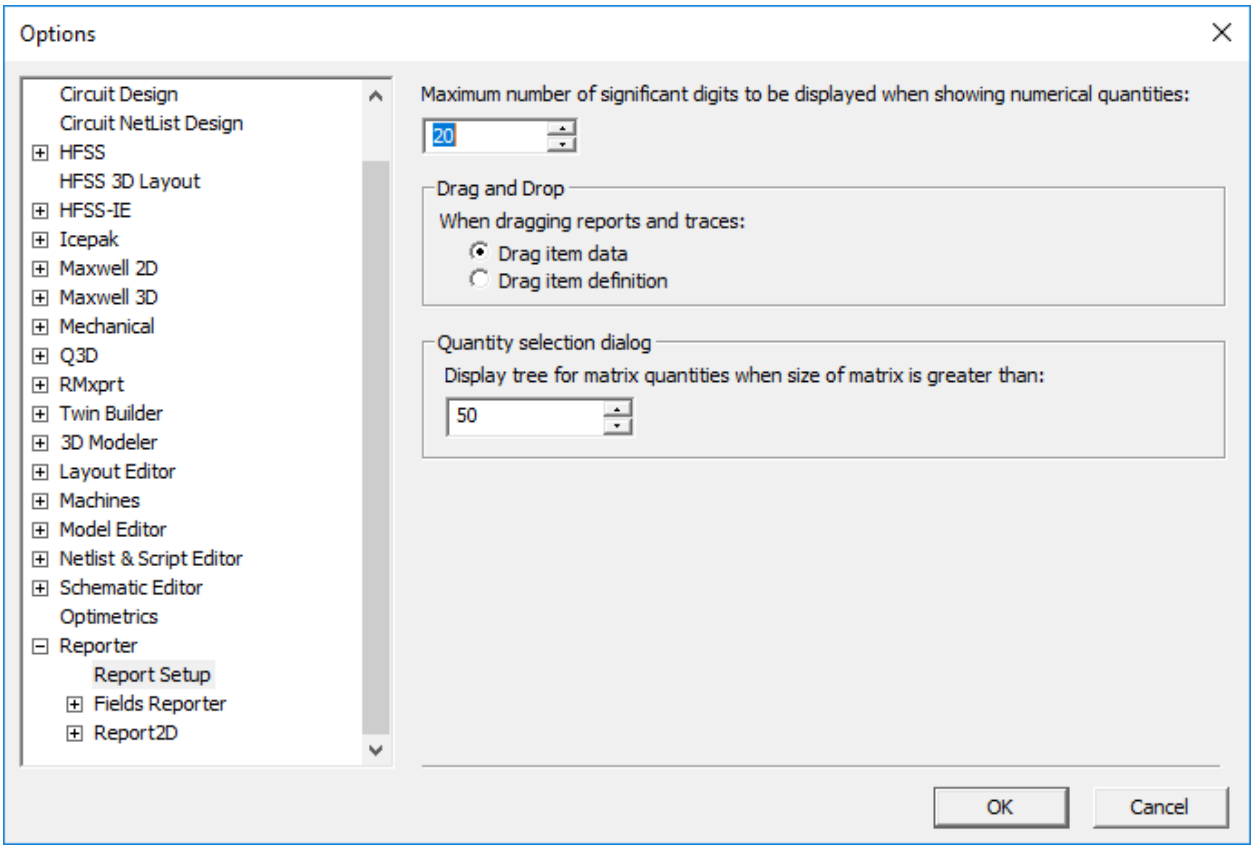
To set Reporter options in Ansys Electronics Desktop:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Tools** > **Options** > **General Options**.
2. In the left pane, select **Reporter**.
3. Select the entries below **Reporter** to display the associated options:
   - Report Setup
   - Fields Reporter
   - Report2D
4. Click each entry and make the desired selections.
5. Click **OK** to apply your preferences.

Reporter Options: Report Setup Options

Under Reporter Options, **Report Setup** options allow you to customize significant digits, drag-and-drop, and quantity selection options for reports.



Set the **Maximum number of significant digits to be displayed when showing numerical quantities**. The value must be between 0 and 20.

In the **Drag and Drop** area, select how Electronics Desktop behaves when you drag reports and traces. You can either **Drag item data** or **Drag item definition**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Quantity Selection Dialog** area, select a **Display tree for matrix quantities when size of matrix is greater than** value. When the number of matrix elements is larger than the selected number, the Quantities field uses a tree structure to divide matrix quantities into groups by their first element name. The initial display shows groups, without initially listing group members. This is useful when dealing with large matrices.

Reporter Options: Fields Reporter Options

Under Reporter Options, **Fields Reporter** options fall into three categories:

- Animation Options
- Mesh Plot Options
- Streamline Plot Options

In the left pane, select a category to view those options.

From the **Animation** options, you can set the following:

- **Group Field Overlays by Type** – If selected, field overlays are grouped in the Project Tree. If deselected, they are not.
- **Default Phase Animation Settings** – Select the default start and stop angles as well as the number of steps for Scalar and Vector plots. See: Creating Phase Animations.

From the **Mesh Plot** options, you can set the **Clipping of volume mesh plot**. When dragging a clip plane, the plot can update automatically. Select **Never** to disable this feature, **Always** to enable it at all times, or set a maximum number of mesh elements for automatic update. This option allows you to update plots for smaller meshes but avoid automatically updating larger plots that may consume too much memory. See: Using Clip Planes.

From the **Streamline Plot** options, set **Streamline drawing stopping criteria** and **Streamline marker spacing**.

Reporter Options: Report2D Options

Under Reporter Options, **Report2D** options fall into several categories:

- Curve
- Axis
- Grid
- Header
- Note
- Legend
- Marker
- Marker Table
- X/Y Markers

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Stacked
- Digital
- General
- Table

In the left pane, select a category to view those options.

Report2D Options: Curve

Curve settings change how curves are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Curve**:

- **Line Style** – use the drop-down menu to select from: Solid, Dot, ShortDash, DotShortDash, Dash, DotDash, DotDot, DotDotDash, and Long Dash.
- **Color** – double-click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Width** – set the line width by editing the real value in the text field.
- **Arrows** – select the check box to use arrows on curve ends.
- **Show Symbol** – select the check box to have symbols mark the locations of data points on the curve.
- **Sym Freq** – set the symbol frequency by editing the integer value in the text field.
- **Sym Style** – use the drop-down menu to select a symbol from: Box, Circle, Vertical Ellipse, Horizontal Ellipse, Vertical Up Triangle, Vertical Down Triangle, Horizontal Left Triangle, and Horizontal Right Triangle.
- **Fill Sym** – select the check box to set the symbol display as solid. Otherwise it will display as hollow.
- **Sym Color** – double-click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Report2D Options: Axis

Axis settings change how axes are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Axis**:

- **Axis Name** – this describes the axis to which a row's settings apply. You cannot change this field.
- **Color** – double-click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Font Color** – double-click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Edit Font** – click **Edit Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Min Gutter %** – sets the amount of empty space (gutter) around the axis.
- **Font Description** – describes the settings applied from the **Font** window. You cannot change this field. To change the font, use **Edit Font**.

Report2D Options: Grid

Grid settings change how grids are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Grid**:

- **Grid Name** – this describes the grid to which a row's settings apply. You cannot change this field.
- **Line Style** – use the drop-down menu to select from: Solid, Dot, ShortDash, DotShortDash, Dash, DotDash, DotDot, DotDotDash, and Long Dash.
- **Line Color** – double-click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Report2D Options: Header

Header settings change how headers are displayed in reports. You can separately style the Title and Subtitle.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Header**:

- **Color** – click a color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Font** – click **Edit Title Font** or **Edit Subtitle Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Company Name** – Enter a company name to appear on all reports.
- **Show Ansys Logo** – Remove the check from the box to hide the Ansys logo that appears on all reports (checked by default).

Report2D Options: Note

Note settings change how notes are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Note**:

- **Note Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Note Font** – click **Edit Note Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Background Visibility** – select the check box to make the background color visible. Deselect to make it transparent.
- **Border Line Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Border Visibility** – select the check box make the note border visible. Deselect to make it transparent.
- **Border Line Width** – set the line width by editing the real value in the text field.

Report2D Options: Legend

Legend settings change how legends are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Legend**:

- **Legend Name** – Default is no name. When non-empty, a header row for the Legend in plot shows up with that string.
- **Show Trace Name** – select the check box to show the trace name; deselect it to hide the trace name.
- **Show Solution Name** – select the check box to show the solution name; deselect it to hide the solution name.
- **Show Variation Key** – select the check box to show the variation key; deselect it to hide the variation key.
- **Highlight Curve on Hover** – select the check box to highlight a curve when you hover the cursor over it; deselect it to leave the curve as-is when you hover the cursor.
- **Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Text Font** – click **Edit Text Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Border Line Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Border Line Width** – set the line width by editing the real value in the text field.
- **Grid Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

You can set different **Text Color** and **Text Font** settings for the Header Row.

Report2D Options: Marker

Marker settings change how markers are displayed in reports.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Marker**:

- **Marker Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Marker Font** – click **Edit Marker Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Arrow Direction** – use the drop-down menu to set the arrow direction to Up, Down, Left, or Right.

Report2D Options: Marker Table

Marker Table settings change how marker tables are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Marker Table**:

- **Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Text Font** – click **Edit Text Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Border Line Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Border Line Width** – set the line width by editing the real value in the text field.
- **Grid Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Grid Line Width** – set the line width by editing the real value in the text field.

You can set a different **Text Color** and **Text Font** for the header row.

Report2D Options: X-Y Markers

X-Y Marker settings change how X-Y markers are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **X-Y Markers**:

Background Colors

- **Marker [#] Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Properties

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **On-screen intersection** – select the check box to enable on-screen intersection; deselect to disable.
- **Marker Font** – click **Edit Marker Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Line Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Line Style** – use the drop-down menu to select from: Solid, Dot, ShortDash, DotShortDash, Dash, DotDash, DotDot, DotDotDash, and Long Dash.
- **Line Width** – edit the text field to specify a line width.
- **Show Name** – select the check box to show X-Y marker names; deselect to hide.
- **Snap to Vertex** – select the check box to snap markers to the vertex; deselect to disable snapping.

Inter Marker Deltas

- **Show Delta** – select the check box to show inter-marker deltas; deselect to hide.
- **Delta Font** – click **Edit Delta Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Delta Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Line Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Line Style** – use the drop-down menu to select from: Solid, Dot, ShortDash, DotShortDash, Dash, DotDash, DotDot, DotDotDash, and Long Dash.
- **Line Width** – edit the text field to specify a line width.

Report2D Options: Stacked

Stacked settings change how stacked plots are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Stacked**:

- **Auto Fit Mode** – select the check box to enable auto fit; deselect to turn off autofit.
- **Stack Height in Pixels** – set the default stack height.
- **Curve Grouping Strategy** – can be **Single**, **By Trace**, or **By Units**. **Single** means that a new stacked plot shows a single curve per stack. **By Trace** means that all curves are grouped by their trace. **By Units** means that all curves are grouped by their unit type.

If you change the **Curve Grouping Strategy** set by default, existing stacked plots remain unaffected. The new default will apply only to new stacked plots.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report2D Options: Digital

Digital settings change how digital stack heights are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Digital**:

- **Digital Literal Foreground** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Expand Arrays/Records** – select the check box to automatically expand arrays and records; deselect it to disable.
- **Digital Stack Height in Pixels** – enter the stack height for Analog, Digital, Enum, Event, and Literal stacks.

Report2D Options: General

These settings contain general report options.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **General**:

- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Plot Area Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Highlight Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Accumulate Depth** – enter a value; the default is 4.
- **Enable Y Axis Stripes** – select the check box to add stripes to the Y axis; deselect to remove stripes.
- **Auto Scale Fonts** -on by default and when enabled scales text in plots and colorkey (contour plot, field plots in 3D modeler) for high resolution screens.

Curve Tooltip

- **Show Trace Name** – select the check box to display trace names in the tooltip when hovering the cursor over a curve.
- **Show Variation Key** – select the check box to display the variation key in the tooltip when hovering the cursor over a curve.
- **Show Solution Name** – select the check box to display the solution name in the tooltip when hovering the cursor over a curve.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clipboard Option

- **Capture Aspect Size Ratio** – select either **As Shown** or **Full Screen**.
- **Capture Background Color** – select either **As Shown** or **White**.

Format

- **Field Width** – enter a field width value.
- **Precision** – enter a precision value.
- **Use Scientific Notation** – select the check box to use scientific notation; deselect to disable scientific notation.

Report2D Options: Table

Table settings change how tables are displayed in reports.

From the **Options** dialog box, the following options are set in **Reporter** > **Report 2D** > **Table**:

- **Rows Per Page** – enter the number of rows you would like to display per page; the default is 2500.
- **Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Text Font** – click **Edit Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Border Width** – enter a value for the border width.
- **Border Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Grid Width** – enter a value for the grid width.
- **Grid Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Page Link Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.
- **Arrow Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Header Row

- **Text Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Text Font** – click **Edit Font** to open the **Font** dialog box, where you can select a font, font style (for example, italic), and font size. Then click **OK**.
- **Background Color** – click the color box to display the **Color** dialog box. Select a default or custom color and click **OK**.

Format

- **Field Width** – enter a value for the field width.
- **Precision** – enter a value for the precision.
- **Use Scientific Notation** – select the check box to enable scientific notation; deselect it to disable scientific notation.

Copy to Clipboard

- **With Header** – select to include header when copying the table to a clipboard; deselect to remove it.
- **With Tab Separator** – select to include tab separator when copying the table to a clipboard; deselect to remove it.

Setting HPC and Analysis Options

All analysis parameters are accessed via a single window. The machine list and options settings have been integrated into analysis configurations. The default configuration is for solving on a single, local machine. You can create many analysis configurations for remote and distributed solutions, and switch between them depending on the job being solved. Multiprocessing has been integrated into the machine lists.

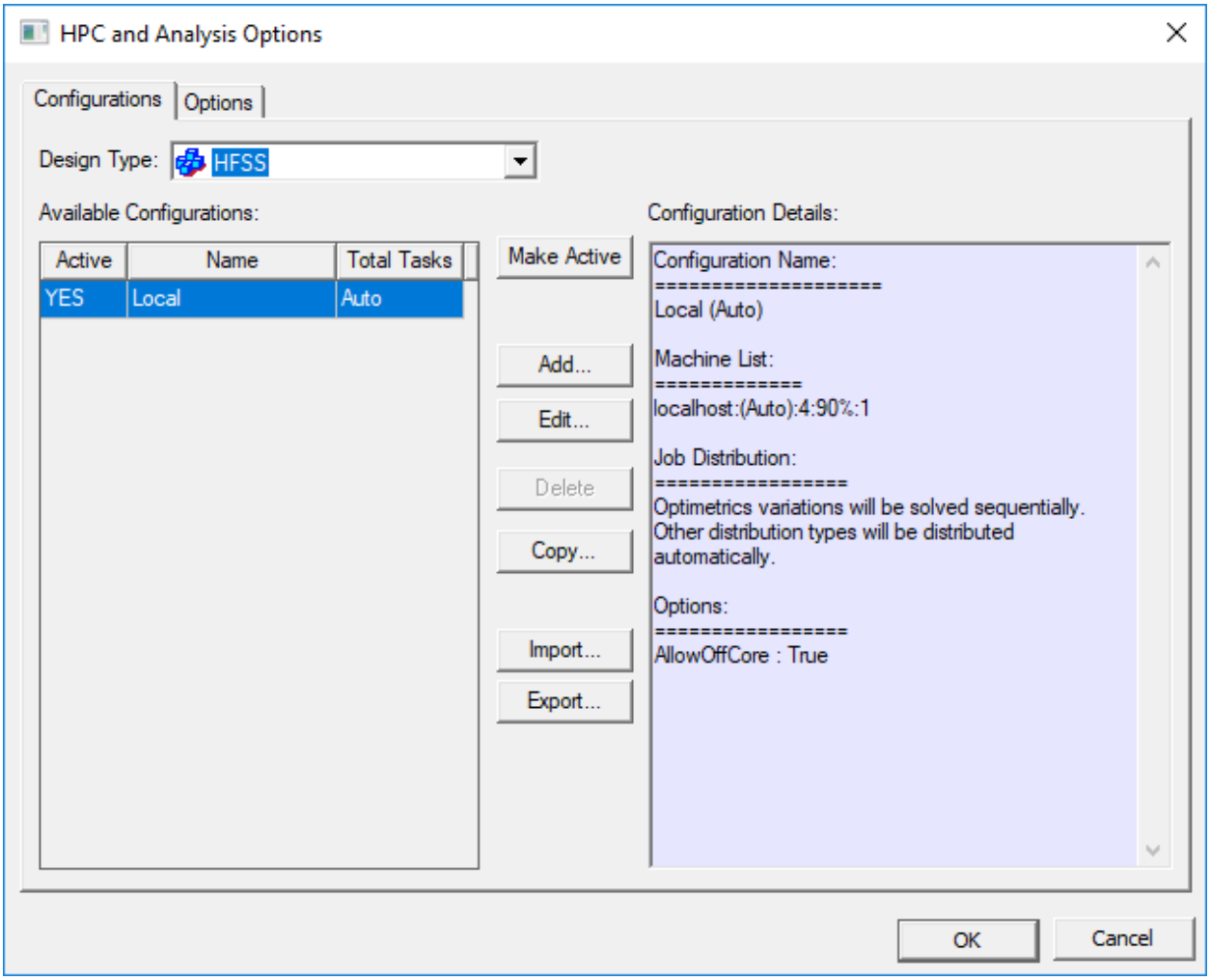
To set HPC and Analysis Options:

1. Click **Tools** > **Options** > **HPC and Analysis Options**.

   You can also access HPC and Analysis Options using the **HPC Options** icon on the **Simulation** ribbon.

   The **HPC and Analysis Options** window appears, displaying the **Configurations** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



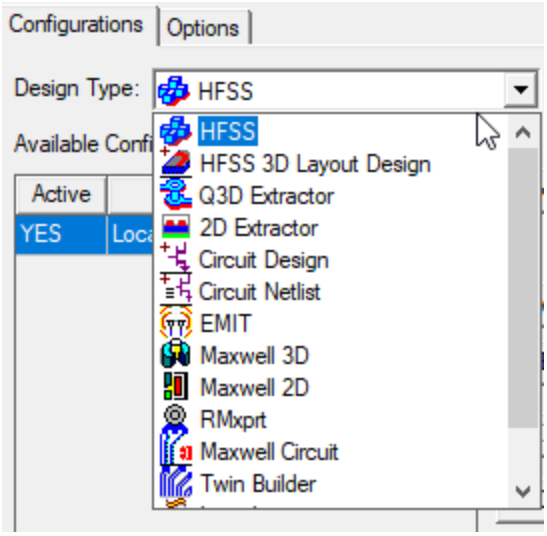
Configurations Tab

Available Configurations are described by **Name**, whether or not they are **Active**, and the **Total Tasks** the configuration can execute.Selecting a configuration from the list displays the details of that configuration in the **Configuration Details** panel.

From the **Configurations** tab, select the **Design Type** to display a list of available configurations for that type.

Configurations must be defined for all design types separately. To use similar analysis parameters for different design types, create separate analysis configurations for each design type. The active configuration is used when solving an analysis for that design type.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selecting an Available Configuration

To activate a configuration, select it from the **Available Configurations** list and click **Make Active**. The active configuration will be indicated by a **YES** in the **Active** column.

Additional options include:

- **Add** – launches a dialog box to create a new analysis configuration.
- **Edit** – launches a dialog box to edit the currently selected analysis configuration.
- **Delete** – deletes the currently selected analysis configuration(s).

  > **Note:** You cannot delete the Local configuration.

- **Copy** – creates a new analysis configuration, and launches a dialog box to edit it. If the dialog box is canceled, the new analysis configuration is not created.
- **Import** – allows you to import an *.acf file to create an analysis configuration.

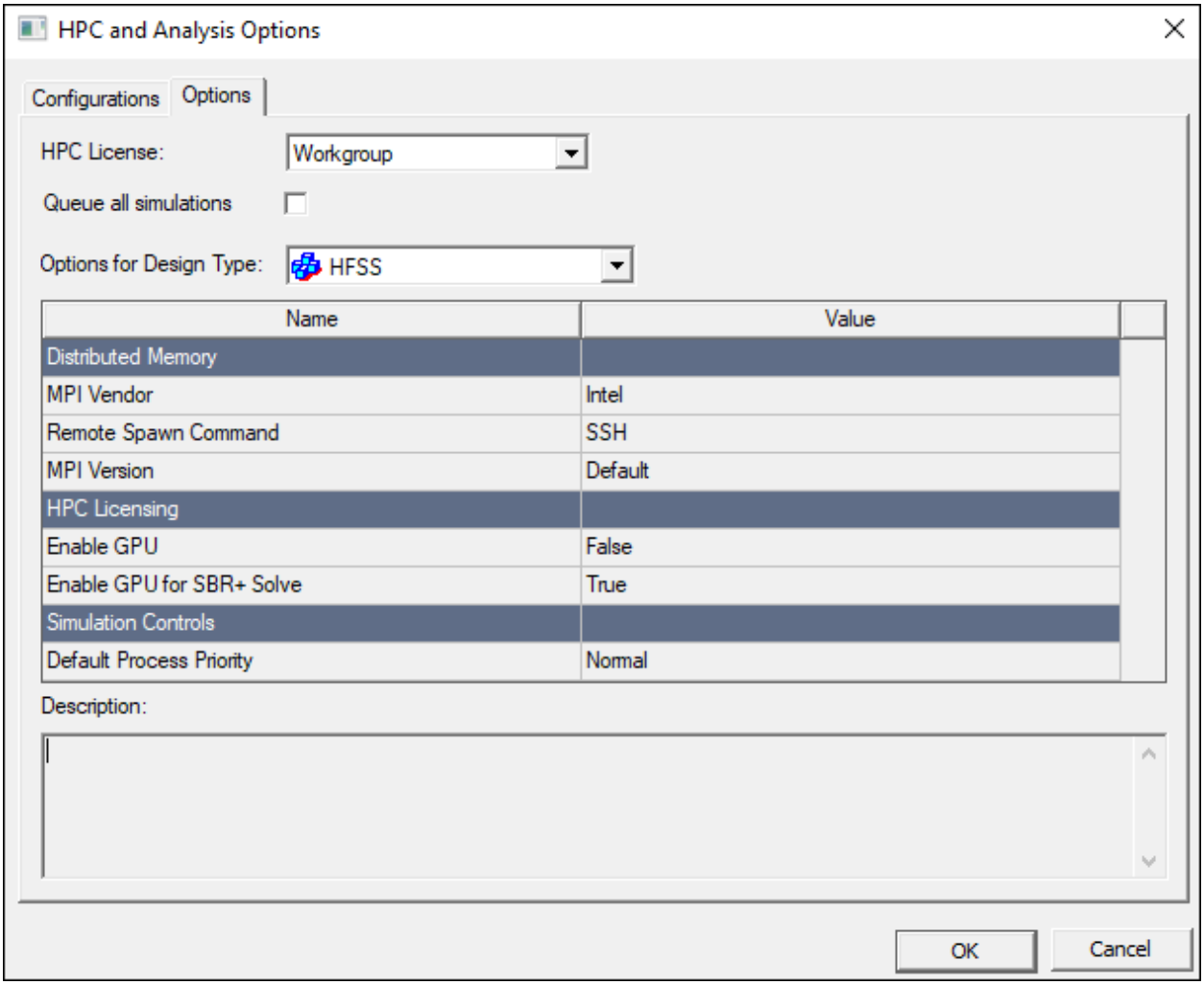
  > **Note:** Importing analysis configurations always adds the imported analysis configurations to the current design type. If there is a name conflict between an imported analysis configuration and an existing analysis configuration, the imported configuration is renamed and a notification appears.

- **Export** – allows you to export the selected analysis configurations to an *.acf file. You can then import the configurations into a different design type, or import them on a different machine.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options Tab

The **Options** tab in the **HPC and Analysis Options** dialog box contains general and product-specific settings.



> **Important:**
>
> Available options vary by design type. Certain items discussed on this page may not be visible, depending on the currently selected design type.

These options are not specified for, or saved as part of, the current analysis configuration. Instead, they are global and are always in effect for the given design type when both of the following conditions are true:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A design of the matching design type is being solved.
- You have not specified corresponding overriding batch options on the command line.

From the **Options** tab, you can:

- Choose the HPC license type
- Enable queuing
- Specify Distributed Memory settings (for example, MPI for certain solvers)
- Set licensing options
- Enable GPU (for Transient, Matrix and SBR+ solves)
- Set the Default Process Priority

> **Note:**
>
> Solving with Intel MPI on a single Windows machine does not require MPI installation, but registration of the user's password with Intel MPI on the host is required if using password authentication. Solving with Microsoft MPI on a single Windows machine requires Microsoft MPI to be installed and the MS-MPI Launch Service to be started. These steps are not required for Microsoft MPI analysis on a Windows HPC cluster. Solving with Microsoft MPI on multiple Windows hosts is only supported on a Windows HPC cluster.
>
> Users running on Linux do not need to install MPI manually.

For **HPC License**, select **Auto**, **Workgroup** or **Pack**.

HPC licensing enables the use of cores and GPUs to accelerate simulations. In general, each core requires one unit of HPC, while each GPU requires eight units. The selected HPC license type determines which license is used, and how units of HPC are converted to license counts.

- **Workgroup** (formerly "pool") – One HPC workgroup license enables one unit of HPC.
- **Pack** – One HPC pack license enables eight units of HPC. Additional packs multiply by four, enabling 32, 128, 512,... , in the context of a single simulation.
- **Auto** – delegate the choice of Workgroup or Pack licensing to the Ansys Licensing Settings tool which is a separate application installed along with Electronics Desktop.

Electronics Desktop products include four units of HPC for each licensed simulation. This means that up to four units can be used without requiring HPC licenses; license counting begins with the fifth unit. For example, a simulation that uses 36 cores requires 32 HPC units after subtracting the four included cores. This simulation will check out 32 HPC workgroup licenses, or two HPC pack licenses.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HPC licenses enable all parallel and distributed simulations, including distributed variations. Distributed variations require a single set of solver licenses, plus HPC to enable the variations.

For HPC Workgroup, distributing *N* variations requires 8*(*N*–1) workgroup licenses and, together with the solver licenses, enables up to four HPC units per variation. Each additional set of *N* workgroup licenses will enable one additional HPC unit per variation. For HPC Pack, distributing *N* variations requires *N*–1 pack licenses and, together with the solver licenses, enables up to four HPC units per variation. Each additional set of *N* pack licenses will enable 8, 32, 128,... additional HPC units per variation.

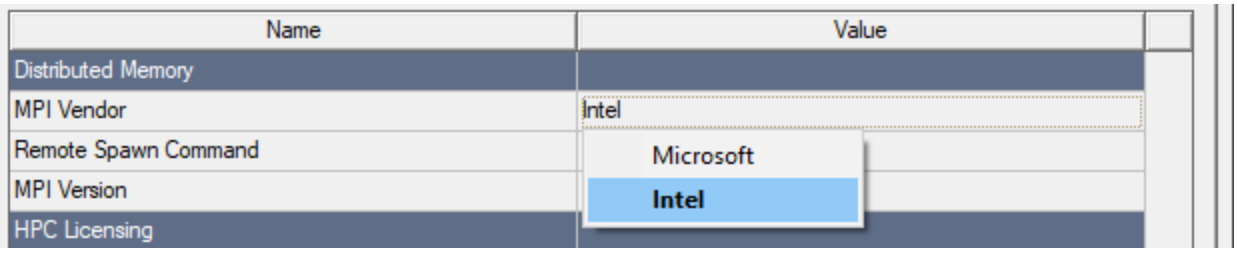
Ansys licensing supports distributed simulations when Electronics Desktop is called from other Ansys tools, such as optiSLang and Workbench. In such cases, distributed design points (variations) generally use HPC counts as described above.

> **Note:**
>
> Licensing for some calling products may include some distributed design points, in which case the total required HPC will be reduced.

If the **Queue all simulations** check box is selected, the Desktop queues any active simulations for design types that have **Save before solving** turned off in the **General Options** and then processes them in order. You can view and change the queue by using Show Queued Simulations.

To configure options:

1. Under **Distributed Memory**, for solvers that use MPI (HFSS, HFSS 3D Layout, Icepack, Maxwell and Q3D) use the drop-down menu to select the MPI Vendor for the selected design type.



   The solvers use the industry standard Message Passing Interface (MPI) and can perform solutions that distribute memory use across machines in a cluster or network. Memory used by the MPI-enabled solver is therefore limited by the set of machines that is available rather than the shared memory available on any single machine. This allows you to simulate larger structures and to optimally reconfigure the cluster of machines for the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

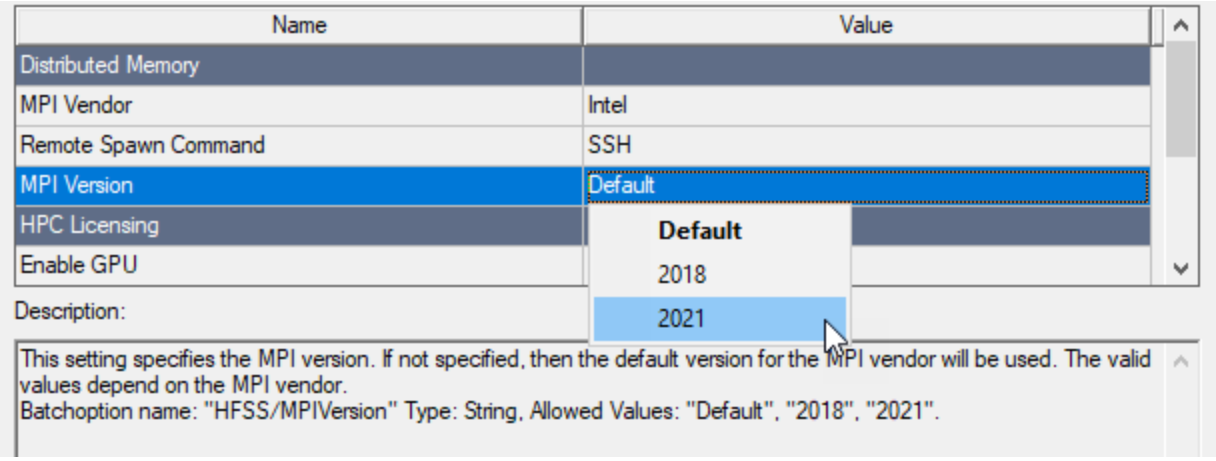
problem at hand. For solving on a single machine, MPI is not required, nor does it provide an advantage.

To use the distributed memory solution you will need to install MPI software from a supported third party vendor on all the machines you intend to use.

Depending on the MPI vendor, you may need to set passwords for authentication on the machines. Settings within each design type turn on distributed memory solutions and define the list of machines you intend to use. Detailed instructions about how to get distributed memory HFSS solutions up and running are outlined in Distributed Memory Solutions with HFSS.

You can specify the MPI version. If not specified, the default version for the MPI vendor will be used. This setting is ignored if there is only one supported version for the selected MPI Vendor. Multiple versions are only supported for Intel MPI, as follows:

- The value "Default" indicates that the default Intel MPI version should be used. This is Intel MPI 2021 in most cases.
- The value "2018" indicates that Intel MPI 2018 should be used.
- The value "2021" indicates that Intel MPI 2021 should be used.



Also see Setting Intel MPI Interconnect for more details.

InfiniBand Support for Windows

By default, MPI vendors use the fastest interconnect by default (typically InfiniBand is faster than Ethernet). If you want to override the default behavior and force the use of Ethernet, you can set the ANSOFT_MPI_INTERCONNECT environment variable to "eth" for the job. Also see Setting Intel MPI Interconnect for more details

2. For Linux authentication, specify the Remote Spawn command as RSH or SSH (the default).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Electronics Desktop supports GPU acceleration for transient, frequency domain, SBR+, and Maxwell 3D eddy current matrix solutions.

   See: Enabling GPU.

   For details on the requirements for GPU use, see Transient GPU Acceleration

4. Optionally, you can select one of the following from the **Default Process Priority** drop-down menu:
   - **Critical** (highest) (Not recommended)
   - **Above Normal** (Not recommended)
   - **Normal** (Default)
   - **Below Normal**
   - **Idle** (lowest)

You can also set these values using VB or Python scripts.

For information on editing configurations, see Editing Distributed Machine Configurations.

High Performance Computing (HPC) Integration

You can use HPC for the following example projects:

- **Helical Antenna** (Examples\HFSS\Antennas\helical_antenna.aedt) – You can enable Domain Decomposition method and setup HPC to solve this large problem.
- **Bandpass Filter** (Examples\HFSS\RF Microwave\bp_filter.aedt) – You can set up HPC to distribute the frequencies in the sweep.
- **Connector** (Examples\HFSS\Signal Integrity\connector.aedt) – Since the project uses a frequency sweep, you can set up HPC.

> **Note:**
>
> Click **File** > **Open Examples** to access the example projects. For more information about the application of HPC in the Bandpass Filter, see the section **Add HPC Analysis and Options** in *Getting Started with HFSS: Bandpass Filter*.

Specifying the Remote Spawn Command (Linux)

An important step in using a high performance cluster is setting up authentication across machines so that the machines can be accessed without a password. By default, HFSS uses SSH authentication on Linux to spawn commands on the remote machines but also supports RSH. The selection of which to use is made on the **Options** tab of the **Tools** > **Options** > **HPC and Analysis**.

SSH

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You will need to set up passwordless access to use HFSS on a Linux cluster with SSH or RSH. In general, for SSH, this is accomplished by:

1. Verifying that you have working SSH servers and clients on your machines.
2. Verifying that the server will accept passwordless logins. You may need to edit the /etc/ssh/ssh_d file to allow RSAAuthentication and PubkeyAuthentication.
3. Generating keys on the client system using the ssh-keygen program. Do not use a passphrase so that you can access the machine without a password.
4. Copying the public key generated in step 1 from the ~/.ssh directory to the server. The easiest way to transfer the keys is to use the ssh-copy-id program. Alternately, you can use any file transfer utility. If the server already has a list of existing keys for other clients add the new public key to the list.
5. Testing the connection. Log in to the client machine using the username that you used to create the identity keys. Open a new shell terminal and attempt to open an SSH login session. For example, type `ssh 192.168.0.4` (where the IP address is the address of the machine you are attempting to connect to). The server should allow you to log in without requesting a password.

Consult the documentation for your machines and network for detailed instructions.

RSH

If you choose to use RSH, you will need to make sure RSH is installed on all the machines and set up the machines so that you are not prompted for a password. There are different ways to set up passwordless RSH, so be sure to consult the documentation for your machines and network for detailed instructions.

Machine access using RSH without a password is often set up by editing the /etc/hosts.equiv file and adding entries for the hosts you would like to use without a password. This file lists hosts and users that are granted "trusted" access to the system.

If you look at the /etc/hosts.equiv file you should have something similar to the following:

Contents of the /etc/hosts.equiv file:

```
job1.n1.com
job2.n1.com
job3.n1.com
```

The machines job1, job2 and job3 can connect without a password. You may also need to verify that the files /etc/hosts.allow and /etc/hosts.deny are empty. Consult your local documentation for detailed instructions and troubleshooting suggestions.

Scheduler

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Remote Spawn Command setting is only meaningful when running on the Linux Operating System. The value 'Scheduler' is valid if the job is a scheduler job running under an LSF, SGE or SLURM scheduler, and only if the MPI Vendor is "Intel".

When submitting a job using the AnsysEM job submission GUI, the Remote Spawn Command for an analysis may be specified using the batchoption with pathname 'DesignType/RemoteSpawnCommand', where DesignType is the Design Type to analyze. The Remote Spawn Command setting is only meaningful when running on the Linux Operating System. The value 'Scheduler' is valid if the job is a scheduler job running under an LSF, SGE or SLURM scheduler, and only if the MPI Vendor is "Intel". To specify the value 'Scheduler' for this option for a scheduler job, the Remote Spawn Command must be specified using the 'DesignType/RemoteSpawnCommand' batchoption in the product command line when the product is launched. In addition, the 'DesignType/MPIVendor' batchoption must be specified with value "Intel" in the product command line when the product is launched. For interactive scheduler jobs, the Remote Spawn Command and the MPI Vendor may be specified with batchoptions or as design type options in the **HPC and Analysis Options** dialog box.

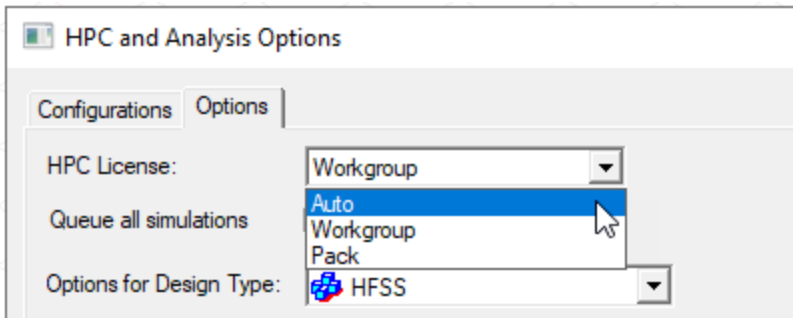
Setting MPI Version

When submitting a job using the Ansys Electromagnetics job submission GUI, the MPI Version may be specified using the batchoption with pathname DesignType/MPIVersion, where DesignType is the type of design to analyze (e.g. HFSS). It allows selection of which Intel MPI version to use, for both Windows and Linux. Valid values are "Default", "2018", and "2021".

For interactive solves, the MPI version may be specified with the batchoption in the command line used to launch the product, or as a design type option in the HPC and Analysis Options dialog box.

Licensing Settings Tool

HPC licensing includes a choice called “Auto”, along with Workgroup and Pack. The “Auto” choice is available in the HPC License combobox found on the **Options** tab of the **HPC and Analysis Options** dialog box. If you make changes in the License Settings Tool, we recommend that you restart Electronics Desktop.



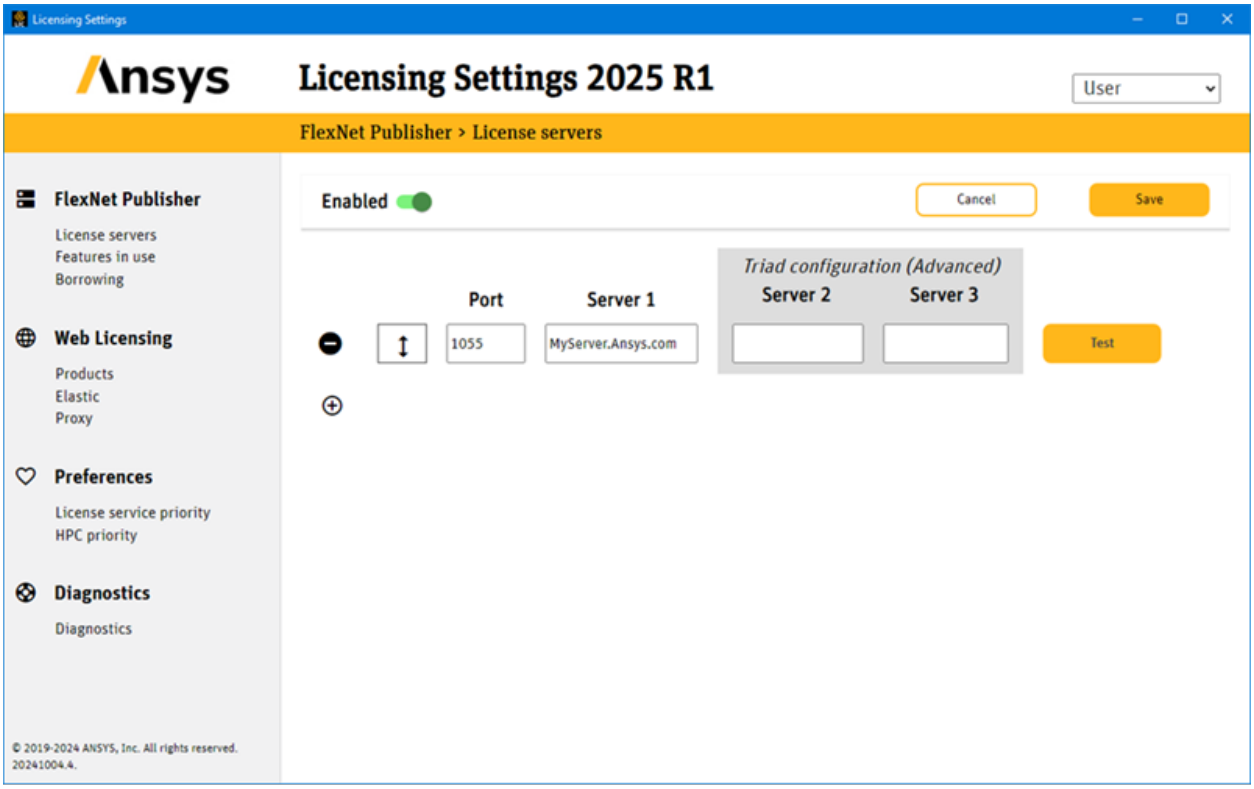
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

"Auto" means delegate the choice of Workgroup or Pack licensing to the Ansys Licensing Settings tool which is a separate application installed along with Electronics Desktop, accessible from the **Tools** menu. The HPC settings specified by the Licensing Settings tool are shared by other Ansys products. This provides you a single place where you can set the HPC preferences.

> **Note:** Important: HPC licensing preferences in the Licensing Settings tool are used only when Auto is selected in Electronics Desktop.

Launching the Licensing Settings Tool from the Main Menu

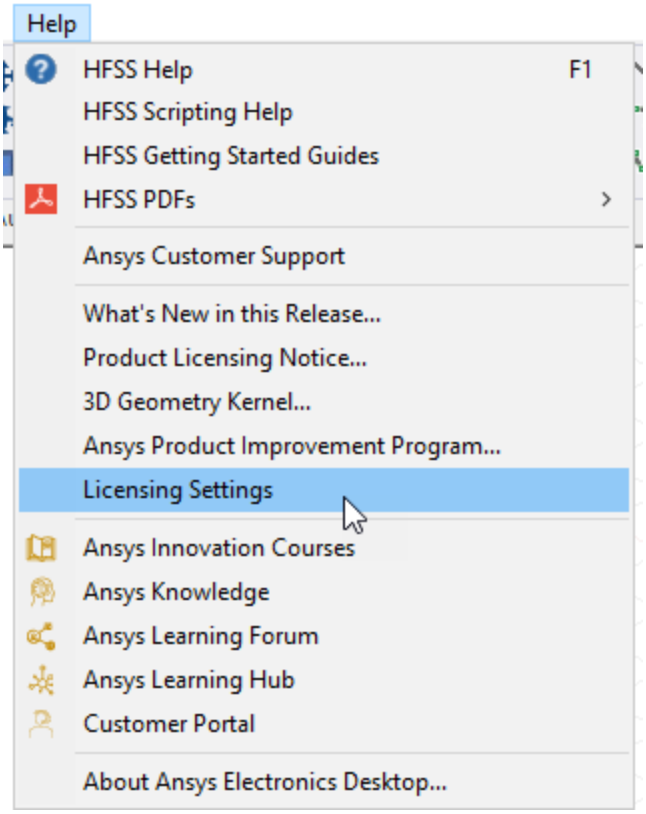
Click **Tools** > **Licensing Settings** to launch the Licensing Settings tool.



Licensing Settings Help

You can obtain help with the Licensing Settings tool from the **Licensing Settings** menu item in the **Help** menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



License Fallbacks

When using Auto HPC licensing, in some cases if there aren't enough available licenses of the highest priority HPC license type, the lower priority HPC license type will be used. For instance, assume the user has set the priority order to "Ansys HPC, Ansys HPC Pack" in the Licensing Settings tool. If the user requests a nominal solve using 14 cores (10 Ansys HPC), but 10 Ansys HPC licenses aren't available, 2 Ansys HPC Pack licenses will be used if available.

Setting Options via Configuration Files

In addition to setting options from the Electronics Desktop user interface (UI), you can also set options in several configuration files. Option settings in configuration files may apply to all users or only to a specific user, and they may apply to all hosts or only to specific hosts. There are four levels of settings, listed below from most specific (highest precedence) to most general (lowest precedence):

- Host-dependent user options – apply to the specified user on the specified host only
- Host-independent user options – apply to the specified user on all hosts
- Host-dependent default options – apply to all users on the specified host
- Installation default – default for all users on all hosts

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the list above, settings at any level override settings at lower levels . If there is no setting in any file, the application default value is used. See UpdateRegistry for instructions on selecting these levels.

> **Important:**
>
> Options set from the Desktop UI override and update user settings in configuration files. Otherwise, the existing configuration file settings are used.

Behavior Examples

Consider running an application as user **jsmith** on host **host123**. If there is no host-dependent user setting for the *Expand Project Tree on Insert* option in the host-dependent user options config file for user **jsmith** on host **host123**, but there is a setting for the "Expand Project Tree on Insert" option in the host-independent user options config file for user **jsmith**, then the latter setting will be used if it is not overridden using the Desktop UI. Any settings in the host-dependent default options config file or the installation default config file are ignored.

As another example, consider running an application as user **jdoe** on host **host123**. If there is no setting for the *Expand Project Tree on Insert* option in the host-dependent user options config file for **jdoe** on **host123** or in the host-independent user options config file for user **jdoe** or in the host-dependent default options config file for host **host123**, then the value from the installation default config file are used, if present.

Rules for Modifying Option Settings

Option settings displayed in the Desktop UI follow the above rules. That is, if there is a setting in any of the option config files, then the setting from the highest priority config file is displayed in the Desktop UI. If there is no setting in any of the option config files, then the global default value is used. You can modify settings using the Options dialog boxes in the Desktop UI. If the dialog box is closed with the **Cancel** button, changes made on any of the tabs are discarded. If the dialog box is closed with the **OK** button, any settings that have been changed by the user are written to the host-dependent user options config file. The changed values written to this file are then used the next time that the application is run by the same user on the same host. The Desktop UI option settings are not written to any of the other option config files.

Configuration File Locations

The configuration files for host-dependent and installation **default** options reside at: "<*installation-directory*>\ANSYS Inc\v251\AnsysEM\config". The configuration files for host-dependent and host-independent **user** options reside in a subfolder of the user's Documents folder (for Windows) or the user's HOME folder (for Linux). See the tables below for specific Windows and Linux file names and paths.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Products with Multiple Desktop Versions

For products that have multiple Desktop versions, each will have a separate user-specific config directory, with a different value for the <*ApplicationName&Version*> directory name.

Table of Directories and Files

The following table shows the directories and files, where the *Level Name* is the name used to describe an options config file when using the UpdateRegistry tool.

Config File	Level Name	File Name	Windows Directory Path	Linux Directory Path
host-dependent user options	user_machine	<*hostname*>_user.XML	%UserProfile%\Documents\Ansoft\<*ApplicationName&Version*>\config	$HOME/Ansoft/<*ApplicationName&Version*>/config
host-independent user options	user	user.XML		
host-dependent default options	install_machine	<*hostname*>.XML	*"<Installation_Directory>*\ANSYS Inc\v251\AnsysEM\config"	*"<Installation_Directory>*/ansys_inc/v251/AnsysEM/config"
installation default	install	default.XML		

Note:

- **<*hostname*>** is the name of the computer on which the Electronics Desktop software is installed
- **$HOME** is the user's home directory on Linux
- **<*ApplicationName&Version*>** is the product name (without spaces) followed by the four-digit year of the version, a decimal point, and the minor release number (such as ElectronicsDesktop2025.1)
- **%UserProfile%** is a Windows variable that represents the currently active user's profile (for example, C:\Users\JohnDoe)
- **<*Installation_Directory*>** is the root folder where the Ansys software is installed (typically, C:\Program Files, on Windows, or /opt, on Linux)
- **<*Version*>** is the last two digits of the product version's year followed by the minor release number, without a decimal point (such as 251)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following table shows an example of specific file names and directory names for a typical Ansys Electronics Desktop installation on Microsoft Windows and on Linux. These are the files that apply to software version **2022 R1**, user "**jsmith**," and hostname "**host123**":

Config File	Level Name	File Name	Windows Directory Path	Linux Directory Path
host dependent user options	user_machine	host123_user.XML	C:\Users\jsmith\Documents\Ansoft\ ElectronicsDesktop2025.1\config	/home/jsmith/Ansoft/ ElectronicsDesktop2025.1/config
host independent user options	user	user.XML		
host dependent default options	install_machine	host123.XML	"C:\Program Files\ANSYS Inc\v251\ AnsysEM\config"	/opt/ansys_inc/v251/AnsysEM/config
installation default	install	default.XML		

Note:

As with the temporary file location configuration files, the settings in these options files have precedence in the following sequence: user_machine (highest precedence), user, install_machine, install (lowest precedence). The first time you start and then exit the application, the file at the "user_machine" level is created (<*hostname*>_user.XML). The other files are only created if you use the UpdateRegistry tool to specify an option at the "user," "install_machine," or "install" level. If the temporary directory is set to an empty string in a configuration file, then that setting is ignored.

Setting or Removing Option Values in Configuration Files: UpdateRegistry Command

UpdateRegistry is a command line tool used to modify option settings in the options config files. You can use this command to add, change or remove settings from any of the option config files. This tool is included in the installation directory of each product. This feature makes it easier for different users to use Ansys Electromagnetics tools installed on shared directories or network drives.

The UpdateRegistry command has multiple command line formats, as shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following command line options are *mutually exclusive*:

- -Set
- -Get
- -GetKeys
- -Delete
- -FromFile

UpdateRegistry -Set Command

This command is used to add or modify an option setting in an option config file. If the option config file does not exist, it will be created. If the setting does not exist in the specified config file, it will be added. If the setting already exists in the specified config file, then the value will be changed to the specified value.

Example:

```
UpdateRegistry -Set -ProductName <name> -
RegistryKey <keyPath> -RegistryValue <value> [
-RegistryLevel <level> ]
```

Required:

<name>	The application or product name and version. For example, ElectronicsDesktop2025.1. If the name contains spaces, it must be quoted. The name can be found in the ProductList.txt file in the install directory: "...\ANSYS Inc\v251\AnsysEM\config\"
<keyPath>	The pathname of the option setting. This includes the same analysis-related registry keys and values that are displayed by the -batchoptions help. For example, Desktop/Settings/ProjectOptions/AnimationMemory
<value>	The new value of the option, typically a string or a number. If the value contains spaces, it must be quoted.

Optional:

<level>	When specifying -RegistryLevel, this is a string denoting which config file to modify. One of: install, install_machine, user, and user_machine. If the level is not specified, the user_machine (host-dependent user options) file is modified.

UpdateRegistry -Get Command

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command is used to view an option value in an option config file. If the setting exists in the specified config file or files, then the value, the value type and the config file where the value was found will be reported. If no value is found, then that will also be reported.

Example:

```
UpdateRegistry -Get -ProductName <name> -
RegistryKey <keyPath> [ -RegistryLevel <level>
]
```

Required:

`<name>`	The application or product name and version. For example, ElectronicsDesktop2025.1. If the name contains spaces, it must be quoted. The name can be found in the ProductList.txt file in the install directory: "...\ANSYS Inc\v251\AnsysEM\config\"
`<keyPath>`	The pathname of the option setting. This includes the same analysis-related registry keys and values that are displayed by the -batchoptions help. For example, Desktop/Settings/ProjectOptions/AnimationMemory

Optional:

`<level>`	When specifying `-RegistryLevel`, this is a string denoting which config file to modify. One of: install, install_machine, user, and user_machine. If the level is not specified, then all config files are searched in order of precedence.

UpdateRegistry -GetKeys Command

This command is used to view the allowed key names for all of the option settings, or to view a subset of the key names that match a string. For each key displayed, the current value, if any, is also reported. If a key has a value in multiple config files, then only the highest precedence value is reported.

Example:

```
UpdateRegistry -GetKeys [ <pattern> ] -
ProductName <name> [ -Case ]
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Required:

`<name>`	The application or product name and version. For example, ElectronicsDesktop2025.1. If the name contains spaces, it must be quoted. The name can be found in the ProductList.txt file in the install directory: "...\ANSYS Inc\v251\AnsysEM\config\"

Optional:

`<pattern>`	If no pattern is specified, then all allowed key names are reported. If a pattern is specified, then only keys that match the pattern are shown. For example, Settings/Project. If the name contains spaces, it must be quoted. By default, the pattern match is case insensitive.
`-Case`	If this command line option is specified, then the pattern match is case sensitive.

UpdateRegistry -Delete Command

This command is used to remove an option setting from an option config file. If the setting does not exist in the specified config file, the file will not be changed. If the setting exists in the specified config file, then it will be removed. A setting may need to be removed from an option config file, to allow the setting from a lower priority file to be used by the application.

Example:

```
UpdateRegistry -Delete -ProductName <name> -
RegistryKey <keyPath> [ -RegistryLevel <level>
]
```

Required:

`<name>`	The application or product name and version. For example, ElectronicsDesktop2025.1. If the name contains spaces, it must be quoted. The name can be found in the ProductList.txt file in the install directory: "...\ANSYS Inc\v251\AnsysEM\config\"
`<keyPath>`	The pathname of the option setting. This includes the same analysis-related registry keys and values that are displayed by the -batchoptions help. For example, Desktop/Settings/ProjectOptions/AnimationMemory

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optional:

`<level>`	When specifying `-RegistryLevel`, this is a string denoting which config file to modify. One of: install, install_machine, user, and user_machine. If the level is not specified, the user_machine (host-dependent user options) file is modified.

UpdateRegistry -FromFile Command

You can use this form of the UpdateRegistry command to set multiple key-value pairs from a file with a single UpdateRegistry command. You specify the -FromFile command line option. This option must be followed by a filename. The file may contain multiple entries, where each entry contains a registry key and a registry value. The key-value pairs are added to the registry level specified by the -RegistryLevel command line option; if no -RegistryLevel is specified, then the default registry level (user_machine) is used.

UpdateRegistry File Format

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

The file format is similar to the -batchoptions file format. An example UpdateRegistry file is shown below:

```
$begin 'AddEntries'
  'TempDirectory'='C:/temp/AnsysEM'
  'Desktop/Settings/ProjectOptions/HPCLicenseType'='Pool'
$end 'AddEntries'
```

Additional notes on the file format:

- The file may contain an arbitrary number of entries, one per line.
- Leading whitespace on each line is ignored. Spaces or tabs may be used to make the file more readable.

Registry key pathname:

- The registry key pathname appears before the equal sign "=" on each line.
- Each registry key pathname must be enclosed in single quotes.
- This includes the same analysis-related registry keys and values that are displayed by the -batchoptions help.

Registry value:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The registry value appears after the equal sign on each line.
- Integral registry values must not be enclosed in quotes.
- All other registry values are treated as strings, and must be enclosed in single quotes.
- The forward slash "/" may be used as a directory separator on Windows and Linux. The back slash "\" may be used as a directory separator on Windows only.
- The back slash "\" is used as an escape character in the value string. That it, this character removes the special meaning of the following character.
- The single quote character normally ends the value string. The back slash may be used to remove this special meaning, and include a single quote in the string.
- To use a back slash as a directory separator on Windows, it must be escaped. That is, a double back slash "\\" is used to denote a single directory separator.

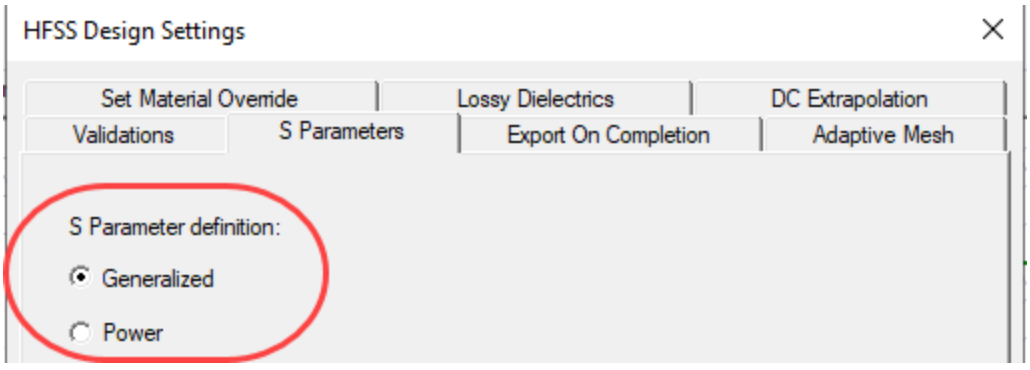
Alternative UpdateRegistry File Format:

- Analysis Configuration File format, which is exported from the HPC and Analysis Options dialog box.

> **Note:**
>
> If a current registry does not exist, Ansys Electronics Desktop can detect earlier minor versions of same application on the same machine. If such a registry exists (and does not involve -help, -batchoptionhelp, IsBatchMode(), -regserver, -unregserver, running a script, or non graphical mode), a prompt displays allowing you to port the registry from an earlier version.

Workaround for HFSS Design Setting for Power S

Use Power S Parameter Definition in HFSS Design Settings can be set using UpdateRegistry but requires the following workaround.



Add one line in ElectronicsDesktopRegistrySyntax.xml.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
-----------------------------------------------------------------
<KEY ObjectName="Hfss" RegObjectType="0">
...
<KEY ObjectName="Preferences" RegObjectType="0">
...
<VALUE ObjectName="UseFerriteMaterials" Value="1" ValueType="4"/>
<VALUE ObjectName="UsePowerS" Value="0" ValueType="4"/> ←add this
line
<VALUE ObjectName="UseWizard" Value="1" ValueType="4"/>
...
-----------------------------------------------------------------
```

Then execute

```
"C:\Program Files\ANSYS Inc\v251\AnsysEM\UpdateRegistry.exe" -Set -
ProductName ElectronicsDesktop2025.1 -RegistryKey
"Hfss\Preferences\UsePowerS" -RegistryValue 1
```

Example Uses for Export Options Features

The **Tools** > **Options** > **Export Options** feature is intended to make it easier for different users to use Ansys Electromagnetics tools installed on shared directories or network drives. This section outlines some use cases enabled by this feature.

> **Note:**
>
> Functionality featured in the examples in this topic apply to multiple design types.

Options that Apply to All Users

In many cases, an Ansys Electromagnetics tool installation is administered and maintained by a single user or group and used by a number of other users or groups. The permissions of the Ansys Electromagnetics tool installation may be set so that the administrator may add, delete or modify files, but other users may only read or execute these files. The administrator may set the recommended option settings in the installation default config file and/or the host dependent default options config file. These config files reside within the installation directory hierarchy, and should generally have the same permissions as other Ansys Electromagnetics tool installation files. This allows that administrator to control these settings, but does not allow other users to add, remove, or change these settings.

Each user can override any of these settings, if needed. This may be done using the Desktop UI, which affects the host-dependent user options config file. It may also be done using the host-independent user options config file. If user has overridden an option setting in either of the user

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

files, the user may revert back to the option settings provided by the administrator by removing the setting of the same option in the host-dependent user option config file and/or the host-independent user option config file.

For global defaults, the administrator may set a value in the installation default config file. These settings will to apply to all users on all hosts.

In some cases, there are significant differences between the capabilities of different hosts. The host-dependent default config file may be used to specify different default values on some hosts. Any setting in a host dependent default config file would affect all users running on the specified host. The installation default value is used if there is no value specified for the setting in the host-dependent default config file for the current host. Note that the host-dependent default config file is named <*hostname*>.xml, where *hostname* is the name of the host computer.

Example: Searching for a Registry Key Pathname

Both administrators and ordinary users may occasionally use the UpdateRegistry command line tool to add, change or delete settings. To use this tool, the registry key pathname must be known by the user. The -GetKeys option may be used to quickly search for a key pathname if some information is known about it. For example, if the administrator knows that there is a setting related to issuing warning messages when available disk space is low, but does not know the exact key name, the following command may list some of the keys related to disk space:

```
UpdateRegistry -GetKeys disk -ProductName ElectronicsDesktop2025.1
```

This will display a list of all keys that match the string "disk" case insensitively.

Typical output may look like the following:

```
Registry keys matching pattern <disk> case insensitively:
Desktop/Settings/ProjectOptions/DiskLimitForAbort: value is <0> at
level <user_machine>
```

Example: Setting an Installation Default Value

The normal default for the Options/General/Desktop Performance/Warn when available disk space is less than setting is 0 MB. If the administrator is concerned that running out of disk space might be a common problem, the administrator could set the installation default for the warn setting setting to 1000 MB, for example. This limit would then apply to all users running on all hosts. The administrator could use the following command to change this setting for Ansys Electronics Desktop:

```
UpdateRegistry -Set -ProductName ANSYSElectronicsDesktop2025.1
 -RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort
 -RegistryValue 1000
 -RegistryLevel install
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example: Setting a Host-Dependent Default Value

For this example, assume that all hosts have two cores, except for three hosts: bighost1, bighost2, and bighost3, which have eight cores each. The administrator has set the Desktop/Settings/ProjectOptions/NumberOfProcessors option value to 2 in the installation default config file. The administrator may set the Desktop/Settings/ProjectOptions/NumberOfProcessors option value to 8 in the host-dependent default config files for the three hosts having 8 cores: bighost1, bighost2 and bighost3. The administrator may log in to host bighost1 and run the following command to change this setting for the host-dependent default options config file for host bighost1:

```
UpdateRegistry -set -ProductName ElectronicsDesktop2025.1
 -RegistryKey Desktop/Settings/ProjectOptions/NumberOfProcessors
 -RegistryValue 8
 -RegistryLevel install_machine
```

To make this change for the other two hosts, the administrator would log in to bighost2 and bighost3, and run the same command on each of those hosts.

Example: Reverting from a User-Defined Option Value to the Administrator Default

Consider the case in which Electronics Desktop was installed and the administrator initially did not set a value for the Desktop/Settings/ProjectOptions/DiskLimitForAbort setting in the default installation config file. User jsmith (who always uses host jshost) wanted to be warned before disk space dropped to zero, so he set the Desktop/Settings/ProjectOptions/DiskLimitForAbort to 100 MB using the UI. This setting is recorded in the host dependent user options config file for host jshost and user jsmith. Now the administrator learns that many users are running into disk space issues, so that administrator sets the installation default value for the setting Desktop/Settings/ProjectOptions/DiskLimitForAbort to 1000 MB, as in the above example.

When user jsmith runs Electronics Desktop on host jshost, the disk limit is 100 MB, not 1000 MB, because the host-dependent user options config file overrides all of the other config files. User jsmith may revert to the administrator provided default by removing this setting from the host dependent user options config file for host jshost and user jsmith. The following command may be run by user jsmith on host jshost to remove this setting:

```
UpdateRegistry -Delete -ProductName ElectronicsDesktop2025.1
 -RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort
 -RegistryLevel user_machine
```

If user jsmith had added a value for this setting to the host-independent user options config file, then user jsmith would also run the following command to remove this setting from the host-independent user options config file:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
UpdateRegistry -Delete -ProductName ElectronicsDesktop2025.1
 -RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort
 -RegistryLevel user
```

User Options and the UpdateRegistry Tool

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

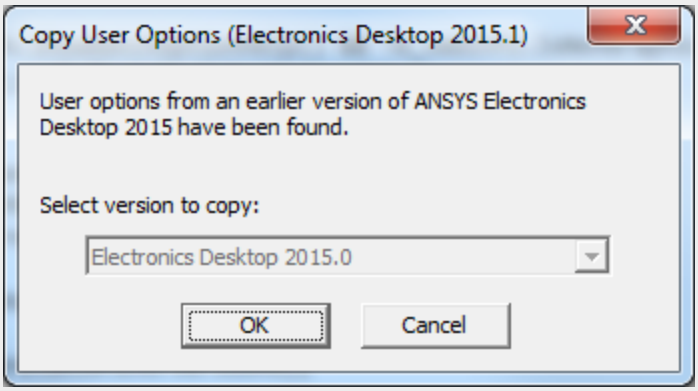
When you change an options value using the Desktop UI, the new value is stored in the host-dependent user options config file. You can use the UpdateRegistry tool to add or modify settings in the host-dependent user options config file; however, you cannot use the Desktop UI to remove settings from the host-dependent user options config file. *You must use the UpdateRegistry tool to remove settings from the host-dependent user options config file.*

If a user has not explicitly created a host-dependent user options config file or a host-independent user options config file, when they first run an Ansys Electromagnetics tool on a host, all settings will come from the host-dependent default options config file or the installation default options config file. Any settings for another host in a host-dependent user options config file will not be carried over to the new host. This may be inconvenient if the user has preferred option settings that differ from the settings that apply to all users, especially if the user runs the Ansys Electromagnetics tool on a number of different hosts. In this case, the user may set these option values in the user's host-independent user options config file. Then, these option values will be used on all new hosts, overriding any values set by the administrator to apply to all users. Any changes made in the UI will only affect the user's host-dependent user options config file for the current host.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

If a current registry does not exist, the Ansys Electronics Desktop can detect earlier minor versions of same application on the same machine. If such a registry exists (and does not involve -help, -batchoptionhelp, IsBatchMode(), -regserver, -unregserver, running a script, or non graphical mode), a prompt displays from which you can select an earlier version from which the registry will be ported.



Copy over registry entries (both Windows and Ansys .xml files).

Example: Removing a Host-Dependent User Option Setting

For this example, user jsmith always uses host jshost to run Ansys Electronics Desktop. At some point, jsmith set the Autosave interval in **General Options** > **Project Options** tab to 1000 edits, and this value was written to the jsmith's host-dependent user options config file for host jshost. Now, jsmith wants to remove this setting and return to the default value of 10. User jsmith may run the following command on host jshost to remove the Desktop/Settings/ProjectOptions/AutoSaveInterval option value from this config file:

```
UpdateRegistry -Delete -ProductName ElectronicsDesktop2025.1
-RegistryKey Desktop/Settings/ProjectOptions/AutoSaveInterval
-RegistryLevel user_machine
```

Example: Getting a Value from a Specific Configuration File

In the previous example, the user jsmith may decide to check the Desktop/Settings/ProjectOptions/DiskLimitForAbort setting in the host-independent user configuration file before making any changes to this setting. The following command may be used to quickly view this setting before making the change:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
UpdateRegistry -Get -ProductName ElectronicsDesktop2025.1 -
RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort -
RegistryLevel user
```

Example: Getting a Value Using Precedence Rules

In may cases, the user is more interested in the value of a setting that will be applicable when running the product than in the setting in a single configuration file. If the -Get option is used with no -RegistryLevel specified, then the value reported is the value found in the highest precedence configuration file. If the user jsmith is interested in the highest precedence value for the Desktop/Settings/ProjectOptions/DiskLimitForAbort setting, then the following command may be used to report this information:

```
UpdateRegistry -Get -ProductName ElectronicsDesktop2025.1 -
RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort
```

Example: Adding a Host-Independent User Option Setting

Consider the case in which there is no value set for the Desktop/Settings/ProjectOptions/DiskLimitForAbort setting for all users for Ansys Electronics Desktop. The default is then 0 MB. User jsmith uses a variety of hosts and wants to be warned whenever disk space drops to 250 MB on any host. User jsmith may use the following command to set the Desktop/Settings/ProjectOptions/DiskLimitForAbort option value to 250 MB for all hosts:

```
UpdateRegistry -set -ProductName ElectronicsDesktop2025.1
-RegistryKey Desktop/Settings/ProjectOptions/DiskLimitForAbort
-RegistryValue 250 -RegistryLevel user
```

Example: Setting a RegistryKey not defined in ElectronicsDesktopRegistrySyntax.xml

```
"C:\Program Files\ANSYS Inc\v251\AnsysEM">UpdateRegistry -Set -
ProductName ElectronicsDesktop2025.1 -RegistryKey "3D
Editors/Preferences/Geometry3D/q" -RegistryValue "m"

registry key is not defined in this product

Register value <3D Editors/Preferences/Geometry3D/q> is created with
value <m>
```

Example: Change a Newly Set Registry Key

```
"C:\Program Files\ANSYS Inc\v251\AnsysEM\">UpdateRegistry -Set -
ProductName ElectronicsDesktop2025.1 -RegistryKey "3D
Editors/Preferences/Geometry3D/q" -RegistryValue "n"
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Register value <3D Editors/Preferences/Geometry3D/q> is set to <n>
```

If you attempt to set a registry key to the wrong data type, a message like the following appears.

```
Cannot set the registry value to different data type
The work around is delete it, then create it again
error on to set registry value at <3D
Editors/Preferences/Geometry3D/q>
```

Setting the Temporary Directory

The temporary directory may be viewed or set using the Electronics Desktop user interface (UI), or from the command line.

To set the directory via the UI:

Navigate to **Tools** > **Options** > **General Options**. In the tree at the left side of the dialog box, expand the **General** branch and select **Directories**. Then, use the **Temp** field and **Override** check box to enter a desired directory path. Values set in this manner are written to the user_machine level configuration file for the temporary directory. If the **Override** check box is cleared, clicking **OK** changes the user_machine level setting for the temporary directory to an empty string. This enables settings from the next highest precedence configuration file. The file that provides the currently active temporary directory setting is shown under the **Temp** edit box.

To set the temporary directory from the command line, using the -batchoptions command line option. See: Running Ansys Electronics Desktop from a Command Line and -Batchoptions Command Line Examples.

The temporary directory may be configured with an installation default value, as well as a host-dependent default value, a host-independent user-specified value and a host-dependent user-specified value. Temporary directory settings are stored in different files from the other option settings. These files are located in the same directories as the configuration files for the other option settings. The following table shows the directories and files used to store temporary directory settings.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Config File	Level Name	File Name	Windows Directory Path	Linux Directory Path
Host-dependent, user-specific temporary directory	user_machine	<*hostname*>.cfg	%UserProfile%\Documents\Ansoft\<*ApplicationName&Version*>\config	$HOME/Ansoft/<*ApplicationName&Version*>/config
Host-independent, user-specific temporary directory	user	default.cfg		
Host-dependent, default temporary directory	install_machine	<*hostname*>.cfg	"<*InstallationDirectory*>\ANSYS Inc\v251\AnsysEM\config"	<*InstallationDirectory*>/ansys_inc/v251/AnsysEM/config
Installation default temporary directory	install	default.cfg		

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- ***<hostname>*** is the name of the computer on which the Electronics Desktop software is installed
- **$HOME** is the user's home directory on Linux
- ***<ApplicationName&Version>*** is the product name (without spaces) followed by the four-digit year of the version, a decimal point, and the minor release number (such as ElectronicsDesktop2025.1)
- **%UserProfile%** is a Windows variable that represents the currently active user's profile (for example, C:\Users\JohnDoe)
- ***<InstallationDirectory>*** is the root folder where the Ansys software is installed (typically, C:\Program Files, on Windows, or /opt, on Linux)
- ***<Version>*** is the last two digits of the product version's year followed by the minor release number, without a decimal point (such as 251)

As with other options, the settings in these files have precedence in the following sequence: user_machine (highest precedence), user, install_machine, install (lowest precedence). The installer creates the file at the "install" level (default.cfg). The first time you start and then exit the application, the file at the "user_machine" level is created (<*hostname*>.cfg). The other files are only created if you use the UpdateRegistry tool to specify an option at the "user" or "install_machine" level. If the temporary directory is set to an empty string in a configuration file, then that setting is ignored.

Temporary Directory Configuration File Format

This section describes the format of temporary directory configuration files. The format is the same for files at all four levels: user_machine, user, install_machine, and install. These files are text files, so any text editor may be used to modify or create them.

An example temporary directory configuration file is shown below:

```
$begin 'Config'
tempdirectory='C:/TEMP/AnsysEM'
$end 'Config'
```

The temporary directory specified by this configuration file is C:/TEMP/AnsysEM.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

- The string containing the pathname of the temporary directory must be enclosed in single quotes.
- The forward slash (/) may be used as a directory separator on Windows and Linux. The backslash (\) may be used as a directory separator on Windows only.
- The backslash (\) is used as an escape character in the tempdirectory string. That it, this character removes the special meaning of the following character.
- The single quote character normally ends the tempdirectory string. The backslash may be used to remove this special meaning, and include a single quote in the string.
- To use a backslash as a directory separator on Windows, it must be escaped. That is, a double backslash "\\" is used to denote a single directory separator.
- On Windows, a UNC path normally begins with two backslash characters. In a tempdirectory string, each of these backslash characters must be doubled, so four consecutive backslashes "\\\\" are used in the config file.

Example: Config File with UNC

```
$begin 'Config'
tempdirectory='\\\\hostxyz\\TEMP\\abc'
$end 'Config'
```

Here, hostxyz is a host with a sharename TEMP having subdirectory abc used as the temporarydirectory. This shows that four backslashes are required for UNC names and that backslashes used as directory separators must be doubled.

Example: Config File with Single Quote

```
$begin 'Config'
tempdirectory='C:/TEMP/ab\'cd'
$end 'Config'
```

Temporary directory is C:/TEMP/ab'cd. This shows how to include a single quote in a tempdirectory pathname. It also shows that forward slashes may be used as directory separators on Windows.

UpdateRegistry: Setting or Removing Temporary Directory Values in Configuration Files

The UpdateRegistry command line tool, described above, may be used to view, add, change or remove the temporary directory setting from any of the temporary directory config files. The registry key for viewing or modifying the temporary directory is TempDirectory.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The -Get, -Set, and -Delete options are valid for viewing, adding, changing, or deleting a temporary directory setting.

The -GetKeys option does not list the temporary directory key.

Batchoptions Command Line Examples

The `-batchoptions` entries command line argument may be used to specify one or more batchoptions settings from the command line. To specify multiple entries using a single `-batchoptions` argument, the entries should be enclosed in double quotes. Alternatively, the batchoptions may be specified in a file using the `-batchoptions <filename>` command line argument format. In this case, the filename is an absolute or relative pathname of the file containing the batchoptions, as described above.

> **Important:**
>
> The two approaches may not be combined: either all batchoptions must be in a file or all batchoptions must be specified explicitly on the command line.

Batchoptions File Format

> **Note:**
>
> Functionality featured in the example in this section applies to multiple design types.

An example batchoptions file is shown below:

```
$begin 'Config'
   'Desktop/ProjectDirectory'='C:/test/projects'
   'Desktop/Settings/ProjectOptions/NumberofProcessors'=2
$end 'Config'
```

Additional notes on the file format:

- The file may contain an arbitrary number of batchoption entries, one per line.
- Leading whitespace on each line is ignored. Spaces or tabs may be used to make the file more readable.
- The **Registry Key** appears before the equals sign (=) on each line and must be enclosed in single quotes ('). The registry key includes the key path and the name of the registry value.
- The registry value (or option value) appears after the equals sign on each line.
- Registry keys are case-insensitive.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- There are two supported types of registry values—string and integer:
  - Each **string** value must be enclosed in single quotes (').
  - Do **not** enclose **interger** values in quotes.
- For file paths within string values, the forward slash (/) may be used as a directory separator on both Windows and Linux systems.
- Alternatively, on Windows only, the customary backslash (\ ) may be used as a directory separator. However, in string values, the backslash is used as an escape code for indicating special characters that cannot be typed directly (such as \n for a new line). Therfore, if you use the backslash as a directory path separator, each instance must be doubled (\\ ). An example is: '\\\\host3\\temp\\Ansoft'. In this case, each double backslash is interpreted as a single backslash (\\host3\temp\Ansoft).
- The single quote character (') normally ends a string value. If you need to include a single quote (or apostrophe) within a string, use the backslash-apostrophe (**\'**) escape sequence. For example, the string '%UserProfile%/Documents/Ansoft/John\'s_Files', is interpreted as: %UserProfile%/Documents/Ansoft/John's_Files.

Example -BatchOptions with -Remote (Windows)

> **Note:**
>
> Functionality featured in the examples in this section applies to multiple design types.

In this example, we run a batch HFSS analysis of project file project1.aedt. We want all temporary files and directories to be created in C:\temp\HFSS instead of using the default temporary directory. We decide that the analysis will be done on a remote host, at IP address 12.34.56.78. Because of limited memory on the remote host, we decide to run the analysis using only a single COM engine. Because the remote host has four cores, we decide to use four four processors for of the analysis. We can use the **-Remote** option to specify that there will be a single remote COM engine.

Here is a sample command line for this analysis, where the project file \\somehost\projects\project1.aedt is located in a shared directory specified using a UNC path:

```
ansysedt -BatchSolve -Remote -Machinelist list=12.34.56.78
 -batchoptions "'TempDirectory'='C:/temp/HFSS'
'Desktop/Settings/ProjectOptions/NumberofProcessors'=4"
\\somehost\projects\project1.aedt
```

An alternative is to use the **-Distributed** command line option. Because the -Machinelist list contains only one host, there is a single remote COM engine in this case also.

```
ansysedt -BatchSolve -Distributed -Machinelist list=12.34.56.78
 -batchoptions "'TempDirectory'='C:\\temp\\HFSS'
 'Desktop/Settings/ProjectOptions/NumberofProcessors'=4"
 \\somehost\projects\project1.aedt
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The second line of the first example shows that you can use the forward slash "/" as a Windows directory separator within option value 'strings'. In this case, it is used in the TempDirectory path. You can also use the customary backslash "\" as a Windows directory separator, but it must be doubled to "\\" because the backslash is also an escape character within parameter strings. This usage is demonstrated in the second line of the second example, again in the TempDirectory path.

Example -Batchsolve with -Machinelist (Windows)

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

Suppose that we want to run a batch HFSS analysis of project file project1.aedt. Because all of our hosts have multiples of 2 cores, we specify that we will use two threads for multiprocessing for both the distributed (HFSS/NumCoresPerDistributedTask) and non-distributed (Desktop/Settings/ProjectOptions/NumberOfProcessors) parts of the job. The analysis contains a sweep that will be distributed across three hosts: ahmed, bill, and catherine. The hosts ahmed and bill have four cores each, so we run two distributed COM engines on each of these hosts, each using two threads. Host catherine has only two cores, so we specify only one distributed COM engine on this host. This COM engine will also use two threads. We specify a desired RAMLimitPercent of 75 for this analysis.

Here is a sample command line for this analysis, where the project file \\dennis\projects\project1.aedt is located in a shared directory specified using a UNC path:

```
ansysedt -BatchSolve -Distributed
-Machinelist list=ahmed,ahmed,bill,bill,catherine
-batchoptions "'Hfss/RAMLimitPercent'=75
'Desktop/Settings/ProjectOptions/NumberOfProcessors'
'Hfss/NumCoresPerDistributedTask'=2"
\\dennis\projects\project1.aedt
```

Example -Batchsolve with -Machinelist (Linux)

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

In this example, we run a batch HFSS analysis of project file project2.aedt. We have four identical host computers—host1, host2, host3, and host4 for analysis, and each host has 4 cores. We do not use multiprocessing for the distributed analysis, so NumCoresPerDistributedTask=1. As each host has four cores, we specify multiprocessing using

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4 threads for the non-distributed part of the analysis, so NumberOfProcessors=4. Because we do not use multiprocessing for the distributed analysis, we will run four distributed COM engines on each host, with a singe core available for each engine. We will specify a RAM limit of 75% for this analysis.

Here is a sample command line for this analysis, where the project file /home/jsmith/projects/project2.aedt is located in a shared directory:

```
hfss -BatchSolve -Distributed
-Machinelist file=/home/jsmith/hosts/list2
-batchoptions "HFSS/'RAMLimitPercent'=75
'Desktop/Setings/ProjectOptions/NumberOfProcessors'=4
'HFSS/NumCoresPerDistributedTask'=1"
/home/jsmith/projects/project2.aedt
```

For this example, the hostnames are in the text file /home/jsmith/hosts/list2. The file contents are as follows:

```
host1
host1
host1
host1
host2
host2
host2
host2
host3
host3
host3
host3
host4
host4
host4
host4
```

Example -Batchsolve for Local (Windows)

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this example, we run a batch analysis of project file testproject.adsn on the local host. We want all temporary files and directories created in directory C:\temp\ansysedt instead of using the installation default temporary directory. Because the local host has four cores, we decide to use four threads for multiprocessing, for both distributed and non-distributed parts of the analysis.

Here is a sample command line for this analysis, where the project file \\host123\projects\testproject.adsn is located in a shared directory specified using a UNC path:

```
ansysedt -BatchSolve -Local -batchoptions
"TempDirectory='C:/temp/ansysedt'
'Planar EM/SolverOptions/NumProcessors'=4
'Planar EM/SolverOptions/NumProcessorsDistrib'=4"
\\host123\projects\testproject.adsn
```

Note that the batchoptions pathnames 'Planar EM/SolverOptions/NumProcessors' and 'Planar EM/SolverOptions/NumProcessorsDistrib' must be in single quotes because they both contain embedded spaces.

Batch Options and Analysis Configurations in the Registry

Analysis configurations are used to specify machines and options for local, remote, and distributed analysis, including capabilities that are enabled by HPC licenses.

How Analysis Configurations are Stored in the Registry

A configuration contains information beyond the machine or machines to use for a solution. Examples are the number of processors to allocate to the analysis for each machine in the list, memory limits, directory locations for personal libraries and temporary files, and many other preferences.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- Options are arranged as keys and values (in a structure similar to the *Windows Registry*). However, these options are not a part of the *Windows Registry* but are separately stored and maintained by the Ansys Electronics Desktop software. For more information concerning the configuration files comprisisng the options registry, see the following topics:

  Setting Options Via Configuration Files
  Setting the Temporary Directory

- For access to options and functionality beyond what is directly accessible via the user interface or batch options, refer to the documentation of the **UpdateRegistry** tool. This tool is discussed in the following help topic and in the topics that follow it in the same branch of the product help:

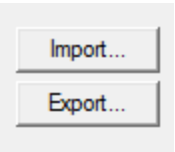
  Setting or Removing Option Values in Configuration Files: UpdateRegistry Command

General settings are associated with the Desktop application or 3D Editor. "HPCLicenseType" and "tempdirectory" values are at the root level of the registry. Other options are specific to a particular product or design type. For example, certain mesh, boundary, and memory limit settings as well as many other preferences are product-specific and are therefore associated only with the applicable design types. Such options appear with a consistent value name but in a different registry path for each applicable design type.

Copying a Configuration from one Design Type or Product to Another

To copy a configuration from one design type (or product) to another:

1. Click **Tools** > **Options** > **HPC and Analysis Options**. The *HPC and Analysis Options* dialog box appears.
2. On the **Configurations** tab of the **HPC and Analysis Options** dialog box, use the **Export…** button to export the configuration to a file.



3. Switch to the destination design type (or product) and use the **Import…** button to import the configuration data.

   Any data that is not applicable to the destination design type is ignored; any settings present in the destination design type that were not present in the source configuration will be assigned default values. The user may then edit the copy, as desired.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using HPC and Analysis for Configurations

Due to the complexity of the registry values for the configurations we do not recommend directly editing these values using the *UpdateRegistry* tool. Instead, use the **HPC and Analysis Options** dialog box to edit or create a configuration. (See: Setting HPC and Analysis Options.) Configurations created or edited using the GUI are stored in the **user_machine** level of the registry. You can create a configuration for one of the other registry levels using the following steps:

1. Create the configuration using the Analysis Configurations GUI, then export the configuration to a file.
2. Delete the configuration using the GUI so that it will not be present in the **user_machine** level. Then, exit the GUI.
3. Use the **UpdateRegistry** tool to import the data into the desired registry level using the **-FromFile** option to specify the file exported via the GUI, and using the **-RegistryLevel** option to specify the registry level where the configuration is to be stored. For example, an administrator may use this approach to create a configuration at the **install** level that may be used by any user on any machine.

Batch Options

There is a large number of both general and product-specific options supported by the software. These options have evolved over time. Therefore, older batchoptions files may no longer be valid, and the options listed in the user interface of the current software may differ from earlier versions you have used. Additionally, there are options beyond those listed within the user interface (that is, the UI provides a subset of the available options). You can generate a more complete list of options by running the UpdateRegistry tool with the -GetKeys switch and piping the output to a text file, as detailed in the following Windows procedure:

1. In a command window, navigate to the following folder:

   "<*installation_directory***>\ANSYS Inc\v251\AnsysEM"**

2. Type and enter the following command:

   UpdateRegistry -GetKeys -ProductName ElectronicsDesktop20*xx.y* **>** <*text_file_path*>**\Batchoptions.txt**

   Substitute the last two digits of the installed product version year and the minor release number for *xx.y* (such as **25.1**). Also, substitute the desired *text_file_path* (such as **%UserProfile%\Desktop**).

   The resulting **Batchoptions.txt** file will contain a nearly complete list of available options.

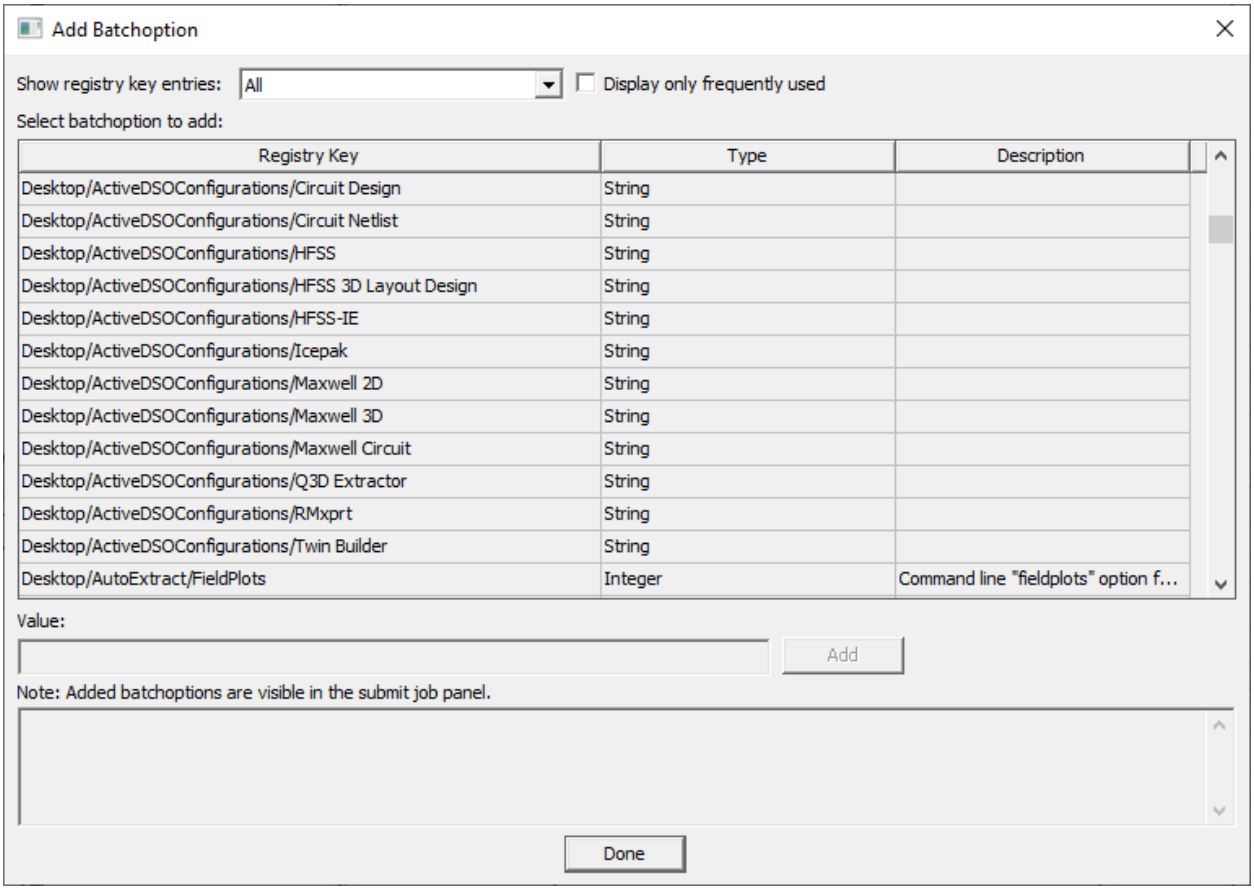
This procedure also works from the Linux <*installation_directory*>, which has the same downstream path (for example, under "opt/") as a Windows installation (typically under "C:\Program Files\").

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

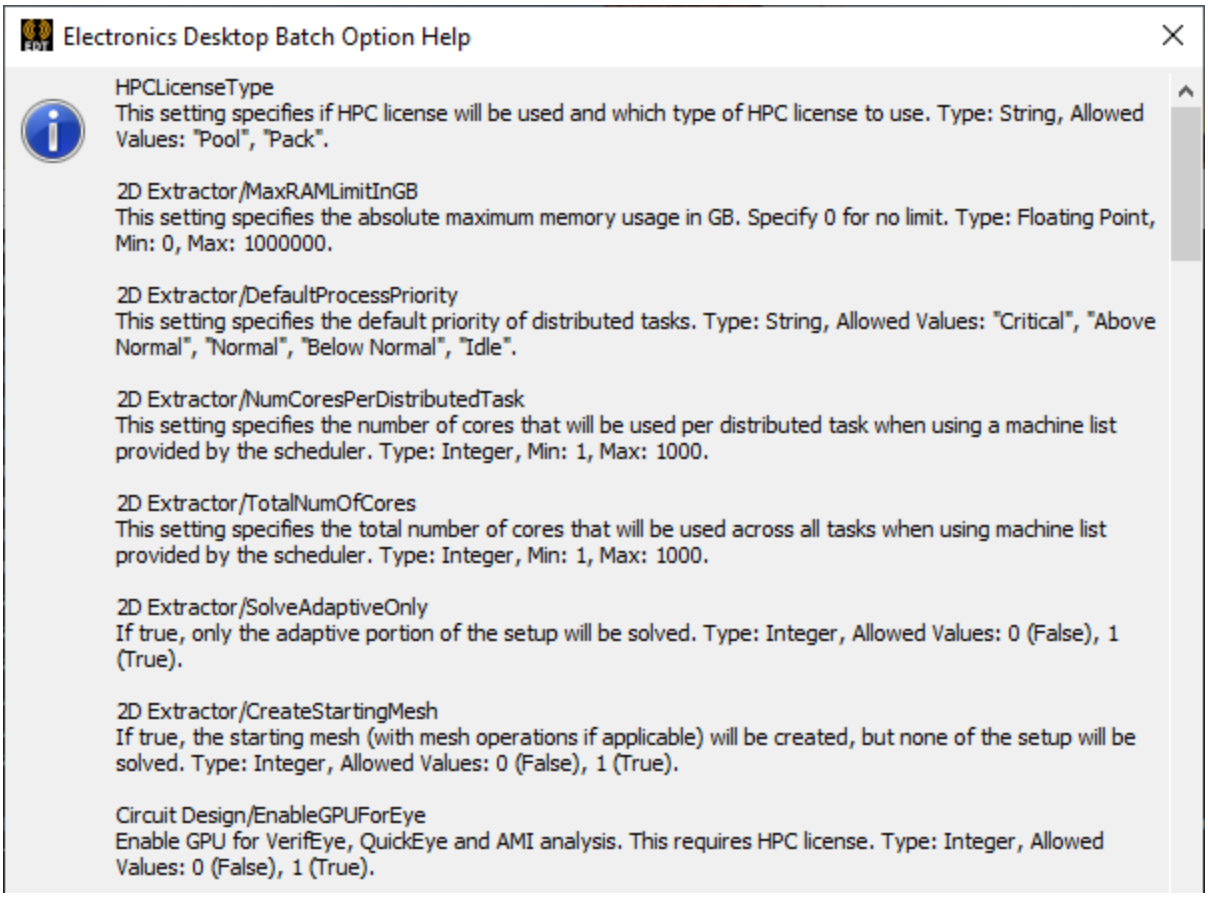
For special product-specific options that are neither available from the GUI nor listed by the UpdateRegistry tool, see Special Batch Options. Even though the -GetKeys switch does not list these special options, you can still use the UpdateRegistry tool to set them.

When you submit jobs to a remote computer or cluster, you can specify batch options using the job submission GUI. When using the GUI, you select the batch options from a list and therefore avoid typographical errors. For the most commonly used batch options, there is detailed information about the allowed values. Click **Submit** on the **Simulation** ribbon tab to access the *Submit Job To* dialog box. Then, click **Add** to access the **Add Batchoption** dialog box pictured below:



To assist users who need to specify batch options and are unable to use the job submission GUI, a new help option has been added. If the Electronics Desktop application is launched with the **-batchoptionhelp** command line argument, a message box is displayed which lists and describes the most common design type-specific batch options:

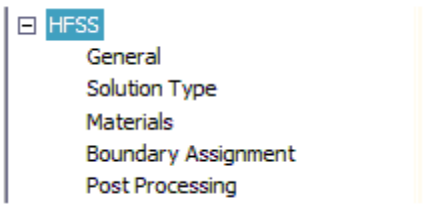
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Setting HFSS Options

To set HFSS options:

1. Click **Tools>Options> General Options** to display the **Options** dialog box. Select the HFSS entry in the hierarchical window, and press the + sign to show the choices under HFSS.



2. Under **General** options, select or clear the following check boxes:
   - Save before solving
   - Apply variation deletions immediately.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Checking this saves disk space.

- Save Optimetrics Field solutions.

Checking this uses more disk space.

3. Under Solution Type, you can set the default solution type when you initially insert a project. Select one of the following from the Default solution type drop-down menu:

- Eigenmode
- Driven Modal
- Driven Terminal
- Transient
- Transient Network
- Driven Modal (Composite Excitation)
- Driven Terminal (Composite Excitation)

4. In the **Material Options** section:

- Check or uncheck whether to **Include ferrite materials**
- Set the **Solve Inside threshold** values in Siemens/m.

5. In the **Boundary Assignment section**, select or clear the following check boxes and settings.

- Use Wizards for data input when creating new boundaries

When this is checked, the creation of boundaries and excitations use Wizard to guide you through the process. When this is not checked, the creation of boundaries and excitations displays a Properties dialog with tabs for different kinds of information.

- Duplicate boundaries/mesh operations with geometry

When this is checked, you can duplicate a boundary or excitation when its geometry is pasted or duplicated. See Duplicating Boundaries and Excitations with Geometry.

- Visualize Boundaries on geometry.

When this is checked, boundaries on geometries are displayed. Unchecking this turns off boundary visualization, and speeds up the display for complex models.

- Auto assign terminals on ports

When this is checked, the commands to assign wave or lumped ports will automatically assign terminals. See Assigning Wave Ports for Terminal Solutions.

- Automatic PEC creation on wave port. See Assigning Wave Ports for Modal Solutions and Assigning Wave Ports for Terminal Solutions.

In HFSS, A wave port must have 'solve inside' objects on one side only. An internal wave port would require a PEC object to be attached to and cover the opposite side before the solution

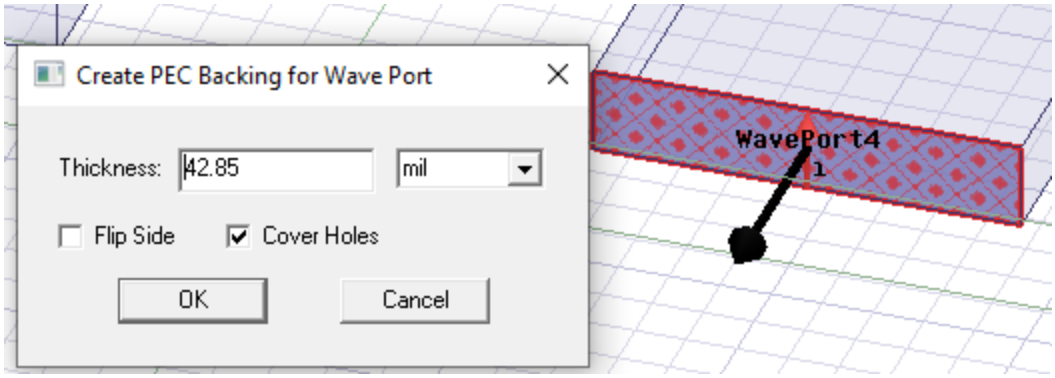
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

can proceed. This feature automatically adds a PEC backing to every newly created wave port when necessary.

A dialog for creating wave port PEC backing will show up if the current wave port assignment is not valid. The general rule is that the dialog for creating wave port PEC backing appears if the wave port is internal to the design and does not touch any non-solve-inside component. If a project has open region defined, all geometries are considered internal. A wave port is not valid if these conditions are true:

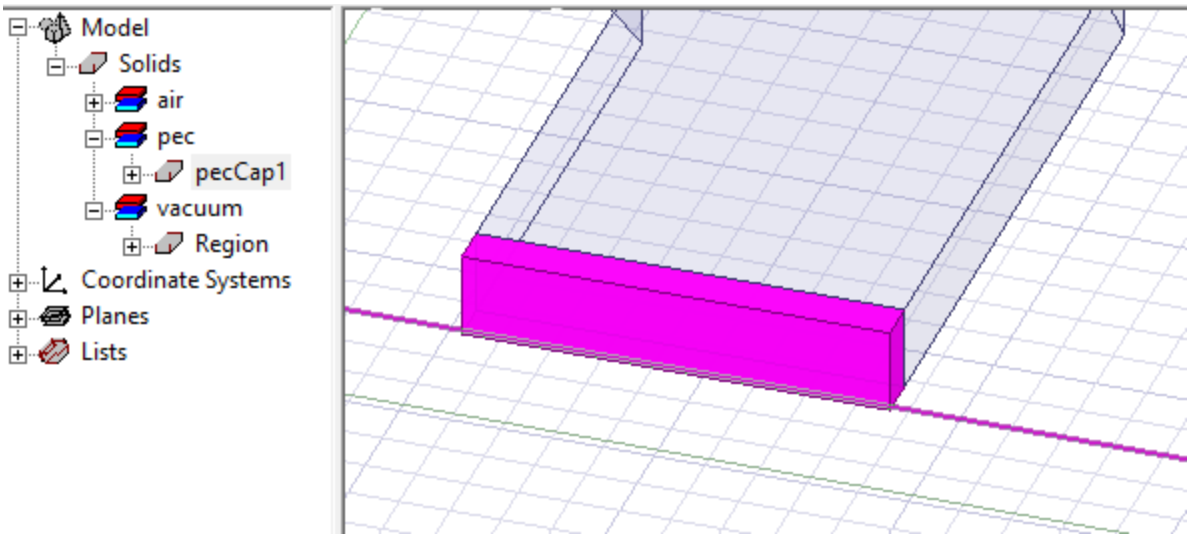
- the wave port is contained by objects that have maximum volume or FEBI Hybrid region or Radiation boundary (also called internal to the design).
- if a project has open region defined, all geometries are considered internal.
- the wave port does not have a non-solve-inside backing (in this case, does not touch any non-solve-inside object).

When a new wave port is created and requires a PEC backing, a dialog window opens to let users create such backing by extruding the port face with specified thickness. You have the option to flip the side of the PEC backing. You can also set the thickness as a variable.



After clicking OK, the PEC backing is created and the new PEC object appears in the History tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



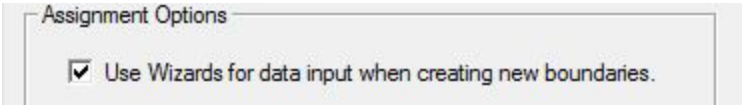
6. In the **Post Processing Options** section:

- Set the default Matrix sort order. This affects the order of the Matrix Data, and is of interest depending on how port names are assigned for that design. The default is ascending alphanumeric. This can also be a User Specified order that defaults to creation order.

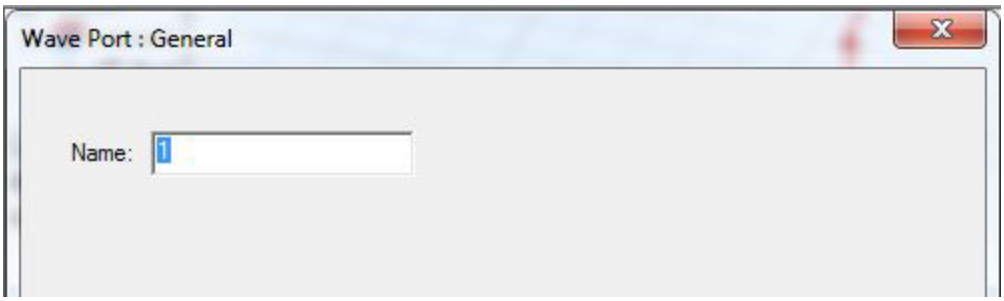
Wave Port Wizard or Multi-tab Dialog Menu

When assigning a wave port HFSS displays either a wave port wizard or a multi-tab dialog menu depending upon the settings on the **Boundary Assignment levels** of the **HFSS** selections in the **Options** dialog box. Consider the following cases:

Case 1: The option **Use Wizards for data input when creating new boundaries** is checked.

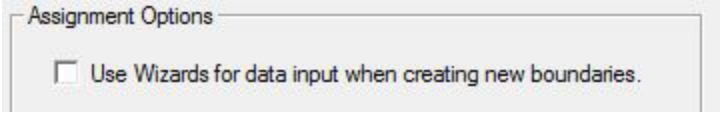


When assigning an excitation, the **Wave Port: General** wizard will appear as shown below.

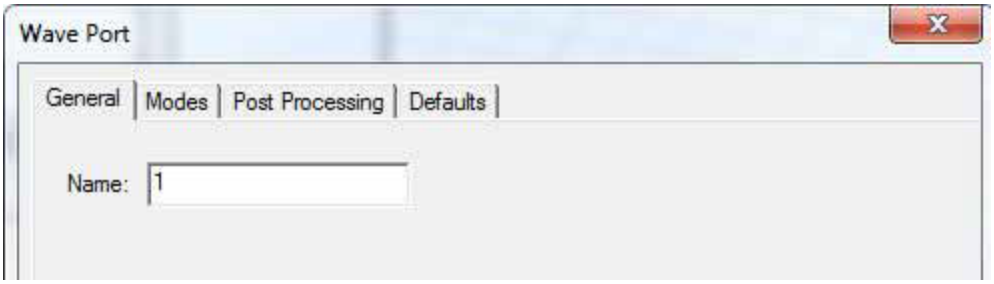


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Case 2: The option **Use Wizards for data input when creating new boundaries** is unchecked.



When assigning the excitation the multi-tab **Wave Port** dialog box appears with the **General** tab selected as shown in the figure below.



Type the port's name in the **Name** text box or accept the default name.

> **Note:**
>
> To change the default base name, see Setting Default Boundary/Excitation Base Names.

Managing Projects and Designs

An Ansys Electronics Desktop *project* is essentially a folder that includes one or more models, or *designs*.

Some basic tasks that can be performed on projects and designs include:

- Opening Projects
- Creating Projects
- Saving Projects
- Importing and Exporting Projects and Data
- Setting Project Options
- Validating Projects
- Copying and Pasting Projects and Designs
- Renaming Projects
- Deleting Projects or Designs

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

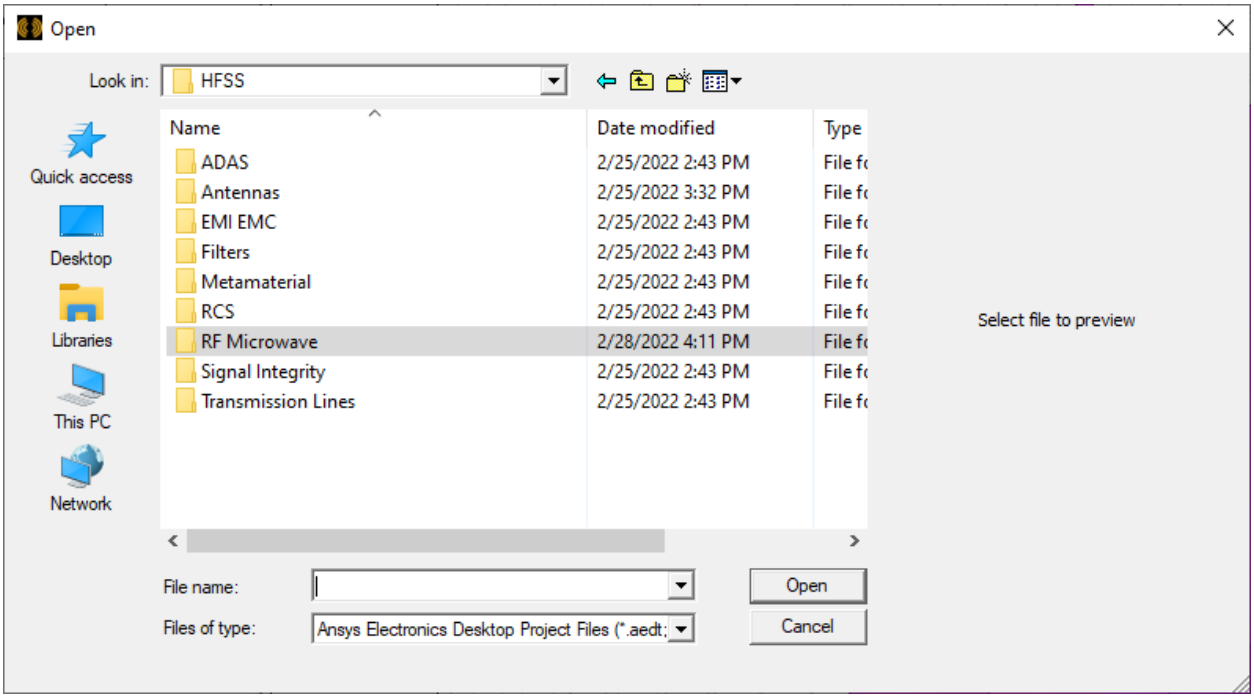
- Setting Read Only Designs
- Updating Design Components
- Undoing and Redoing Commands
- Inserting a Documentation File
- Saving Project Notes
- Printing

Opening Projects

Open a previously saved project using the **File** > **Open** command.

1. Click **File** > **Open** or click the **Open** icon on the **Desktop** tab of the ribbon.

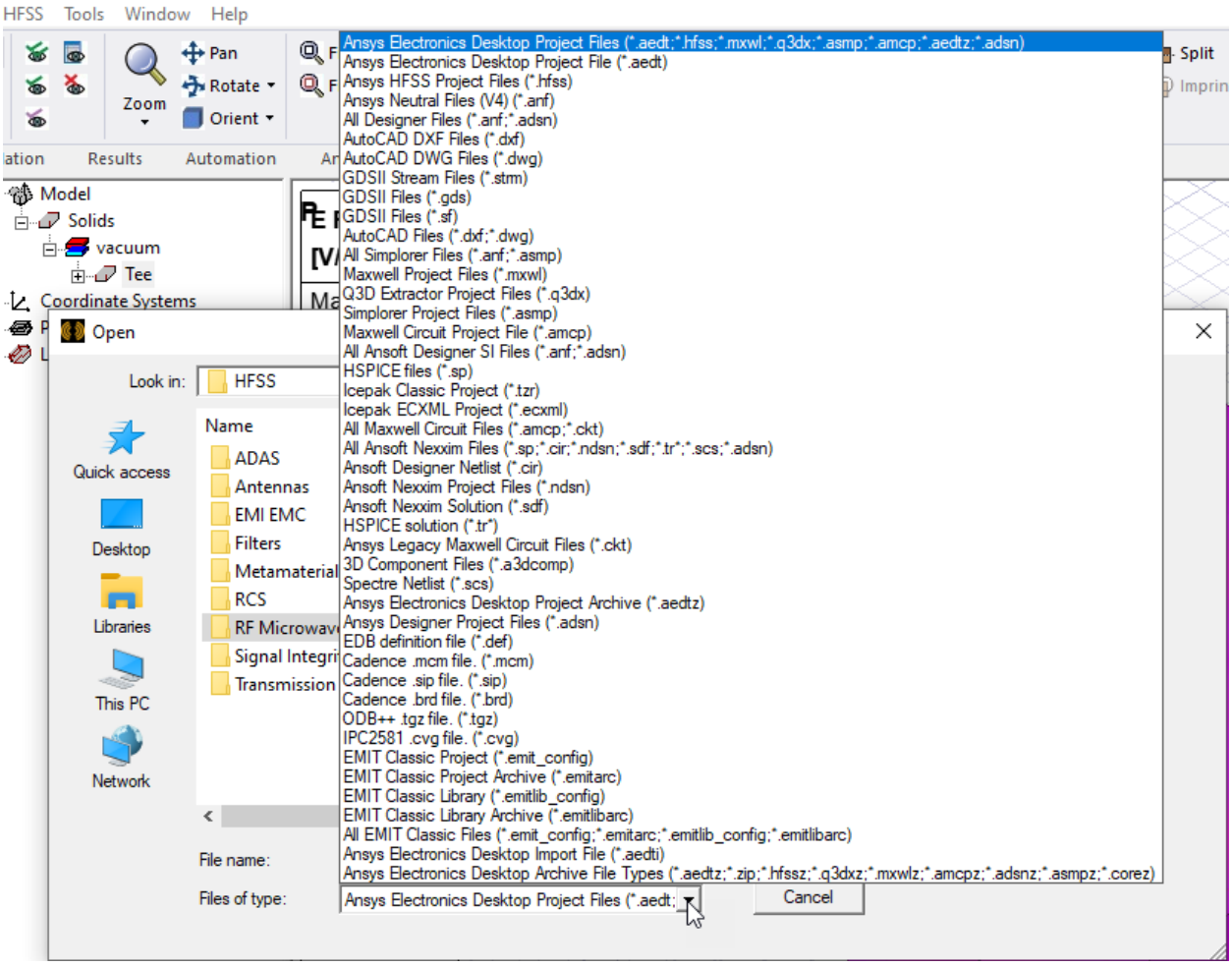
2. Use the file browser to find the project file.



By default, only files that can be opened or translated display. *.aedt is the main default, although, you can also view several types of legacy files. The dropdown for Files of type

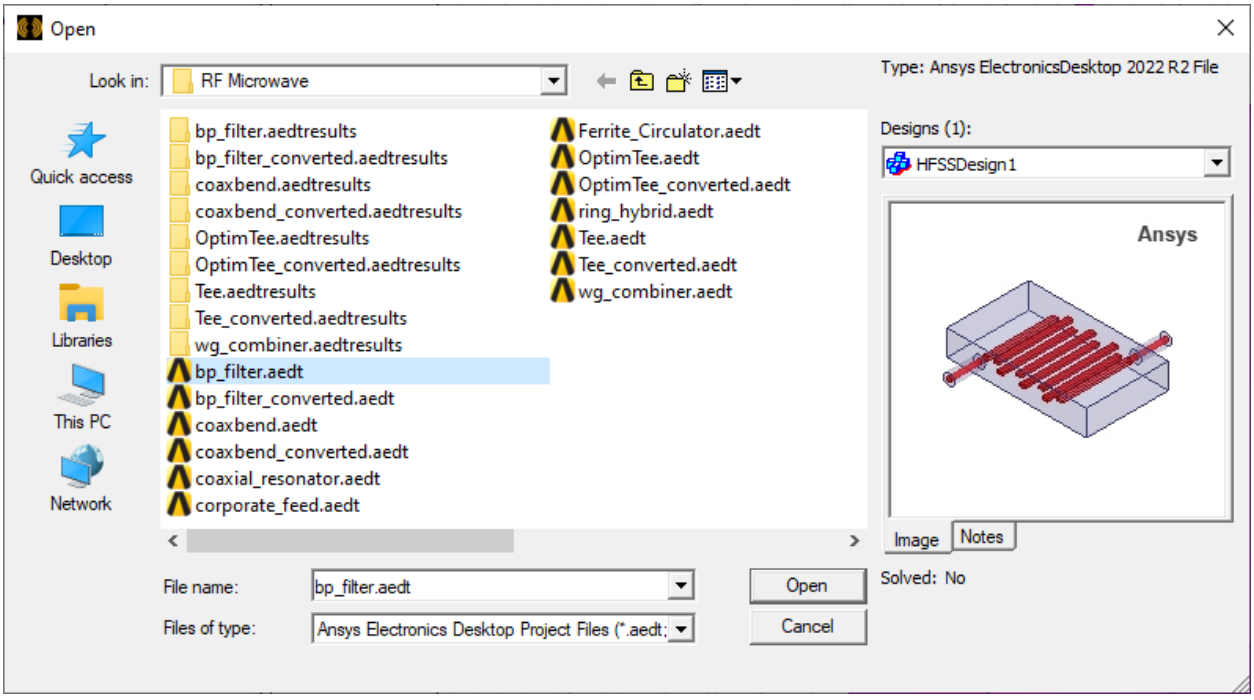
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

lists the kinds file and archive formats for which you can select filters.



3. Most of these will be familiar to those seeking them. The Ansys Electronics Desktop Import file (*.aedti) format is for importing exported projects from ANSYS Discovery, an ANSYS Workbench tool. This format allows import of side by side *.aedti and the correponding geometry definition *.sat into the Ansys Electronics Desktop, and creates a ready-to-solve HFSS project.
4. Select the file you want to open.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Click **OK**.

   The project information appears in the Project tree.

If you open another project without editing the automatically created project, the automatically created project is removed.

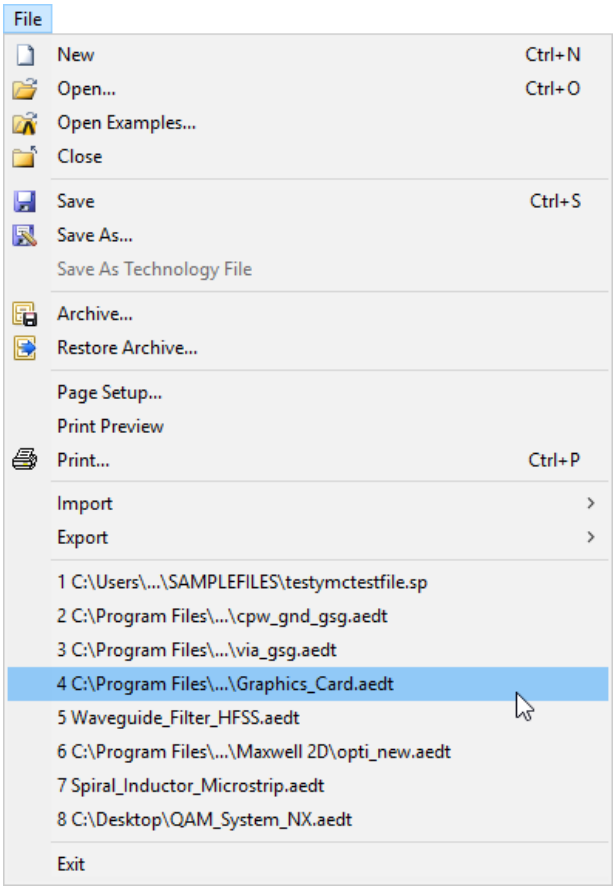
You can also open a saved project by:

- Dragging a project file icon to the Open icon
- Dragging a project file icon to the desktop
- Double-clicking a project file icon

Opening Recent Projects

Recently opened projects appear at the bottom of the **File** menu. To open one of these files, click its file name.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



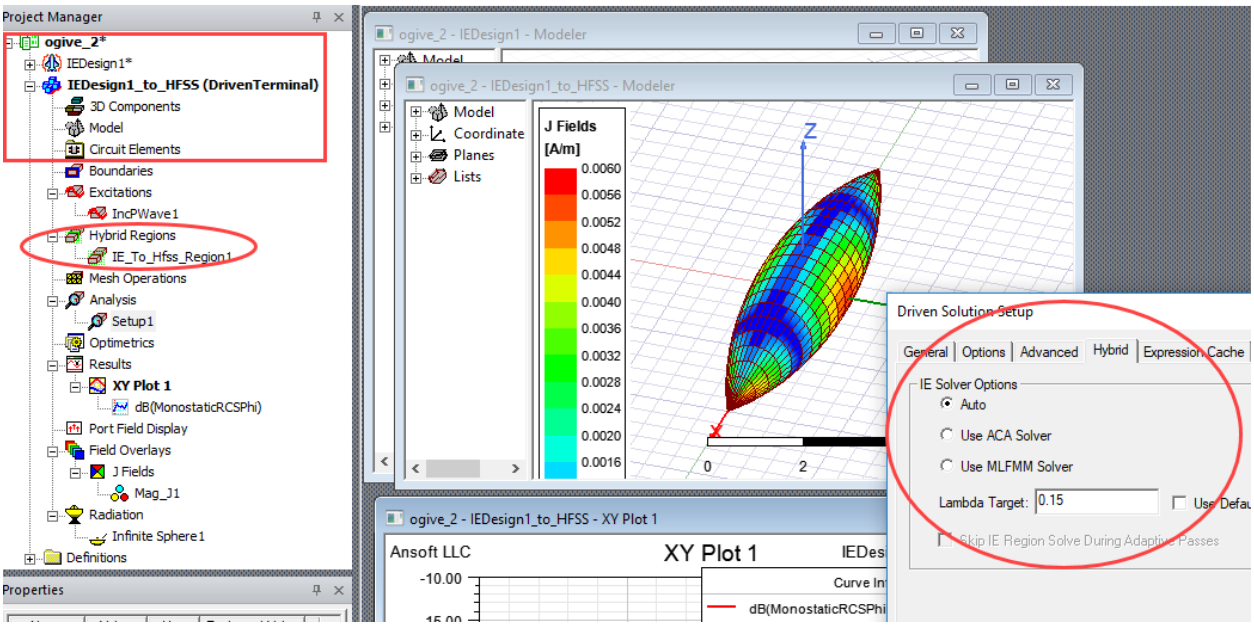
Opening Example Projects

You can directly access and open example projects included with product installation using **File** > **Open Examples**, or by clicking **Open Examples** on the **Desktop** tab of the ribbon. See Example Projects for additional description of these files.

Opening Legacy HFSS-IE Projects

If you open an existing HFSS-IE project in Ansys Electronics Desktop, it is automatically converted to an equivalent HFSS project with IE Regions assigned.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The primary changes in the converted project are that an IE Region is assigned to the objects, and that the **Hybrid** tab of the Setup has the IE Solver Options. All Excitations, Plots, Field Overlays, and Radiation are all equivalent to those in the original project. If you select the original project, the HFSS-IE menus will still appear for that project. However, going forward, HFSS-IE will not be supported. IE Regions should be used.

HFSS-IE included a Physical Optics (PO) solver for use with metallic designs excited by an incident plane wave, a far field wave, or a near field wave. The PO functionality can be achieved using PO Regions. This solver provides first-order scattering information, such as an approximate RCS. If that result is not accurate enough (for example, for a back lobe calculation), you can use the regular IE solver. The types of reports are the same; it is the accuracy that changes. When selected, it solves for only one pass, performing no adaptive refinement.

Opening Legacy Projects

HFSS cannot open projects created in Ansoft HFSS version 8.5 or earlier. HFSS 10 files can be opened, but saving them in the current format renders them unable to be reopened in version 10.

To open a legacy project:

1. Click **File** > **Open**.
2. From the **Look in** drop-down menu, select the location of the project. In the folder list, double-click folders to find the one that contains the project.
3. Double-click the project you want to open.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Legacy Project Translation

When you open a legacy project, virtually all of the project's pre-processing data is translated. Solution results and Optimetrics setup data are unavailable; however, the nominal model created for Optimetrics is translated.

The following table contains additional notes about the translation of legacy project information.

Model Geometry	• The translated geometry's construction history is unavailable; therefore, the original object properties you defined cannot be modified in the **Properties** window. • For units unavailable in the older version, such as yards, the nearest available units are used; the model will be scaled slightly to fit the new units. • View visualization settings apply to the saved design. If these have been changed from the default (15 deg), this affects the memory and CPU required to open the project.
Excitations and Boundaries	• If the legacy project contained both port impedance and calibration lines, the impedance lines are translated to integration lines, and calibration lines are ignored. If the project contained both impedance and terminal lines, both are translated to integration lines. The impedance lines are ignored for Driven Terminal solutions and terminal lines are ignored if the project is changed to a Driven Modal solution. • Boundaries assigned to named interface selections or rectangle selections are not translated. • For a boundary assigned to the intersection of two faces, a new 2D sheet object is created from the intersecting area and assign the boundary to that object.
Hybrid Regions	• Radiation Boundaries with FEBI become Hybrid Regions assigned as FEBI.
Materials	• Functions defined in legacy projects become project variables; therefore, functional material properties are translated. • Perfect conductors become regular materials with conductivity values of 1E30. • Object coordinate systems are created for objects assigned anisotropic materials in legacy projects. The coordinate system is defined at the same origin as the global coordinate system with the same orientation defined when the anisotropic material was assigned to the object in the legacy project. • Nonlinear materials from legacy projects that have magnetic saturation values greater than zero are treated as ferrite materials. Their properties

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	are not modified.
Mesh Operations	• Mesh refinement operations performed on arbitrary boxes in legacy projects are ignored. • Area and volume-based mesh operations are translated as length-based mesh operations by taking their square roots and cube roots, respectively.
Optimetrics	• Setup information, including design variables, is not supported; however, the nominal model can be translated. • Parameterizing a translated model is limited because geometry construction history is unavailable.
Solution Types	• Driven solver projects that contained terminal lines are translated to Driven Terminal solution types.
Solution Setup	• Impedance-only and emissions-only solutions are not supported; therefore, these selections in legacy projects are ignored. • The design's initial mesh is used for the solution. Current meshes are not translated. • Saving dominant-only or higher-order-only mode S-matrix entries are not supported; therefore, these mode selections in legacy projects are ignored. • For frequency sweeps, the **Number of Steps** value specified in the legacy project is converted to the corresponding **Step Size** value. • The total number of requested adaptive passes in the legacy project becomes the **Maximum Number of Passes** value. For example, if you request 3 adaptive passes, solve them, and then request 2 adaptive passes, 5 will be the value specified for the **Maximum Number of Passes**.
Solutions	• Solution data is not translated; therefore, you must solve legacy projects again.

Note:

Legacy Circuit (Serenade) projects (**.ssp** extension) do not open correctly from the **File Open** dialog box. Contact Ansys Technical Support for assistance in converting legacy projects. Opening an Ansys Neutral File project (**.anf** suffix) begins with a conversion dialog box. See Importing ANF Design Data. The Planar EM simulator can use ANF data.

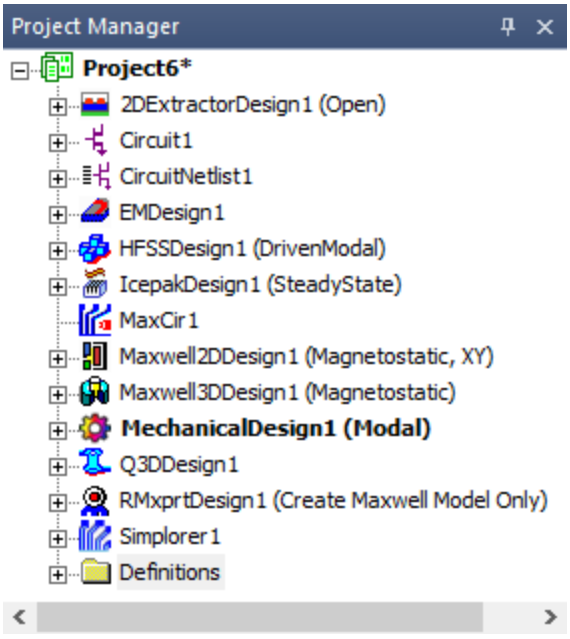
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating Projects

To create a project, click **File** > **New**, or click the **New** icon on the **Desktop** tab of the ribbon.

A new project is listed in the project tree. It is named Project*n* by default, where *n* is the order in which the project was added to the current project folder.

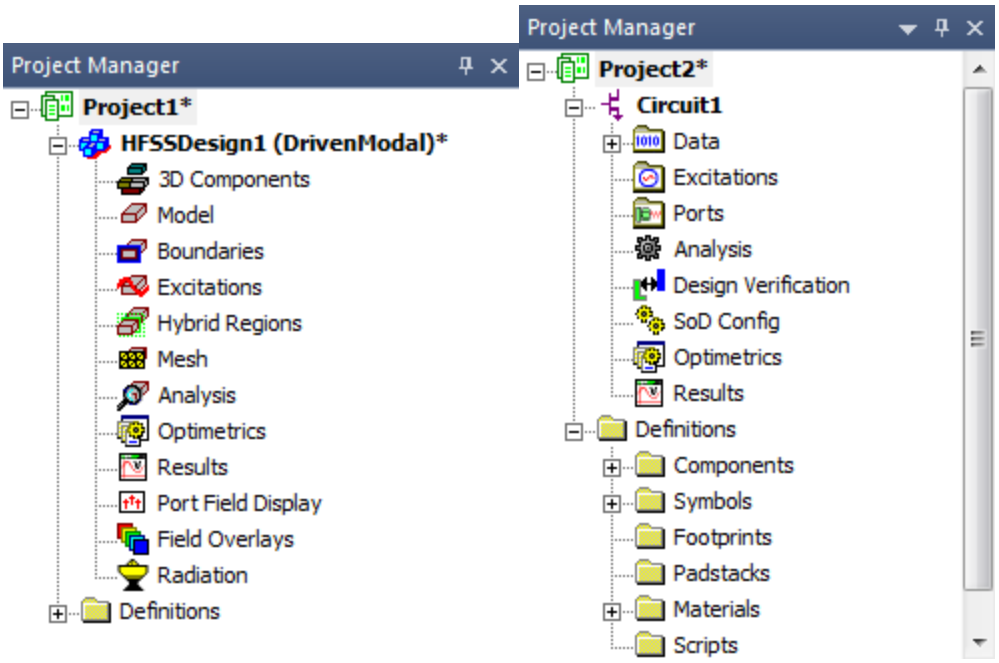
You can insert designs of any type into the project, where they are stored in the project tree.



The default name for each inserted project is <designType>*n*. You can also specify the name of the project when you save it using the **File** > **Save** or **File** > **Save As** commands.

You can view the contents of a project by clicking the plus sign (+) for each level of the hierarchy in the project or design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Creating a Target Design from 3D Designs

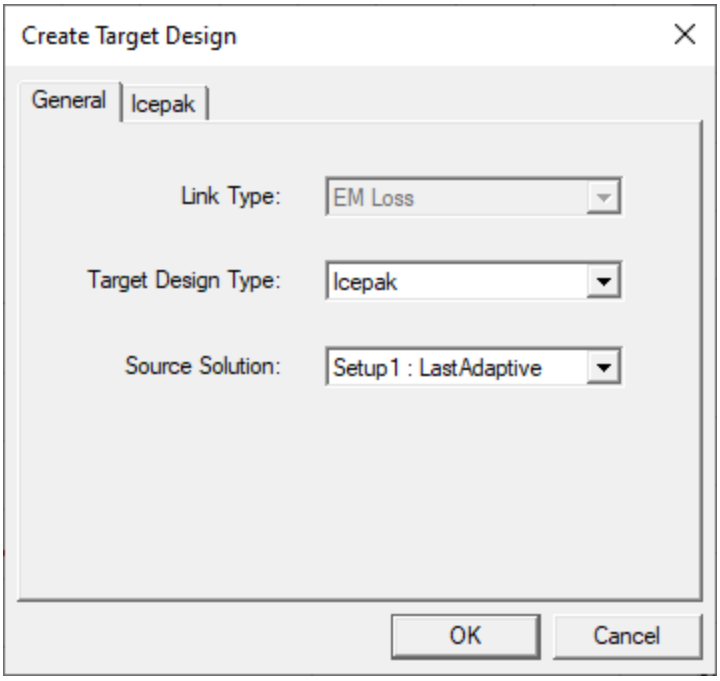
In an HFSS, Q3D, or Maxwell 3D design, **Create Target Design** automatically generates and inserts a thermal design (Icepak or Mechanical Thermal) into your project.

To create a target design from an existing **HFSS** design:

1. Click **HFSS** > **Create Target Design**.

   The **Create Target Design** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select the **Target Design Type** to be created.
3. Select the **Source Solution** to specify the solution data to be linked to the target design.
4. For Icepak designs, specify **Forced Convection** or **Natural Convection** on the **Icepak** tab.
    - For **Forced Convection**, specify **Flow Speed** and **Flow Direction**.
    - For **Natural Convection**, specify the **Gravity Vector Direction**.
5. Click **OK**.

Icepak Designs

The Icepak design is generated based on the source design and selections in the **Create Target Design** dialog box:

- Model geometry is created with corresponding material assignments. Vacuum objects are not created in target Icepak designs.
- Any assigned materials that do not contain thermal properties are assigned default thermal properties in Icepak.
- The Region (computational domain) is created with the appropriate dimensions.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- An EM loss excitation is assigned to the geometry, and variable mapping and intrinsic variables are linked to the source design.
- Design settings are specified.
    - Solution type: **Steady State**
    - Problem type: **Temperature and Flow**
- For forced convection simulations, the following are defined:
    - A free opening boundary condition with a velocity specified is assigned to the Region face to create forced flow.
    - A free opening boundary condition is assigned to the face of the Region opposite to the opening with velocity specified.
    - In the **Icepak Solve Setup** dialog box, forced convection **Solve Setup Defaults** are defined.
- For natural convection simulations, the following are defined:
    - **Gravity Vector**
    - Free opening boundary conditions are assigned to each face of the air Region.
    - In the **Icepak Solve Setup** dialog box, natural convection **Solve Setup Defaults** are defined.

Mechanical Designs

The Mechanical Thermal design is generated based on the source design:

- Model geometry is created with corresponding material assignments. For any vacuum objects, **Solve Inside** is disabled.
- Any assigned materials that do not contain thermal properties are assigned default thermal properties in Mechanical.
- A convection boundary is assigned to the faces of all 3D objects (not 2D sheet objects). **A Film Coefficient** is specified as **Uniform** at 10 w_per_m2kel. Objects with **Solve Inside** disabled and those with air assigned as the material are not assigned to the convection boundary.
- An EM loss excitation is assigned to the geometry, and variable mapping and intrinsic variables are linked to the source design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Saving Projects

Use the **File** > **Save As** command or select the **Desktop** tab of the Ribbon and click the **Save As** icon to:

- Save a new project.
- Save the active project with a different name or in a different location.
- Save the active project in another file format for use in another program.

Use the **File** > **Save** command or select the **Desktop** tab of the Ribbon and click the **Save** icon to save the active project.

> **Warning:**
>
> Be sure to save models periodically. Saving frequently helps prevent the loss of work if a problem occurs.
>
> Although Ansys Electronics Desktop has an auto-save feature, it may not save frequently enough for your needs.

Each solver has a Save Before Solving setting located in the **Tools** > **Options** window. This setting is enabled by default.

A prompt appears when you attempt to save a previously versioned file. If you agree to the prompt, the file is upgraded to the Ansys Electronics Desktop version in which you are running the software. In this case the file may no longer be compatible with previous versions. If you do not agree to the prompt, the file is not saved, so the file retains its previous compatibility.

If you have a simulation running, you see a warning that if you continue, Ansys Electronics Desktop will abort the simulation. If you OK the warning, Ansys Electronics Desktop aborts the simulation and saves the project.

Path Name Length Issues for Windows

For most Windows programs, the current directory pathname length is limited to 259 characters on startup. Thus, the pathname length of the directory containing installed programs should be limited to no more than 259 characters because double-clicking on an application in Windows

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Explorer will set the working directory to the directory containing the application when the application is started.

Win32 Long Paths Not Enabled

If win32 long paths are not enabled on the analysis host, then essentially all files are limited to a maximum absolute pathname length of 259 characters. Directories may be limited to a maximum absolute pathname length of about 246 characters.

Win32 Long Paths Are Enabled

If win32 long paths are enabled on the analysis host, then many files are not subject to the maximum absolute pathname length limit of 259 characters. Below is a partial list of files that are still subject to this limit even if win32 Long Paths are enabled:

- Project files
- Project Archive files
- Ansys EDB (Electronics Database) files, typically stored in the `ProjectName.aedb` folder
- The temporary directory and most temporary files

  Because some temporary files and directories have automatically generated names that are long, it is best to use a short pathname for the temporary directory.

  > **Important:**
  >
  > Although most temporary files are created within the temporary directory, there may be some temporary files within the project directory. As a result, the project file directory pathname should be well below the 259 character limit.

To enable Long Path support on Windows, both of the following requirements must be met:

- Long paths must be enabled on the machine

  Use either of two ways to do this:

  - **Via Registry Setting:** `HKLM\SYSTEM\CurrentControlSet\Control\FileSystem : LongPathsEnabled=1`
  - **Via Group Policy Tool:** The “Enable Win32 long paths” setting is in the folder: `Computer Configuration\Administrative Templates\System\Filesystem`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Changing Auto-Save Settings

Recent actions performed on the active project are stored in an auto-save file in case of a sudden workstation crash or other unexpected problem. The auto-save file is stored in the same directory as the project file and is named Project*n*.aedt.auto by default. Ansys Electronics Desktop automatically saves all data for the project to the auto-save file, except solution data.

By default, Ansys Electronics Desktop automatically saves project data after every 10 edits. An edit is any action performed that changes data in the project or the design, including actions associated with project management, model creation, and solution analysis.

After a problem occurs, you may be able to choose to re-open the original project file in an effort to recover the solution data, or to open the auto-save file. If the original file is not available, attempting to open the file provides a message that the auto-save is being used. If neither file is available, an error message displays.

To modify auto-save settings:

1. Click **Tools** > **Options** > **General Options**.

   The **Options** window appears.

2. Under **Desktop Configuration**, verify that **Do Autosave** is selected.

   This option is selected by default.

3. In the **Autosave interval** box, enter the number of *edits* that you want to occur between automatic saves. By default, this option is set to 10.

   > **Note:**
   >
   > Auto-save *always* increments forward; therefore, even when you undo a command, Ansys Electronics Desktop counts it as an edit.

4. Click **OK** to apply the specified auto-save settings.

   Once the specified number of edits is carried out, a "model-only" save will occur. This means that Ansys Electronics Desktop does not save solutions data or clear any undo/redo history. When Ansys Electronics Desktop auto-saves, an ".auto" extension is appended to the original project file name. For example, Project1.aedt will automatically be saved as Project1.aedt.auto.

> **Warning:**
>
> When you close or rename a project, Ansys Electronics Desktop deletes the auto-save file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Save Before Solving Option

The **Save before solving** option forces a full save before running the solve and is enabled by default. For efficiency reasons, the project is saved only if it has been modified since its last save.

To change this setting, click **Tools** > **Options** to open the Options window. Select the **General** tab and use the **Save before solving** check box to toggle the option.

You can start a new solve while running another without having to abort the running solve. If you start a solve while another solve is running and the Save before solving option is set, Ansys Electronics Desktop asks if you want solve without saving first. This lets you perform multiple solves, and if you have not edited the project in between solves, crash recovery will work.

Recovering Project Data from an Auto-Save File

Following a sudden workstation crash or other unexpected problem, you can recover project data from its auto-save file.

> **Warning:**
>
> If you choose to recover the auto-save file, you cannot recover the original project file that has been overwritten; recovering data in an auto-save file is *not* reversible.

Importing and Exporting Projects and Data

Ansys Electronics Desktop can import and export a variety of file types.

Importing Files

You can import the following types of files to HFSS:

- 2D model (GDSII) files
- 3D model files
- Solution data files
- Data table files
- HFSS Plot Data
- Plot Data (comma delimited files)

Object, material, and parameter names with non-ASCII characters are not allowed, and therefore, not allowed for data transfer. Such transfers fail and produce an error message.

For certain file types, the import dialog contains **Validation and Healing Options**, which are enabled by default.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These types are:

- ACIS SAB files (*.sab)
- ACIS SAT files (*.sat)
- Ansys 3D Modeler files (*.sm3)
- Autodesk Inventor files (*.ipt, *.iam)
- CATIA V4/V5 files (*.exp, *.model, *.CATpart, *.CATproduct)
- Creo Parametric files (*.prt, *.asm)
- IGES files (*.iges, *.igs)
- JT files (*.jt)
- Nastran files (*,nas)
- NX files (*.prt)
- Parasolid files (*.x_t, and *.x_b)
- SOLIDWORKS Files (*.SLDPRT, *.SLDASM)
- STEP files (*.step, *.stp)

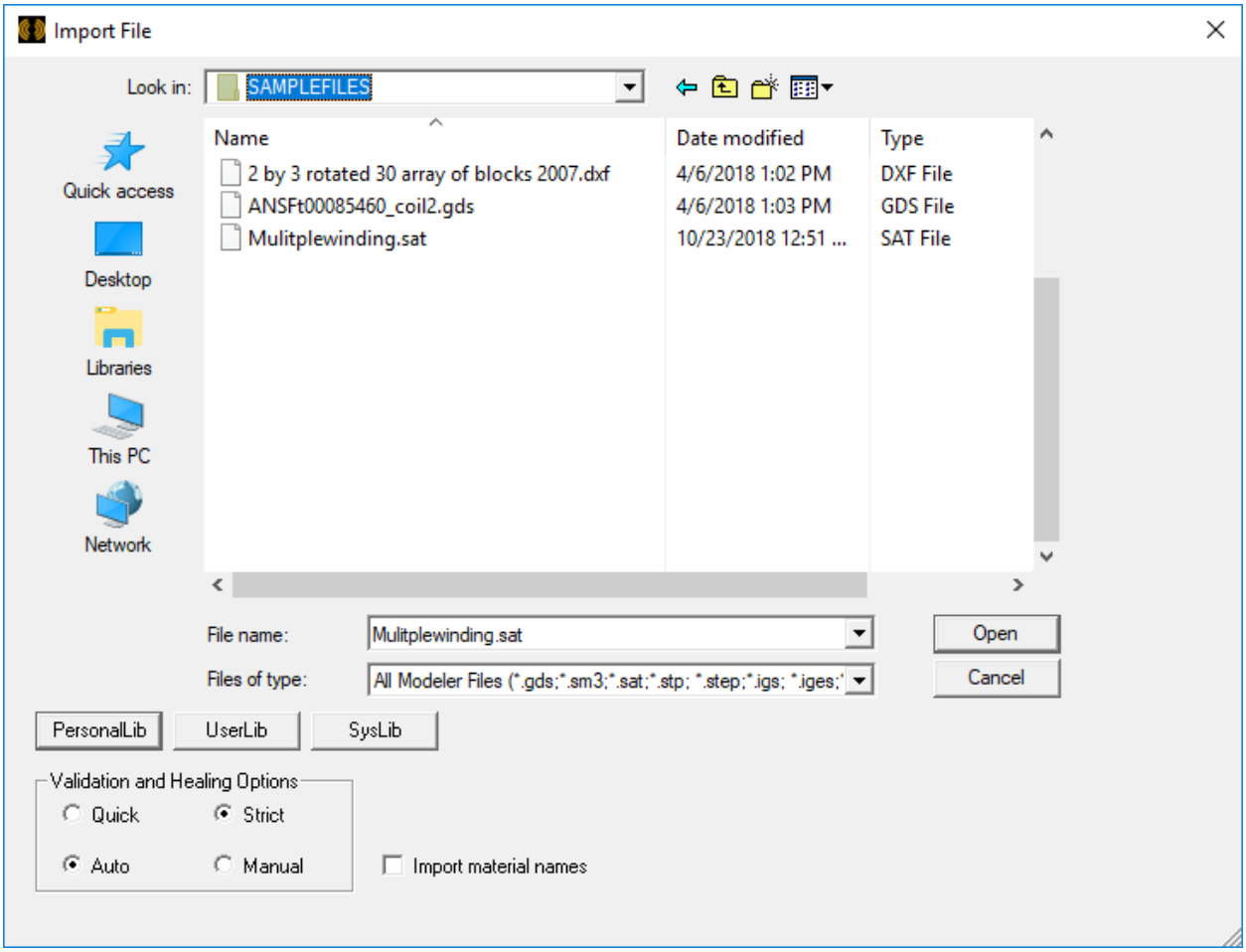
For more information, see Heal.

To import a file:

1. Click **Modeler > Import**.

   The **Import File** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If applicable to the file type, **Validation and Healing Options** are enabled by default. You can change them as desired.

You can also use the check box to choose whether to **Import material names**.

2. Click **Open**.
3. Depending on the file type, additional options may appear. Set these as appropriate.
4. Click **OK**.

   The file loads.

Importing 2D Model (GDSII) Files

2D model files can be imported directly into the active **Modeler** window. GDSII is the industry standard file format for 2D graphical design layout data.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

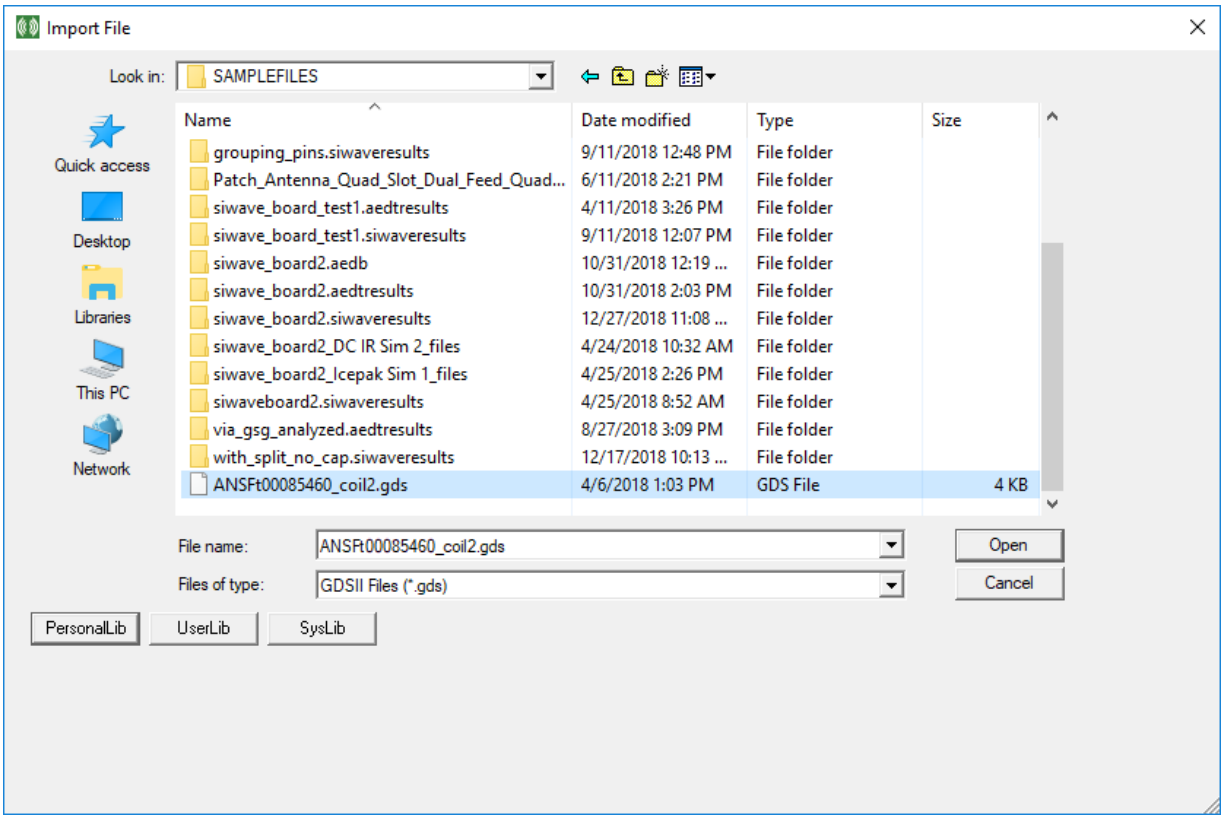
Note:

- Object, material, and parameter names with non-ASCII characters are not allowed for data transfer. Such transfers fail and produce an error message.
- If you import a file into an active **Modeler** window that contains an existing model, the file is added to the existing model; it will not replace it.

To import a 2D model file:

1. Click **Modeler** > **Import**.

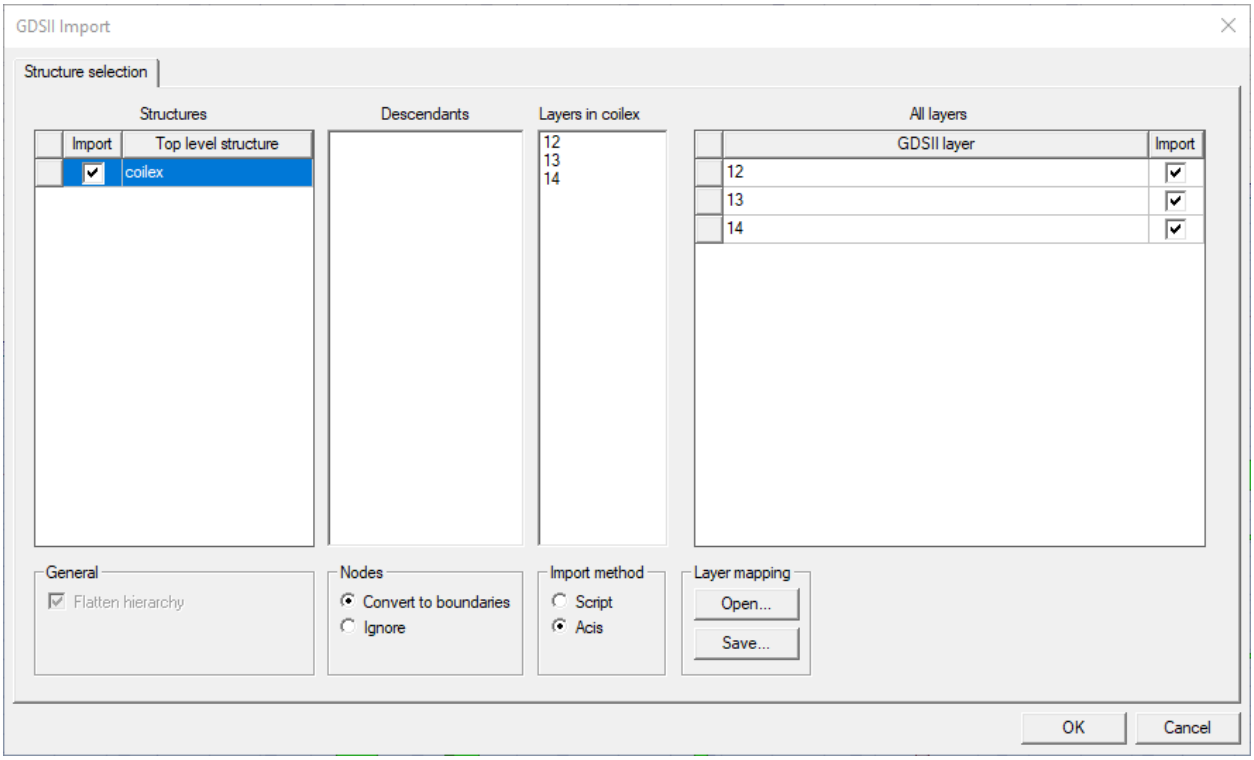
   The **Import File** dialog box appears.



2. Select a file type from the **Files of type** drop-down menu. For 2D model files, this is **GDSII Files (*.gds)**.
3. Use the file browser to find and select the file you want to import.
4. Click **Open**.

   The **GDSII Import** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Use the panes to select your desired options:
    - **Structure Selection** – a GDSII file may contain several top-level structures. Use this pane to select structure(s) for import. When multiple structures are imported, HFSS creates multiple designs under the current project, one for each of the GDSII structures.
    - **Descendants** – GDSII files are hierarchical and may contain many sub-layouts. This panel shows the sub-layouts in the selected top-level designs.
    - **Layers in [Structure Name]** – this panel's name changes based on the name of the selected structure, and displays that structure's layers, by layer number.
    - **All Layers** – lists all layers from all structures in the file. Use the **Import** check boxes to select or deselect layers for import.
    - **General** – the **Flatten hierarchy** check box is selected by default. HFSS always flattens any hierarchical geometry in the GDSII.
    - **Nodes** – GDSII supports nodes and boundaries as separate data types. Normally, boundaries represent polygons. HFSS can either convert objects that use the nodes data type to boundary types, or ignore them. Use the radio buttons to select either **Convert to boundaries** (default) or **Ignore**.
    - **Import Method** – select either Script or Acis.
    - **Layer Mapping** – if desired, create a mapping of GDSII layer numbers to layer names in the design stackup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Click **OK**.

   The file is imported into the active **Modeler** window.

Creating GDSII Layer Mapping Files

While importing a 2D model (GDSII) file, you can create a mapping of the GDSII layer numbers to layer names in the design stackup.

To create and use the mapping file:

1. Use a text editor to create a text file that maps GDSII layer numbers to layer names in the stackup. The layer mapping file must have a ***.tech** suffix.

   In *.tech format:

   - / is the comment character
   - Units may be specified with a line UNITS <string> before the lines of layer information. <string> is any of the allowed desktop length units. The default unit is nm.
   - Each layer is specified by a line that contains <*import layer*> <*product layer*> <*layer color*> <*layer elevation*> <*layer thickness*>

   where:

   - <*import layer*> is the name of the DXF layer
   - <*destination layer*> is the name to map the DXF layer to
   - <*layer color*> is a string from the choices listed here
   - <*layer elevation*> is specified in nms
   - <*layer height*> is specified in nms

   For example:

```
/ ------------------------------------------------------------
/ import#  destination     Color    Elevation  Thickness
/ ------------------------------------------------------------
S12         signal12         blue      1100       530
TR          trace            red       6620       530
S3          signal3          yellow    8150       2000
```

2. Save the *.tech file locally.
3. During import, click **Open** in the **Layer mapping** panel to locate and open the existing layer mapping file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **OK**.

   The file is imported into the active **Modeler** window.

Creating GDSII Layer Mapping Files

While importing a 2D model (GDSII) file, you can create a mapping of the GDSII layer numbers to layer names in the design stackup.

To create and use the mapping file:

1. Use a text editor to create a text file that maps GDSII layer numbers to layer names in the stackup. The layer mapping file must have a ***.tech** suffix.

   In *.tech format:

   - / is the comment character
   - Units may be specified with a line UNITS <string> before the lines of layer information. <string> is any of the allowed desktop length units. The default unit is nm.
   - Each layer is specified by a line that contains <*import layer*> <*product layer*> <*layer color*> <*layer elevation*> <*layer thickness*>

   where:

   - <*import layer*> is the name of the DXF layer
   - <*destination layer*> is the name to map the DXF layer to
   - <*layer color*> is a string from the choices listed here
   - <*layer elevation*> is specified in nms
   - <*layer height*> is specified in nms

   For example:

```
/ ------------------------------------------------------------
/ import#  destination     Color    Elevation  Thickness
/ ------------------------------------------------------------
S12         signal12         blue      1100       530
TR          trace            red       6620       530
S3          signal3          yellow    8150       2000
```

2. Save the *.tech file locally.
3. During import, click **Open** in the **Layer mapping** panel to locate and open the existing layer mapping file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **OK**.

   The file is imported into the active **Modeler** window.

Tech File Colors with RGB Values

Consult the table below for color string choices and their corresponding RGB values.

> **Note:**
>
> - Colors listed as two words can also be entered in Pascal case (e.g., GhostWhite is a valid substitute for ghost white).
> - Gray can be spelled using either the American or British spelling (Grey).

Color String(s)	RGB
snow1, snow	{ 255, 250, 250 }
snow2	{ 238, 233, 233 }
snow3	{ 205, 201, 201 }
snow4	{ 139, 137, 137 }
ghost white	{ 248, 248, 255 }
white smoke	{ 245, 245, 245 }
gainsboro	{ 220, 220, 220 }
floral white	{ 255, 250, 240 }
old lace	{ 253, 245, 230 }
linen	{ 250, 240, 230 }
antique white	{ 250, 235, 215 }
AntiqueWhite1	{ 255, 239, 219 }
AntiqueWhite2	{ 238, 223, 204 }
AntiqueWhite3	{ 205, 192, 176 }
AntiqueWhite4	{ 139, 131, 120 }
papaya whip	{ 255, 239, 213 }
blanched almond	{ 255, 235, 205 }
bisque1, bisque	{ 255, 228, 196 }
bisque2	{ 238, 213, 183 }
bisque3	{ 205, 183, 158 }
bisque4	{ 139, 125, 107 }
PeachPuff1, peach puff	{ 255, 218, 185 }
PeachPuff2	{ 238, 203, 173 }
PeachPuff3	{ 205, 175, 149 }
PeachPuff4	{ 139, 119, 101 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
NavajoWhite1, navajo white	{ 255, 222, 173 }
NavajoWhite2	{ 238, 207, 161 }
NavajoWhite3	{ 205, 179, 139 }
NavajoWhite4	{ 139, 121, 94 }
moccasin	{ 255, 228, 181 }
cornsilk1, cornsilk	{ 255, 248, 220 }
cornsilk2	{ 238, 232, 205 }
cornsilk3	{ 205, 200, 177 }
cornsilk4	{ 139, 136, 120 }
ivory1, ivory	{ 255, 255, 240 }
ivory2	{ 238, 238, 224 }
ivory3	{ 205, 205, 193 }
ivory4	{ 139, 139, 131 }
LemonChiffon1, lemon chiffon	{ 255, 250, 205 }
LemonChiffon2	{ 238, 233, 191 }
LemonChiffon3	{ 205, 201, 165 }
LemonChiffon4	{ 139, 137, 112 }
seashell1, seashell	{ 255, 245, 238 }
seashell2	{ 238, 229, 222 }
seashell3	{ 205, 197, 191 }
seashell4	{ 139, 134, 130 }
honeydew	{ 240, 255, 240 }
honeydew2	{ 224, 238, 224 }
honeydew3	{ 193, 205, 193 }
honeydew4	{ 131, 139, 131 }
mint cream	{ 245, 255, 250 }
azure1, azure	{ 240, 255, 255 }
azure2	{ 224, 238, 238 }
azure3	{ 193, 205, 205 }
azure4	{ 131, 139, 139 }
alice blue	{ 240, 248, 255 }
lavender	{ 230, 230, 250 }
LavenderBlush1, lavender blush	{ 255, 240, 245 }
LavenderBlush2	{ 238, 224, 229 }
LavenderBlush3	{ 205, 193, 197 }
LavenderBlush4	{ 139, 131, 134 }
MistyRose1, misty rose	{ 255, 228, 225 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
MistyRose2	{ 238, 213, 210 }
MistyRose3	{ 205, 183, 181 }
MistyRose4	{ 139, 125, 123 }
DarkSlateGray1	{ 151, 255, 255 }
DarkSlateGray2	{ 141, 238, 238 }
DarkSlateGray3	{ 121, 205, 205 }
DarkSlateGray4	{ 82, 139, 139 }
dark slate gray	{ 47, 79, 79 }
SlateGray1	{ 198, 226, 255 }
SlateGray2	{ 185, 211, 238 }
SlateGray3	{ 159, 182, 205 }
SlateGray4	{ 108, 123, 139 }
slate gray	{ 112, 128, 144 }
light slate gray	{ 119, 136, 153 }
midnight blue	{ 25, 25, 112 }
navy, navy blue	{ 0, 0, 128 }
cornflower blue	{ 100, 149, 237 }
slate blue	{ 106, 90, 205 }
medium slate blue	{ 123, 104, 238 }
light slate blue	{ 132, 112, 255 }
SlateBlue1	{ 131, 111, 255 }
SlateBlue2	{ 122, 103, 238 }
SlateBlue3	{ 105, 89, 205 }
SlateBlue4	{ 71, 60, 139 }
dark slate blue	{ 72, 61, 139 }
medium blue	{ 0, 0, 205 }
royal blue	{ 65, 105, 225 }
RoyalBlue1	{ 72, 118, 255 }
RoyalBlue2	{ 67, 110, 238 }
RoyalBlue3	{ 58, 95, 205 }
RoyalBlue4	{ 39, 64, 139 }
blue1, blue	{ 0, 0, 255 }
blue2	{ 0, 0, 238 }
blue3	{ 0, 0, 205 }
blue4, dark blue	{ 0, 0, 139 }
DodgerBlue1, dodger blue	{ 30, 144, 255 }
DodgerBlue2	{ 28, 134, 238 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
DodgerBlue3	{ 24, 116, 205 }
DodgerBlue4	{ 16, 78, 139 }
DeepSkyBlue1, deep sky blue	{ 0, 191, 255 }
DeepSkyBlue2	{ 0, 178, 238 }
DeepSkyBlue3	{ 0, 154, 205 }
DeepSkyBlue4	{ 0, 104, 139 }
sky blue	{ 135, 206, 235 }
SkyBlue1	{ 135, 206, 255 }
SkyBlue2	{ 126, 192, 238 }
SkyBlue3	{ 108, 166, 205 }
SkyBlue4	{ 74, 112, 139 }
light sky blue	{ 135, 206, 250 }
LightSkyBlue1	{ 176, 226, 255 }
LightSkyBlue2	{ 164, 211, 238 }
LightSkyBlue3	{ 141, 182, 205 }
LightSkyBlue4	{ 96, 123, 139 }
steel blue	{ 70, 130, 180 }
SteelBlue1	{ 99, 184, 255 }
SteelBlue2	{ 92, 172, 238 }
SteelBlue3	{ 79, 148, 205 }
SteelBlue4	{ 54, 100, 139 }
light steel blue	{ 176, 196, 222 }
LightSteelBlue1	{ 202, 225, 255 }
LightSteelBlue2	{ 188, 210, 238 }
LightSteelBlue3	{ 162, 181, 205 }
LightSteelBlue4	{ 110, 123, 139 }
light blue	{ 173, 216, 230 }
LightBlue1	{ 191, 239, 255 }
LightBlue2	{ 178, 223, 238 }
LightBlue3	{ 154, 192, 205 }
LightBlue4	{ 104, 131, 139 }
powder blue	{ 176, 224, 230 }
pale turquoise	{ 175, 238, 238 }
PaleTurquoise1	{ 187, 255, 255 }
PaleTurquoise2	{ 174, 238, 238 }
PaleTurquoise3	{ 150, 205, 205 }
PaleTurquoise4	{ 102, 139, 139 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
dark turquoise	{ 0, 206, 209 }
medium turquoise	{ 72, 209, 204 }
turquoise	{ 64, 224, 208 }
Turquoise1	{ 0, 245, 255 }
Turquoise2	{ 0, 229, 238 }
Turquoise3	{ 0, 197, 205 }
Turquoise4	{ 0, 134, 139 }
cyan1, cyan	{ 0, 255, 255 }
cyan2	{ 0, 238, 238 }
cyan3	{ 0, 205, 205 }
cyan4, dark cyan	{ 0, 139, 139 }
LightCyan1, light cyan	{ 224, 255, 255 }
LightCyan2	{ 209, 238, 238 }
LightCyan3	{ 180, 205, 205 }
LightCyan4	{ 122, 139, 139 }
cadet blue	{ 95, 158, 160 }
CadetBlue1	{ 152, 245, 255 }
CadetBlue2	{ 142, 229, 238 }
CadetBlue3	{ 122, 197, 205 }
CadetBlue4	{ 83, 134, 139 }
medium aquamarine	{ 102, 205, 170 }
aquamarine1, aquamarine	{ 127, 255, 212 }
aquamarine2	{ 118, 238, 198 }
aquamarine3	{ 102, 205, 170 }
aquamarine4	{ 69, 139, 116 }
dark green	{ 0, 100, 0 }
dark olive green	{ 85, 107, 47 }
DarkOliveGreen1	{ 202, 255, 112 }
DarkOliveGreen2	{ 188, 238, 104 }
DarkOliveGreen3	{ 162, 205, 90 }
DarkOliveGreen4	{ 110, 139, 61 }
dark sea green	{ 143, 188, 143 }
DarkSeaGreen1	{ 193, 255, 193 }
DarkSeaGreen2	{ 180, 238, 180 }
DarkSeaGreen3	{ 155, 205, 155 }
DarkSeaGreen4	{ 105, 139, 105 }
SeaGreen1	{ 84, 255, 159 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
SeaGreen2	{ 78, 238, 148 }
SeaGreen3	{ 67, 205, 128 }
SeaGreen4, sea green	{ 46, 139, 87 }
medium sea green	{ 60, 179, 113 }
light sea green	{ 32, 178, 170 }
pale green	{ 152, 251, 152 }
PaleGreen1	{ 154, 255, 154 }
PaleGreen2	{ 144, 238, 144 }
PaleGreen3	{ 124, 205, 124 }
PaleGreen4	{ 84, 139, 84 }
SpringGreen1, spring green	{ 0, 255, 127 }
SpringGreen2	{ 0, 238, 118 }
SpringGreen3	{ 0, 205, 102 }
SpringGreen4	{ 0, 139, 69 }
medium spring green	{ 0, 250, 154 }
lawn green	{ 124, 252, 0 }
light green	{ 144, 238, 144 }
green1, green	{ 0, 255, 0 }
green2	{ 0, 238, 0 }
green3	{ 0, 205, 0 }
green4	{ 0, 139, 0 }
chartreuse1, chartreuse	{ 127, 255, 0 }
chartreuse2	{ 118, 238, 0 }
chartreuse3	{ 102, 205, 0 }
chartreuse4	{ 69, 139, 0 }
green yellow	{ 173, 255, 47 }
lime green	{ 50, 205, 50 }
yellow green	{ 154, 205, 50 }
forest green	{ 34, 139, 34 }
olive drab	{ 107, 142, 35 }
OliveDrab1	{ 192, 255, 62 }
OliveDrab2	{ 179, 238, 58 }
OliveDrab3	{ 154, 205, 50 }
OliveDrab4	{ 105, 139, 34 }
dark khaki	{ 189, 183, 107 }
khaki	{ 240, 230, 140 }
khaki1	{ 255, 246, 143 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
khaki2	{ 238, 230, 133 }
khaki3	{ 205, 198, 115 }
khaki4	{ 139, 134, 78 }
LightYellow1, light yellow	{ 255, 255, 224 }
LightYellow2	{ 238, 238, 209 }
LightYellow3	{ 205, 205, 180 }
LightYellow4	{ 139, 139, 122 }
yellow1, yellow	{ 255, 255, 0 }
yellow2	{ 238, 238, 0 }
yellow3	{ 205, 205, 0 }
yellow4	{ 139, 139, 0 }
gold1, gold	{ 255, 215, 0 }
gold2	{ 238, 201, 0 }
gold3	{ 205, 173, 0 }
gold4	{ 139, 117, 0 }
pale goldenrod	{ 238, 232, 170 }
light goldenrod yellow	{ 250, 250, 210 }
light goldenrod	{ 238, 221, 130 }
LightGoldenrod1	{ 255, 236, 139 }
LightGoldenrod2	{ 238, 220, 130 }
LightGoldenrod3	{ 205, 190, 112 }
LightGoldenrod4	{ 139, 129, 76 }
goldenrod	{ 218, 165, 32 }
goldenrod1	{ 255, 193, 37 }
goldenrod2	{ 238, 180, 34 }
goldenrod3	{ 205, 155, 29 }
goldenrod4	{ 139, 105, 20 }
dark goldenrod	{ 184, 134, 11 }
DarkGoldenrod1	{ 255, 185, 15 }
DarkGoldenrod2	{ 238, 173, 14 }
DarkGoldenrod3	{ 205, 149, 12 }
DarkGoldenrod4	{ 139, 101, 8 }
rosy brown	{ 188, 143, 143 }
RosyBrown1	{ 255, 193, 193 }
RosyBrown2	{ 238, 180, 180 }
RosyBrown3	{ 205, 155, 155 }
RosyBrown4	{ 139, 105, 105 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
indian red	{ 205, 92, 92 }
IndianRed1	{ 255, 106, 106 }
IndianRed2	{ 238, 99, 99 }
IndianRed3	{ 205, 85, 85 }
IndianRed4	{ 139, 58, 58 }
saddle brown	{ 139, 69, 19 }
sienna	{ 160, 82, 45 }
sienna1	{ 255, 130, 71 }
sienna2	{ 238, 121, 66 }
sienna3	{ 205, 104, 57 }
sienna4	{ 139, 71, 38 }
peru	{ 205, 133, 63 }
burlywood	{ 222, 184, 135 }
burlywood1	{ 255, 211, 155 }
burlywood2	{ 238, 197, 145 }
burlywood3	{ 205, 170, 125 }
burlywood4	{ 139, 115, 85 }
beige	{ 245, 245, 220 }
wheat	{ 245, 222, 179 }
wheat1	{ 255, 231, 186 }
wheat2	{ 238, 216, 174 }
wheat3	{ 205, 186, 150 }
wheat4	{ 139, 126, 102 }
sandy brown	{ 244, 164, 96 }
tan	{ 210, 180, 140 }
tan1	{ 255, 165, 79 }
tan2	{ 238, 154, 73 }
tan3	{ 205, 133, 63 }
tan4	{ 139, 90, 43 }
chocolate	{ 210, 105, 30 }
chocolate1	{ 255, 127, 36 }
chocolate2	{ 238, 118, 33 }
chocolate3	{ 205, 102, 29 }
chocolate4	{ 139, 69, 19 }
firebrick	{ 178, 34, 34 }
firebrick1	{ 255, 48, 48 }
firebrick2	{ 238, 44, 44 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
firebrick3	{ 205, 38, 38 }
firebrick4	{ 139, 26, 26 }
brown	{ 165, 42, 42 }
brown1	{ 255, 64, 64 }
brown2	{ 238, 59, 59 }
brown3	{ 205, 51, 51 }
brown4	{ 139, 35, 35 }
dark salmon	{ 233, 150, 122 }
salmon	{ 250, 128, 114 }
salmon1	{ 255, 140, 105 }
salmon2	{ 238, 130, 98 }
salmon3	{ 205, 112, 84 }
salmon4	{ 139, 76, 57 }
LightSalmon1, light salmon	{ 255, 160, 122 }
LightSalmon2	{ 238, 149, 114 }
LightSalmon3	{ 205, 129, 98 }
LightSalmon4	{ 139, 87, 66 }
orange1, orange	{ 255, 165, 0 }
orange2	{ 238, 154, 0 }
orange3	{ 205, 133, 0 }
orange4	{ 139, 90, 0 }
dark orange	{ 255, 140, 0 }
DarkOrange1	{ 255, 127, 0 }
DarkOrange2	{ 238, 118, 0 }
DarkOrange3	{ 205, 102, 0 }
DarkOrange4	{ 139, 69, 0 }
coral	{ 255, 127, 80 }
coral1	{ 255, 114, 86 }
coral2	{ 238, 106, 80 }
coral3	{ 205, 91, 69 }
coral4	{ 139, 62, 47 }
light coral	{ 240, 128, 128 }
tomato1, tomato	{ 255, 99, 71 }
tomato2	{ 238, 92, 66 }
tomato3	{ 205, 79, 57 }
tomato4	{ 139, 54, 38 }
OrangeRed1, orange red	{ 255, 69, 0 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
OrangeRed2	{ 238, 64, 0 }
OrangeRed3	{ 205, 55, 0 }
OrangeRed4	{ 139, 37, 0 }
red1, red	{ 255, 0, 0 }
red2	{ 238, 0, 0 }
red3	{ 205, 0, 0 }
red4, dark red	{ 139, 0, 0 }
hot pink	{ 255, 105, 180 }
HotPink1	{ 255, 110, 180 }
HotPink2	{ 238, 106, 167 }
HotPink3	{ 205, 96, 144 }
HotPink4	{ 139, 58, 98 }
DeepPink1, deep pink	{ 255, 20, 147 }
DeepPink2	{ 238, 18, 137 }
DeepPink3	{ 205, 16, 118 }
DeepPink4	{ 139, 10, 80 }
pink	{ 255, 192, 203 }
pink1	{ 255, 181, 197 }
pink2	{ 238, 169, 184 }
pink3	{ 205, 145, 158 }
pink4	{ 139, 99, 108 }
light pink	{ 255, 182, 193 }
LightPink1	{ 255, 174, 185 }
LightPink2	{ 238, 162, 173 }
LightPink3	{ 205, 140, 149 }
LightPink4	{ 139, 95, 101 }
maroon	{ 176, 48, 96 }
maroon1	{ 255, 52, 179 }
maroon2	{ 238, 48, 167 }
maroon3	{ 205, 41, 144 }
maroon4	{ 139, 28, 98 }
pale violet red	{ 219, 112, 147 }
PaleVioletRed1	{ 255, 130, 171 }
PaleVioletRed2	{ 238, 121, 159 }
PaleVioletRed3	{ 205, 104, 137 }
PaleVioletRed4	{ 139, 71, 93 }
medium violet red	{ 199, 21, 133 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
violet red	{ 208, 32, 144 }
VioletRed1	{ 255, 62, 150 }
VioletRed2	{ 238, 58, 140 }
VioletRed3	{ 205, 50, 120 }
VioletRed4	{ 139, 34, 82 }
magenta1, magenta	{ 255, 0, 255 }
magenta2	{ 238, 0, 238 }
magenta3	{ 205, 0, 205 }
magenta4, dark magenta	{ 139, 0, 139 }
violet	{ 238, 130, 238 }
plum	{ 221, 160, 221 }
plum1	{ 255, 187, 255 }
plum2	{ 238, 174, 238 }
plum3	{ 205, 150, 205 }
plum4	{ 139, 102, 139 }
orchid	{ 218, 112, 214 }
orchid1	{ 255, 131, 250 }
orchid2	{ 238, 122, 233 }
orchid3	{ 205, 105, 201 }
orchid4	{ 139, 71, 137 }
medium orchid	{ 186, 85, 211 }
MediumOrchid1	{ 224, 102, 255 }
MediumOrchid2	{ 209, 95, 238 }
MediumOrchid3	{ 180, 82, 205 }
MediumOrchid4	{ 122, 55, 139 }
dark orchid	{ 153, 50, 204 }
DarkOrchid1	{ 191, 62, 255 }
DarkOrchid2	{ 178, 58, 238 }
DarkOrchid3	{ 154, 50, 205 }
DarkOrchid4	{ 104, 34, 139 }
dark violet	{ 148, 0, 211 }
blue violet	{ 138, 43, 226 }
purple	{ 160, 32, 240 }
purple1	{ 155, 48, 255 }
purple2	{ 145, 44, 238 }
purple3	{ 125, 38, 205 }
purple4	{ 85, 26, 139 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
medium purple	{ 147, 112, 219 }
MediumPurple1	{ 171, 130, 255 }
MediumPurple2	{ 159, 121, 238 }
MediumPurple3	{ 137, 104, 205 }
MediumPurple4	{ 93, 71, 139 }
thistle	{ 216, 191, 216 }
thistle1	{ 255, 225, 255 }
thistle2	{ 238, 210, 238 }
thistle3	{ 205, 181, 205 }
thistle4	{ 139, 123, 139 }
gray0, black	{ 0, 0, 0 }
gray1	{ 3, 3, 3 }
gray2	{ 5, 5, 5 }
gray3	{ 8, 8, 8 }
gray4	{ 10, 10, 10 }
gray5	{ 13, 13, 13 }
gray6	{ 15, 15, 15 }
gray7	{ 18, 18, 18 }
gray8	{ 20, 20, 20 }
gray9	{ 23, 23, 23 }
gray10	{ 26, 26, 26 }
gray11	{ 28, 28, 28 }
gray12	{ 31, 31, 31 }
gray13	{ 33, 33, 33 }
gray14	{ 36, 36, 36 }
gray15	{ 38, 38, 38 }
gray16	{ 41, 41, 41 }
gray17	{ 43, 43, 43 }
gray18	{ 46, 46, 46 }
gray19	{ 48, 48, 48 }
gray20	{ 51, 51, 51 }
gray21	{ 54, 54, 54 }
gray22	{ 56, 56, 56 }
gray23	{ 59, 59, 59 }
gray24	{ 61, 61, 61 }
gray25	{ 64, 64, 64 }
gray26	{ 66, 66, 66 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
gray27	{ 69, 69, 69 }
gray28	{ 71, 71, 71 }
gray29	{ 74, 74, 74 }
gray30	{ 77, 77, 77 }
gray31	{ 79, 79, 79 }
gray32	{ 82, 82, 82 }
gray33	{ 84, 84, 84 }
gray34	{ 87, 87, 87 }
gray35	{ 89, 89, 89 }
gray36	{ 92, 92, 92 }
gray37	{ 94, 94, 94 }
gray38	{ 97, 97, 97 }
gray39	{ 99, 99, 99 }
gray40	{ 102, 102, 102 }
gray41, dim gray	{ 105, 105, 105 }
gray42	{ 107, 107, 107 }
gray43	{ 110, 110, 110 }
gray44	{ 112, 112, 112 }
gray45	{ 115, 115, 115 }
gray46	{ 117, 117, 117 }
gray47	{ 120, 120, 120 }
gray48	{ 122, 122, 122 }
gray49	{ 125, 125, 125 }
gray50	{ 127, 127, 127 }
gray51	{ 130, 130, 130 }
gray52	{ 133, 133, 133 }
gray53	{ 135, 135, 135 }
gray54	{ 138, 138, 138 }
gray55	{ 140, 140, 140 }
gray56	{ 143, 143, 143 }
gray57	{ 145, 145, 145 }
gray58	{ 148, 148, 148 }
gray59	{ 150, 150, 150 }
gray60	{ 153, 153, 153 }
gray61	{ 156, 156, 156 }
gray62	{ 158, 158, 158 }
gray63	{ 161, 161, 161 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
gray64	{ 163, 163, 163 }
gray65	{ 166, 166, 166 }
gray66	{ 168, 168, 168 }
dark gray	{ 169, 169, 169 }
gray67	{ 171, 171, 171 }
gray68	{ 173, 173, 173 }
gray69	{ 176, 176, 176 }
gray70	{ 179, 179, 179 }
gray71	{ 181, 181, 181 }
gray72	{ 184, 184, 184 }
gray73	{ 186, 186, 186 }
gray74	{ 189, 189, 189 }
gray	{ 190, 190, 190 }
gray75	{ 191, 191, 191 }
gray76	{ 194, 194, 194 }
gray77	{ 196, 196, 196 }
gray78	{ 199, 199, 199 }
gray79	{ 201, 201, 201 }
gray80	{ 204, 204, 204 }
gray81	{ 207, 207, 207 }
gray82	{ 209, 209, 209 }
light gray	{ 211, 211, 211 }
gray83	{ 212, 212, 212 }
gray84	{ 214, 214, 214 }
gray85	{ 217, 217, 217 }
gray86	{ 219, 219, 219 }
gray87	{ 222, 222, 222 }
gray88	{ 224, 224, 224 }
gray89	{ 227, 227, 227 }
gray90	{ 229, 229, 229 }
gray91	{ 232, 232, 232 }
gray92	{ 235, 235, 235 }
gray93	{ 237, 237, 237 }
gray94	{ 240, 240, 240 }
gray95	{ 242, 242, 242 }
gray96	{ 245, 245, 245 }
gray97	{ 247, 247, 247 }

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color String(s)	RGB
gray98	{ 250, 250, 250 }
gray99	{ 252, 252, 252 }
gray100, white	{ 255, 255, 255 }

Importing 3D Model Files

3D model files can be imported directly into the active **Modeler** window. Supported 3D model file formats are listed in the table that follows.

Type	Extension(s)	Supported Version(s)
ACIS SAB	*.sab	Up to 2024
ACIS SAT	*.sat	Up to 2024
Ansoft Geometry	*.AnstGeom	All
Ansys 3D Modeler	*.sm3	Up to 2022R2 See UDMs and CAD integration with Workbench.
AutoCAD	*.dxf, *.dwg	**Linux**: Versions 2.5 (AC1002) through 2012 (AC1024) **Windows**: Versions 2.5 (AC1002) through 2017 (AC1027) See Importing DXF and DWG Format Files.
Autodesk Inventor	*.ipt, *.iam	Up to 2025
CATIA	*.exp, *.model, *.CATPart, *.CATproduct	CATIA V4 – Up to 4.2.5 CATIA V5 – Up to V5_6R2024
Creo Parametric	*.prt, *.asm	Pro/Engineer 19.0 to Creo 11.0
IGES	*.iges, *.igs	5.1, 5.2, 5.3
JT	*.jt	Up to v10.9
NASTRAN	*.nas	CROD, CBEAM, CTRIA3, and CQUAD4 element types
Siemens NX	*.prt	UG11 to UG18, UG NX, NX5 to NX12, NX1847 to NX2406
Parasolid	*.x_t, *.x_b	9.0 through 36.1.185
SOLIDWORKS	*.SLDPRT, *.SLDASM	From 97 up to 2024
STEP	*.step, *.stp	AP 203 E1/E2, AP 214, AP 242 (geometry only)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type	Extension(s)	Supported Version(s)
STL	*.stl	2.0 (see Lightweight Geometry in SBR+)

Note:

- Object, material, and parameter names with non-ASCII characters are not allowed for data transfer. Such transfers fail and produce an error message.
- If you import a file into an active **Modeler** window that contains an existing model, the file is added to the existing model; it will not replace it.

To import a 3D model file:

1. Click **Modeler** > **Import**.

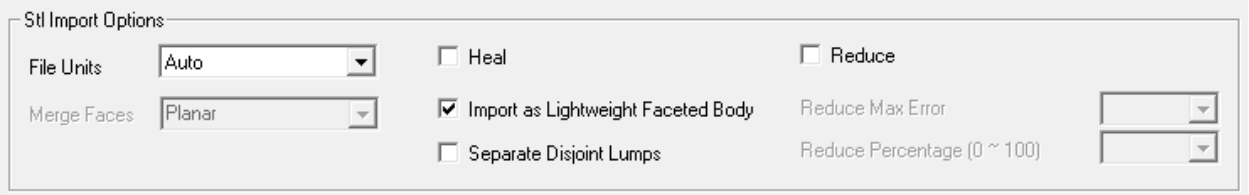
   The **Import File** dialog box appears.

2. Select the file type you want from the **Files of type** drop-down menu.
3. Set any import options available for the selected file type. Options may include:
   - **Validation and Healing Options** – see Healing an Imported Object.
   - **Import Material Names** – use the check box to import material names.
   - **Import Free Surfaces** – for Creo Parametric files, use the check box to import surfaces as well as parts.
   - **Stitch Tolerance and Units** – for STEP and IGES files, specify stitch tolerance and units. The default value (auto) comes from Healing options.
   - **STL Import Options** – for STL files for Modal or Terminal solutions, select the modeling units to which the imported model is scaled and whether to merge faces that are on the same plane, and whether to Heal on import. Heal attempts to remove common issues while importing stl files, including closing solids when they are open due to minor cracks in triangulation and removing self-intersections.

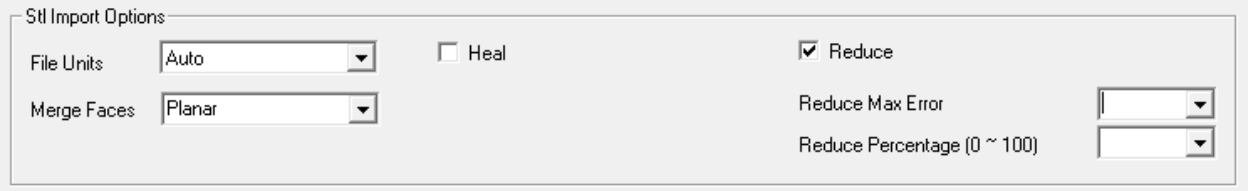
   The generic format specification for STL does not include units. When “Auto” is chosen for the file units, the current Model units are normally used.

   If the solution type is HFSS SBR+, the STL import also includes options to **Import as Lightweight Faceted Body** and **Separate Disjoint Lumps**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For STL files, you can choose to **Reduce** the number of facets in faceted objects while keeping the overall shape of the object.



Checking **Reduce** enables fields for specifying Reduce Max Error and Reduce Percentage.

For details on these features and their use with SBR+, see Lightweight Geometry in SBR+.

- **Create Groups for Sub Assembly** – for formats that allow importing MCAD Assemblies as groups, use the check box to retain the assembly structure of objects using groups. For every subassembly in the model, a group is created and it retains hierarchical information by creating group hierarchy. See: Group Commands for Modeler Objects.

4. Use the file browser to find and select a file for import.
5. Click **Open**.

   The file is imported into the active **Modeler** window

> **Note:**
>
> While objects created in HFSS can always be classed in the history tree as either a solid, sheet, or wire, some imported objects may have mixture of these. HFSS places such objects in an **Unclassified** folder in the history tree.

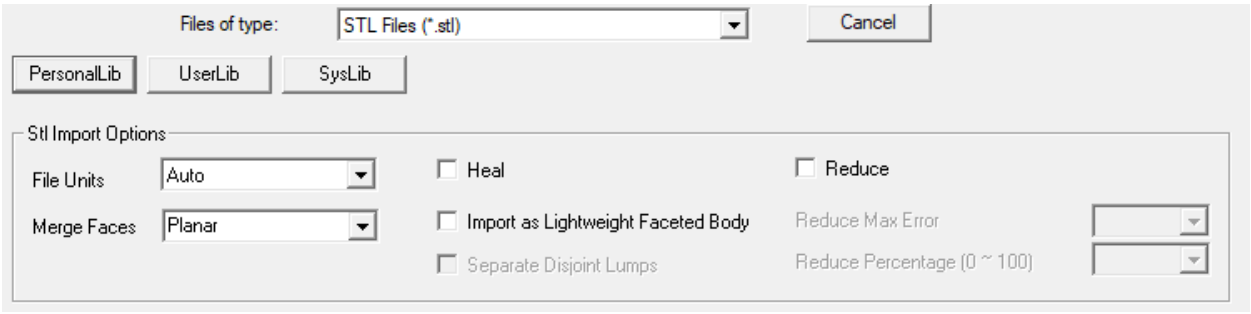
Lightweight Geometry for HFSS SBR+ Designs

Importing large stl files into Ansys Electronics Desktop can be time consuming. STL files contain triangulations of objects, and importing them to the modeler creates a face out of every triangle,

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and an edge out of every triangle edge. This creates a large number of faces and edges. which can slow down operations. The lightweight geometry feature gives you the option to import objects as one face, without edges or vertices in them. Internally, the modeler does not create bodies for these objects.

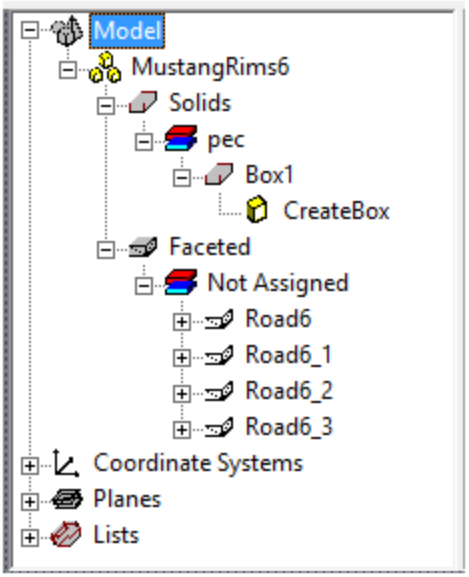
When the solution type is SBR+, the Import File dialog for STL files contains several options.



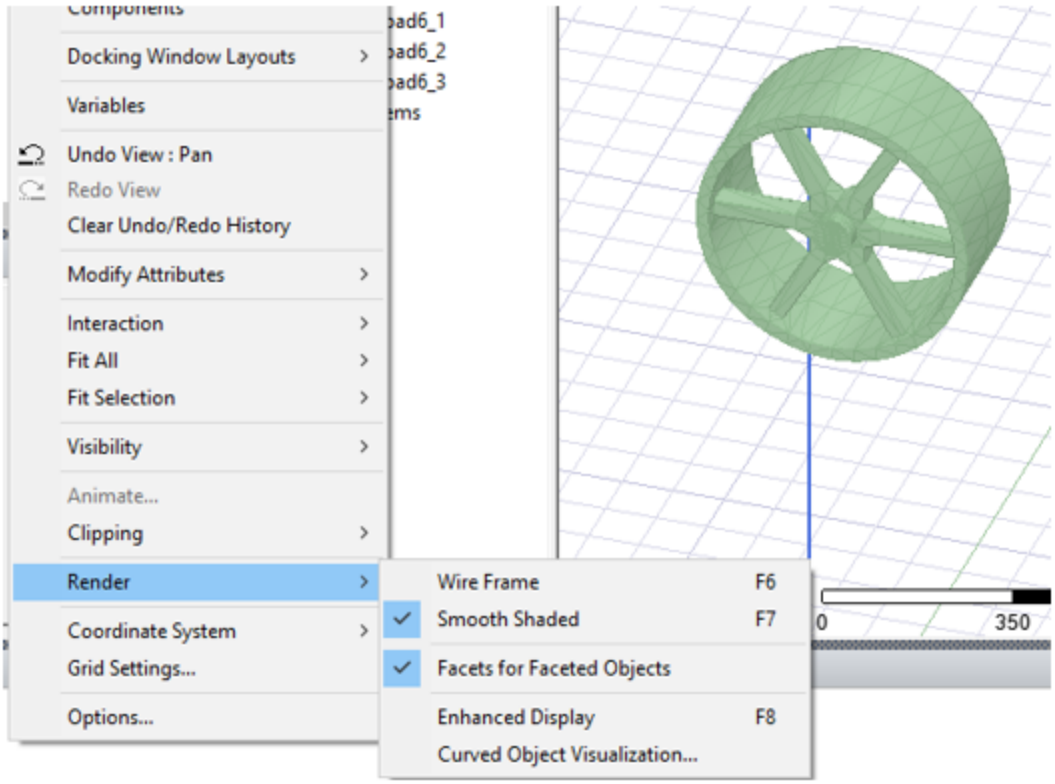
- **Heal** attempts to remove common issues while importing stl files, including closing solids when they are open due to minor cracks in triangulation and removing self-intersections.
- **Import as Lightweight Faceted Body**, if checked, creates faceted bodies from imported objects and enables the Separate Disjoint Lumps option.
- **Separate Disjoint Lumps** option, if checked, create multiple parts if the .stl files have objects having disjoint lumps.
- **Reduce** enables lets you reduce the number of facets in faceted objects while keeping the overall shape of the object. Checking **Reduce** enables fields for specifying Reduce Max Error and Reduce Percentage.

Once imported, faceted objects have distinct icons in history tree. They can also be grouped together, as shown in the following figure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A new option, **Facets for Faceted Objects**, is provided in **View>Render** menu for SBR+ solution types. When this option is check on, the modeler renders edges of triangle contained in the faceted bodies. These edges cannot be selected. See the following figure:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initial Mesh Operations for Lightweight Geometries

Importing a lightweight geometry into an SBR+ design disables editing for the **Use Dynamic Meshing** option for the Initial Mesh Settings. After lightweight geometry import, Use Dynamic Meshing is always On, and eliminates mesh file generation during simulation.

> **Important:**
>
> The consolidation of surfaces into a conformal mesh is skipped for dynamic and tolerant meshing, including for light weight geometries. This can lead to overlapping surfaces in SBR+ simulations. The user should carefully avoid overlapping surfaces or objects as SBR+ can produce unexpected results.

Permitted Modeler Operations on Faceted Objects

- Arrange operations such as Move, Rotate, Mirror.
- Duplicate along line, around axis, with mirror.
- Scale.
- Copy-Paste.
- Create object CS - you can snap to vertices of facets to create object CS.
- Create face CS - you can pick any point, determine the triangle on which the picked point lies, and store the position relative to the triangle vertices. **Note:** this capability is particularly useful for antenna placement on lightweight geometry models in SBR+ designs.
- Create 3D components out of faceted objects.
- Selecting faces for assigning boundaries, coordinate systems, excitations or hybrid regions. **Note:** the entire object boundary is selected.

Other Characteristics of Faceted Objects

- Solid and sheet objects are identified in the History tree.
- Object Measure dialog shows the correct volume, and surface area of faceted objects.
- Faceted objects are skipped during intersection checks and containment checks done during geometry validation.
- Faceted objects cannot be exported from the Modeler.

Importing DXF and DWG Format Files

You can import AutoCAD versions 2.5 (AC1002) through 2012 (AC1024) on Linux and versions 2.5 (AC1002) through 2017 (AC1027) on Windows. The entities are imported as 2D, not 3D. The

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

types of entities imported are:

- 2D Polyline
- Polyline
- Line
- Arc
- Circle
- Ellipse
- Solid
- Block

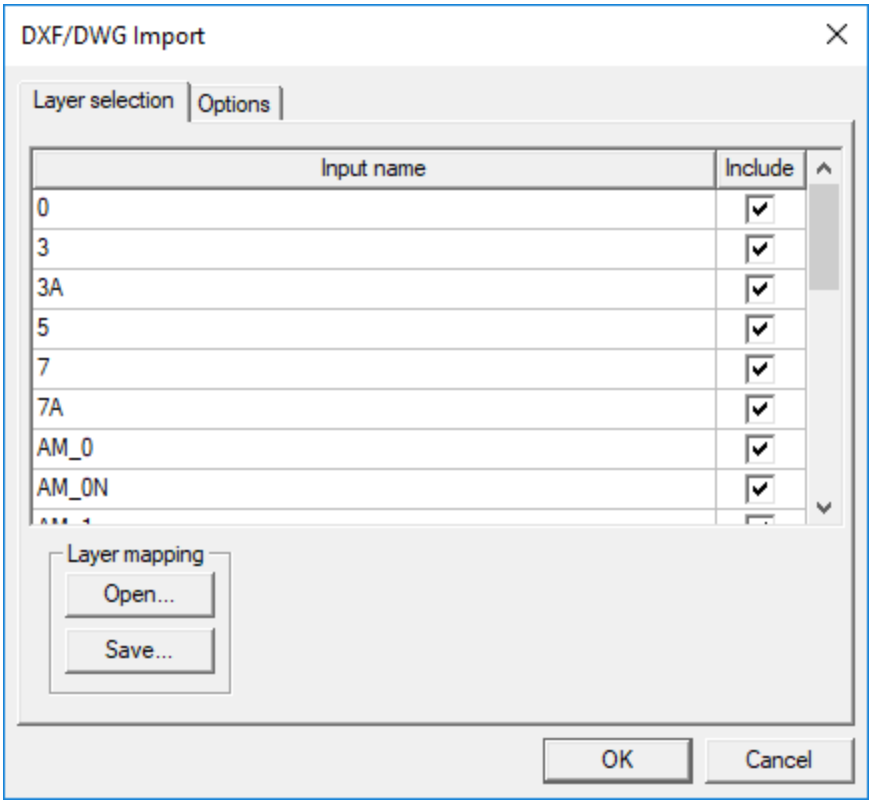
To import a ***.dxf** or ***.dwg** model file (which may use an associated *.tech file):

1. Click **Modeler** > **Import**.

   The **Import File** dialog box appears.

2. Use the file browser to find and select an AutoCAD file for import.
3. Click **Open**.

   The **DXF/DWG Import** dialog box opens with the **Layer Selection** tab selected.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

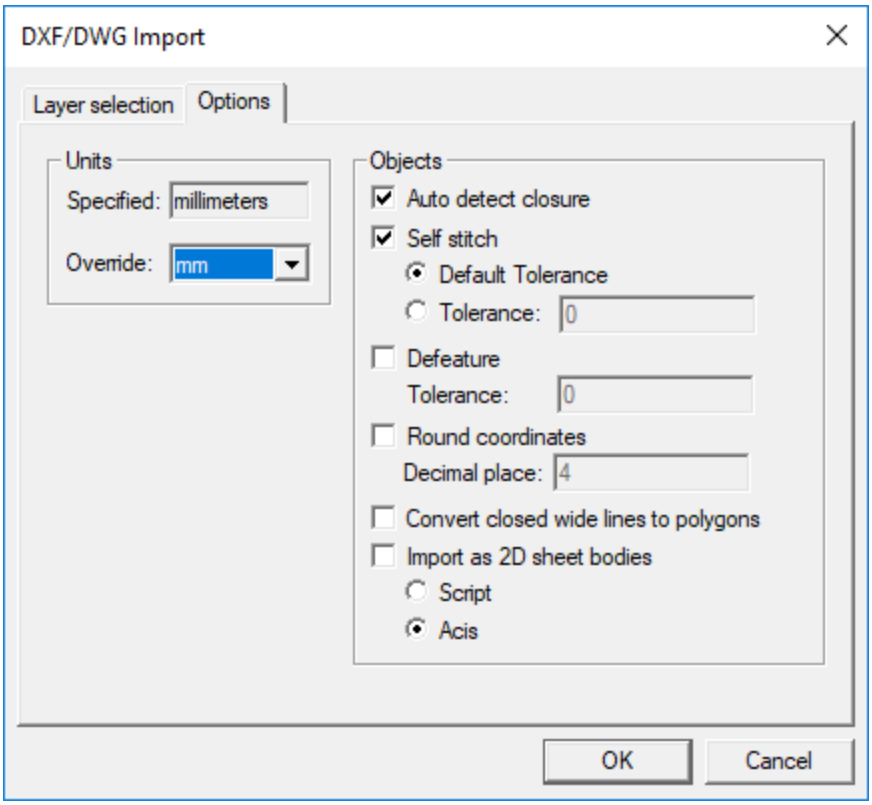
The **Input Name** field shows the name of the layer in the DXF/DWG file. This field is not editable.

4. Use the **Include** check boxes to specify which layers to import from the selected file.
5. If there is an associated *.tech file, click **Open** to find and select it. A *.tech file is a plain text file that includes units, layer names, color, elevation, and thickness information. For example:

```
units um

//Layer_Name Color Elevation Thickness

BOTTOMLAYER purple 0 200

MIDLAYER green 500 200

TOPLAYER blue 1000 200
```

6. Click **Options**.

   The **Options** tab displays.



7. In the **Units** area, the **Specified** field displays the layout units in the file for import. Use the **Override** drop-down menu to select a different unit of measure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. In the **Objects** area, use the check boxes to fine tune the import:
    - **Auto-detect closure** causes polylines to be checked to see whether or not they are closed. If a polyline is closed, the modeler creates a polygon in the design.
    - **Self-stitch** joins multiple straight line segments to form polylines. If the resulting polyline is closed, a polygon is created.
        - **Default Tolerance** uses default values to whether two coordinates should be considered for joining.
        - **Tolerance** allows you to specify a specific value for tolerance. This is useful if particular features in a model are outside of a normal tolerance allowance.
    - **De-feature Tolerance** removes certain small features in the imported geometry to reduce complexity. The features that are removed include: multiple points placed within the specified distance; thin or narrow regions ("thins" and "spikes"); and extraneous points along straight line segments.
    - **Round Coordinates Decimal Place** rounds all imported data to the specified number of decimal points.
    - **Convert closed wide lines to polygons** imports wide polylines as polygons. This affords more flexibility in changing the shape of such an object.
    - **Import as 2D sheet bodies** causes imported objects to be organized in terms of 2D sheets.
9. Select either **Script** or **Acis** for the import method.
10. Click **OK**.

    The file is imported into the active **Layout** window.

Importing Solution Data

1. Click **HFSS > Results > Import Solutions**.

    The **Imported Data** dialog box appears.

2. Click **Import Solution**.

    The **S Parameter Import** dialog box appears.

3. In the **File Name** text box, type the name of the solution file you want to import or click **Browse** and use the file browser to locate the file.
4. Selecting the file with the file browser loads it, but note that the file has not been imported yet.
5. Optionally, type a new name in the **Source Name** box or accept the default name.
6. Click the solutions you want to import in the **Available Solutions** list, and then click **Import**.

    You return to the **Imported Data** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Click the solution data you want to import, and then click **OK**.

Importing Data Tables

You can import data table files that contain data in the following formats:

- Tab-separated – HFSS will recognize complex data if the values are separated by a comma (e.g., *real, imaginary*).
- Comma-separated – HFSS will recognize complex data if the values are separated by a space (e.g., *real imaginary*).

1. Click **HFSS** > **Results** > **Import Solutions**.
   - Alternatively, right-click **Results** in the project tree and then click **Import Solutions** on the shortcut menu.

   The **Imported Data** dialog box appears.
2. Click **Import Table**.

   The **Table Import** dialog box appears.
3. In the **File Name** text box, type the name of the data table file you want to import or click **Browse** and use the file browser to locate the file.
4. If the data in the table is complex, select the format — real/ imaginary, or magnitude/ phase — in which to import the data.

   If the data is simple, this option will be ignored.
5. Click **Load File**. Note that the file has not been imported yet.
6. Optionally, type a new name in the **Source Name** box that indicates the origin or the data table, or accept the default name.
7. Optionally, type a new name in the **Table Name** box that describes the data in the table, or accept the default name
8. In the **All Columns** list, the headings of each column in the data file are listed. Optionally, specify a new name for a column heading by doing the following:
   a. In the **All Columns** list, click the heading you want to change. The heading appears in the **Column Name** box.
   b. Type a new name in the **Column Name** box, and then click **Set Column Name**.

   The heading is changed to the new name in every place it appears in the **Imported Data** dialog box.
9. In the **Independent Data Columns** list, the first heading in the data table file is listed by default. In the **Dependent Data Columns** list, the second and subsequent headings in the data table file are listed by default. Optionally, click a heading name and then click an arrow button to move it from one column to another.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. If the data in the **Dependent Data Columns** list contains matrix data, select **Matrix Data**. If it contains field data, select **Field Data**.
11. Click **Import**.

    You return to the **Imported Data** dialog box.

12. Click the data you want to import in the **Current Imports** list, and then click **OK**.

The solution data is now available for postprocessing.

Importing Ansys Electronics Desktop Plot Data

Import Solutions can also import plot data.

1. On the solver menu click **Results > Import Solutions**.

   The **Imported Data** dialog box appears.

2. Click the Import Plot Data button

   The **Read Plot Data** file dialog box opens.

3. Use the file browser to select the plot data file (*.dat) to open.

   You can choose to specify the file as Read Only.

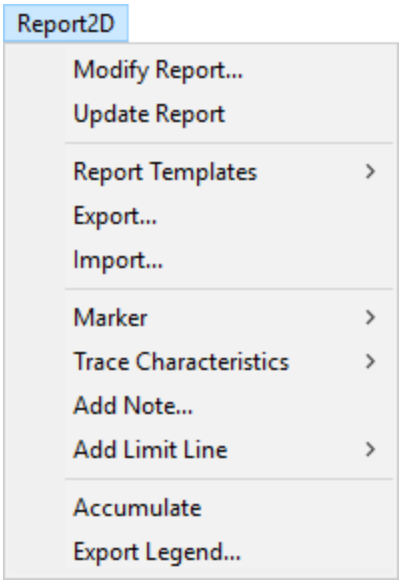
4. Click the Open button to import the file.

   The imported files are listed in the **Imported Data** dialog box.

Importing Plot Data

When a report is open in Ansys Electronics Desktop, the **Report2D** menu appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> You *must* have a report open in order to import plot data.

The **Report2D**> **Import Data** command lets you import plot data from comma delimited files (*.csv), tab delimited files (*.tab), Ansoft PlotData files (*.dat), post-processor files (*.txt), or Ansoft ReportData files (*.rdat).

1. From the top menu bar, click **Report2D** > **Import**.

   The **Open** window appears.

2. Browse for and select the desired file.
3. Click **Open** to import the file into the current report.

   The imported traces appear in the Project tree under the current report.

   > **Note:**
   >
   > For a report trace where the primary sweep is NOT the same as the x-component, an export report then import report may not produce the same curve. In this circumstance, only *.rdat import the same curve/trace. For other formats, the import produces two separated traces.

Exporting Files

You can export the following types of files from Ansys Electronics Desktop projects:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 2D model files
- 3D model files
- Graphics files
- Reports as data or graphics files
- Array Metadata
- Project Metadata

Exporting 2D Model Files

HFSS exports 2D models in AutoCAD *.dxf format. In this format, the geometry located within the XY plane is exported.

> **Note:**
>
> If you want to export a plane that does not coincide with the global xy plane, you must create a relative coordinate system to redefine the location of the origin.

To export a file to ***.dxf** format:

1. Click **Modeler** > **Export**.

   The **Export File** window appears.

2. Browse to and select a location to save your file.
3. In the **File Name** field, name the file.
4. From the **Save as Type** drop-down menu, select **AutoCAD DXF Files (*.dxf)**.
5. Click **Save**.

   The file is exported to the specified location with the appropriate file format.

Exporting 3D Model Files

You can export 3D models to 3D model file formats.

Supported 3D model file formats include the following:

Type	Extension(s)
ACIS SAB (exported with version 5.0)	*.sab
ACIS SAT (exported with version 5.0)	*.sat
AutoCAD	*.dxf
GDSII	*.gds
IGES	*.iges, *.igs
OBJ Files (for importing models to Ensight.)	*.obj

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type	Extension(s)
Parasolid	*.x_t, *.x_b
STEP	*.step, *.stp
STL	*.stl
GLTF (only available on Windows)	.gltf or .glb

To export a file to a 3D model format:

1. Click **Modeler** > **Export**.

   The **Export File** window appears.

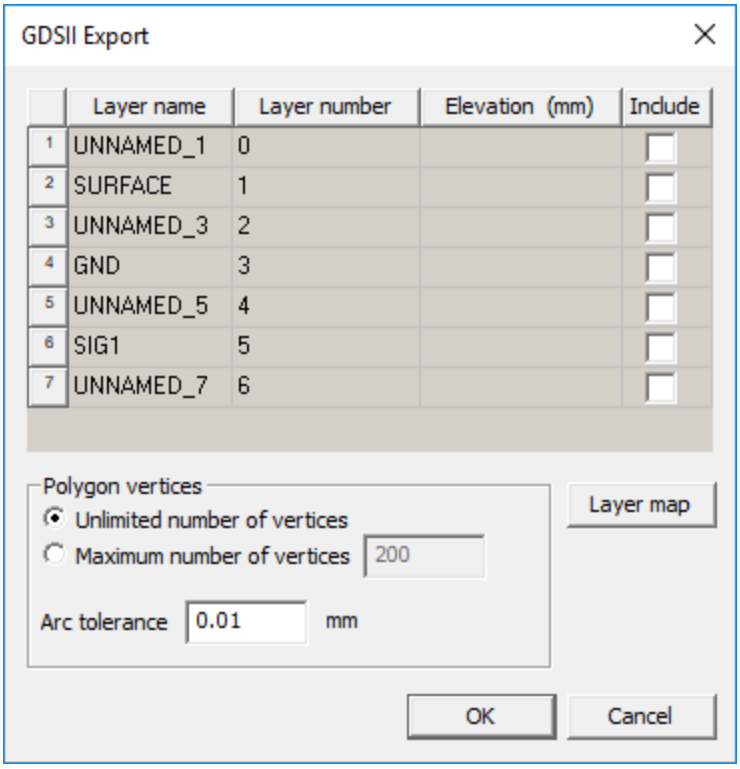
2. Browse to and select a location to save your file.
3. In the **File Name** field, name the file.
4. From the **Save as Type** drop-down menu, select the desired 3D file type.
5. Click **Save**.

   The file is exported to the specified location with the appropriate file format. Some file types launch an additional dialog box for specifying settings prior to export. These are described below.

GDSII Files

There are additional options for *.gds files:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

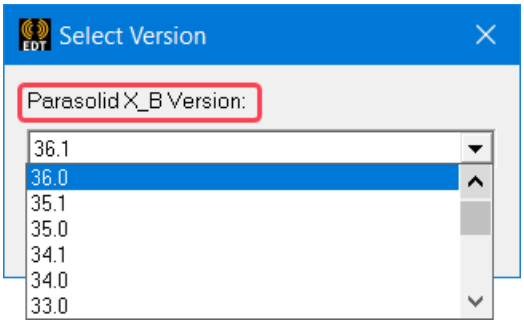


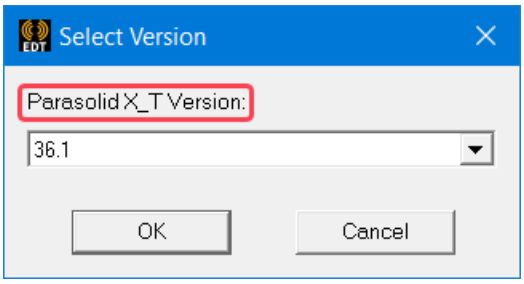
- Use the **Include** check boxes to select layers for export.
- Click **Layer map** to define a *.layermap file, if desired. This is a text file that maps GDSII layer numbers to layer names in the stackup. The *.layermap file can have the same format as the .tech file used in GDSII import, but it only needs the layer name and number. Any other information is ignored.
- In the **Polygon Vertices** area, select either **No Limit to the number of vertices** or **Limit the number of vertices** and specify a value.
- For **Arc Tolerance**, leave the default or specify a value.

Parasolid Files

There is one additional option for Parasolid files:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Use the drop-down menu to select a Parasolid version.

Exporting Graphics Files

You can export to the following graphics formats:

Extension	Contents
.bmp	Bitmap files
.gif	Graphics Interchange Format files
.jpg	Joint Photographics Experts Group files
.png	Portable Network Graphics format files
.tiff	Tagged Image File Format files.
.wrl	Virtual Reality Modeling Language (VRML) files.
.gltf or **.glb**	Graphics Language Transmission Format (GLTF); only available on Windows

To export a Modeler image to a graphics format:

1. Click **Modeler** > **Export**.

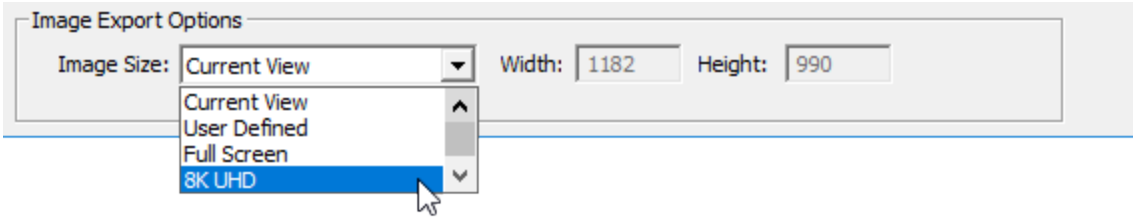
   The **Export File** window appears.

2. Browse to and select a location to save your file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **File Name** field, name the file.
4. From the **Save as Type** drop-down menu, select the desired image format.

   When you select an image format, **Image Export Options** appear. Note that this window does not appear for .gltf, .glb, or .wrl files, which are not image formats; if you are saving to one of these formats, skip to Step 6.



5. Choose your **Image Size**.

   Options include: Current View, User Defined (which allows you to specify **Width** and **Height**), Full Screen, 8K UHD, 4K UHD, 1080p HD, 720p HD, and 480p SD.

6. Click **Save**.

   The file is exported to the specified location with the appropriate file format.

You can also export an image file to a specified resolution using scripting commands. Fonts and line thickness are not scaled, only the image. You will have to iteratively increase font sizes until you find a suitable output.

Example Script:

```
oEditor.ExportModelImageToFile("C:/Users/Documents/highresexample_
image.jpg", 7680, 4320,

[

"NAME:SaveImageParams",

"ShowAxis:="        , "True",

"ShowGrid:="        , "True",

"ShowRuler:="       , "True",

"ShowRegion:="      , "Default",

"Selections:="      , ""

])
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

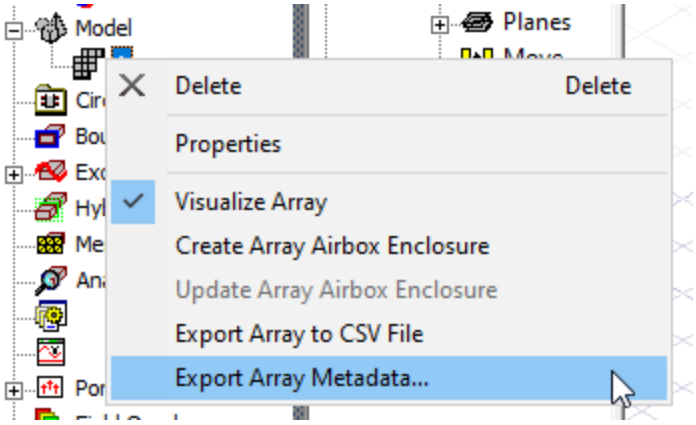
Exporting Array Metadata

In HFSS, you can assign a virtual array based on a unit cell object (native geometry or 3D component) and simulate it using distributed processing, treating the instances as parent and child objects. This allows for faster definition, visualization, and simulation of array-based designs, such as antenna arrays.

However, compared to array definition and simulation, array post-processing may require more effort from the user. For example, result data (such as far-field element patterns and S-parameters) commonly used in antenna array analysis needs to be exported to different workflows and file formats. The **Export Array Metadata** command allows you to export the array metadata you require.

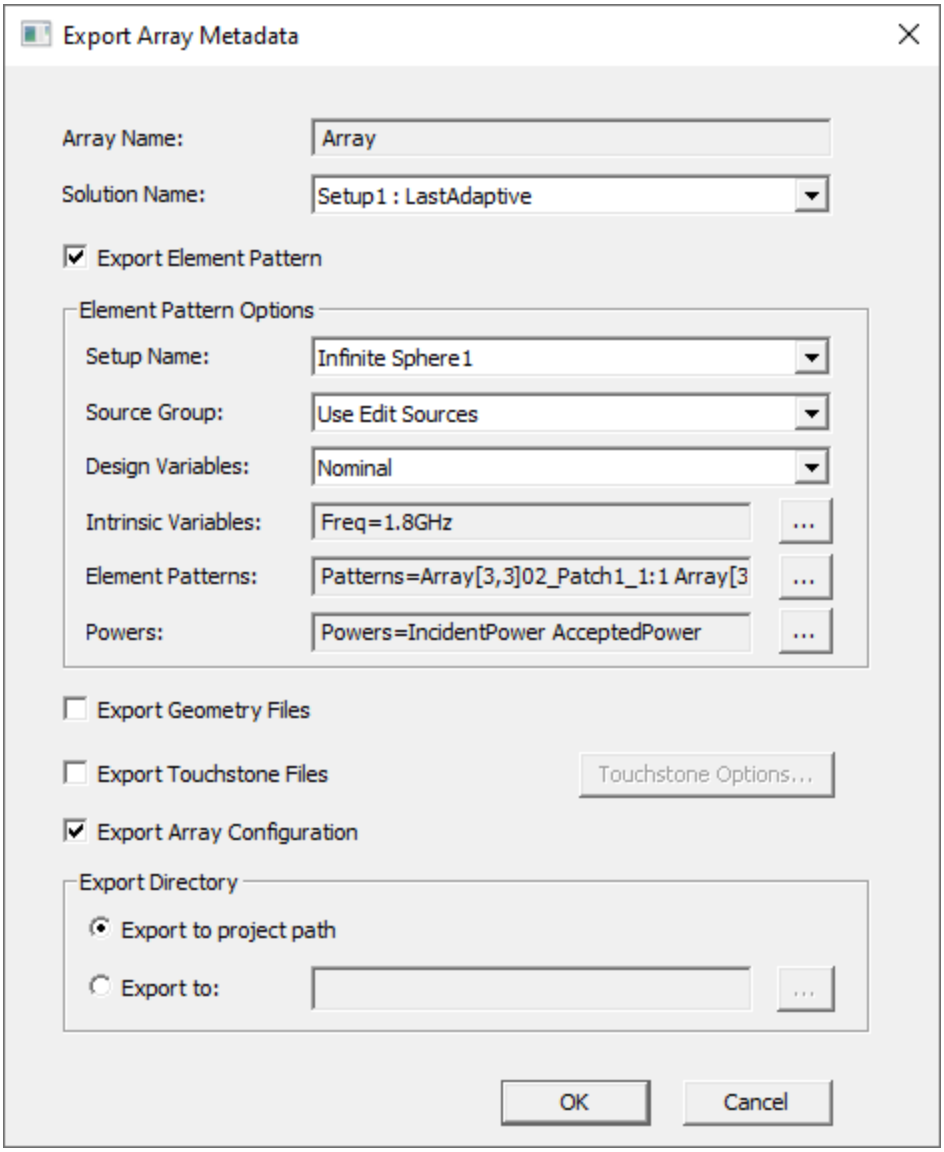
To export array metadata:

1. From the **Project Manager**, right-click and select **Export Array Metadata**.



The **Export Array Metadata** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. This window allows you to define export context, such as solution setup and variations, and to select metadata for export. Options include:

   - **Array Name**
   - **Solution Name**
   - **Export Element Pattern** – this check box must be enabled to select additional Element Pattern Options:
     - Setup Name
     - Source Group
     - Design Variables

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - Intrinsic Variables
    - Element Patterns (full or subset)
    - Powers – select from Incident, Accepted, System, and Radiated power.
  - **Export Geometry Files** – export *.obj files.
  - **Export Touchstone Files** – export *.sNp files. The **Touchstone Options** button launches the **SYZ Data Export Options** window, which allows you to select either Touchstone or Broadband export and specify additional export options.
  - **Export Array Configuration**
  - **Export Directory Options** – either generate metadata files in the project directory, or browse and select a location to save the file.
3. When you have selected the appropriate options, click **OK**.

   The metadata is exported to the specified location.

Export File Specifications

After defining export settings in the **Export Metadata** window, the data is exported to a metadata folder based on export directory settings. The overall file structure inside the exported directory is shown below:

```
${METADATA_DIR}
   └── ${DESIGN_NAME}.xml
   └── ${DESIGN_NAME}.metadata/
      └── ${DESIGN_NAME}.obj
      └── ${DESIGN_NAME}.mtl
      └── ${SOLUTION_NAME}.s4p
      └── element_pattern
        ├── exportfield_0.ffd
        ├── exportfield_1.ffd
```

Examples of an XML header file, and *.s4p file and an *.obj file follow.

Example XML Header File:

```
<ElementPatterns>
  <Source name='Array[4,4]3DC_Cell_Radome_In1_1:1'>
    <Filename>exportfield_0.ffd </Filename>
      <PortLocation>0.000857,0.001142,0.000127 </PortLocation>
      <ReferenceLocation>0.000000,0.000000,0.000000
      </ReferenceLocation>
                <PowerInfo>
          <Power Freq='77GHz'>
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
            <IncidentPower>1.000000 </IncidentPower>
            <AcceptedPower>0.837657 </AcceptedPower>
          </Power>
               </PowerInfo>
      </Source>
      <Source name='Array[4,5]3DC_Cell_Radome_In1_1:1'>
               <Filename>exportfield_1.ffd </Filename>
               <PortLocation>0.000857,-0.000572,0.000127
        </PortLocation>
               <ReferenceLocation>0.000000,-0.001714,0.000000
        </ReferenceLocation>
      <PowerInfo>
               <Power Freq='77GHz'>
          <IncidentPower>1.000000 </IncidentPower>
          <AcceptedPower>0.725554 </AcceptedPower>
               </Power>
      </PowerInfo>
    </Source>
  </ElementPatterns>
  <GeometryInfo>
    <ObjectFilename>2X2_MultiCell_CA-Array.obj </ObjectFilename>
    <MTLFilename>2X2_MultiCell_CA-Array.mtl </MTLFilename>
    <ModelUnit>mm </ModelUnit>
  </GeometryInfo>
  <TouchstoneInfo>
    <Filename>Setup1_DV400.s4p </Filename>
  </TouchstoneInfo>
</Metadata>
```

Example Touchstone .s4p File:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
! Touchstone file exported from HFSS 2024.1.0
!       File:           C:/Users/chjiang/OneDrive - ANSYS, Inc/Workspace/Feature/24R1/Array_meta/test_base1.aedt
!       Generated:      1:03:21 PM  May 09, 2023
!       Design:         2X2_MultiCell_CA-Array
!       Project:        test_base1
!       Setup:          Setup1
!       Solution:       LastAdaptive
!
! Variables:
!       ScanAnglePhi = 0deg
!       ScanAngleTheta = 30deg
!       ScanFrequency = 77000000000Hz
!       ScanMag1 = 1W
!       ScanPhase1 = 0deg
!       ScanPhaseShiftA = -((ScanFrequency*-3.59287710049e-11*sin(ScanAngleTheta)*cos(ScanAnglePhi)) + (ScanFrequency*0.0*sin(ScanAngleTheta)*sin(ScanAnglePhi))) rad
!       ScanPhaseShiftB = -((ScanFrequency*0.0*sin(ScanAngleTheta)*cos(ScanAnglePhi)) + (ScanFrequency*-3.59287710049e-11*sin(ScanAngleTheta)*sin(ScanAnglePhi))) rad
!
# GHz S MA R 50.000000
! Terminal data exported
! Port[1] = Array[4,4]3DC_Cell_Radome_Inl_1
! Port[2] = Array[4,5]3DC_Cell_Radome_Inl_1
! Port[3] = Array[5,4]3DC_Cell_Radome_Inl_1
! Port[4] = Array[5,5]3DC_Cell_Radome_Inl_1
77                              0.366064159766482 39.9213181550772 0.183993466485272 124.190379299152 0.200486982990943 116.476502049391 0.150813734783173 -34.0773205827
                                0.183993450987674 124.190374706296 0.257681803192874 40.2112353272733 0.153136852819726 -34.9441149862702 0.231204204488984 117.137287560294
                                0.200486967476455 116.476502344132 0.153136846503967 -34.9441136306449 0.368203660628769 40.1210892525329 0.103223659647427 124.226399056572
                                0.150813730638676 -34.0773260510378 0.231204200538537 117.137287602124 0.183223658330948 124.226399946906 0.258651818000565 39.959130486095
! Gamma          0               0               0               0               0               0               0               0               0
! Port Impedance         50              0               50              0               50              0               50              0
```

Example *.obj File:

```
#
mtllib obj_Test.mtl
usemtl ff8faf8f
v -3.428571e+00 -3.428571e+00 2.427352e+00
v -5.142857e+00 -3.428571e+00 2.427352e+00
v -3.428571e+00 -5.142857e+00 2.427352e+00
v -5.142857e+00 -5.142857e+00 2.427352e+00
vn 0.000000 0.000000 1.000000
vn 0.000000 0.000000 1.000000
vn 0.000000 0.000000 1.000000
vn 0.000000 0.000000 1.000000
f -4//-4 -3//-3 -2//-2
f -2//-2 -3//-3 -1//-1
v -5.142857e+00 -5.142857e+00 -9.733521e-01
v -5.142857e+00 -3.428571e+00 -9.733521e-01
v -3.428571e+00 -5.142857e+00 -9.733521e-01
v -3.428571e+00 -3.428571e+00 -9.733521e-01
vn -0.000000 -0.000000 -1.000000
vn -0.000000 -0.000000 -1.000000
vn -0.000000 -0.000000 -1.000000
vn -0.000000 -0.000000 -1.000000
f -4//-4 -3//-3 -2//-2
f -2//-2 -3//-3 -1//-1
v -5.142857e+00 -5.142857e+00 2.427352e+00
v -5.142857e+00 -5.142857e+00 -9.733521e-01
v -3.428571e+00 -5.142857e+00 2.427352e+00
v -3.428571e+00 -5.142857e+00 -9.733521e-01
vn -0.000000 -1.000000 -0.000000
vn -0.000000 -1.000000 -0.000000
vn -0.000000 -1.000000 -0.000000
vn -0.000000 -1.000000 -0.000000
```

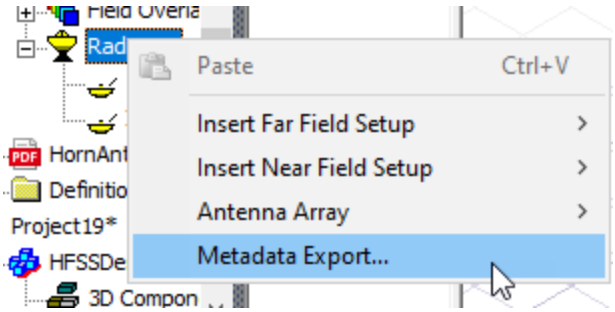

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Exporting Metadata

For cases in which post-processing may require greater effort, the **Metadata Export** command allows you to export project metadata.

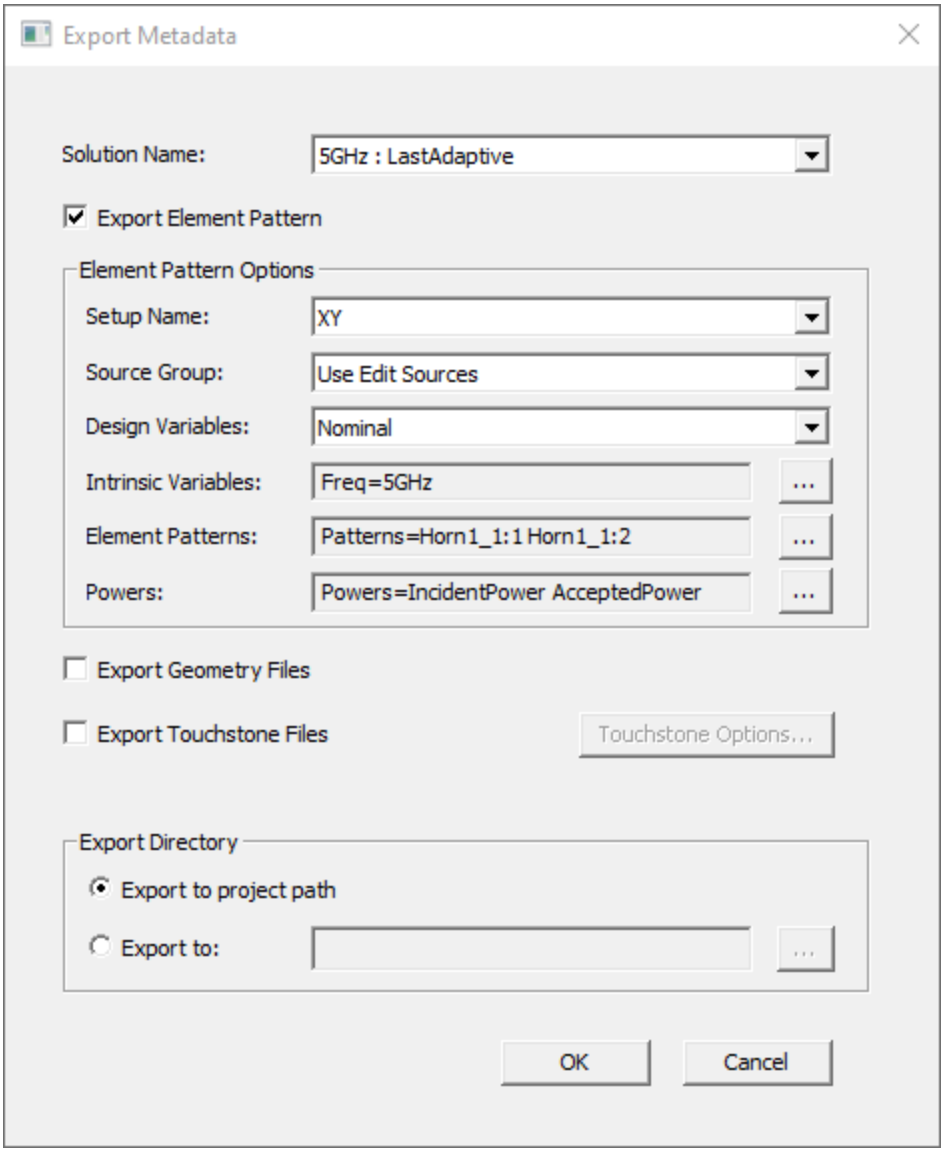
To export metadata:

1. From the **Project Manager**, right-click **Radiation** and select **Metadata Export**.



The **Export Metadata** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. This window allows you to define export context, such as solution setup and variations, and to select metadata for export. Options include:

   - **Solution Name**
   - **Export Element Pattern** – this check box must be enabled to select additional Element Pattern Options:
     - Setup Name
     - Source Group
     - Design Variables
     - Intrinsic Variables

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Element Patterns (full or subset)
- Powers – select from Incident, Accepted, System, and Radiated power.

- **Export Geometry Files** – export *.obj files.
- **Export Touchstone Files** – export *.sNp files. The **Touchstone Options** button launches the **SYZ Data Export Options** window, which allows you to select either Touchstone or Broadband export and specify additional export options.
- **Export Directory Options** – either generate metadata files in the project directory, or browse and select a location to save the file.

3. When you have selected the appropriate options, click **OK**.

   The metadata is exported to the specified location.

Export File Specifications

After defining export settings in the **Export Metadata** window, the data is exported to a metadata folder based on export directory settings. The overall file structure inside the exported directory is shown below:

```
${METADATA_DIR}
   └── ${DESIGN_NAME}.xml
   └── ${DESIGN_NAME}.metadata/
      └── ${DESIGN_NAME}.obj
      └── ${DESIGN_NAME}.mtl
      └── ${SOLUTION_NAME}.s4p
      └── element_pattern
         ├── exportfield_0.ffd
         ├── exportfield_1.ffd
```

Examples of an XML header file, and *.s4p file and an *.obj file follow.

Example XML Header File:

```
<ElementPatterns>
  <Source name='Array[4,4]3DC_Cell_Radome_In1_1:1'>
    <Filename>exportfield_0.ffd </Filename>
      <PortLocation>0.000857,0.001142,0.000127 </PortLocation>
      <ReferenceLocation>0.000000,0.000000,0.000000
      </ReferenceLocation>
                <PowerInfo>
          <Power Freq='77GHz'>
            <IncidentPower>1.000000 </IncidentPower>
            <AcceptedPower>0.837657 </AcceptedPower>
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
            </Power>
                  </PowerInfo>
      </Source>
      <Source name='Array[4,5]3DC_Cell_Radome_In1_1:1'>
                  <Filename>exportfield_1.ffd </Filename>
                  <PortLocation>0.000857,-0.000572,0.000127
        </PortLocation>
                  <ReferenceLocation>0.000000,-0.001714,0.000000
        </ReferenceLocation>
      <PowerInfo>
                  <Power Freq='77GHz'>
            <IncidentPower>1.000000 </IncidentPower>
            <AcceptedPower>0.725554 </AcceptedPower>
                  </Power>
      </PowerInfo>
    </Source>
  </ElementPatterns>
  <GeometryInfo>
    <ObjectFilename>2X2_MultiCell_CA-Array.obj </ObjectFilename>
    <MTLFilename>2X2_MultiCell_CA-Array.mtl </MTLFilename>
    <ModelUnit>mm </ModelUnit>
  </GeometryInfo>
  <TouchstoneInfo>
    <Filename>Setup1_DV400.s4p </Filename>
  </TouchstoneInfo>
</Metadata>
```

Example Touchstone .s4p File:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
! Touchstone file exported from HFSS 2024.1.0
!       File:           C:/Users/chjiang/OneDrive - ANSYS, Inc/Workspace/Feature/24R1/Array_meta/test_base1.aedt
!       Generated:      1:03:21 PM  May 09, 2023
!       Design:         2X2_MultiCell_CA-Array
!       Project:        test_base1
!       Setup:          Setup1
!       Solution:       LastAdaptive
!
! Variables:
!       ScanAnglePhi = 0deg
!       ScanAngleTheta = 30deg
!       ScanFrequency = 77000000000Hz
!       ScanMag1 = 1W
!       ScanPhase1 = 0deg
!       ScanPhaseShiftA = -((ScanFrequency*-3.59287710049e-11*sin(ScanAngleTheta)*cos(ScanAnglePhi)) + (ScanFrequency*0.0*sin(ScanAngleTheta)*sin(ScanAnglePhi))) rad
!       ScanPhaseShiftB = -((ScanFrequency*0.0*sin(ScanAngleTheta)*cos(ScanAnglePhi)) + (ScanFrequency*-3.59287710049e-11*sin(ScanAngleTheta)*sin(ScanAnglePhi))) rad
!
# GHz S MA R 50.000000
! Terminal data exported
! Port[1] = Array[4,4]3DC_Cell_Radome_Inl_1
! Port[2] = Array[4,5]3DC_Cell_Radome_Inl_1
! Port[3] = Array[5,4]3DC_Cell_Radome_Inl_1
! Port[4] = Array[5,5]3DC_Cell_Radome_Inl_1
77                              0.366064159766482 39.9213181550772 0.183993466485272 124.190379299152 0.200486982990943 116.476502049391 0.150813734783173 -34.0773205827
                                0.183993450987674 124.190374706296 0.257681803192874 40.2112353272733 0.153136852819726 -34.9441149862702 0.231204204488984 117.137287560294
                                0.200486967476455 116.476502344132 0.153136846503967 -34.9441136306449 0.368203660628769 40.1210892525329 0.103223659647427 124.226399056572
                                0.150813730638676 -34.0773260510378 0.231204200538537 117.137287602124 0.183223658330948 124.226399946906 0.258651818000565 39.959130486095
! Gamma          0               0               0               0               0               0               0               0               0
! Port Impedance         50              0               50              0               50              0               50              0
```

Example *.obj File:

```
#
mtllib obj_Test.mtl
usemtl ff8faf8f
v -3.428571e+00 -3.428571e+00 2.427352e+00
v -5.142857e+00 -3.428571e+00 2.427352e+00
v -3.428571e+00 -5.142857e+00 2.427352e+00
v -5.142857e+00 -5.142857e+00 2.427352e+00
vn 0.000000 0.000000 1.000000
vn 0.000000 0.000000 1.000000
vn 0.000000 0.000000 1.000000
vn 0.000000 0.000000 1.000000
f -4//-4 -3//-3 -2//-2
f -2//-2 -3//-3 -1//-1
v -5.142857e+00 -5.142857e+00 -9.733521e-01
v -5.142857e+00 -3.428571e+00 -9.733521e-01
v -3.428571e+00 -5.142857e+00 -9.733521e-01
v -3.428571e+00 -3.428571e+00 -9.733521e-01
vn -0.000000 -0.000000 -1.000000
vn -0.000000 -0.000000 -1.000000
vn -0.000000 -0.000000 -1.000000
vn -0.000000 -0.000000 -1.000000
f -4//-4 -3//-3 -2//-2
f -2//-2 -3//-3 -1//-1
v -5.142857e+00 -5.142857e+00 2.427352e+00
v -5.142857e+00 -5.142857e+00 -9.733521e-01
v -3.428571e+00 -5.142857e+00 2.427352e+00
v -3.428571e+00 -5.142857e+00 -9.733521e-01
vn -0.000000 -1.000000 -0.000000
vn -0.000000 -1.000000 -0.000000
vn -0.000000 -1.000000 -0.000000
vn -0.000000 -1.000000 -0.000000
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

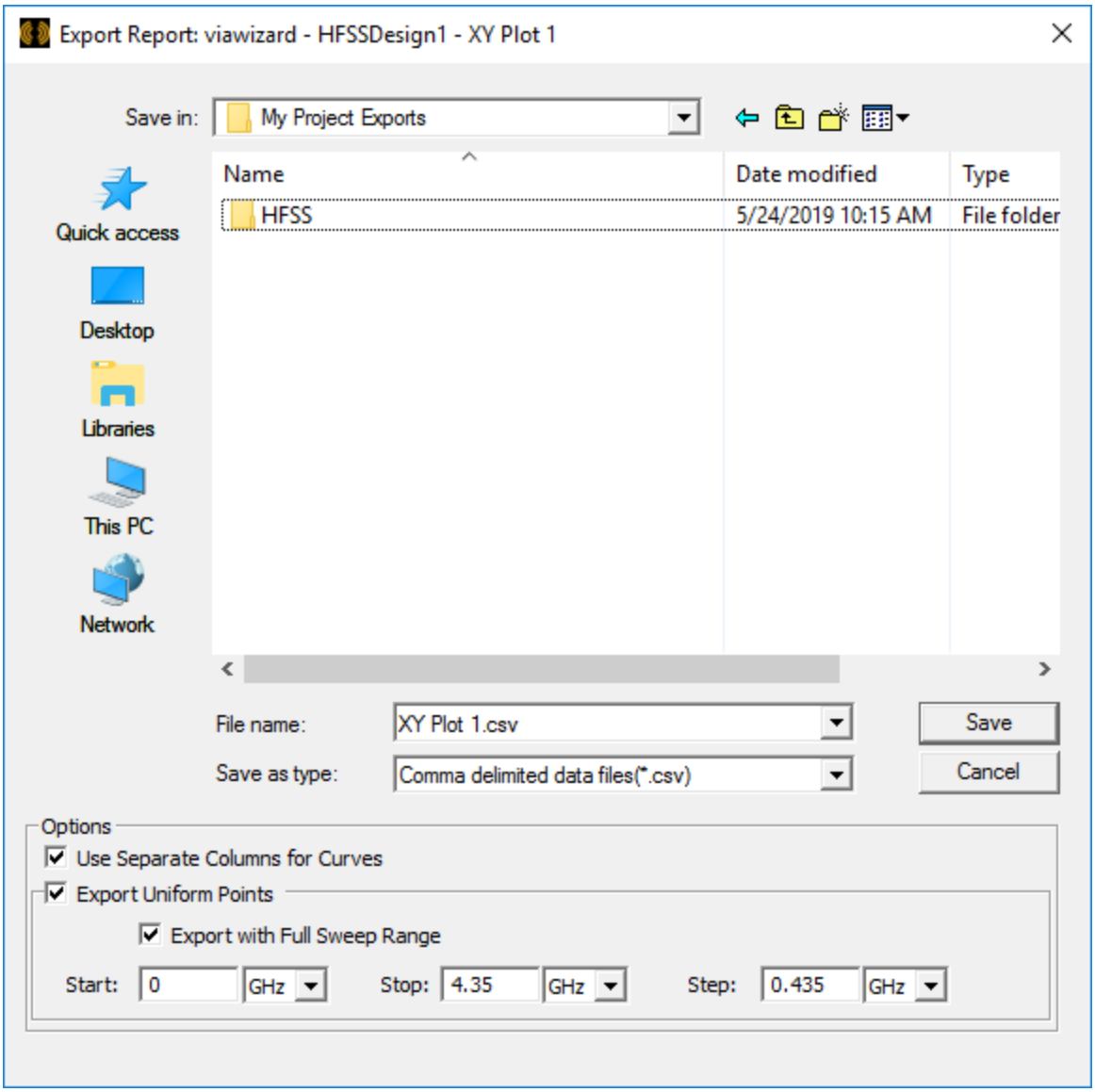
Exporting Reports

You can export reports in a variety of text and graphic formats. You must have an existing plot open to see the corresponding **Report2D** or **Report3D** menu.

1. Click **Report2D**> **Export** or **Report3D**> **Export**.

   Alternatively, right-click the plot and select **Export**.

   The *Export Report* dialog box appears.



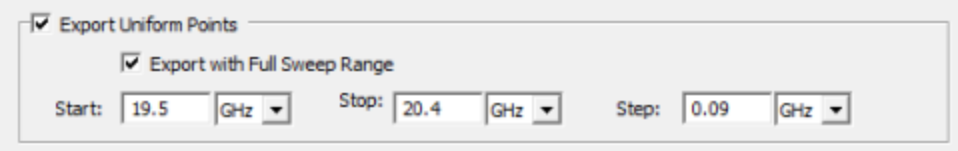
2. Use the file browser to find the directory where you want to save the file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Type the name of the file in the **File name** field.
4. Select one of the following file formats from the **Save as type** drop-down menu:

Extension	Contents
.csv	Comma-delimited data file
.tab	Tab-separated file
.dat	Ansoft plot data file
.txt	Post processor format file
.rdat	Ansoft report data file
.emf	Microsoft EMF files
.gif	GIF files
.bmp	BMP files
.wrl	VRML files
.tif, **.tiff**	TIFF files
.jpg, **.jpeg**	JPEG files

   - When exporting to *.csv or .tab files for an XY Plot, Eye Diagram or Data Table, a **Separate Columns for Curves** check box will be shown. Otherwise the check box will be hidden.
   - When it is checked the data is exported with one column per curve, and variable values appear in the column's header cell.
   - When it is unchecked the data is exported with one column per a trace, each variable has its own column.
   - When the design has many variables, uncheck the check box may keep the column header from becoming too long and hard to read.
   - When trace has more curves than number of sweep values, uncheck the check box to prevent the number of columns from being larger than the number of rows.
   - When a report only has one trace or all trace have same variations, this option is unchecked by default. Otherwise, this feature is checked.
   - For 2D reports, you can select **Export Uniform Points** and specify a full sweep range by editing start, stop, and step values.



5. Click **Save**. The report is exported to the specified location.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

For a report trace where the primary sweep is NOT the same as the x-component, exporting and then importing a report may not produce the same curve. In this circumstance, only *.rdat format imports the same curve/trace. For other formats, the import produces two separated traces.

Exporting Options Files

Ansys Electronics Desktop can export user and host options files in XML format.

Exporting Options Files Using the Desktop UI

Export options files by selecting **Tools** > **Options** > **Export Options Files**. This brings up a browser dialog box to select the destination directory.

Click **Open** to copy all config files for the current user and current host to the specified directory. Config files for the install, install_machine, user, and user_machine levels will be copied, if they exist. One additional file, admin.XML, will also be copied to the destination directory; this file does not contain user configurable options.

Exporting Options Files Using a Script

A script command has been added that exports options config files:

ExportOptionsFiles

Syntax	`ExportOptionsFiles <DestinationDirectory>`
Return Value	None.
Parameters	BSTR <DestinationDirectory>

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VBScript Example	`Dim oAnsoftApp`
	`Dim oDesktop`
	`Set oAnsoftApp = CreateObject ("Ansoft.ElectronicsDesktop")`
	`Set oDesktop = oAnsoftApp.GetAppDesktop()`
	`oDesktop.ExportOptionsFiles "D:/test/export/"`

IPY Example	`oDesktop.ExportOptionsFiles ('D:/test/export')`

Validating Projects

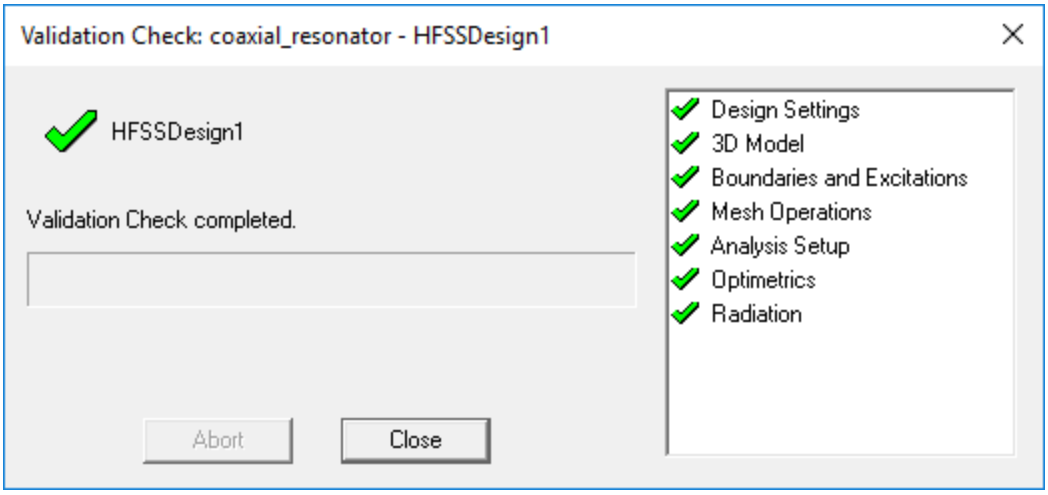
Before running an analysis on a model, it is important to first perform a validation check on the project. During a validation check, the solver examines the setup details of the active project to verify that all the necessary steps have been completed and that their parameters are reasonable.

To perform a validation check on the active project:

1. Click **HFSS** > **Validation Check**. You can also click the **Validate** icon on the **Simulation** ribbon tab.

   Electronics Desktop checks the project setup, and the **Validation Check** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

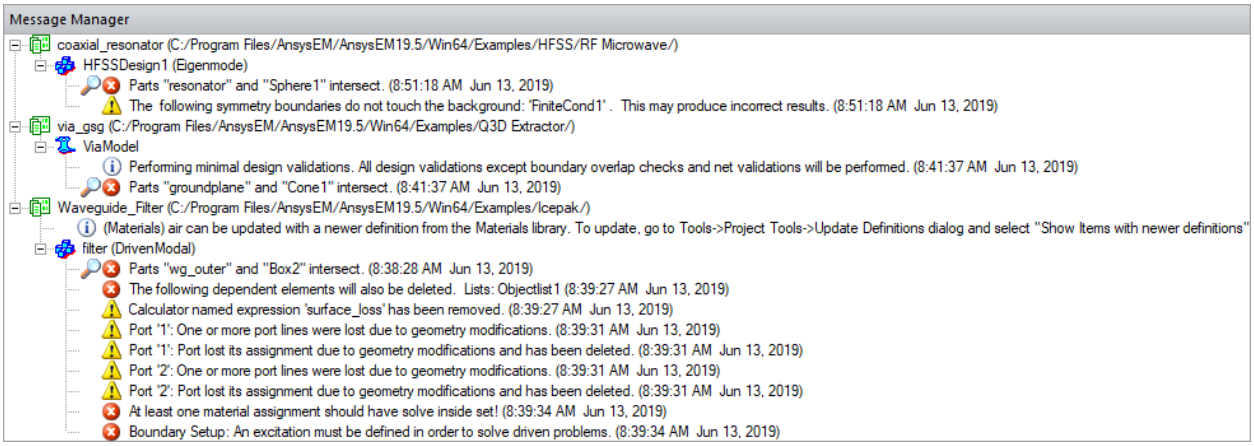


2. View the results of the validation check in the **Validation Check** window.

   The following icons can appear next to an item:

	Indicates the step is complete.
	Indicates the step is incomplete.
	Indicates the step may require your attention.

3. View any messages in the **Message Manager** window.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If the validation check indicates that a step in your project is incomplete or incorrect, carefully review the setup details for that particular step and revise them as necessary.
5. After you have revised any setup details, click **HFSS** > **Validation Check** to run another validation check. Repeat until the Validation Check completes with no issues.

   > **Note:**
   >
   > You can change the strictness with which 3D Model faults are validated.

6. Click **Close**.

Modeler Validation Settings

You can adjust the degree to which the software checks a model for faults that could jeopardize mesh accuracy.

There are three levels of model validation for a given design: **Warning Only**, **Basic**, and **Strict**.

> **Note:**
>
> This setting only affects the "3D Model" stage of a design validation.

- **Warning Only** – allows all models to pass 3D Model validation regardless of any faults that are found (acis_entity check errors). These faults are posted in the message window as warnings.
- **Basic** – allows most models to pass 3D Model validation. This excuses non-manifold errors and most acis_entity_check errors. Some faults are flagged as model errors (basic entity check errors), thereby prohibiting a design from proceeding to the meshing stage of an analysis. You must either correct such errors before attempting to analyze the design under the **Basic** setting, or change the **Model Validation** level to **Warning Only**.
- **Strict** – enforces a tighter tolerance for model faults than the **Warning Only** and **Basic** settings. All model faults found during 3D Model validation are posted to the message window. These errors must be corrected before attempting to analyze the design under the **Strict** setting, otherwise you must change the **Model Validation** level to **Basic** or **Warning Only**.

To set the Model Validation level:

1. Select **HFSS** > **Design Settings**.

   The **Design Settings** dialog box appears. The **Validations** tab lets you set the validation as **Basic**, **Strict**, or **Warning Only** and includes solver-specific selections.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Choose the desired level of validation from the **Entity Check Level** drop-down menu.

   You can also click the **Save as Default** button to make the current selection the default, or select **Restore Default**.

3. Click **OK** to accept the selection and close the dialog box.

> **Note:** For more information on Maxwell-specific settings, see Maxwell Validation Tab.

Copying and Pasting a Project or Design

To **copy** a project or design:

1. In the Project Tree, select a project or design to enable the menu command **Edit** > **Copy**.
2. Click **Edit** > **Copy**.

   The project or design is copied for pasting.

To **paste** a project or design:

1. In the Project Tree, select a project or design to enable the menu command **Edit** > **Paste**.
2. Click **Edit** > **Paste**.

   The project or design is pasted under the selected project, and an icon is added to the project tree.

> **Note:**
>
> You can also use keyboard shortcuts.

Renaming Projects

In general, use the **File** menu commands to manage projects. If you move or change the names of files without using these commands, the software may not be able to find information necessary to solve the model.

To rename an existing, active project:

1. Select the project in the Project Tree.
2. Right-click and select **Rename**.

   This activates the text field for the project name.

3. Type the new project name and press **Enter**.

   The new project name appears in the directory and the project remains in the original location.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Archiving Projects

Use the **File** > **Archive** command to place a project and any other files related to the project that you want to include in a *.<product>z file or *.zip format archive. You can make notes about the contents of the archive and specify whether to include results and solutions files. The **Archive** command attempts to automatically detect the necessary files for linked projects and automatically include them in the archive. You can also add additional files to the archive, including results files, external files and projects. For example, if a project linked to the main project also has linked or associated files, you can add them.

Archive File Types

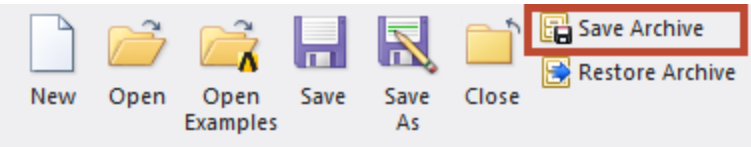
Internally, project archive files are .zip files, and are compatible with any program that can read .zip files. Project archive files have an extension that is unique for each product, and is generated by adding a 'z' to the project file extension (for example, *.aedtz, *.adsnz). This extension displays as the default when saving and restoring archive files.

Archive Preview

The **Archive** command includes a preview feature that lets you review the contents of a planned archive.

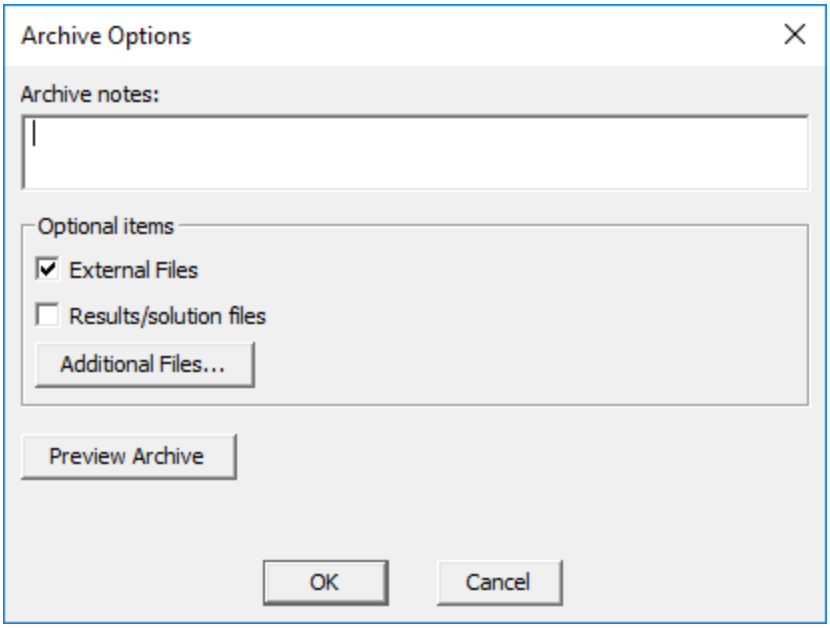
To archive the current project:

1. Click **File** > **Archive** or select the **Desktop** tab of the ribbon and click **Save Archive**.



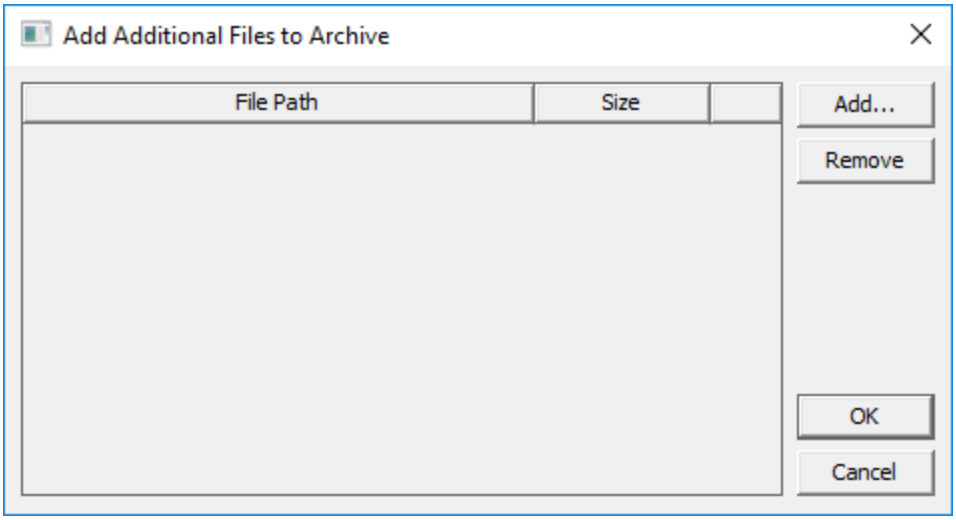
The **Archive Options** dialog box opens.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Archive Notes** – in this field, you can specify notes that will be visible when previewing the archive. These notes can be viewed from the preview dialog box without restoring the archive.
- **External Files** – selecting this check box causes all external files to be included in the archive. These include any existing files associated with the project, such as linked files, or files added through the **Project** > **Insert Documentation File** command or **Project** > **Data Set** command.
- **Results/Solution Files** – selecting this check box causes the entire results directory to be included in the archive. This may greatly increase the size of the archive file.
- **Additional Files** – clicking this opens the **Add Additional Files to Archive** dialog box.

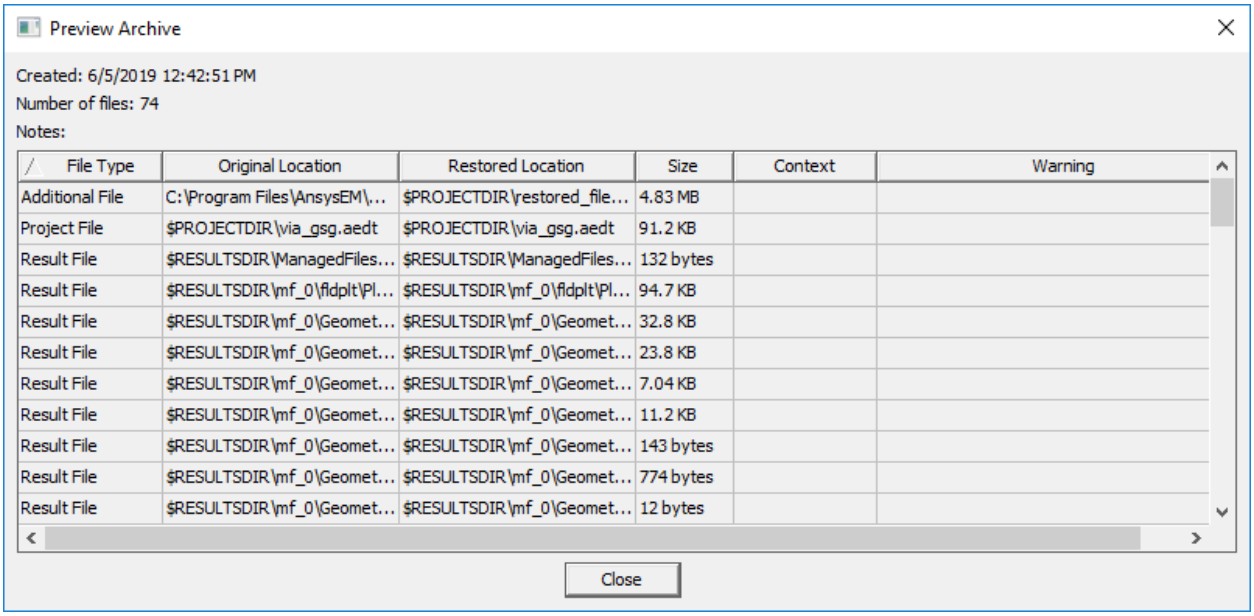
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can click **Add** to browse and locate additional files you want to include in the archive. You can select and the **Remove** any files listed.

You can click **OK** to accept changes or **Cancel** any proposed changes.

2. Select any optional items, and make any desired notes in the text field.
3. When you have made your selections, click **Preview Archive** to view archive contents, file types, the locations of the archive files, and the locations where restoring from archive would place them. Additionally, there may be context or warning information.



To read longer locations, you can drag the column header to expand them.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Previewing an archive before creating the archive can be helpful in order to see exactly what files will be included in an archive, as well as how those files are being relocated. Another purpose of previewing an archive is to view warnings and consider if any additional files need to be added to the archive.

4. When you are ready to create the archive, **Close** the preview and click **OK**.

   A browsing window appears.

5. Specify a name for your file and select the format you want to use from the **Save as type** drop-down menu.
6. Click **OK** to create the archive.

File Relocation

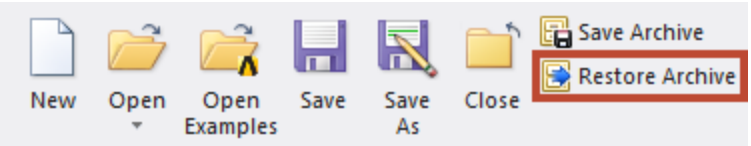
In a project to be archived, external files can be located anywhere on the user's system. One of the goals is for the restored project to be relatively self contained, and to NOT allow the restoring of an archived project to haphazardly write files anywhere on the restoring user's system.

To achieve this, it is sometimes necessary to change the location of files in the archived project so that the external files are located in the project directory. At archive time, any external files not located in the project directory are relocated to the restored_files subdirectory of the project directory in the archived project. Any external files located in the user library or system library will be relocated to the personal library directory. Note that the project file that is written into the archive will be updated to refer to the files at the new locations, and the original project file will remain unaltered.

Restoring an Archived Project

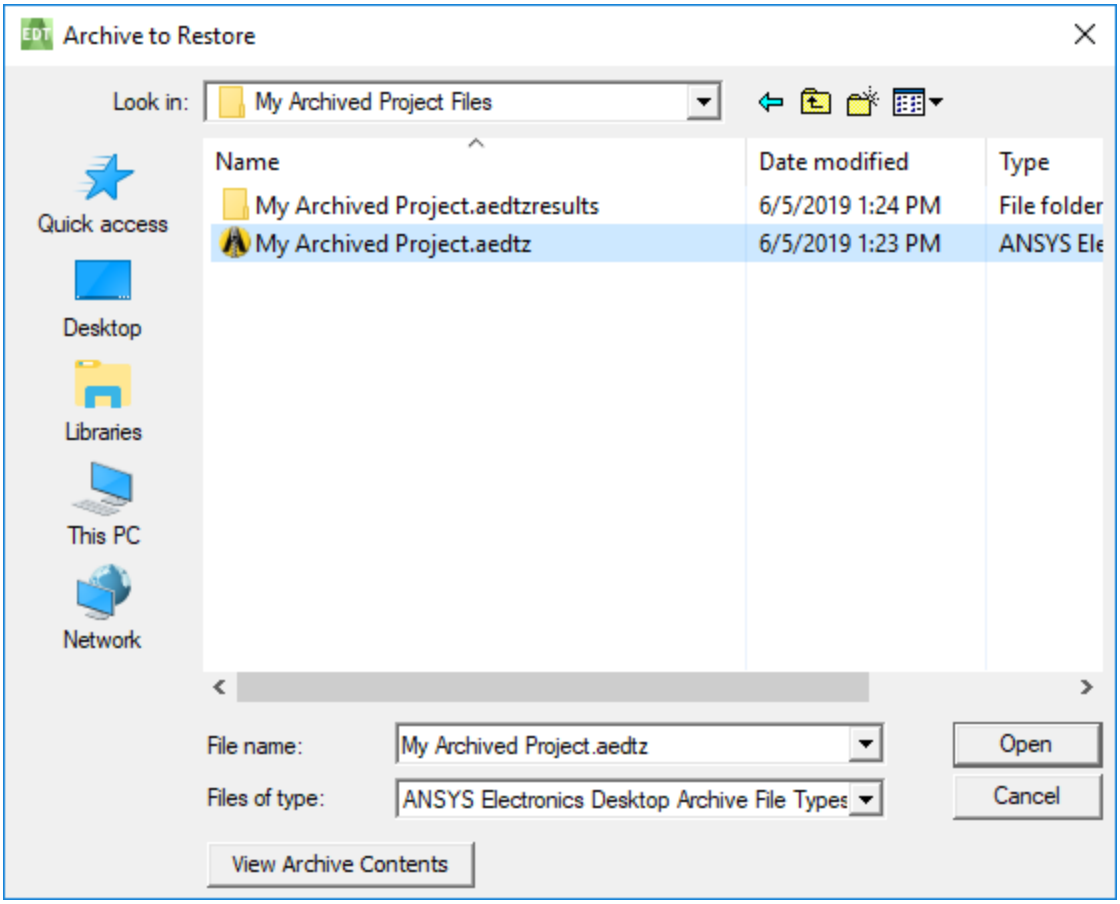
To restore an existing archive created with **File** > **Archive...**, use the **File** > **Restore Archive** command.

1. Click **File**> **Restore Archive** or select the **Desktop** tab of the ribbon and click **Restore Archive**.



   This displays an **Archive to Restore** browser window that lets you navigate your file system for *.aedtz, legacy *.*<solversuffix>z* or *.zip archive files.

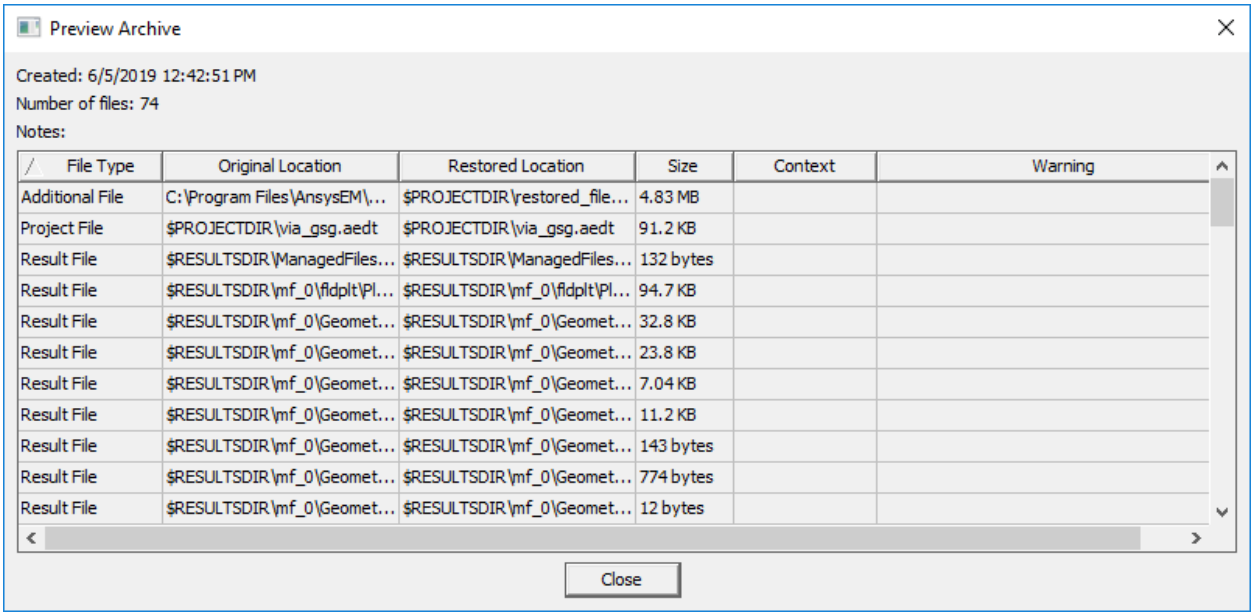
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. If you select a valid archive file, you can click **View Archive Contents** to preview the contents.

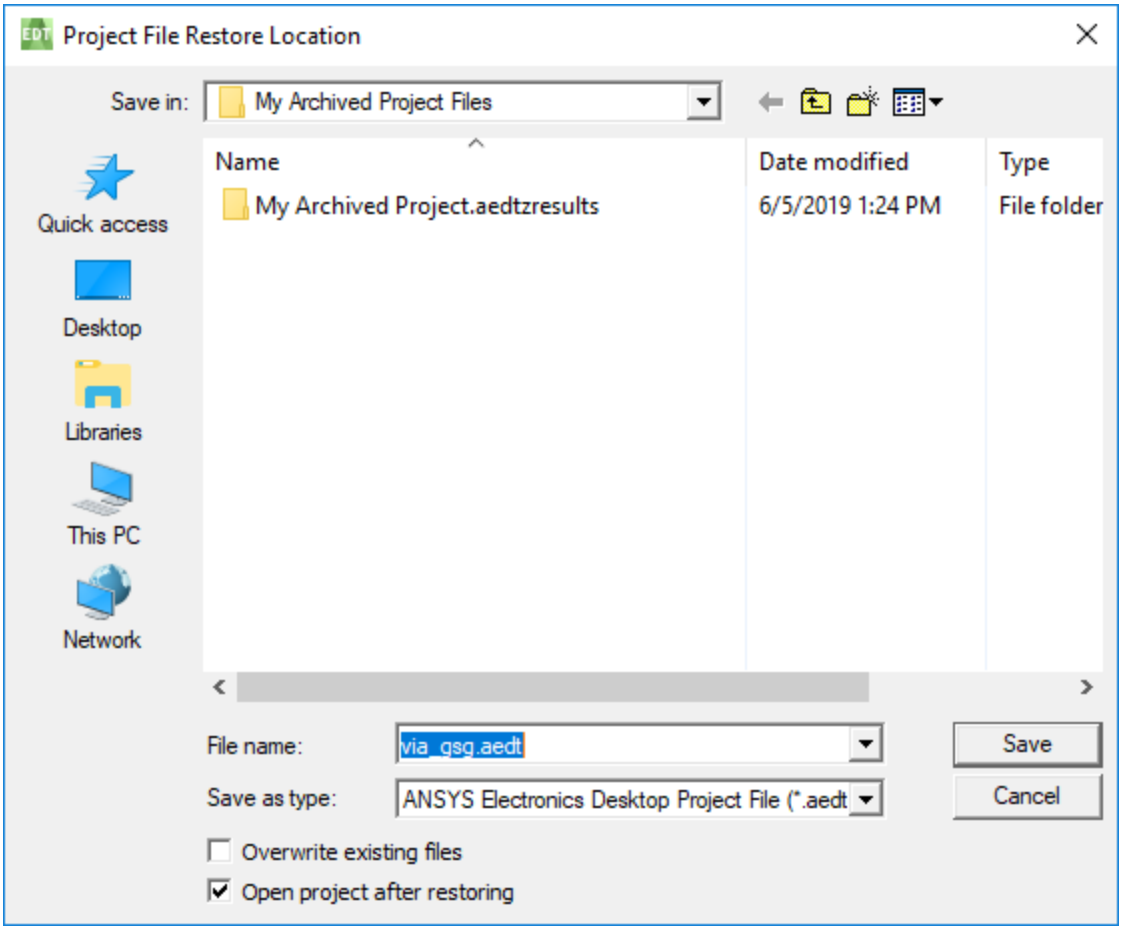
   The **Preview Archive** dialog box lists all files in the archive, notes, the original and restored locations, and any warnings that were generated at archive time. These warnings may be useful to identify additional steps that are needed to update any files to refer to files which had to be manually added to the archive.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click **Close** to exit the preview.
4. In the browsing window, click **Open** to change to a **Project File Restore Location** browser.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



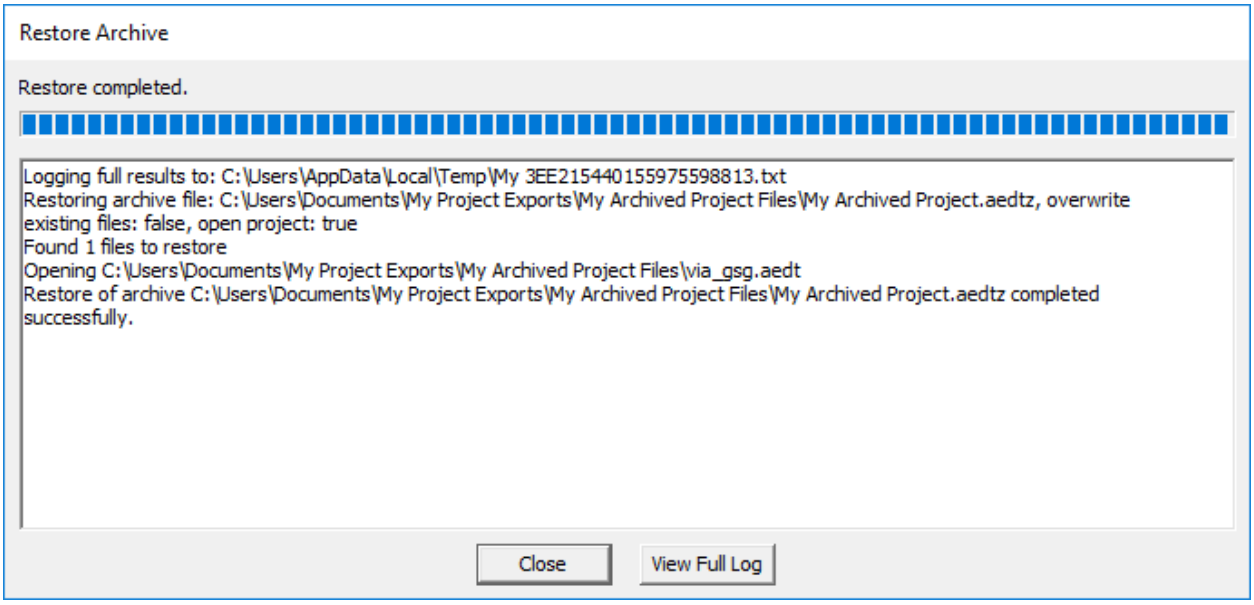
You can edit the file name, and select options:

- **Overwrite existing files** – If this check box is selected, restored files will automatically overwrite existing files during the restore process. If this button is deselected, existing files will not be modified.
- **Open project after restoring** – If this check box is selected, the project will be opened in this instance of the application after all files have been restored.

5. Click **Save** to restore the archived file.

   While restoring an archive, a dialog box displays showing restore results. A progress bar shows the relative progress, and the text window displays important information and warnings.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Electronics Desktop also generates a full log file that contains detailed information about the restore process. The first line in the text window displays the location of the full log file. After the restore has been completed, you can click **View Full Log** to open this log file, or use a text editor and open the file at the specified location.

Deleting a Project or Design

To delete a project or design:

1. In the Project Tree, select a project or design.
2. Click **Edit** > **Delete**, or press the **Delete** key.
3. Confirm the warning box to complete the delete operation.

   The project or design is removed from the Project tree.

Setting Read Only Designs

Designs can be set as either Read Only or Full Access. Full Access is the default status of a design and allows you to modify the design. In Read Only mode, you may only run the design or link it to another design. Setting a design as Read Only protects it from accidental modification.

Read Only designs are marked with a red lock in the Project Manager:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To change a single design's designation:

1. In the **Project Manager**, right-click the design.
2. Select either **Convert to Read Only** or **Convert to Full Access**.

To change the designation of *every design* in a project:

1. In the **Project Manager**, right-click the project.
2. Select either **Convert All Designs to Read Only** or **Convert All Designs to Full Access**.

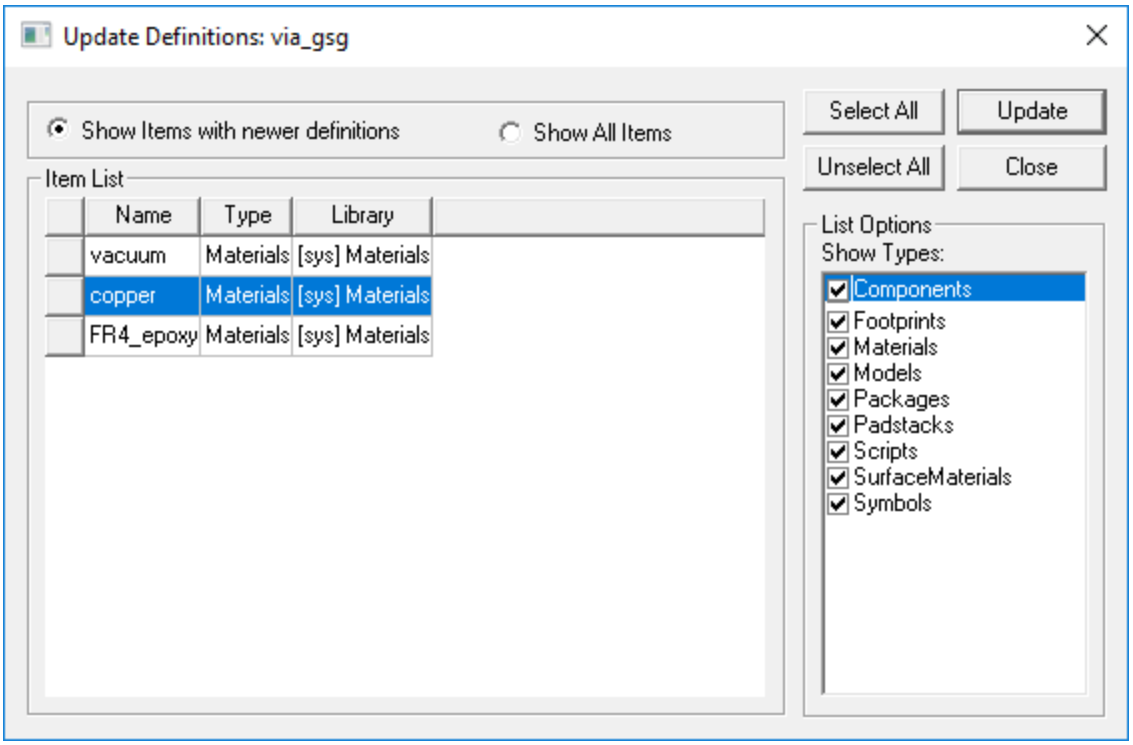
> **Note:**
>
> Read Only settings are not saved when you save the project file.

Updating Design Components

To update components defined in the current design:

1. Click **Tools** > **Project Tools** > **Update Definitions**.

   The **Update Definitions** dialog box appears.



2. Select either **Show Items with newer definitions** or **Show All Items**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. From the **Show Types** list in the **List Options** section, select the types of definitions you want to show in the **Item List**.
4. Select the item(s) you want to update from the **Item List**, or use **Select All.**
5. Click **Update**.

   A message appears telling you the update was successful.

6. Click **OK** to close the message.

When you are finished updating definitions, click **Close**.

Undoing and Redoing Commands

The **Edit** menu contains **Undo and** Redo commands.

Use the **Undo** command to cancel the last action you performed on the active project or design.

Use the **Redo** command to reperform an action you previously undid.

Both commands are useful for dealing with unintended actions related to project management, model creation, and post-processing.

To use **Undo** or **Redo**:

1. In the **Project Manager** window, do one of the following:
   - To undo or redo your last project-level action, such as inserting a design or adding project variables, click the project icon.
   - To undo or redo your last design-level action, such as drawing an object or deleting a field overlay plot, click the design icon.

   > **Note:**
   >
   > You cannot undo an analysis that you've performed on a model—that is, the **HFSS** > **Analyze** command.

2. Click **Edit** > [**Undo/Redo**], or click the **Undo** or **Redo** button ( ) on the **Desktop** tab.

   Your action is undone or redone.

> **Important:**
>
> When you save a project, Electronics Desktop clears the entire undo/redo history for the project and its designs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inserting a Documentation File

You may want to add a documentation file to the project tree.

1. Click **Project** > **Insert Documentation File**.

   This opens a file browser that lets you navigate your file system.

2. Select the documentation file and click **OK**.

   The documentation file is placed in the Project Tree.

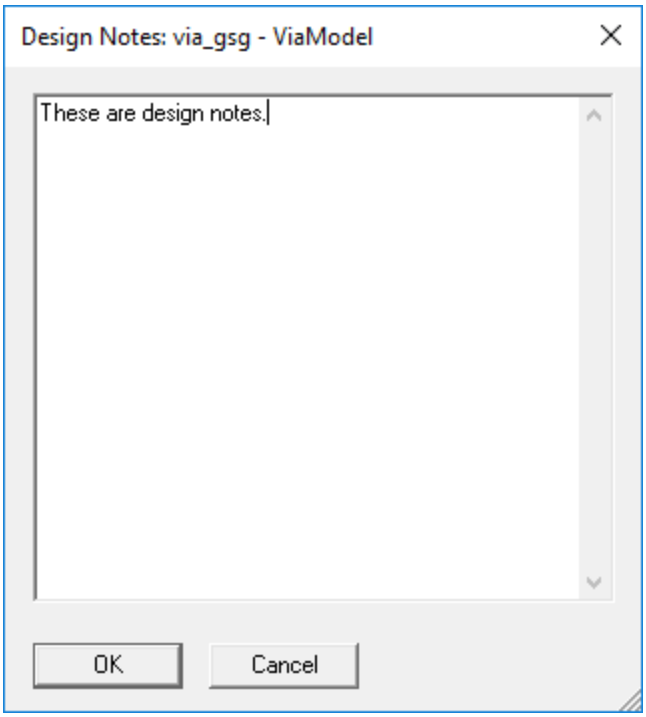
Working with Design Notes

You can save design notes, such as information about the creation date or a description of the device being modeled. This is useful for keeping a running log.

To add notes:

1. Click **HFSS** > **Edit Notes**.

   The **Design Notes** window appears.



2. Click in the window and type your notes.
3. Click **OK** to save the notes with the current project.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To edit notes, use the same window or double-click the **Notes** icon ( ) in the Project Tree.

To delete the existing notes for a design:

1. Select the **Notes** icon in the Project Tree.
2. Click **Edit** > **Delete**, or right-click and select **Delete**.

   The Notes icon is removed from the Project Tree.

> **Note:**
>
> Notes are used to document aspects of designs only.
>
> For project-level documentation, you can insert a documentation file into a project with the **Project** > **Insert Documentation File** command.

Working with Design Notes

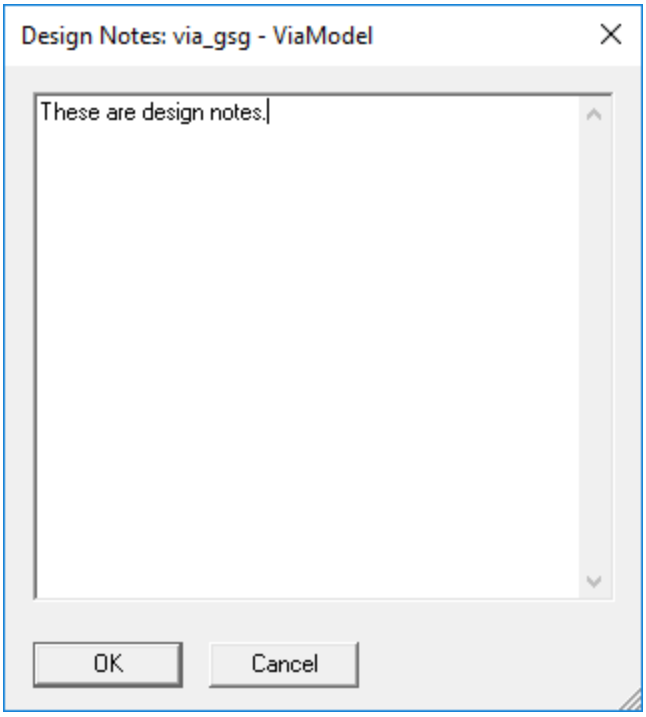
You can save design notes, such as information about the creation date or a description of the device being modeled. This is useful for keeping a running log.

To add notes:

1. Click **HFSS** > **Edit Notes**.

   The **Design Notes** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click in the window and type your notes.
3. Click **OK** to save the notes with the current project.

To edit notes, use the same window or double-click the **Notes** icon (Notes) in the Project Tree.

To delete the existing notes for a design:

1. Select the **Notes** icon in the Project Tree.
2. Click **Edit** > **Delete**, or right-click and select **Delete**.

   The Notes icon is removed from the Project Tree.

> **Note:**
>
> Notes are used to document aspects of designs only.
>
> For project-level documentation, you can insert a documentation file into a project with the **Project** > **Insert Documentation File** command.

Printing

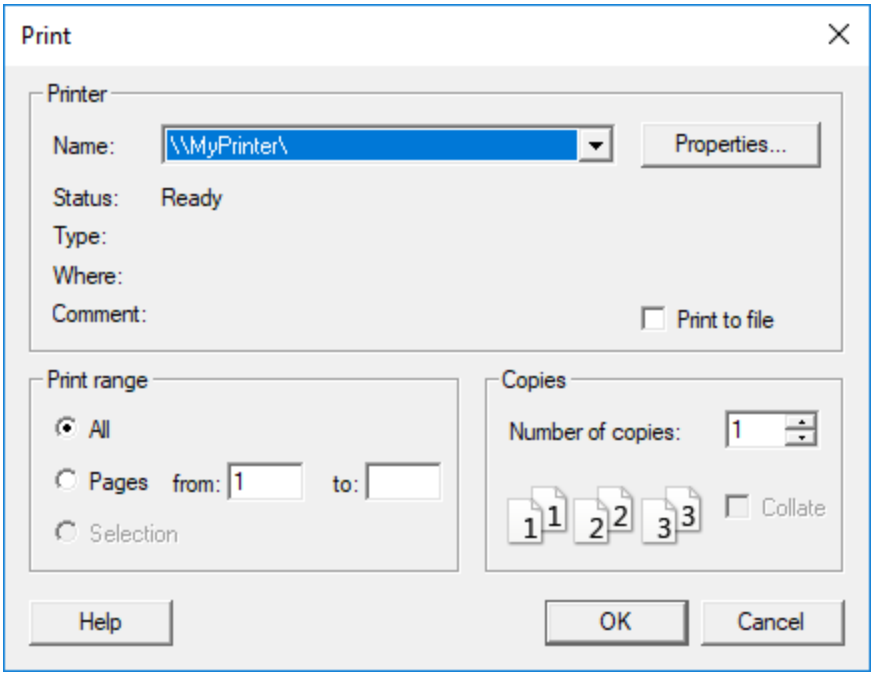
Electronics Desktop printing commands enable you to send an image of the active window to the printer.

To print the project:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **File** > **Print**.

   A dialog box similar to the following one appears:



2. Print options will vary based on your printer. Specify your desired options.
3. Click **OK** to print.

To change additional print settings, click **File** > **Page Setup**.

To preview what your printout will look like, click **File** > **Print Preview**.

> **Important:**
>
> To print from Ansys software on Linux, you must first configure a printer from the MainWin control panel.
>
> For more information, consult documentation for your specific instance of Linux.

Getting Started with HFSS

HFSS™ is an interactive software package for calculating the electromagnetic behavior of a structure. The software includes post-processing commands for analyzing this behavior in detail.

Using HFSS, you can compute:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Basic electromagnetic field quantities and, for open boundary problems, radiated near and far fields.
- Characteristic port impedances and propagation constants.
- Generalized S-parameters and S-parameters renormalized to specific port impedances.
- The eigenmodes, or resonances, of a structure.

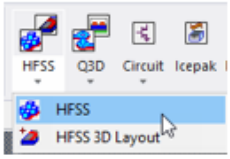
You are expected to draw or import the structure, specify material characteristics for each object, and identify ports and special surface characteristics. HFSS then generates the necessary field solutions and associated port characteristics and S-parameters.

> **Note:**
>
> If you are using the Eigenmode Solution solver, you do not need to specify sources for the problem. HFSS calculates the resonances for the model based on the geometry, materials, and boundaries.

As you set up the problem, HFSS allows you to specify whether to solve the problem at one specific frequency or at several frequencies within a range.

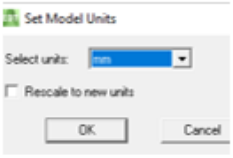
Setting up an HFSS Design

To set up an HFSS design, follow this general procedure. After you insert a design, you do not need to do the optional steps nor perform the intermediate steps sequentially, but you must complete them before you generate a solution.

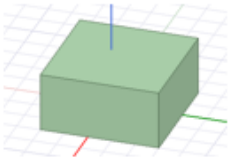


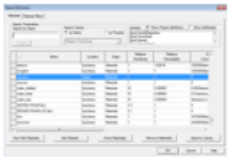
Insert HFSS Design

Select Solution Type

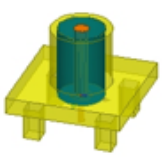


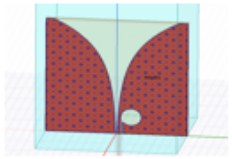
Set Model Units

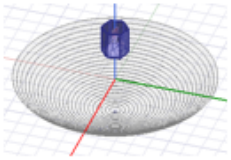

Draw/Import Geometry

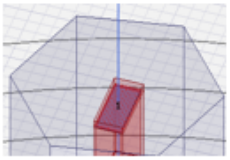

Assign Materials

[Assign Variables]

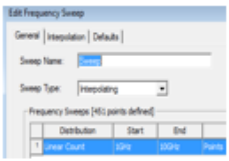

[Add 3D Components]

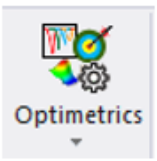

Add Boundaries


[Add Hybrid Regions]

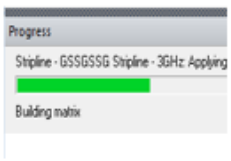

Add Excitations

Setup Solution

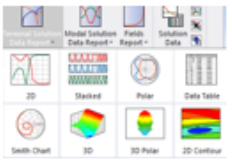

[Setup Frequency or Parametric Sweeps]



[Setup Optimization]



Run Simulation


View and Post Process Results

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Insert an HFSS design into a project.

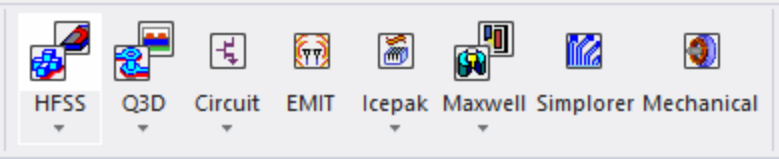
1. Select the solution type.
2. Set the model's units of measurement.
3. Draw the model geometry and assign material characteristics to objects.
4. Optionally, Working with Variables.
5. Assign boundaries, which specify the field behavior at the edges of the problem region and object interfaces.
6. Optionally, assign Hybrid Regions, which invoke other solvers for areas you assign as FEBI, HFSS-IE, Physical Optics, SBR+ or Dielectric Cavities.
7. For Driven solution-type designs, assign excitations - sources of electromagnetic fields and charges, currents, or voltages on objects or surfaces.

   For Composite Excitation Solutions, you provide a complete set of excitations via the **Edit Sources** dialog before solving.
8. Optionally, add 3D Components.
9. Specify how HFSS will compute the solution.
10. Optionally, Set up any Optimetrics you want to run.
11. Run the simulation.
12. View solution results, post-process results, view reports, and create field overlays.

> **Note:**
>
> After a period of idleness of 10 minutes, HFSS gives up its license. A renewal of activity automatically requests a license. Such idle notifications do not occur during solves.

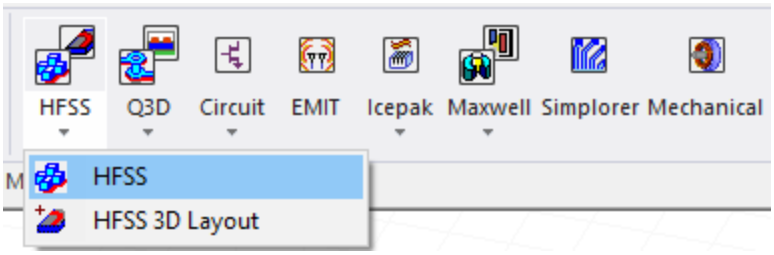
Inserting an HFSS Design

- Click **Project**>**Insert HFSS Design** or on the **Desktop** tab of the ribbon, click the HFSS icon:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can also use the **Desktop** tab ribbon to select HFSS from the drop-down menu under the HFSS icon:



The new design is listed in the project tree. It is named HFSSDesign*n* by default, where *n* is the order in which the design was added to the project.

By option, you can also set General Desktop configuration option to Insert a design of type HFSS by default when creating a new project. See General Options: Desktop Configuration.

The **3D Modeler** window appears to the right of the Project Manager. You can now create the model geometry.

> **Note:**
>
> Click the plus sign to the left of the design icon in the project tree to expand the project tree and view specific data about the model, such as its boundary assignments.

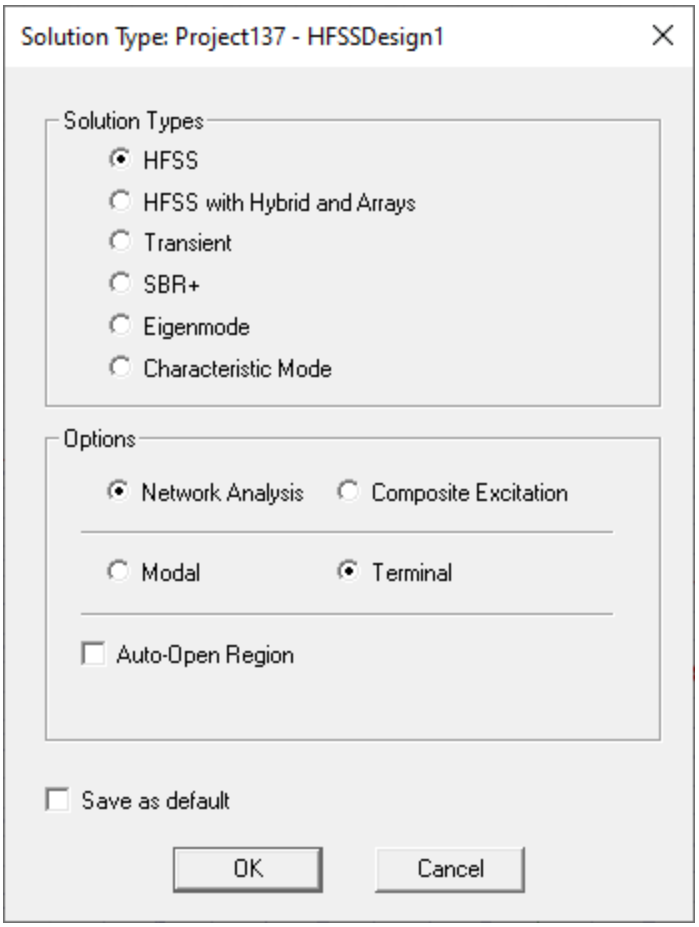
Selecting the Solution Type

Before you draw the model for an HFSS project, specify the design's solution type. As you set up your design, options available in the user interface will depend upon the selected solution type.

1. Click **HFSS**>**Solution Type**.

The **Solution Type** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select one of the following solution types:

Solution Types	
HFSS	This enables the Driven Options Modal and Terminal (the default). **HFSS** selection enables Mesh Fusion for FEM components. You can add padding to the mesh bounding volume. The finite array is disabled, and the Hybrid folder is removed from the project tree. The FEBI boundary becomes a regular boundary instead of hybrid region. When there are components set up for mesh fusion, the solution setup only allows direct solver and no derivatives.
HFSS with Hybrid and Arrays	This enables the Driven Options Modal and Terminal (the default). **HFSS with Hybrid and Arrays** selection enables mesh assembly for hybrid components. Users cannot add paddings to the mesh assembly bounding volume. The finite array is enabled, and the Hybrid folder is added to the project tree, along with the tab for Hybrid in the solution setup.
Transient	For calculating problems in the time domain. It employs a time-domain ("transient") solver. For **Transient** your choice of **Composite Excitation** or **Network Analysis** affects the options for the setup. If you select **Network Analysis** the setup includes an Input Signal tab for the simulation. Typical transient applications include, but are not limited to: • Simulations with pulsed excitations, such as ultra-wideband antennas, lightning strikes, electro-static discharge; • field visualization employing short-duration excitations;

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	• time-domain reflectometry.
SBR+	This option simplifies design creation for SBR+ users. HFSS can use shooting and bouncing ray (SBR) technology to calculate the far field from current sources and defined geometry via a one-way coupling. With this solution type, you do not need to specify explicit SBR+ Hybrid Regions. The Driven option for this solutions is Network Analysis only, with no Auto-Open Region. For details, see Design Flow for SBR+ Solution Type. You can also use parametric antennas in SBR+ solutions. For SBR+ solutions, you see an option in Initial Mesh Settings to **Allow tolerant meshing in SBR+ regions**. For HFSS SBR+solutions, you can also choose to import STL files as Lightweight Geometry.
Eigenmode	For calculating the eigenmodes, or resonances, of a structure. The Eigenmode solver finds the resonant frequencies of the structure and the fields at those resonant frequencies. Eigenmode designs cannot contain design parameters that depend on frequency, for example a frequency-dependent impedance boundary condition.
Characteristic Mode	This option is used for calculating the characteristic modes of a structure. The structure can be metal or dielectric. The solution reports the Number of Modes, the characteristic angle and current (amp/meter), the modal significance and quality factor, and the voltage per port based in edit sources weighting. Selecting Characteristic Modes changes the Solution Setup criteria and dialog. You specify the minimum modal significance (default 0.02). Convergence

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	is based on Max E rather than Max S (default (0.02).
	Only discrete sweeps are supported. Only the CMA solver is supported. Only lossless boundaries are allowed. Finite conductivity boundaries are allowed but are converted to lossless. The half-space boundary is not allowed.

For the Driven Solutions, specify whether to use Network Analysis or Composite Excitation. Network Analysis is the default for driven problems. Composite Excitation provides a method for solving fields in a large frequency domain problem.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. For HFSS Driven solutions, you can select a range of Driven options.

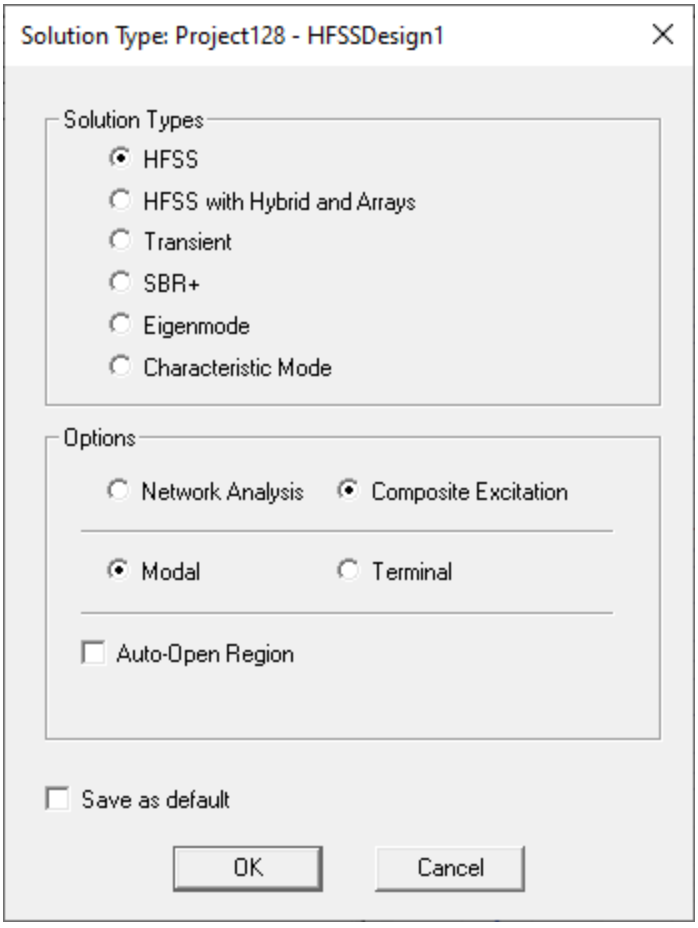
Driven Options	
Modal	For calculating the mode-based S-parameters of passive, high-frequency structures such as microstrips, waveguides, and transmission lines which are "driven" by a source, and for computing incident plane wave scattering. **Network Analysis** is the default and functions as before. Composite Excitation provides a method for solving fields in a large frequency domain problem. See Assigning Wave Ports for Modal Solutions and Assigning Lumped Ports for Modal Solutions.
Terminal	For calculating the terminal-based S-parameters of passive, high-frequency structures with multi-conductor transmission line ports which are "driven" by a source. Terminal is the default. This solution type results in a terminal-based description in terms of voltages and currents. Some modal data is also available. **Network Analysis** is the default and functions as before. Composite Excitation provides a method for solving fields in a large frequency domain problem. See Assigning Wave Ports for Terminal Solutions and Assigning Lumped Ports for Terminal Solutions.
Network Analysis	**Network Analysis** is the default and functions as before.
Composite Analysis	Composite Excitation provides a method for solving fields in a large frequency domain problem.
Auto-Open Region	For open region problems (typically antennas), you can choose **Auto-Open Region**. The option is available for Driven modal, terminal and transient solution type. This automatically creates an open region and a predefined Analysis setup for the project. You can select whether the region is Radiation, FE-BI, or PML. This simplifies the design process. If you do not choose **Auto-Open Region**, you must create an airbox and then assign a radiation boundary, either manually, or using the Create Open Region command. For more information on this Solution setting, see Using Auto-Open Region for the Solution Type for Antenna Designs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: When you opening an old project of solution types “Modal” or “Terminal”, they are converted to the new solution type “HFSS with Hybrid and Arrays” with corresponding options. However, if the old project has components that are not from array and all have FEM mesh assembly turned on, it is converted to solution type “HFSS” with corresponding options.

Composite Excitation Option for HFSS Driven Solution Types

The Composite Excitation option for HFSS - Driven solution types, for Model and Terminal options and Transient Solution Types provides a way to quickly solve fields in a large frequency domain problem.



For Driven Model and Driven Terminal optoins, you provide a complete set of excitations before solving. Those excitations will be accessed via the Edit Sources dialog box, and the resulting excitation vector is passed to the solver. The solver will solve only that set of excitations, and will produce:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- a single field file representing the composite excitation
- a set of surface radiated field files
- a field header file
- a vector of Active S parameters
- the port impedances
- gamma
- the port field display files

Using the **Edit Sources** dialog, you define a set of excitations as input to the solver. The right-click menu on the selected excitation will bring up an **Edit Sources** dialog limited to display excitations to the selected excitation (voltage, current, terminal) or to the terminals and modes of a selected wave port. The right-click menu on the excitation folder in the project tree will bring up the full **Edit Sources** dialog box. Edit Sources is available from the Fields folder.

When the solution type is single excitation, the **Edit Sources** dialog will hide the total voltage and apply port post processing options - these are not supported in this mode. The buttons to load and store excitation vectors are fully supported.

In a single excitation problem:

- changes to the excitation vector are a major edit
- only design and project variables are permitted (no post process variables)
- entries in the excitation vector may be expressions
- the user can sweep those variables in Optimetrics and the reporter
- the excitation vector for each variation to be solved will be passed to the solver
- Total voltage and port post processing effects will be unavailable
- The excitation vector will be validated as part of the boundary module.

The **Edit Sources** dialog can show either the complete excitation vector or be limited to the vector for a single excitation. The latter functionality is available in the network versions of Driven Terminal, Driven Modal, and Transient.

The Load and Save functions in **Edit Sources** will be available in both network and non-network.

The solver will produce a single field file for each solved variation. This field file (and its header) will occupy the same directory structure as was used for network solutions.

The solver will produce a single vector of Active S parameters. That data will occupy a single column of the network data solution which would have been produced by a network solve. The network data will include port Zo and gamma.

Field reports and field overlays will be available as in a Network Analysis solution.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Edit Sources is a major edit. So, it will invalidate all plots.
- Port renormalization, deembedding, and lumped port calibration are also major edits
- Animation of major intrisics (frequency) or design variables is supported.
- Field reports using post process variables (for example, to move a non-model point through the design) are supported.

Port field displays are supported as for Network Analysis. That is, port fields represent a uniform excitation at each port, NOT for the excitations specified by the user in **Edit Sources**. The reason is that we still need to solve the ports individually (even non-excited ports) before applying the user excitations.

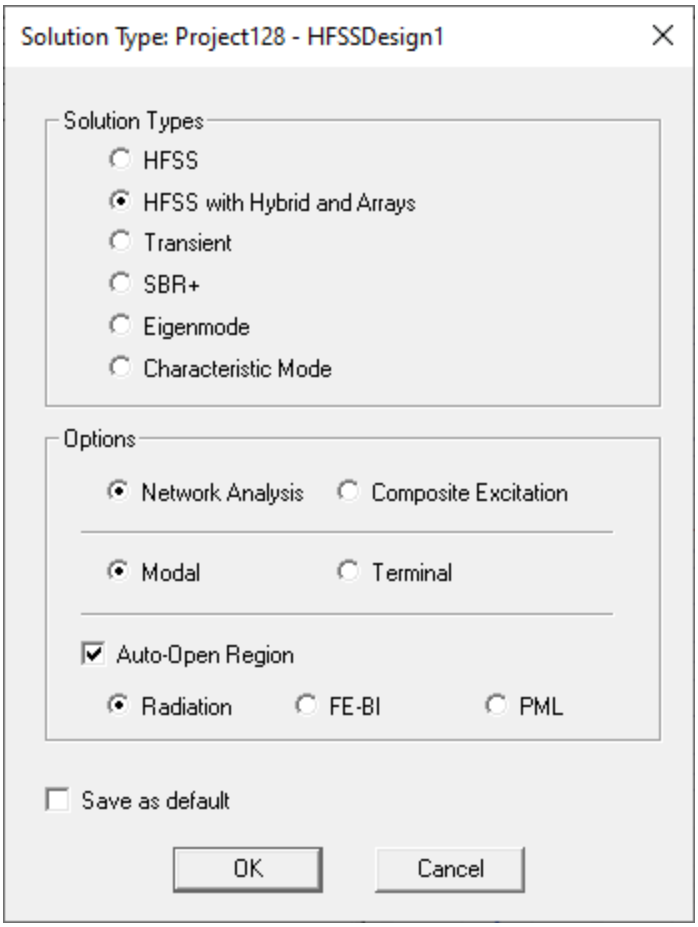
For Composite Excitation Solution types:

- Differential pairs are not supported.
- Derivatives are not supported.
- The reporter cannot display S parameters or any data derived from them (Y, Z, …).
- Gamma, terminal (and modal) port impedance will be available. The Matrix display panel will be limited to Gamma and port impedances. There will be no touchstone export.
- The reporter can plot fields as well as Active S (and Active Y, etc.). However, the Active S is provided by the solver, rather than by post processing. Active Y, Z and VSWR are functions of Active S and port Zo.
- Radiated fields are supported.

Using Auto-Open Region for the HFSS Driven Solution Type for Antenna Designs

In Auto-open region mode, the workflow eliminates as much as possible the required interactions from a user and allows you to get to a robust solution quickly. It is intended for those who do not want to worry of the region and the exterior radiation boundaries. The option is available for HFSS Driven types for the modal, terminal and transient solution options. You can save this option as a user default.

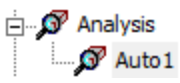
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In an open problem, an air volume encompassing the outer radiating surfaces is modeled by a surrounding object. Radiation boundary conditions (ABC, PML or FEBI) are assigned to the outer radiating surfaces to absorb all outgoing waves

If you have selected **Auto-Open Region**, HFSS automatically creates the region object or PML objects, the radiation boundaries or hybrid region according to your selection as Radiation, FE-BI, or PML, though these objects are hidden (not shown in the Project tree or History tree or listed in the Visibility dialog) unless you subsequently uncheck **Use Auto-Open Region**. In the case of PML, the PML Materials are hidden with Auto-Open (not listed under the Definitions>Materials folder in the Project tree), unless you subsequently uncheck **Use Auto-Open Region**. Rechecking Use Auto-Open Region leaves the PML for the previous auto-open is still visible.

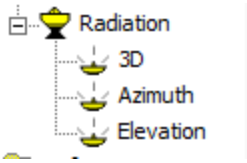
Selecting **Use Auto-Open Region** also creates a Solution Setup called Auto1.



Selecting **Use Auto-Open Region** also creates three far field setups, as follows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- ‘3D’ is a full sphere, Phi = 180 to 180 step 2, Theta = 0 to 180 step 2
- ‘Elevation’ will be Phi = 0 to 90 step 90, Theta = 180 to 180 step 1
- ‘Azimuth’ will be Phi = 180 to 180 step 1, Theta = 90 to 90 step 0



HFSS applies the most appropriate out-of-the-box settings based on the geometries and the operating frequency. You do not need to (and cannot) edit these settings directly.

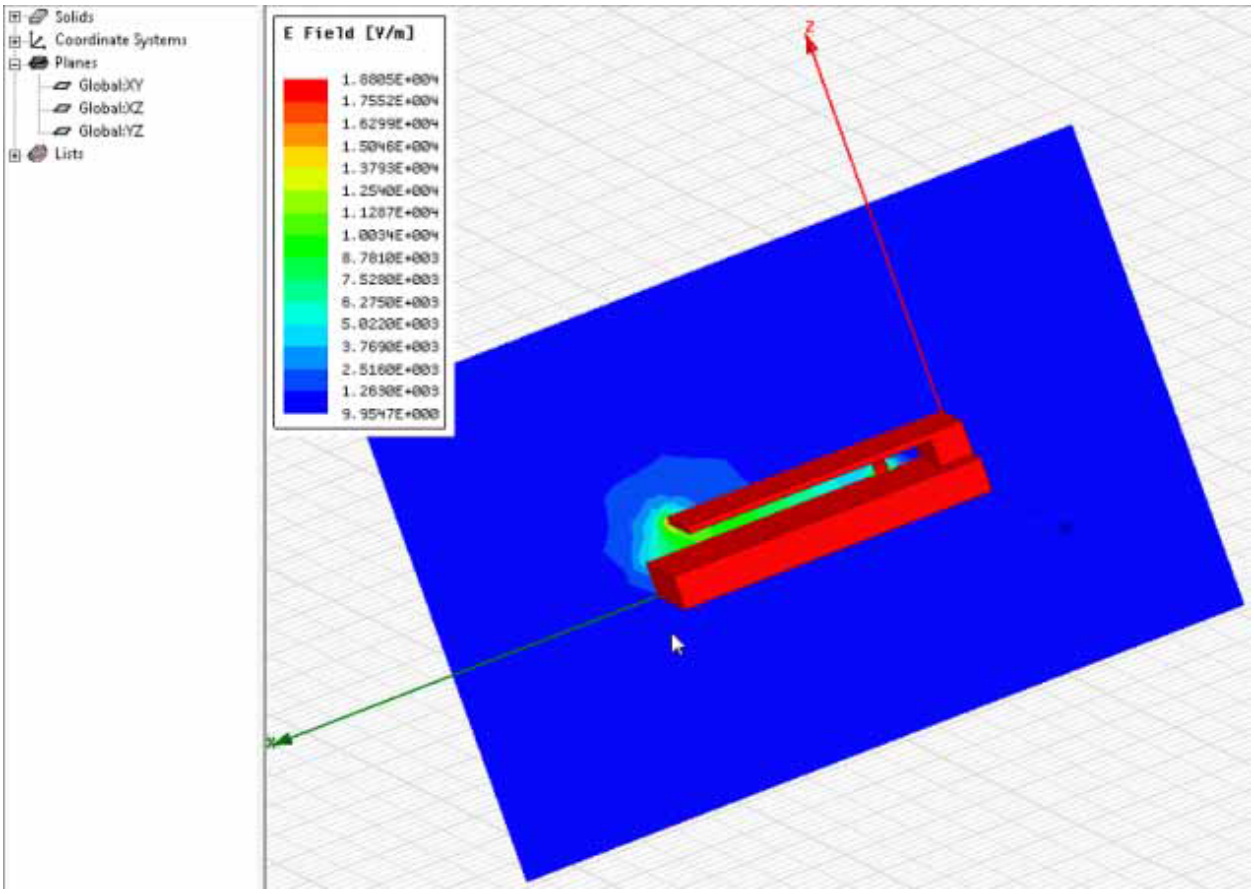
This mode restricts the design to a single solve setup. You cannot manually create the region object, radiation boundaries or PML in this mode. Infinite ground plane, metallic IE region, primary/secondary (Lattice pair) and symmetry boundaries are also not allowed.

- Region padding for ABC is = lambda/3, for PML is lamda/4, and for FEBI (exterior as HFSS-IE Domain) = lambda/8 where lambda is the wavelength of the adaptive frequency of the solve setup. The region is automatically resized when users edit the solve setup's adaptive frequency. Initial meshes and solutions are invalidated when the region is resized. For multi-frequency or broadband setups the lambda is determined by the center frequency of the range.
- ABC assigned on the region object, with the user's selection of 1st order ABC or FEBI (HFSS-IE Domain)
- PML objects are created and resized automatically, invalidating solutions when a region is resized.

The region object is not listed in the command history, not visible in the View window and its definition cannot be edited directly. Similarly, the radiation boundary is not listed in the project tree, not visible in the View window and cannot be edited. This boundary is enforced to the lowest priorities.

Because the region padding is automatically defined and not displayed, you cannot select faces from those regions for plots. However, you are able to create plots on the Global XY, XZ, or YZ planes.

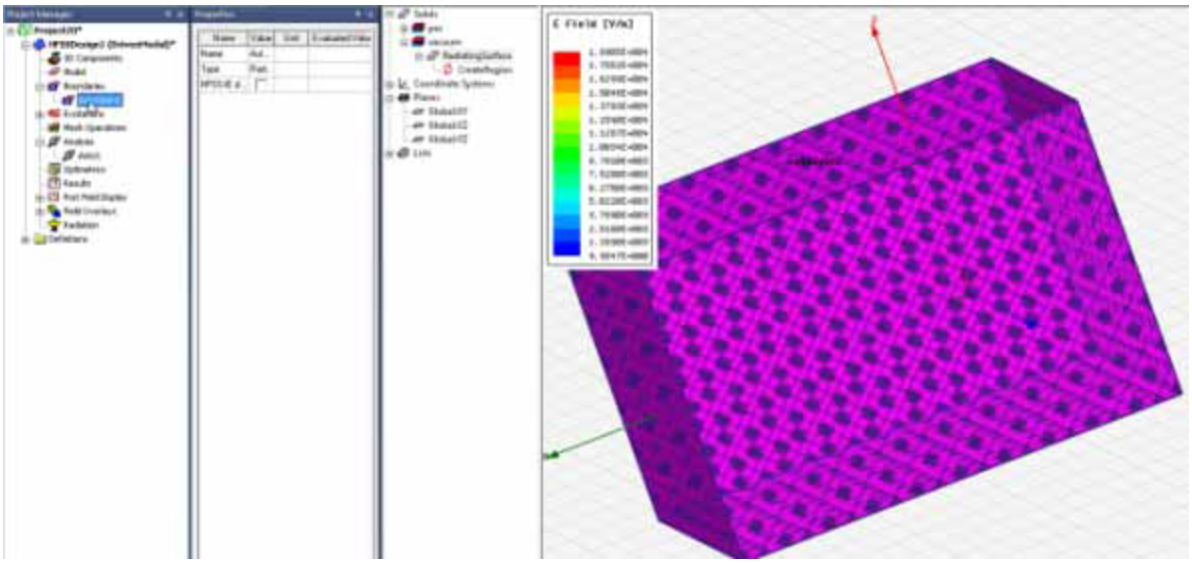
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



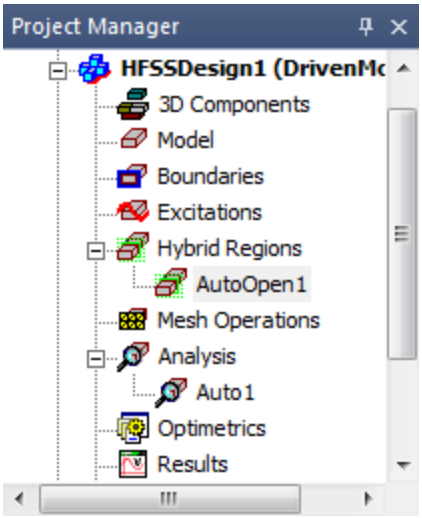
- Changing a design from auto-open to non-auto-open preserves solution.
- Changing a design from non-auto-open to auto-open invalidates solution.

If you turn off the Auto-Open Region setting, the Radiation boundary becomes visible in the modeler window and as a Boundary called AutoOpen1 in the Project tree, and the region appears in the history tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

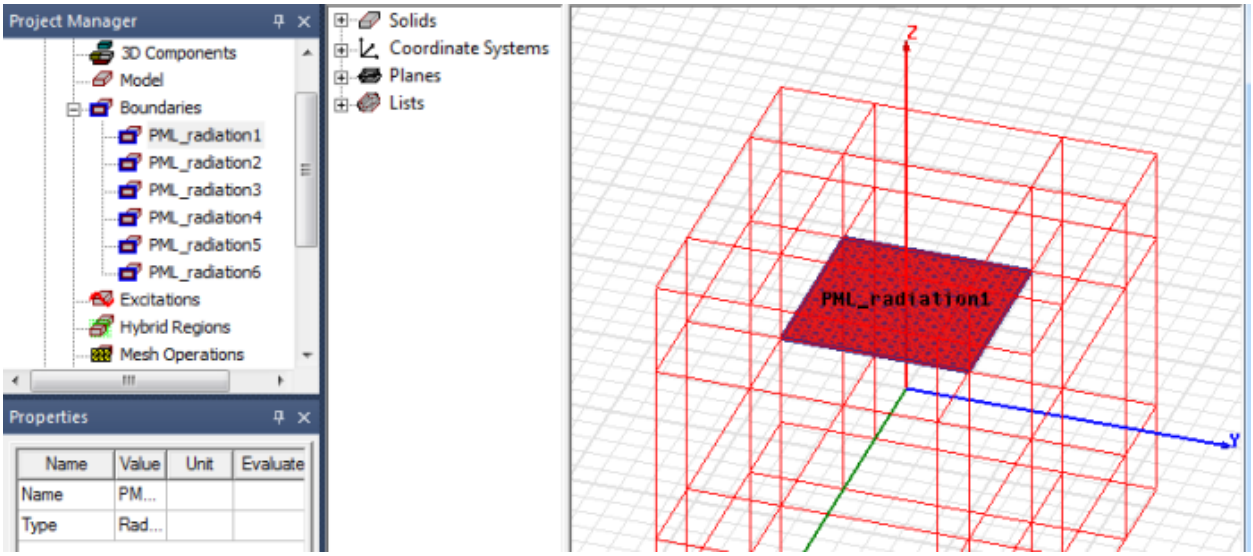


In the case when you turn off the Auto-Open Region setting for a FEBI boundary, the Project tree shows a Hybrid region labeled AutoOpen1.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

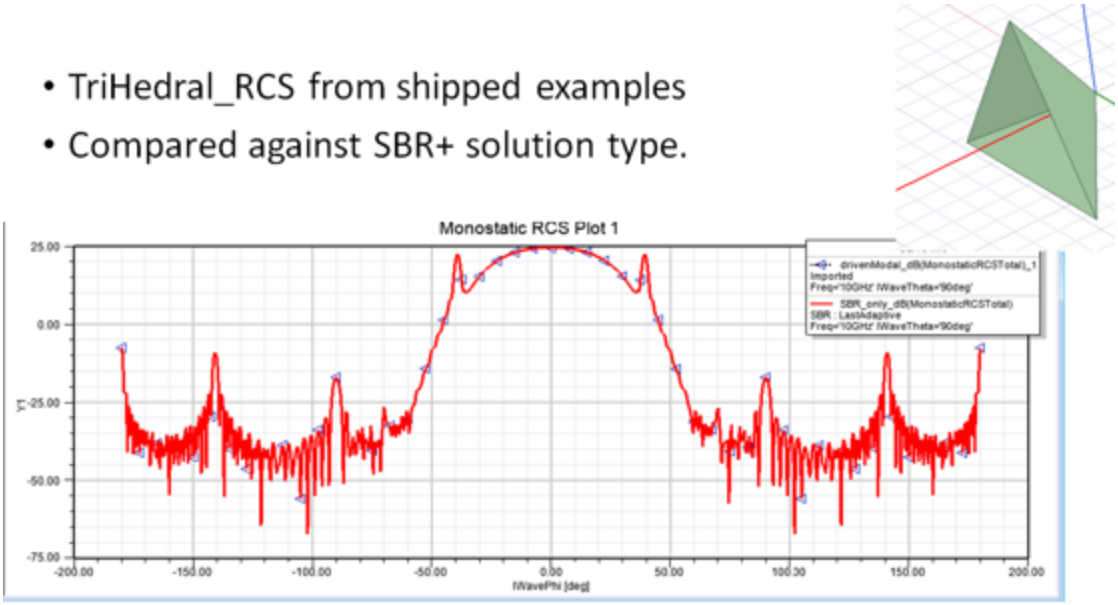
In the case when you turn off the Auto-Open Region setting for a PML boundary, the Project tree shows a set of PML_radiation*n* boundaries.



Design Flow for SBR+ Solution Type

The SBR+ Solution type simplifies design creation for SBR+ designs for use with Shooting Bouncing Ray technology with optional UTD and PTD correction. With this selection, you do not create explicit SBR+ Regions, and there is a reduced Solution Setup that includes a frequency sweep definition. Near field and far field reports, as well as antenna parameters are available. Matrix reports and Network Data Explorer are available if the design includes a parametric- or file-based antenna. 3D fields, near fields, and port displays are not available. Plot fields, and SAR settings do not apply. An SBR+ only design and its equivalent Modal/Terminal design should have comparable if not identical results.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Prerequisites

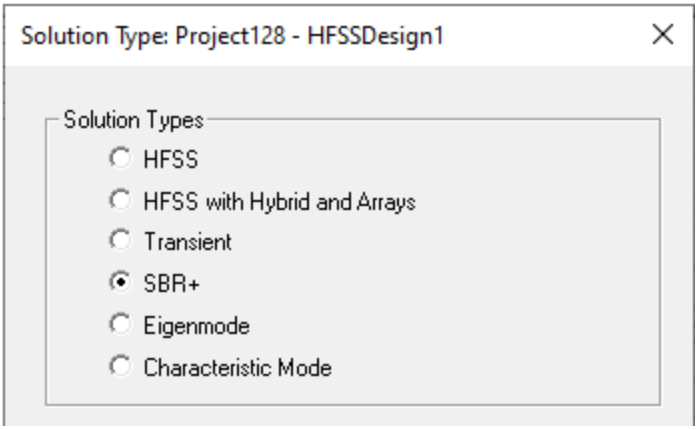
Requirements for SBR+ solutions:

- Must have at least one Incident Plane Wave or Antenna Component excitation. An Antenna Component is either a link to a source design, a built-in parametric antenna, or an imported far-field antenna gain pattern (.ffd) file.
- Ports are not allowed.
- Infinite ground planes are not supported.
- SBR+ can trace rays inside dielectric regions of non-uniform thickness, following Snell's laws of reflection and refraction, including for lossy non-conductive dielectrics. For volumetric SBR+, only conformal meshes are supported. There is no lightweight, dynamic or tolerant mesh support. Meshes must represent volume regions (that is, objects) in HFSS. Sheets cannot be assigned materials, but only boundaries. Supported materials are pure dielectrics, with zero conductivity and without excessive loss.
- No spatially dependent boundaries.
- For near fields and far fields, the setup for sample points must be defined before simulation. This can be done through the Field Observation Domain area of the Options tab. You can create new Near Field Lines, Rectangles, boxes, or Spheres, or Far Field Infinite spheres and edit existing domains.
- SBR+ near field analysis will be aborted if one or more encrypted components are included.
- SBR+ designs that include blockages have an enhanced work flow described in "Configure Simulation with SBR+ Antenna Geometry Blockage (SBR+ Solution Type)" on page 3-213

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using SBR+ Solution Type for HFSS

1. When you select the SBR+ Solution type, you do not need to set SBR+ Regions in the design.



2. SBR+ Solutions support the following boundaries:

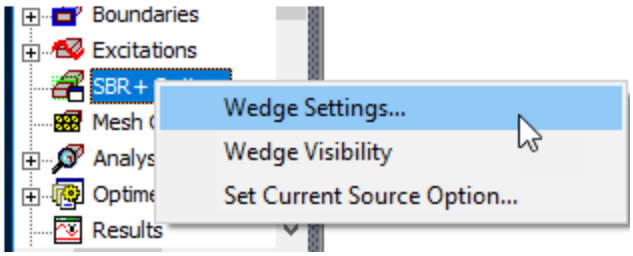
- Perfect E
- Perfect H
- Finite Conductivity
- Layered Impedance
- Impedance

For the supported boundaries, only the material property, impedance, and thickness are used. Surface roughness and shell elements are disabled. If you convert an existing design to SBR+, unsupported excitations and boundaries are deleted, and any unsupported boundary properties are removed. You can **Undo** to restore the original settings.

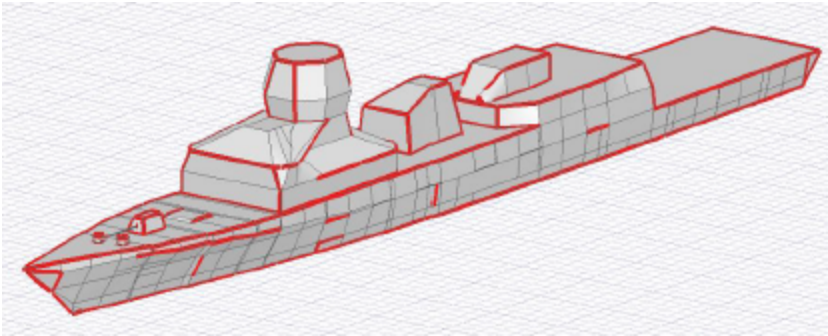
3. Set the SBR+ Options

The Project Tree for the SBR+ Solution type does not include a Hybrid Regions icon, nor a Port Field display. It does include SBR+ Options where you can specify **Wedge Settings** and enable or disable **Wedge Visibility**. These options are described below. See this link for Set Current Source Option.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

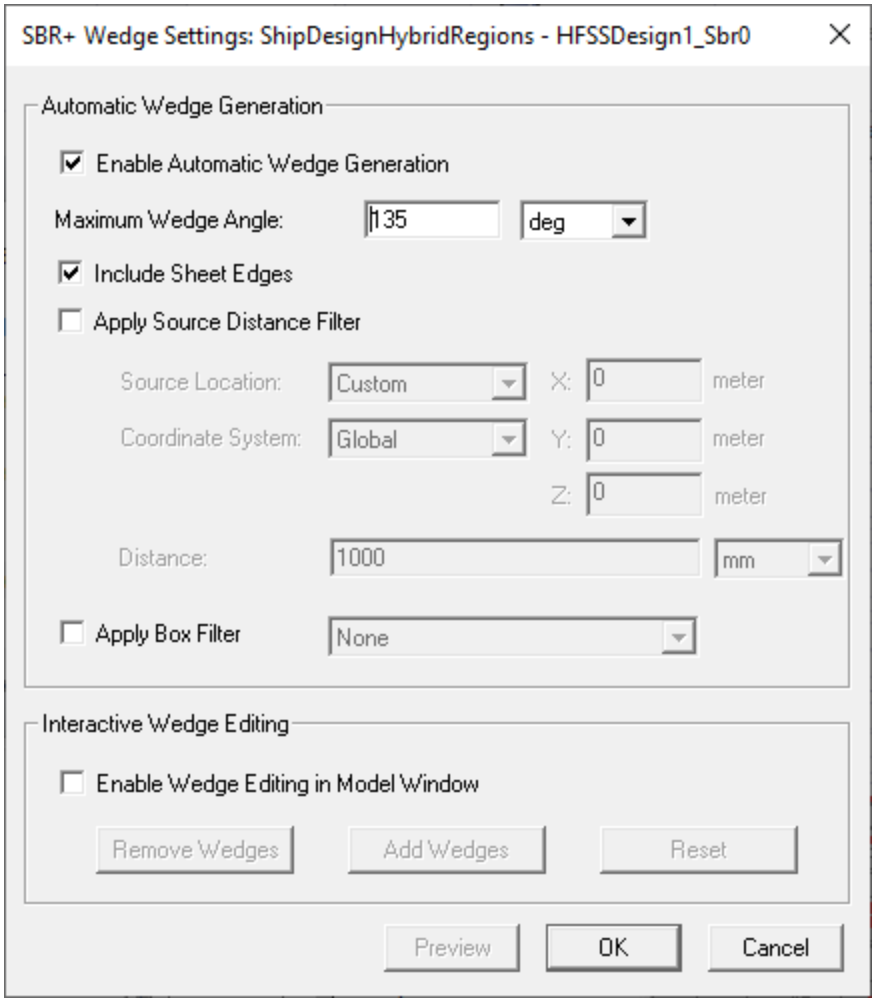


If visible, wedges are shown as thick red lines in 3D model window. Visualization is turned off when there is design edits that invalidate the initial mesh.



The **SBR+ Wedge Settings** dialog contains the Automatic Wedge Generation parameters and the Interactive Wedge Editing feature. Wedges for PTD and UTD are based on the user-specified wedge settings and the initial mesh. The initial mesh is automatically generated on launch of the **SBR+ Wedge Settings** dialog box, if needed. The dialog box will not launch if an initial mesh is not available and cannot be generated.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

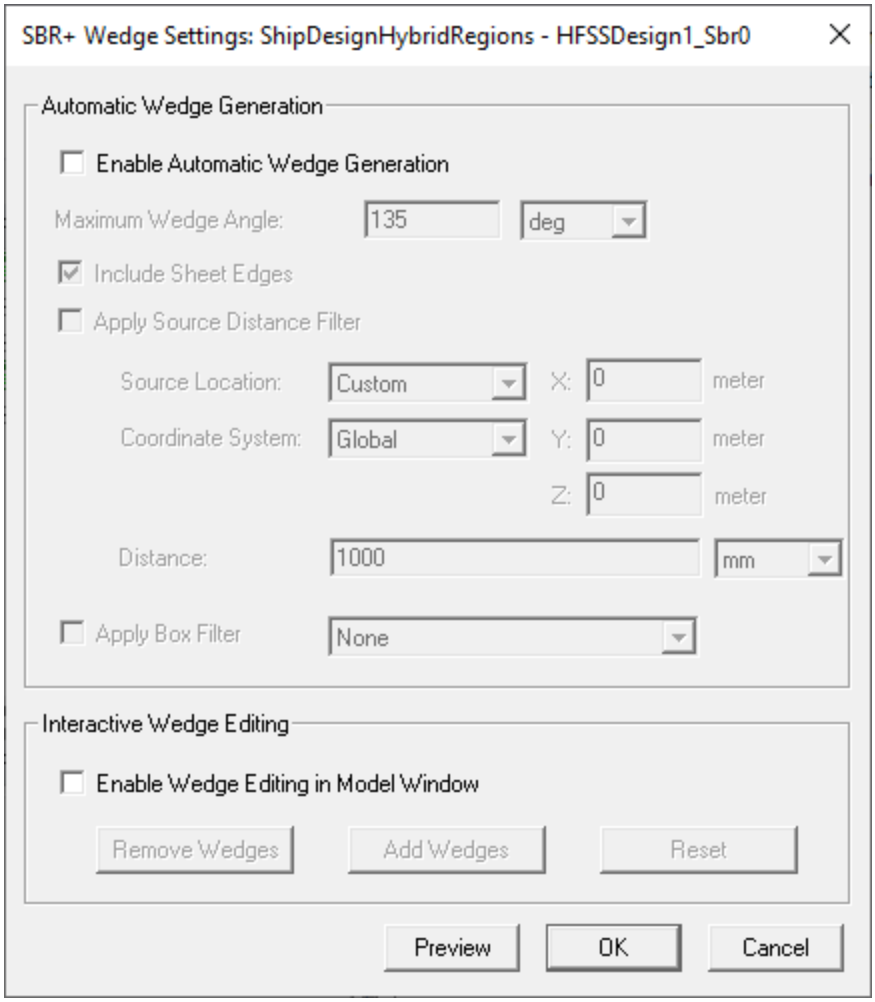


The Wedge Generation parameters are used to determine the candidate set of wedges to be included, with optional filters that can restrict their location on the geometry. The final set of wedges to be used in the SBR+ simulation is the intersection of all the wedge generation criteria, and the effects of any optional Interactive Wedge Editing. complete flexibility in specifying where the wedges are to be used for the simulation. This is important since adding many wedges on very large geometry structures (aircraft, vehicles, etc.) can significantly impact the SBR+ simulation time.

Enable Automatic Wedge Generation.

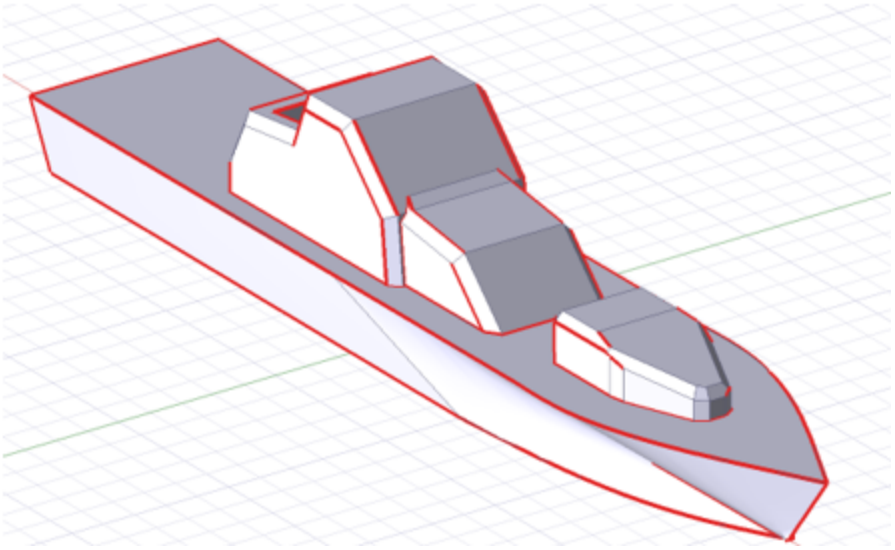
By default, the checkbox is enabled, allowing access to the automatic features and parameters. If you un-check it, the automatic features are disabled. You can independently enable Wedge Editing.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



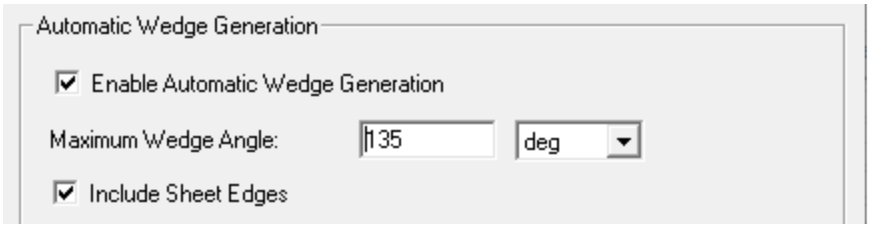
Wedges added through Automatic Wedge Generation are displayed in red.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Maximum Wedge Angle:

An angular criteria can be specified for the maximum wedge angle to help filter the desired set of wedges - default value is 135 deg. This wedge angle ranges from minimum 0 deg to maximum 180 deg. All wedges are included that are less than or equal to the specified maximum angle. An angle equal to 0 deg would be created by a “collapsed” wedge (like a closed hinge), while an angle equal to 180 deg would be created by a completely “open” wedge (like a fully open hinge). You can assign a design variable to Maximum Wedge angle, but you cannot create a design variable within the dialog.



Include Sheet Edges:

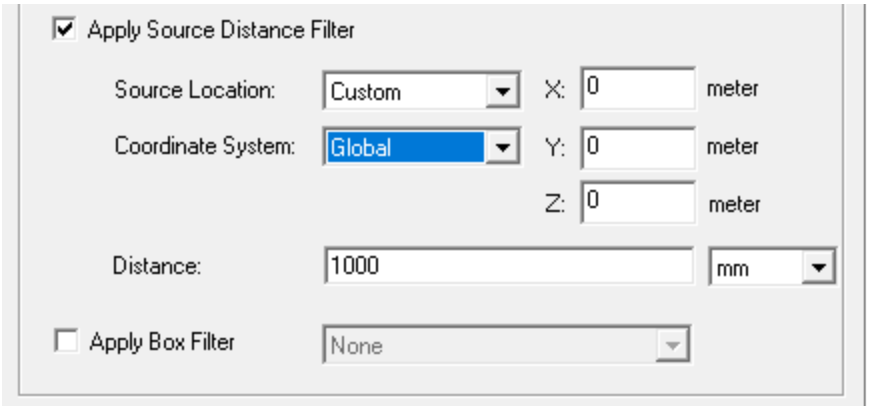
You can choose to include or exclude the edges from sheets (non-connected edges, also known as “knife edges”). The default value is true which includes these edges. Sheet edges are triangle edges with no adjacent/connected triangle along that edge. In the physical object we are modeling these may represent extremely thin surfaces such as metal “fins”, etc.

Optional parameters:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Apply Source Distance filter:

A “source distance filter” is a way to specify the inclusion of wedges within a finite 3D distance from a point (the “source”).



You can choose model Points from drop down as source location, or specify the absolute X, Y, Z in model units. X, Y, Z can be specified in global coordinates or relative to a custom CS. When you select a geometry Point, its coordinate system and X, Y, Z are displayed.

The Distance default is 1 <model unit>. You can assign an existing design variable to the Distance. Default value is false (no Source Distance Filter).

Apply Box Filter:

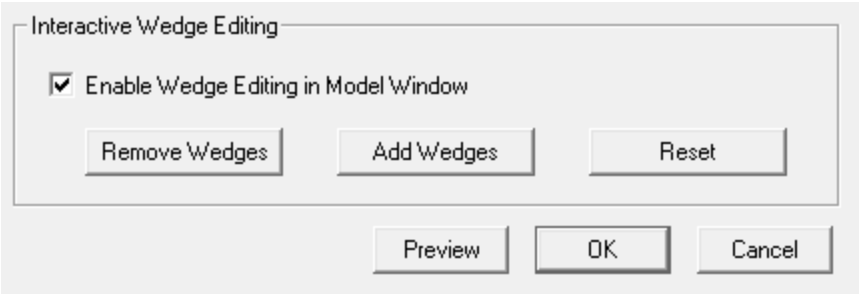
You can select any non-model box that is aligned with the Global Coordinate System axes via a drop-down menu, within which wedges will be included. Default value is false (no Box Filter).

Optional Interactive Wedge Editing

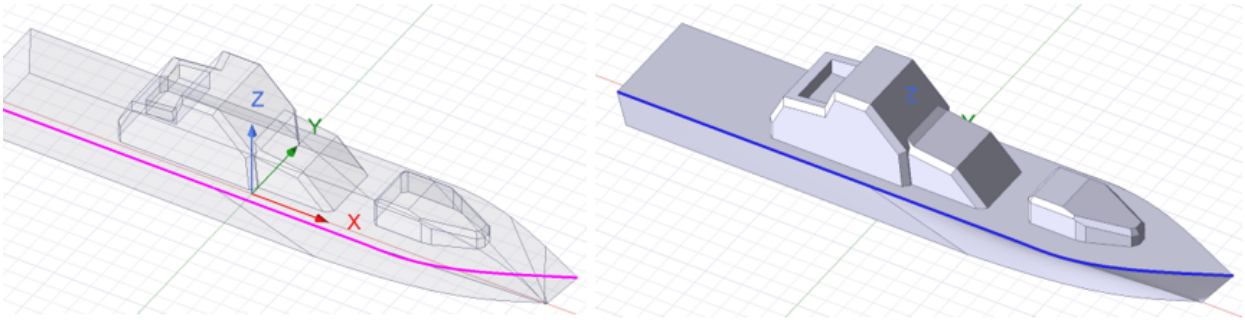
This feature provides complete flexibility in specifying where the wedges are to be used for the simulation. This is important since adding many wedges on very large geometry structures (aircraft, vehicles, etc.) can significantly impact the SBR+ simulation time.

Checking the Enable Wedge Editing the Model Window box enables the buttons for removing wedges from selected geometries (faces, objects, etc.) , adding wedges to selected geometries, and resetting edits.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After you select a geometry and **Add Wedges**, the interactively added wedges appear in blue. Allowed geometry selections include entire geometry objects, faces on objects, and edges on objects.



If you select **Remove Wedges**, all wedges on the selected geometry objects are removed. These can be automatically generated wedges or interactively added wedges.

Click **Reset Wedges** to undo all interactive wedge editing operations. A confirmation dialog appears. Press Yes to reset all wedge editing operations. The reset button is disabled if no wedge edit operations are present.

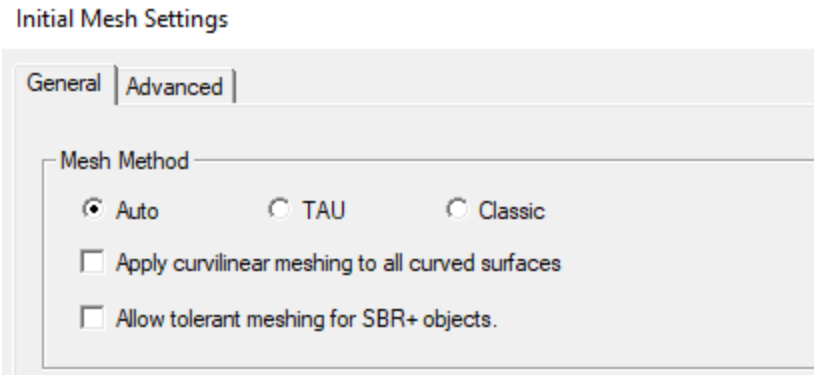
Current Limitations on PTD/UTD Wedge Generation/Editing

Situations where wedges cannot be generated:

Light Weight Imported STL Geometries: STL models imported as light weight geometries are currently not supported for interactive wedge editing (as of R24.1).

In the Initial Mesh Settings for SBR+ Solution type or for HFSS with Hybrid and Arrays that include SBR+ regions, you can turn off tolerant meshing.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

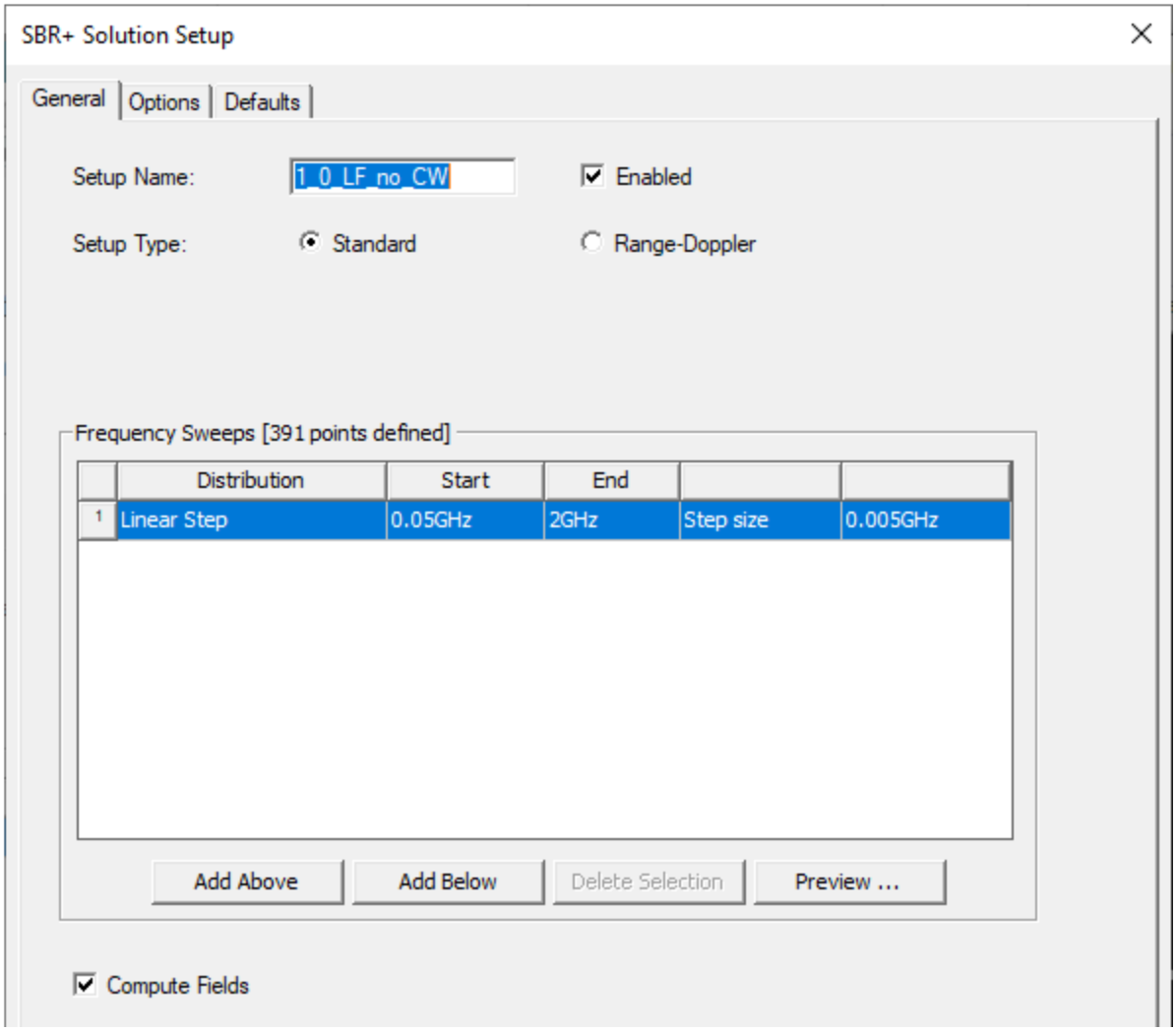


In this case you should consider Current Limitations on PTD/UDT Wedge/Generation Editing.

4. Add an SBR+ Solution Setup by selecting the **Simulation** tab and clicking the Setup icon.

SBR+ solutions offer a reduced solve setup dialog with a frequency sweep definition and SBR+ specific **Options** tab.

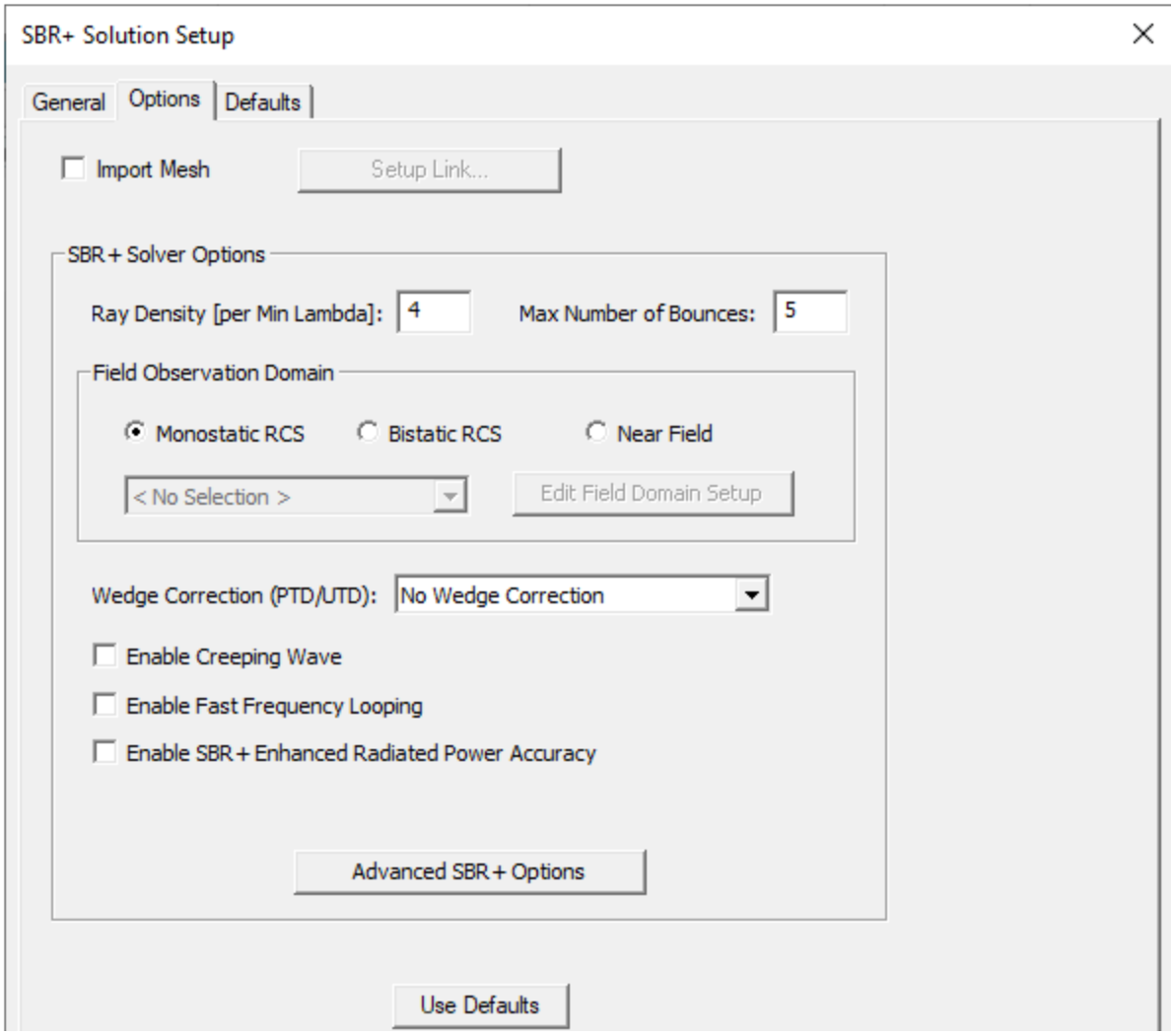
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- There are two options for Setup Type: Standard (default) and Range-Doppler. The settings and specialized capabilities of the Range-Doppler type are described at SBR+ Range-Doppler Solution Setup. The rest of this page pertains only to the Standard setup type.
- Because you can define Frequency Sweeps here, you cannot add Frequency sweeps under the setup.
- Switch on **Compute Fields** to generate installed antenna patterns or perform RCS simulations using SBR+. Switch off **Compute Fields** to restrict simulation results to coupling data (S-parameters) between two or more antenna components configured in the design (i.e., source design links, parametric antennas, imported .ffd far-field gain pattern files).

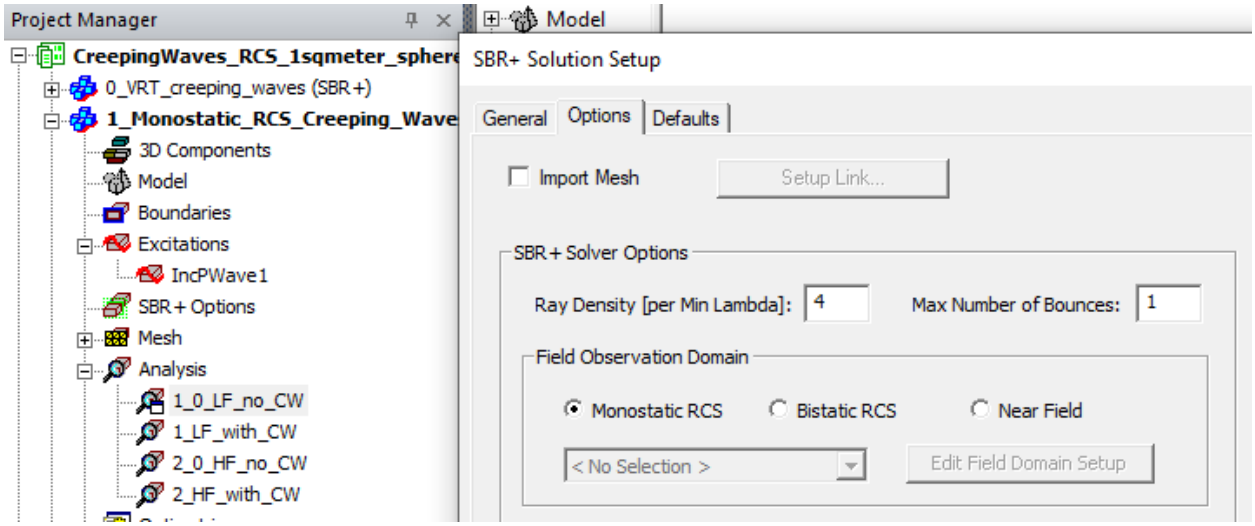
The **Options** tab lets you specify SBR+ Solver Options, including **Ray Density Per Wavelength** and **Maximum Number of Bounces**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



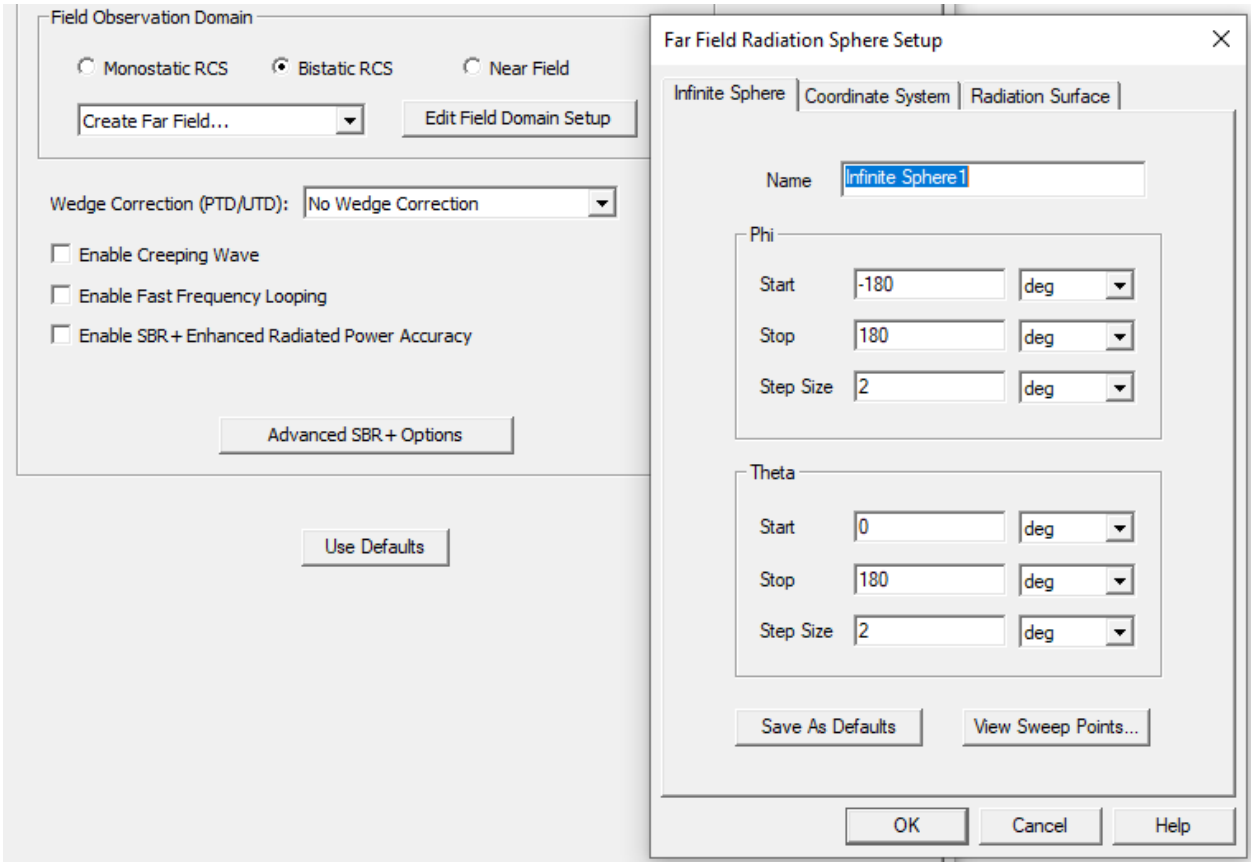
For the RCS Type options for Monostatic and Bistatic to appear, you must include an Incident Plane wave. For Monostatic RCS, you do not need to create a Far Field setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

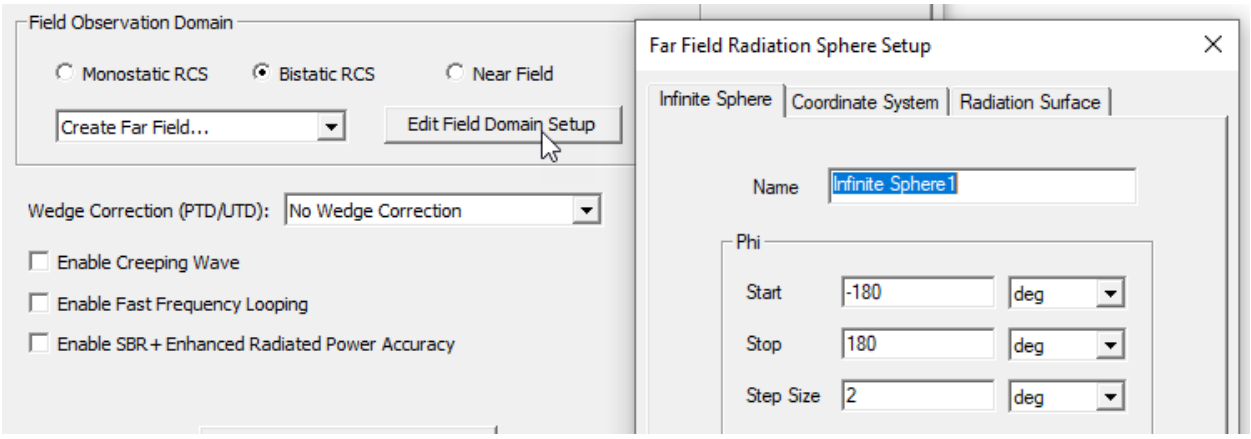


If you have assigned both an SBR+ region and an Incident Plane wave, and select Bistatic as the RCS Type, the Near Field and Far Field Observation domain fields are enabled, and the **Far Field Radiation Sphere Setup** dialog box appears as follows, allowing you to create an infinite sphere for a Far Field or a rectangle for examining near fields. You can edit an existing Far Field Infinite Sphere or Near Field Rectangle setup by selecting from the drop-down menu and clicking **Edit Field Domain Setup**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you have created one or more field observation domains, these are listed on the drop-down menu. You can edit an existing Far Field Infinite Sphere or Near Field Rectangle by clicking **Edit Field Domain Setup**.



For Near Field rectangle setups, you can use the feature described in Overlay Contour Plot of

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

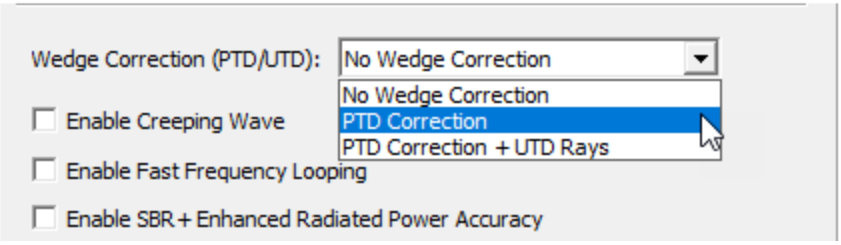
Near Field on Rectangle.

For **Enable Creeping Wave**, see Creeping Waves, Creeping Wave VRT Plots, Creeping Wave for Antenna Placement, and Creeping Wave for RCS of Highly Curved Objects.

The **Fast Frequency Looping** feature is to significantly reduce simulation time for high-density frequency sweeps with tens to thousands of uniform frequency samples. You can select Enable Fast Frequency Looping if you have configured a uniform frequency sweep with at least 2 frequencies. From there, the acceleration is handled automatically, with no other workflow changes in problem setup or post-processing. The acceleration engages for all SBR+ field and signal output types: S-parameters, far-field antenna patterns, near-field E/H-field spatial distributions. See SBR+ Fast Frequency Looping for High-Density Sweeps for details on the benefits and limitations of this acceleration feature.

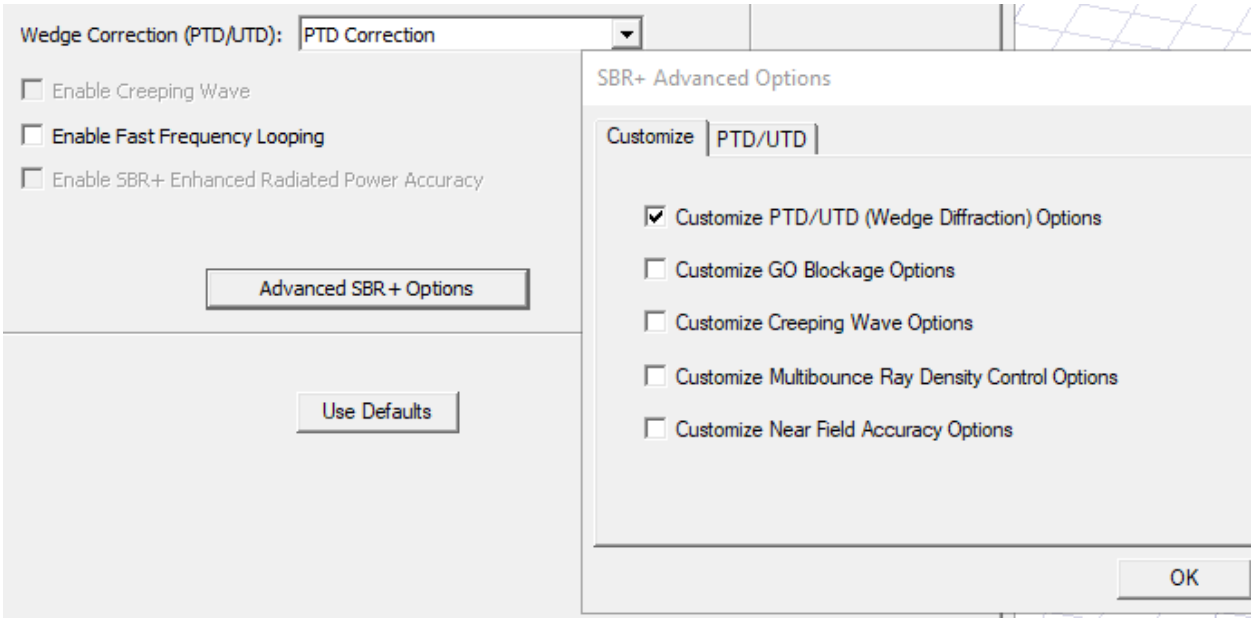
The **SBR+ Enhanced Radiated Power Accuracy** feature can be beneficial when an SBR+ region contains lossy materials (got example, finite conductors, half-space layered impedance boundaries, or absorbers). Enabling this feature produces a more accurate result by having SBR+ compute total fields across the entire far-field sphere. These total field results are then weighted using the configuration specified in the Edit Sources dialog and then integrated over the far-field sphere to determine the total power radiated by the design. The total power is then used to improve the accuracy of the directivity fields and other related antenna parameters such as radiated power.

The **Wedge Correction (PTD/UTD)** simulation settings allow for the inclusion of additional wedge diffraction phenomenology that can improve the accuracy of SBR+ simulations. You can opt out of using the PTD/UTD settings, or select PTD Correction or PTD Correction + UTD Rays. Selecting one of the PTD Correction options enables a field for specifying PTD Edge Density.

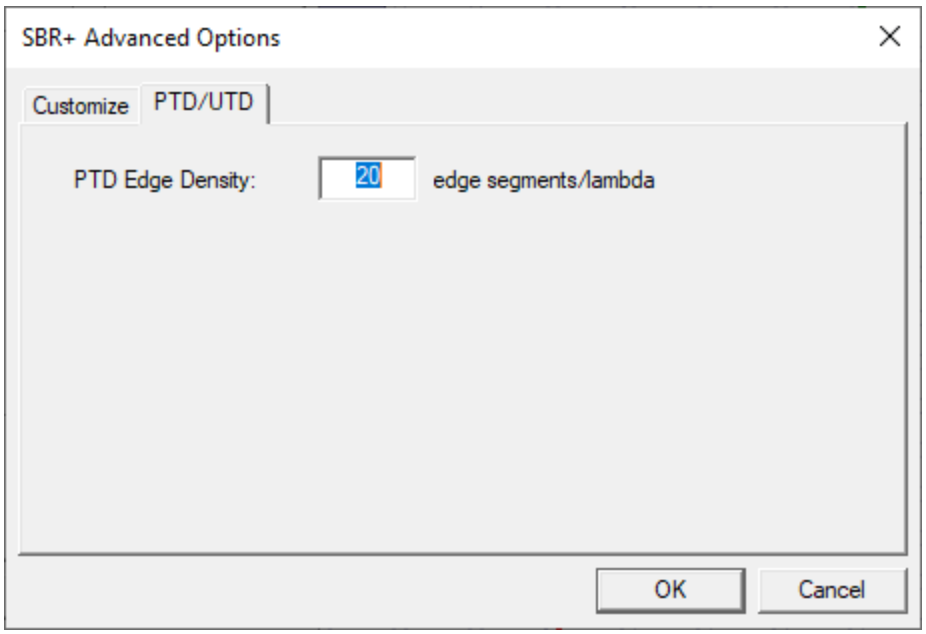


If you select one of the PTD Correction options you can then select the **Advanced SBR+ Options** button to Enable PTD/UTD (Wedge Diffraction) Options.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



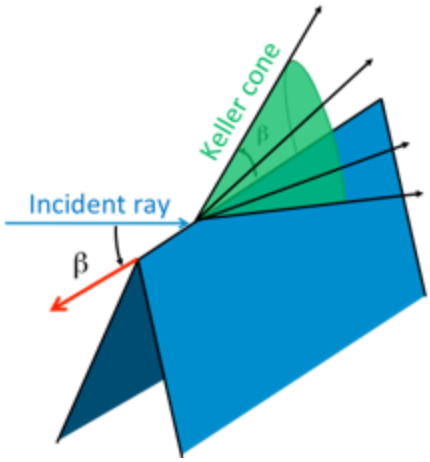
Enabling PTD/UTD (Wedge Diffraction) Options opens **PTD/UTD** tab for specifying PTD Edge Density.



For SBR+ simulations, the Physical Theory of Diffraction (PTD) and Uniform Theory of Diffraction (UTD) features can account for additional phenomenology not well predicted by plain SBR due to truncation of uniform Physical Optics (PO) currents at sharp angular discontinuities (“wedges”) on metallic surfaces and blockage of SBR’s Geometrical Optics

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(GO) rays. PTD is a numeric correction to the scattered fields radiated by PO currents near wedges. UTD launches bundles of edge-diffraction rays from directly illuminated portions of each wedge along the Keller cone. Once launched, the UTD rays behave exactly like regular SBR rays, propagating according to GO and painting PO currents at each bounce that contribute to the scattered field. The UTD rays often illuminate portions of the SBR scattering geometry that are never reached by SBR GO rays launched directly from the field source.



The PTD and UTD wedge features are only deployed for metallic wedges with line-of-sight visibility from the source (Tx) location. If either adjacent surface of the wedge is non-PEC and not within the tolerance for PEC-like, or if the entire edge segment is not visible to the source, the wedge will be ignored in the SBR+ simulation and for visual ray tracing (VRT). For technical details, see Assigning SBR+ Hybrid Regions.

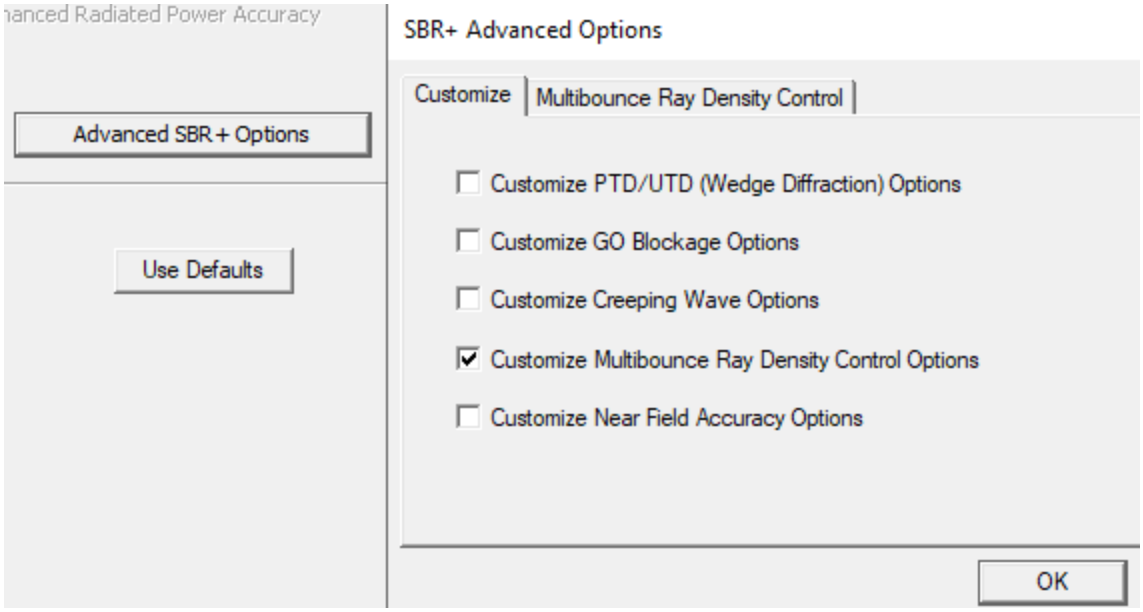
5. Advanced Option for NF Accuracy Settings for SBR+

The NF Accuracy settings (Beta feature, NF Accuracy Controls for SBR+) provide access to methodology settings relating to cases where Tx antennas, Rx antennas, and near-field observation points are in proximity to the scattering geometry (i.e., the platform CAD model). When this feature is switched off, HFSS SBR+ uses default near-field accuracy settings that should be adequate for most situations, including close-proximity Tx/observer conditions that this feature is designed to tune.

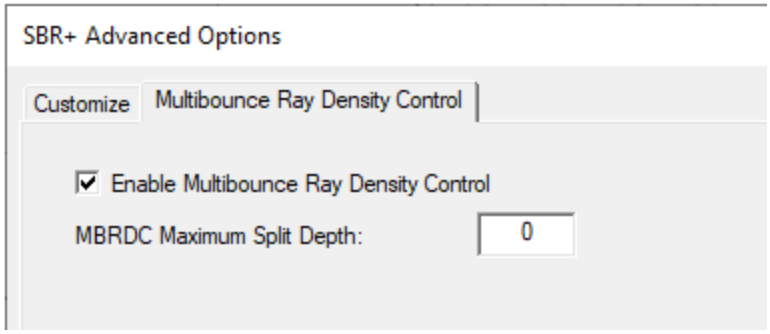
6. Advanced Option for Multibounce Ray Density

By option you can select the **Advanced SBR+ Options** and enable Customize Multibounce Ray Density Control Options.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can then select the **Multibounce Ray Density Control** tab.



Use the checkbox to **Enable Multibounce ray density** and enter the maximum number of subdivisions to be used in the solve. Enabling the feature may increase the solve time.

The **MBRDC Maximum Split Depth** field value increases the number of spawned ray tracks on each ray bounce. The MBRDC algorithm is designed to achieve the ray density per wavelength criterion at any bounce depth. If a footprint at depth *N* in a ray track does not satisfy the ray-density criterion, the MBRDC algorithm attempts a refined ray shoot from an earlier bounce, just for the ray in question. The size of the triggering footprint informs the level of refinement. If the refined shoot is not successful in achieving the desired footprint size, or new footprints are too large, the MBRDC algorithm attempts further, recursive refinements. The **MBRDC Maximum Split Depth** setting limits how aggressively the refinement algorithm is applied by specifying the maximum number of split recursions. For this reason, linear

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

increases in Maximum Split Depth can yield a geometric progression in the total number of rays shot and associated solution time.

When both MBRDC and UTD are enabled, first-bounce UTD rays and bright points on wedges are recomputed during the MBRDC splitting process, but only the original raytrack is rendered up to the 1st bounce. Similarly, length-based ray filters are applied to the original UTD initial ray track up to 1st bounce rather than using the information in the additional MBRDC tracks. This is a known limitation.

7. Advanced Option for Geometrical Optics (GO) Blockage

SBR+ normally applies a physical optics (PO) blockage model where blockage effects emerge from partial cancellation of the incident field by the scattered field. PO blockage not only occurs in connection with the incident field from the Tx antennas, but also within the scattered field itself when reflected fields from the previous bounce are diminished by a blocking surface along the geometrical-optics (GO) reflection path at the current bounce by evaluating the scattering contribution of the incident ray field at that bounce.

In cases of significant blockage by a large obstruction, the PO blockage model requires accurate currents over the extent of the obstruction. Sometimes, this is hard to achieve in the ray tracing approximation. For example, the obstruction may not be well illuminated by direct rays from the Tx antenna or multi-bounce rays after an earlier reflection. An alternative is to use a GO blockage model where scattered field contributions are added to an observation point subject to a line-of-sight (LOS) blockage check performed by the ray tracer. In some cases, this is more accurate than the default PO blockage formulation, while in others it is less accurate. GO blockage also entails a non-trivial computational cost, as the blockage check with the ray tracer must be performed for each observation angle (or point) for each ray hit point.

In addition to GO Blockage for SBR and UTD ray tracks, this feature is also available for PTD and CW.

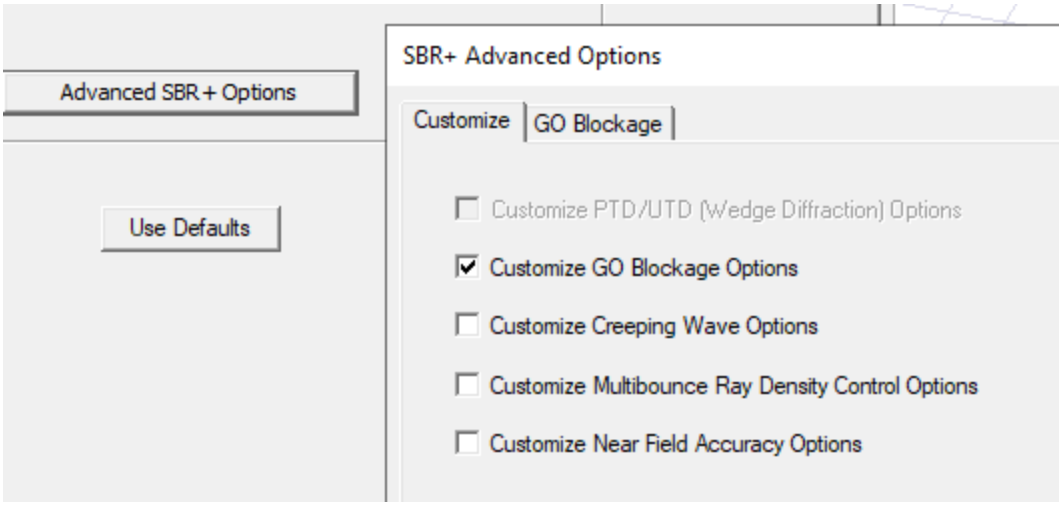
Limitations for GO Blockage:

GO Blockage formulation is suited specifically for SBR+ ray tracks, including UTD ray tracks. While GO Blockage for CW and PTD has been implemented for the first time in this release, results can be surprising and/or inaccurate, as a limitation of the methodology.

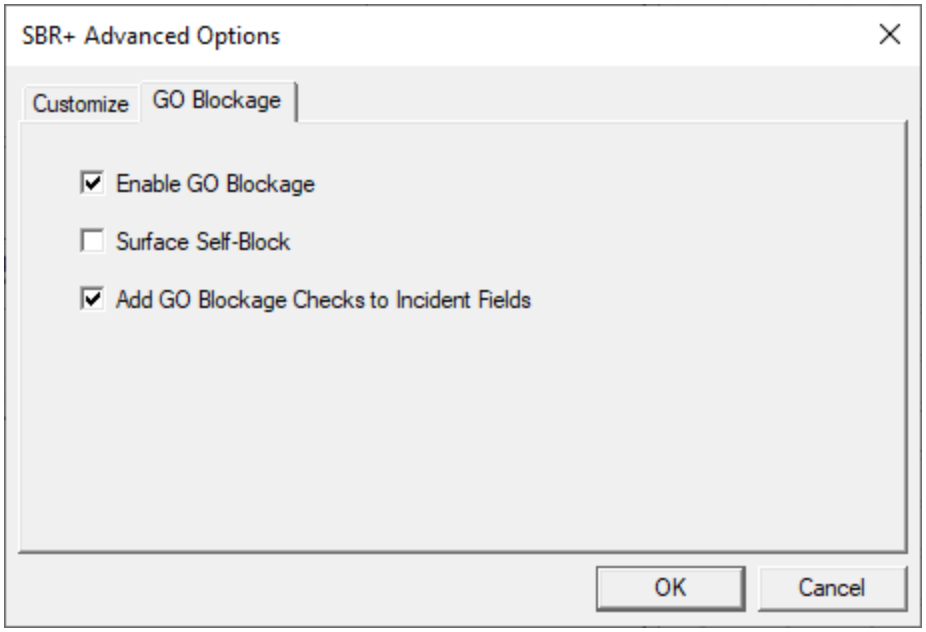
Summary Workflow for GO Blockage:

Click the **Advanced SBR+ Options** button to open the **SBR+ Advanced Options** window. On the **Enable** tab, Check GO Blockage Options to enable the **GO Blockage** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



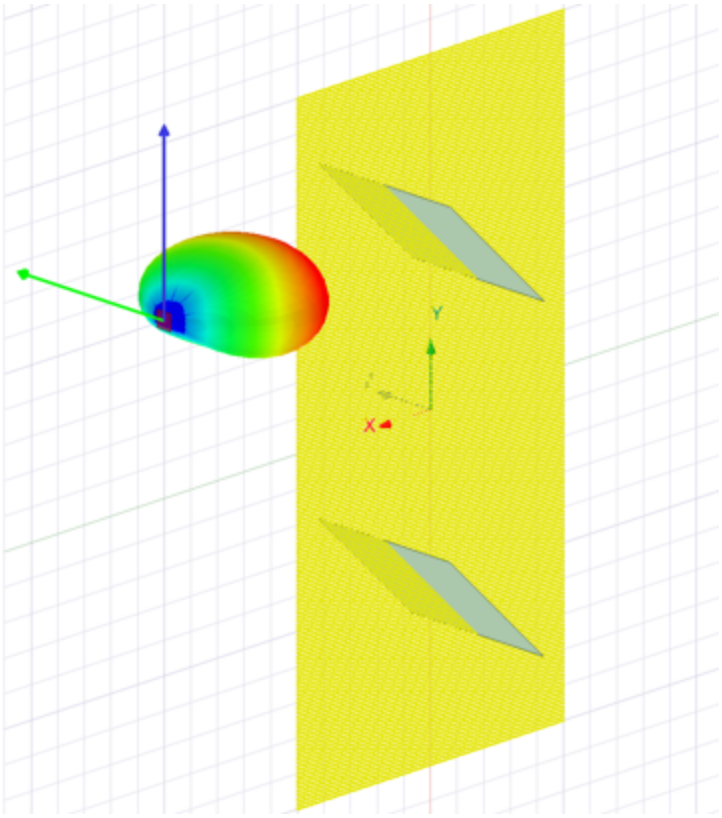
On the **GO Blockage** tab, you have check box options to enable or disable GO Blockage, Surface self-block, and GO Blockage Checks for Incident Fields..



Surface Self-Block: if enabled, the surface where the footprint is radiated from is used for blockage check, that is, anything below the surface with respect to the incoming ray is not in line of sight. If the setting is disabled, the footprint surface is not used for blockage checks (anything below the surface with respect to the incoming ray is in line of sight), but other surfaces are used for blockage.

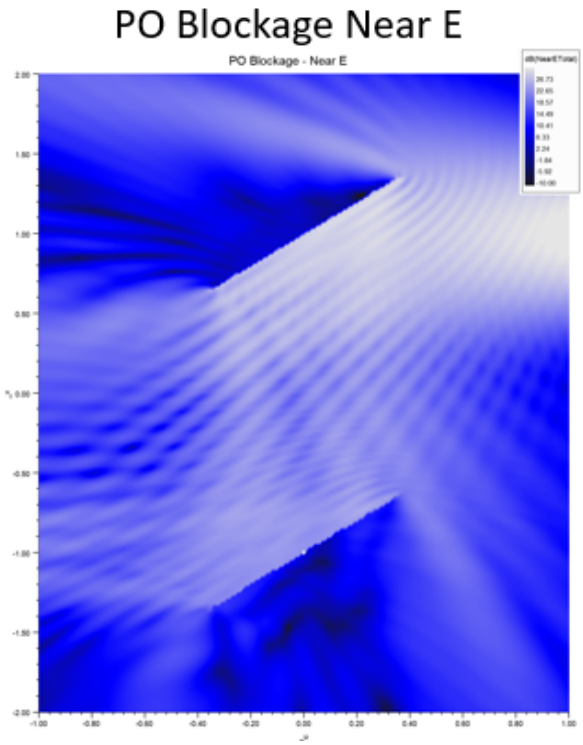
For example, consider a parametric beam antenna illuminating a series of plates.

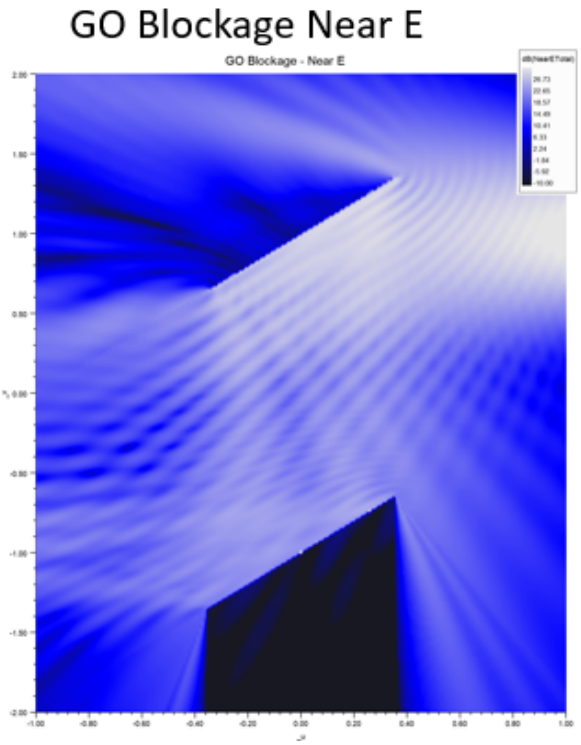
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the plots, the shadows behind the plates are much more pronounced with GO Blockage enabled, but the fields have non-physical discontinuities.

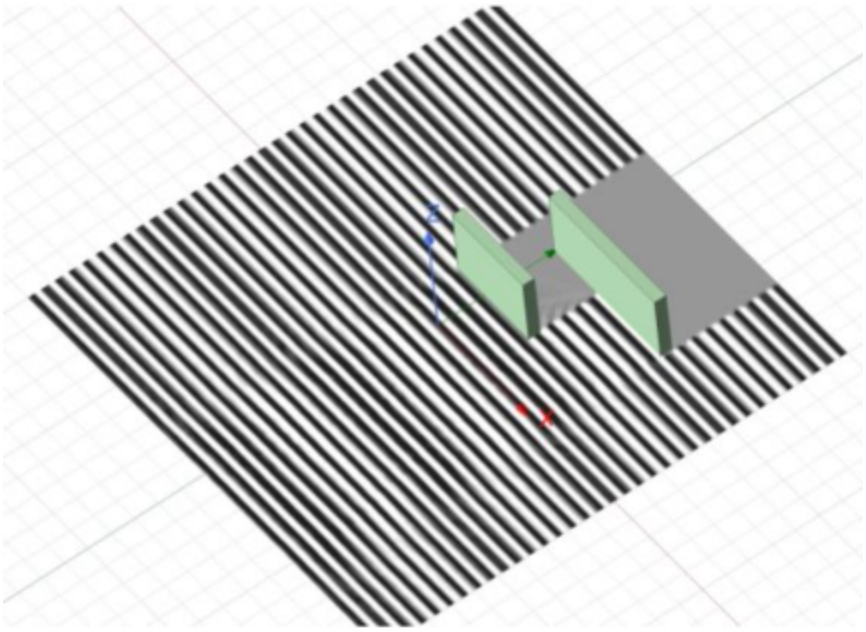
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





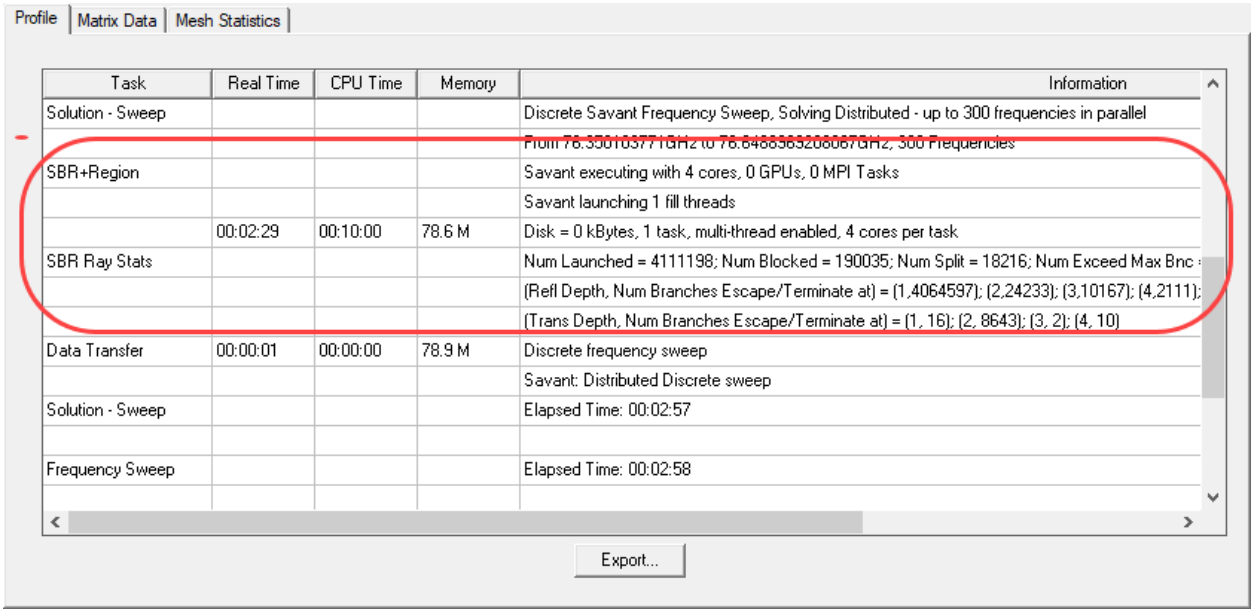
Add GO Blockage Checks to Incident Fields: if enabled,GO blockage effects will also be calculated for the incident field via line-of-sight checks from sources to field points, effectively computing a direct field in the Geometrical Optics sense. For example, the figure below shows the total E-field of a planewave incident on PEC bodies with this option enabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8. Perform Analysis and view Solution Data for SBR+ Solution

After analysis, you can view the **Solution** profile and mesh statistics. Because SBR+ regions cannot contain ports no matrix reports will be provided. Similarly 3D fields, near fields, and port field displays are not available.



SBR+ Ray Stats include:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

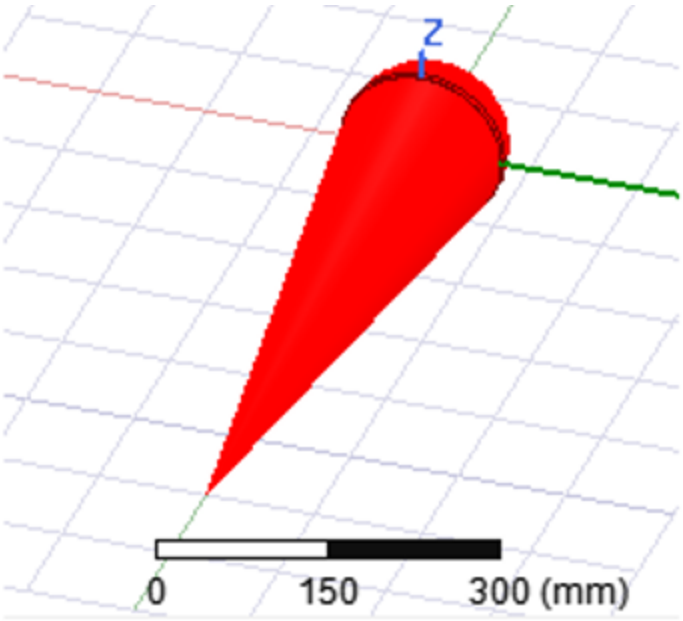
- Number of rays
- Histogram on Reflection
- Histogram on Transmission

If included in the solution, Creeping Wave Stats include

- Number of rays
- Ray lengths
- Source Counts by Initialization Failure Condition
- Ray Counts by Termination Condition
- Histogram On CW ray footprints

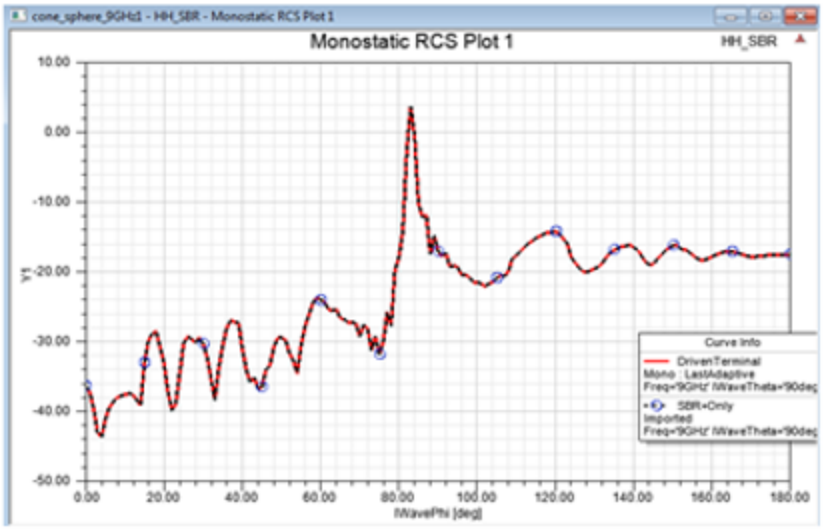
For details see Ray Statistics for SBR+ Analysis.

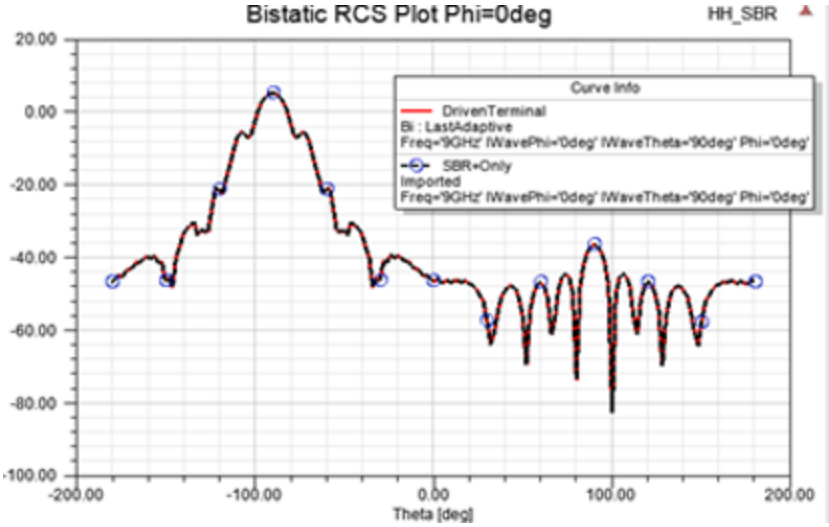
Matrix reports and Network data explorer are available if any parametric antenna or file based antenna is in the design. Far field reports (including antenna parameters) are available. 3D fields, near fields, and port field displays are not available. The following example shows monostatic and bistatic RCS reports for a cone sphere, comparing SBR+ compared to a driven terminal simulation. In a Driven Terminal design, the cone was assigned an SBR+ Region.



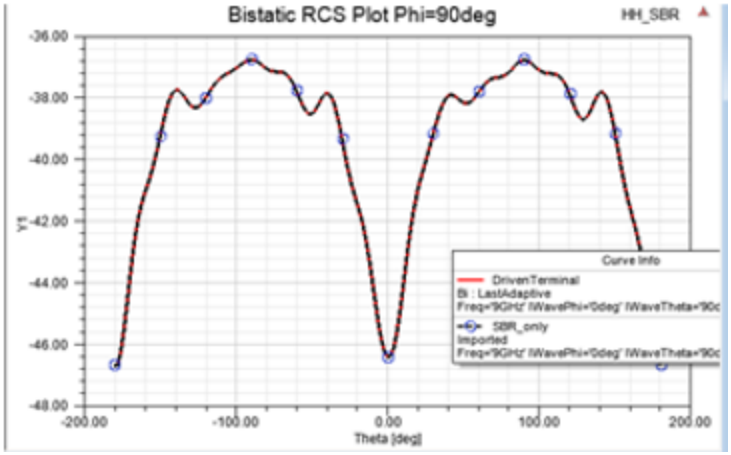
The following plots compare Bistatic RCS and Monostatic RCS in dB. The results match very well.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

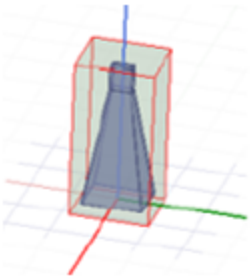




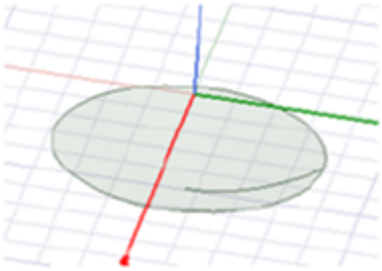
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following design shows Pyramidal Horn Antenna and FEM Parabola: SBR+ vs. HFSS IE Hybrid Region:

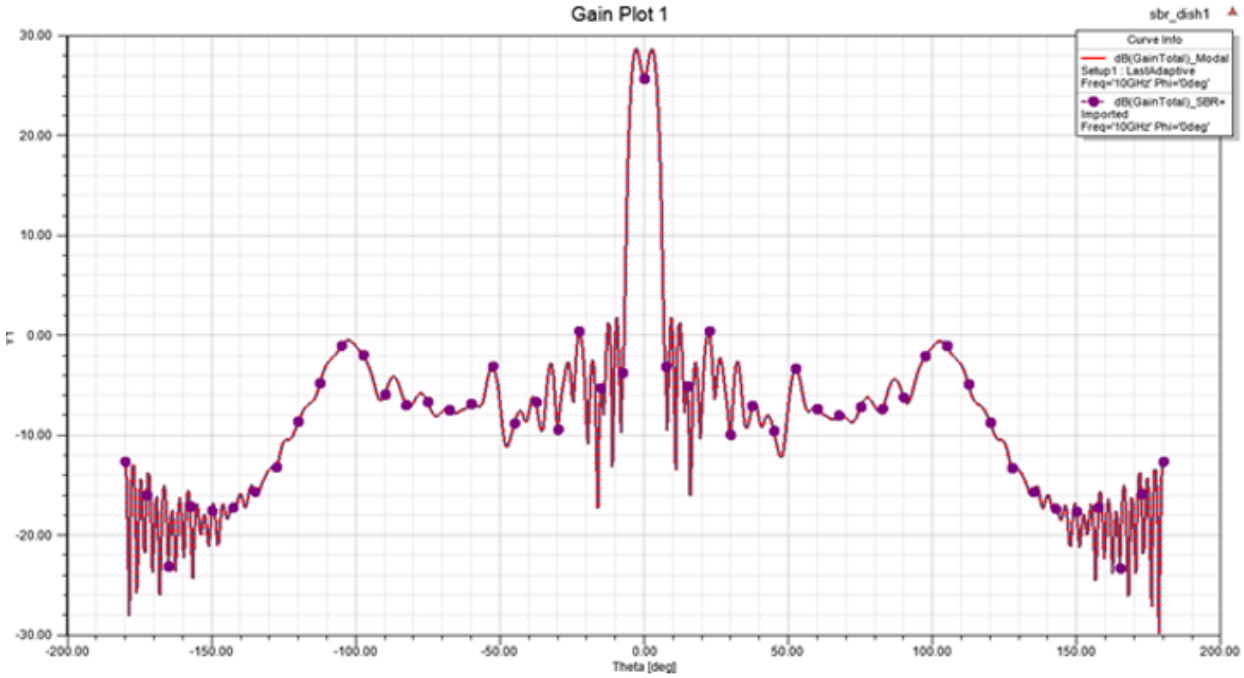


Pyramidal Horn Antenna



Parabola

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3D Components for and from SBR+ Designs

A new SBR+ 3D Component type is available. It can be a component created from an SBR+ design, or from a Driven Model/Terminal design while all solid and sheet geometries included are all assigned SBR+ Regions. These components always have the Mesh Assembly option available.

A 3D component containing only SBR+ region can be inserted into the SBR+ only design, irrespective to the solution type of the source design. A 3D component created from an SBR+ only design can be inserted into any design when SBR+ region is supported in that design's solution type.

If you create a 3D component from an SBR+ Solution type, that component does not contain a **Hybrid Regions** tab. However, if you create a component from an non-SBR+ Solution type, if Hybrid Regions are legal, it will contain a **Hybrid Regions** tab. Editing an SBR+ 3D Component will open the model in the corresponding source Solution Type.

Selecting the SBR+ Solution type also permits you to use Parametric Antennas.

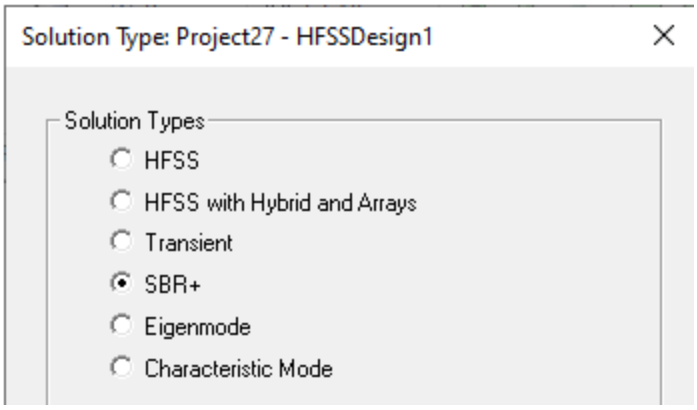
Configure Simulation with SBR+ Antenna Geometry Blockage (SBR+ Solution Type)

The typical configuration begins with a fully integrated model with all geometry. You then select the near antenna region including any nearby geometry interacting with the antenna for analysis

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

by HFSS – either by identifying these objects by hybrid region assignment, or by copying them out to a separate source antenna design to be solved with the HFSS solver, etc. The nearby antenna geometry could even be something different than what is used for the integrated model, such as a flat plate or other structure for considering the nearby antenna interactions. This source antenna design should likely be enclosed in a FE-BI region to allow for easier downstream processing, whether we are in a hybrid or SBR+ solution type. If you use a separate source antenna design, you can re-introduced it to the target design with the full geometry model by creating a 3D component for an antenna that is a linked design to the source design. This means you no longer need to designate any “Model Blockage” when setting up the linked design. You do this as a separate step outside creation of the linked design by selecting geometry from the SBR+ region to be designated as “Model Blockage” associated with a particular source antenna. Note there is no restriction on what objects and/or materials can be designated as “model blockage” geometry. Also note that for either solution type, SBR+ or hybrid HFSS-SBR+, that this selection will be from the available SBR+ region geometry objects, which is different than the previous workflow. Once you select these antenna blockage geometries for each source antenna, HFSS with SBR+ can solve the combined target SBR+ design.The preferred work flow is this.

1. Create/open an HFSS design with SBR+ solution type.



2. Add an Antenna to the design via the menu option for **3D Components > Create Antenna > Link to Source Design...** or **Excitations > Create Antenna Component > Link to Source Design...**

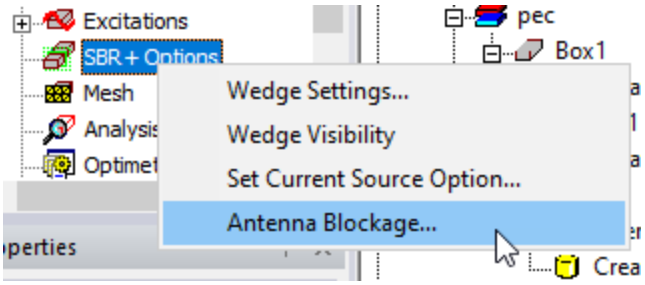
   This linked design should be an HFSS design that contains all the relevant geometry to represent nearby interactions with the antenna, such as a radome, enclosure, ground plane, mounting structure, or other nearby (within a few wavelengths) objects.

3. Add the desired SBR+ geometry objects to the design, configuring all materials/boundaries as desired.

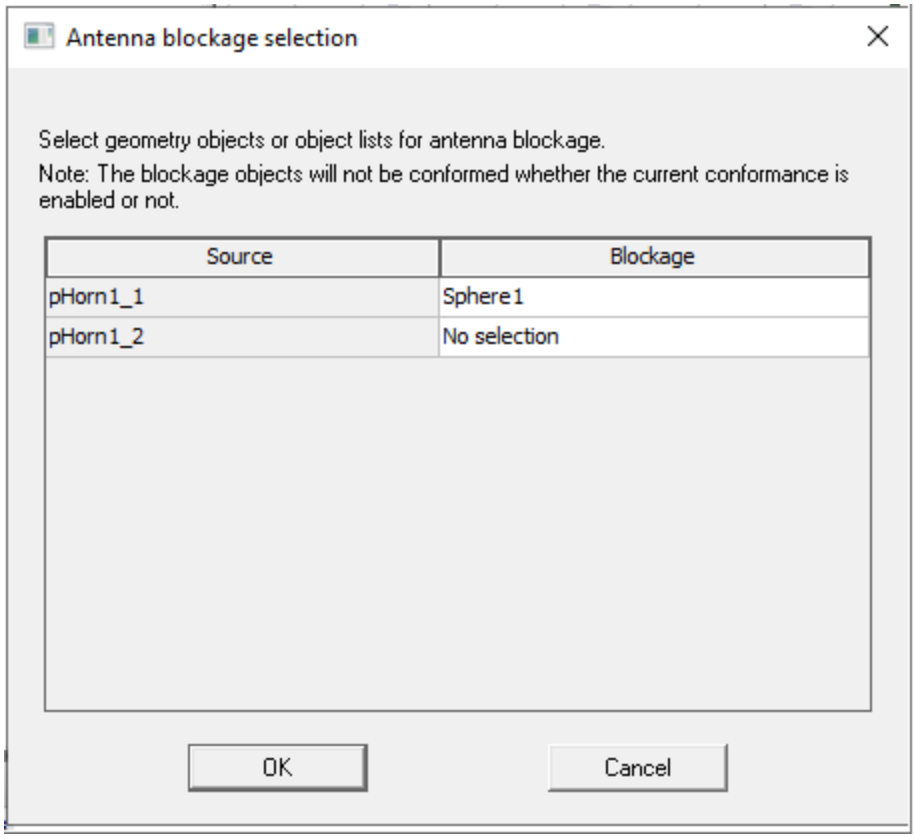
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If you want to designate multiple SBR+ geometry objects as blockage, select them in the 3D Modeler window and create an Object List by selecting **Modeler > List > Create > Object List**. You may want to name the list appropriately. If only a single object is to be designated as a geometry blockage, skip this step.

5. Using the right click menu under **SBR+ Options**, select **Antenna Blockage...**



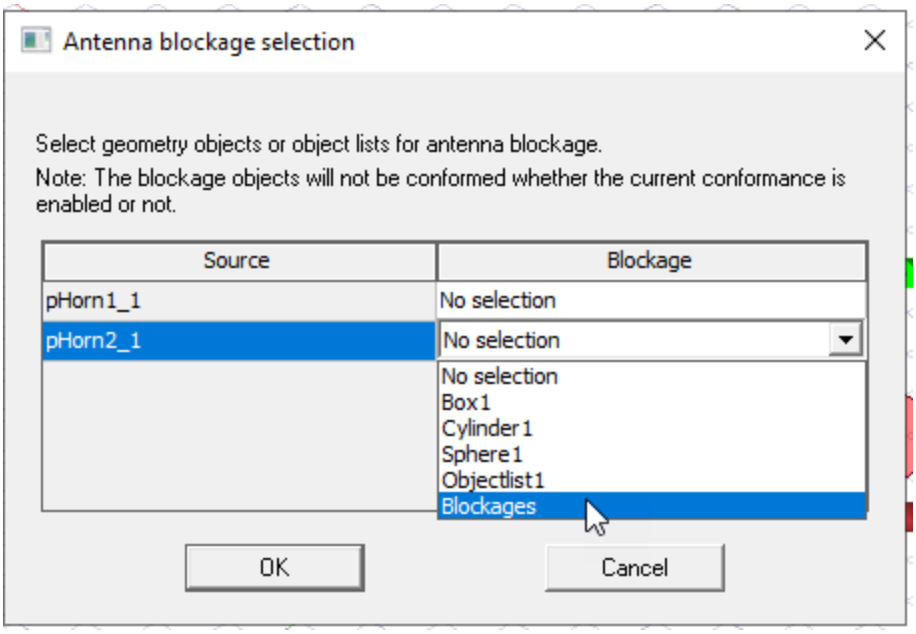
This opens the **Antenna blockage selection** dialog.



6. For each antenna source, use the dropdown menu for each cell in the column to select the geometry objects to associate with that source as an associated blockage. Each cell in the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

column contains a dropdown list of all geometry objects and all user created object lists.



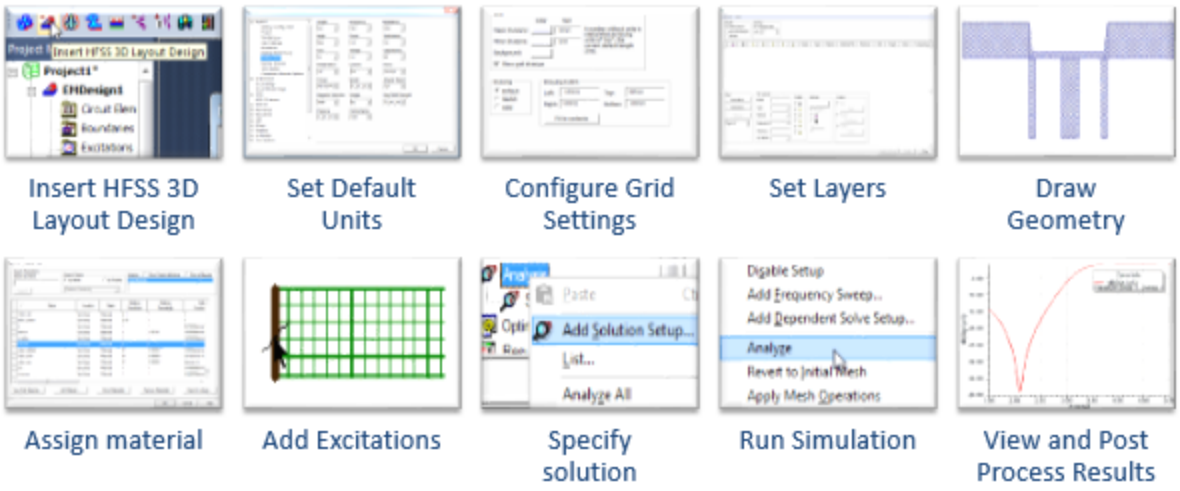
7. Click OK to save the configuration.
8. Create an Analysis Setup.
9. Run the simulation.

Workflow of an HFSS 3D Layout Design

Setting up an HFSS 3D Layout design is very similar to setting up an HFSS design or an HFSS-IE design. To set up an HFSS 3D Layout design, follow this general procedure. After you insert a design, you do not need to perform the intermediate steps sequentially, but you must complete them before you generate a solution.

Defining a solution setup enables HFSS 3D Layout to compute a solution for a design. You can define multiple solution setups for the same design. If you right-click in on the **Project Manager** window and select **Add Nexxim Solution Setup** , select a type of analysis to set up, which opens a solution setup window. You can specify multiple settings in the solution setup window for computing a solution for a design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



1. Insert an HFSS 3D Layout design into a project.
2. Draw the model geometry, specifically:
    a. Create the metal and dielectric layers of the design.
    b. Assign a thickness, roughness, elevation, and material to each layer.
    c. Draw the geometry of the signal layers.
    d. If applicable, create a cavity between two infinite negative signal planes.
3. Assign material characteristics
4. Define the ports or incident waves in the model.
5. If applicable, insert an external device, an *N-port*, with known *S*-parameters into your design.
6. Specify how to compute the solution by adding a solution setup.
7. (Optional) Set up any Optimetrics you want to run.
8. Run the simulation. HFSS 3D Layout computes the current inside the structure.
9. View the solution results, post-process results, and reports, and create field overlays.

Note:

Some of these tasks may not be applicable to your designs. Not all designs require cavities or *N*-ports.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

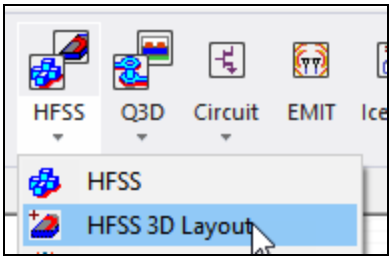
After a period of idleness of 10 minutes, HFSS gives up its license. A renewal of activity automatically requests a license. Such idle notifications do not occur during simulations.

Inserting an HFSS 3D Layout Design

This page briefly describes how to set up and run an HFSS 3D Layout design in the Ansys Electronics Desktop (AEDT)™ electronics systems design platform. The focus is on circuit and system analysis and the framework required to design RF-microwave, signal integrity, and printed circuit board applications. Before beginning, launch **Electronics Desktop** (**Electronics Desktop**), create a new project, and also enable the display of the **Choose Technology** window.

To insert an HFSS 3D Layout design:

1. From the **Project** menu, select **Insert HFSS 3D Layout Design** or in the **Desktop** ribbon, open the menu under the HFSS icon and select **HFSS 3D Layout** to open the **Choose Layout Technology** window.



So long as the check box **Show this window on new design** is selected, this window appears when a new design is inserted into a project. If the check box is cleared, the next time that a new design is inserted, the window does not appear and a design is inserted using the chosen technology on the last insertion.

1. Select an existing EM technology on the list or click **None** to manually add the model's metal and dielectric layers.

   A new design appears in the **Project Manager** window. The design is named HFSS3DLayout*n* by default, where *n* is the order of the design in a potential series of designs. Folders named **Data**, **Excitations**, **Analysis**, **Optimetrics**, and **Results** automatically appear under the design name. These folders store data about the design.

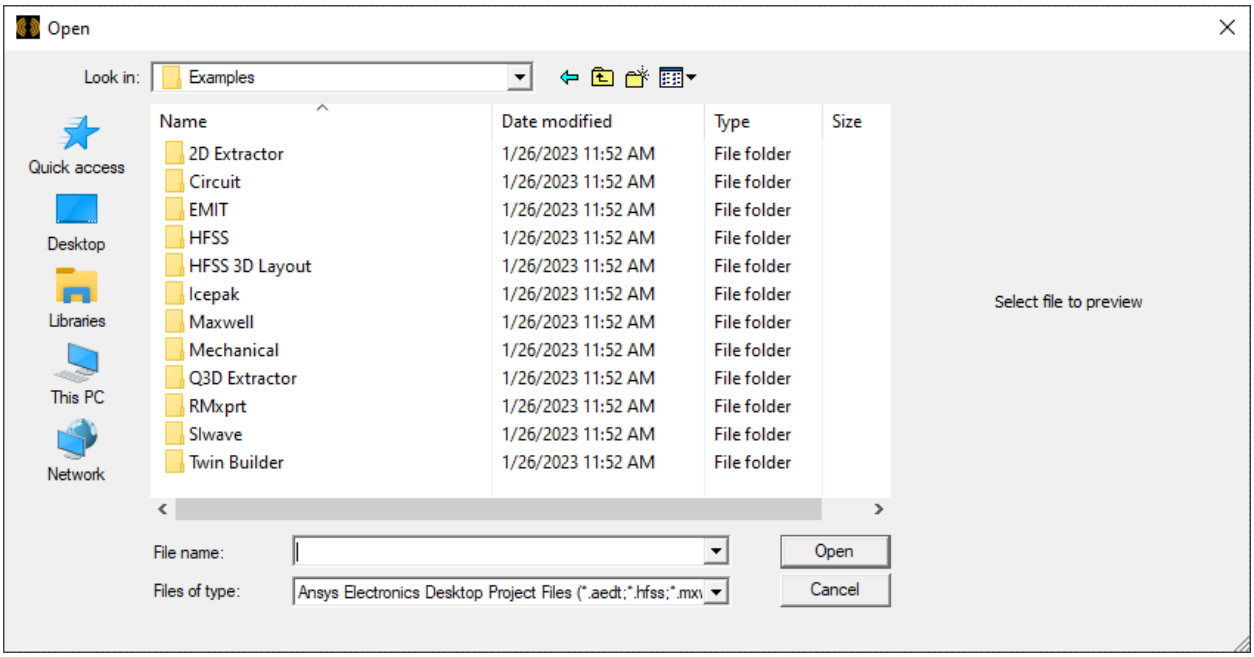
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To automatically insert an HFSS 3D Layout design when creating a new project, see **General Options > Desktop Configuration** to open the **Layout** view window. You can now create or edit the design layers in the Edit Layers window.

> **Note:**
>
> To expand the **Project Tree** and view specific data about the model such as its boundary assignments, click **+** to the left of the design icon in the **Project Manager** window.

Example Projects

Your Ansys Electronics Desktop installation includes an example directory containing projects folders for several kinds of designs.



Example projects are organized by the design type. The examples under Circuit, HFSS, and HFSS 3D Layout include subfolders, organizing the projects according to different applications:

- Circuit has subfolders for Automation, High Speed Digital, Low Noise Amplifier, RF Microwave, and Signal Integrity examples. The High Speed Digital folder has further subfolders for PCIe and USB projects. And the RF Microwave folder has further subfolders for Amplifiers, DC-IV, Filters, Misc, Mixers, Oscillators, and System projects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- HFSS has subfolders for ADAS, Antennas, EMI EMC, Filters, Metamaterial, RCS, RF Microwave, Signal Integrity, and Transmission Lines.
- HFSS 3D Layout has subfolders for Antennas, Component, Filters, FSS, and Signal Integrity.

Several of these projects are associated with detailed getting started guides.

- HFSS Examples Associated with a Getting Started Guide

Many other projects include descriptions in the help.

HFSS Examples

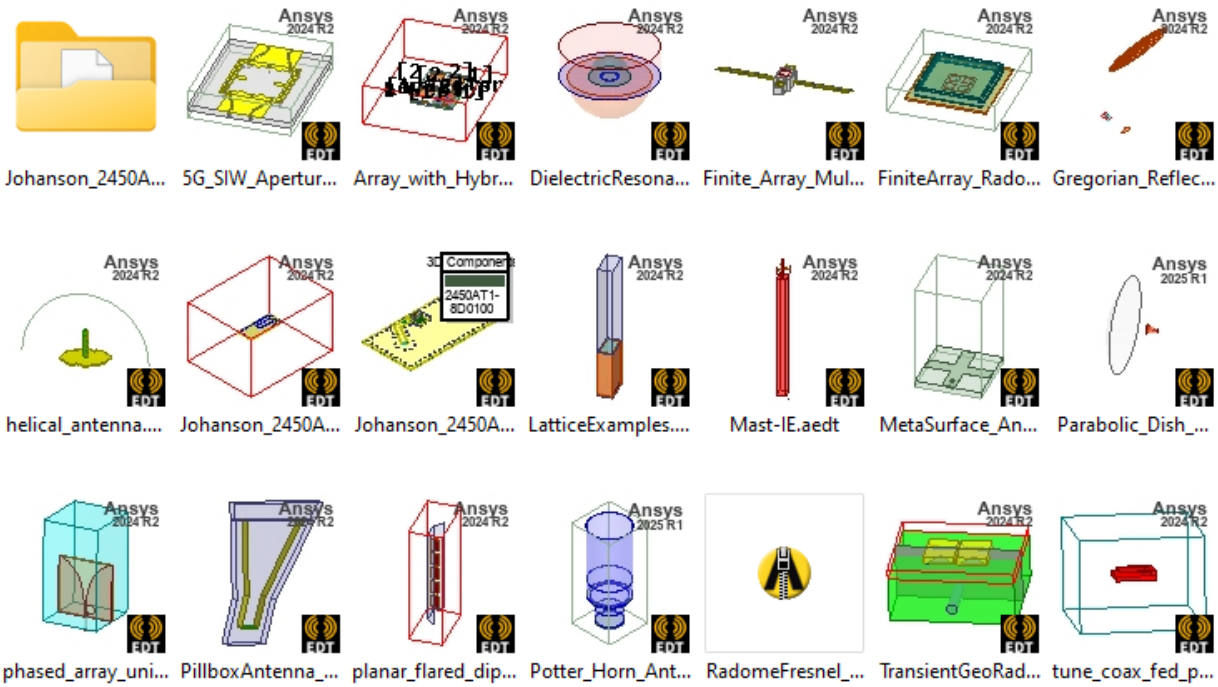
The example models are provided as an additional applications resource and are included with the software installation.

Many, but not all, of the following HFSS example projects have descriptions included in the help. They are organized by category—ADAS, Antennas, EMI EMC, RCS, Signal Integrity, Metamaterial, RF Microwave, and Transmission Lines. For models with included descriptions in the HFSS help, the icons in the figures are linked to the associated topic.

ADAS Examples (Advanced Driver Assistance Systems)

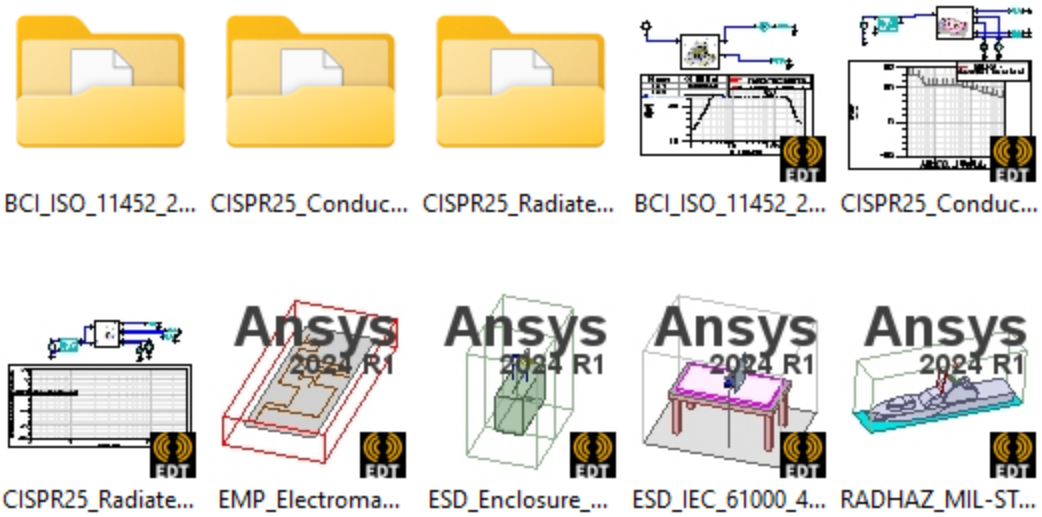
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Antenna Examples

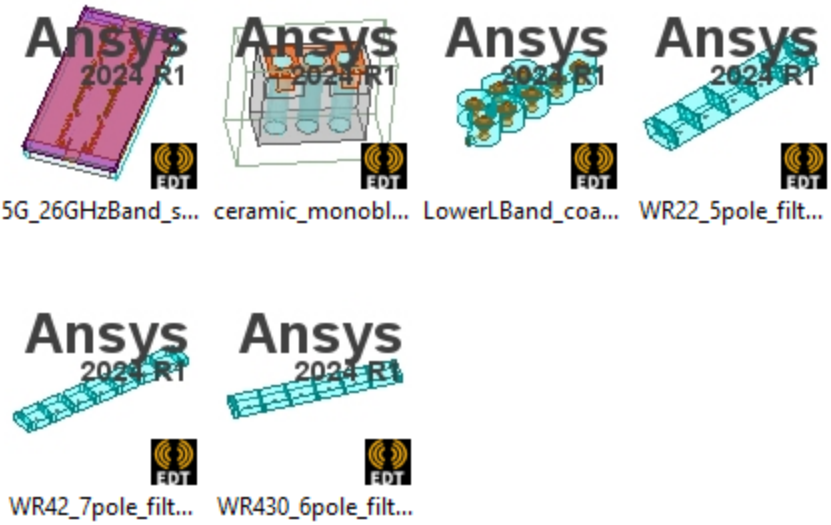


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EMI EMC Examples (Electromagnetic Interference, Electromagnetic Compatibility)



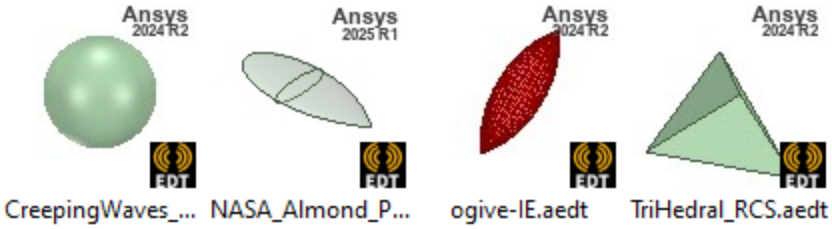
Filter Examples



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

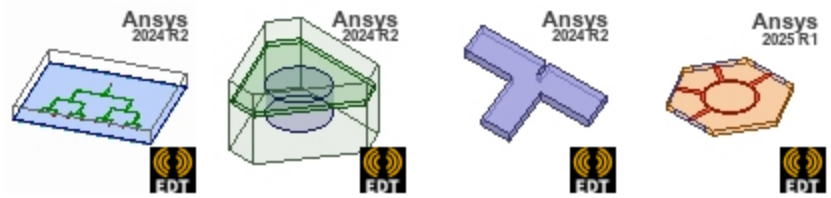
Metamaterial Examples

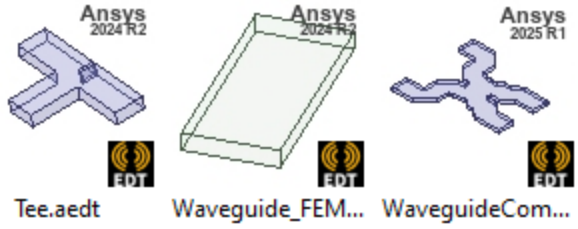
RCS Examples (Radar Cross Section)



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RF Microwave Examples

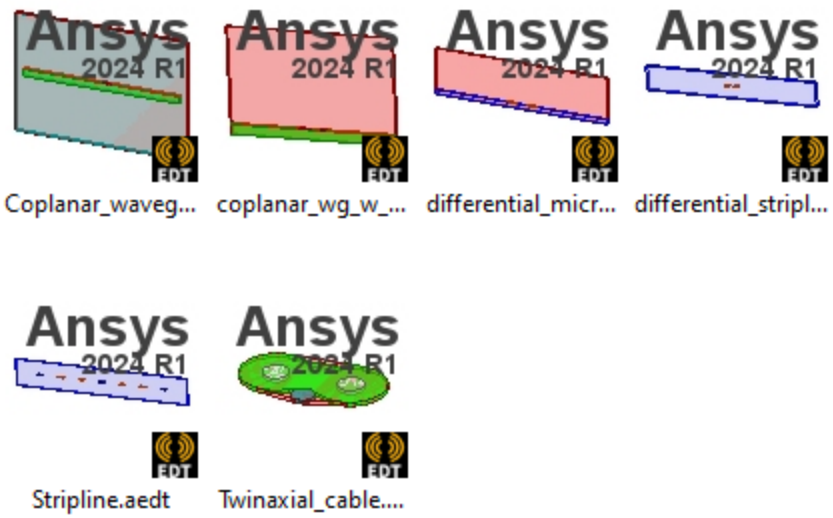




Signal Integrity Examples

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transmission Line Examples



For further examples, see the HFSS Getting Started Guides (link below).

Working with Variables

A variable is a numerical value, mathematical expression, or mathematical function that can be assigned to a design parameter in **HFSS**. You can assign a variable to any dimension, material property, or output value. Variables are useful in the following situations:

- You expect to change a parameter often.
- You expect to use the same parameter value often.
- You intend to run a parametric analysis, in which you specify a series of variable values within a range to solve.
- You intend to optimize a parameter value by running an optimization analysis or using Design of Experiments to generate a response surface.
- You intend to run a convergence on an output variable.
- You intend to calculate derivatives for variables.
- You intend to animate a plot against a variable.

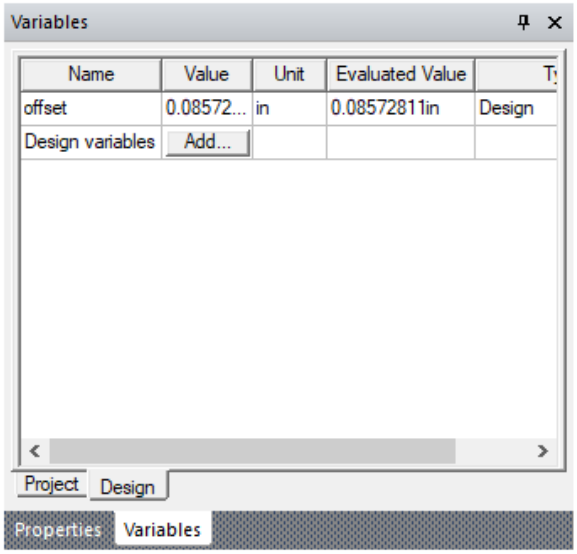
Variables can be regular variables, array index variables, or arrays. A project variable's value can reference other project variables or intrinsics. Values can be a number, variable, or mathematical expression.

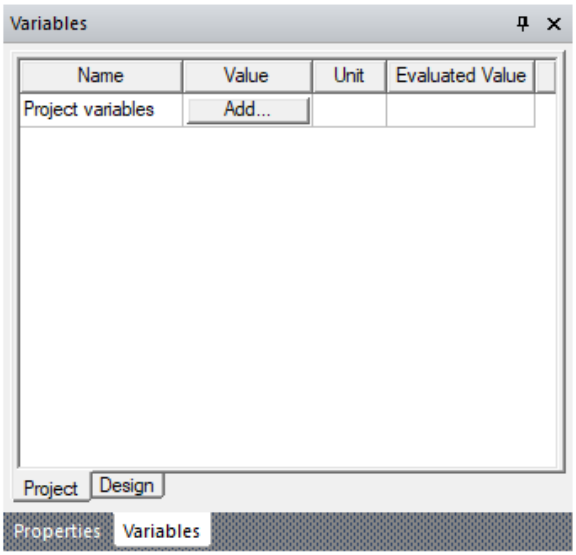
There are two types of variables in HFSS:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Project Variables	A project variable can be assigned to any parameter value in the HFSS project in which it was created. HFSS differentiates project variables from other types of variables by prefixing the variable name with dollar sign symbol ($). You can manually include the $ in the project variable's name, or HFSS will automatically append the project variable's name after you define the variable. Project variables can be designated as Design, ArrayIndex or Separator variables but not as Post-Processing Variables.
Design (Local) Variables	A design variable can be assigned to any parameter value in the HFSS design in which it was created. From the Design Variables **Properties** dialog box, you can Add, Add Array, Edit, or Remove design variables. Design Variables can be designated as Design, ArrayIndex or Separator variables and as Post-Processing Variables.

Clicking **View** > **Variables** brings up a dockable variable window that is associated with the active project and/or design. When there is an active project, there will be a corresponding project variable tab. When there is an active design, there will be a corresponding design variable tab. Each tab contains an **Add...** button allowing creation of new variable of this type. If variables exist for the Project or Design, they are shown in the corresponding tab.

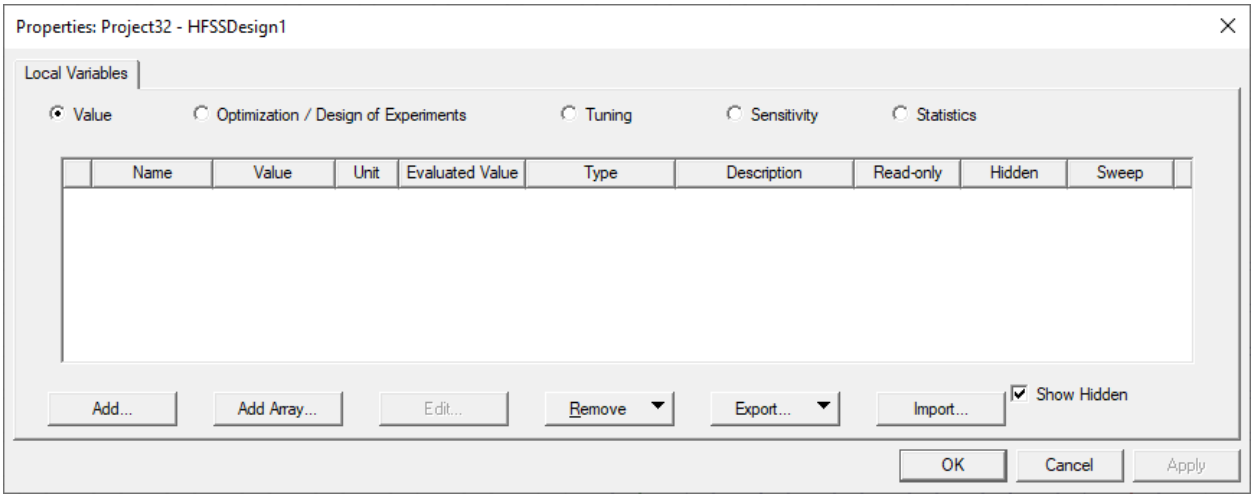




Importing and Exporting Variables

Once you have defined Design and/or Project Variables, you can export them to a csv file, and import them to another project or design. If there are naming conflicts on import, these are flagged as errors. The **Properties** dialogs for Design Variables and Project variables include **Import...** and **Export...** buttons.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Working with Project Variables

Project variables are available across all hierarchical levels of a project. A project variable can be assigned to a parameter value in the HFSS project in which it was created. HFSS differentiates project variables from other types of variables by prefixing the variable name with the dollar sign symbol ($). You can manually include the $ in the project variable's name when you create it, or HFSS will automatically append it after you define the variable.

The **Project Variables** menu item allows you to access three tabs:

- **Project Variables** – variables, which you define, that apply to the entire project.
- **Intrinsic Variables** – pre-defined variables that cannot be changed. See: List of Intrinsic Variables.
- **Constants** – pre-defined constants that cannot be changed. See: List of Constants.

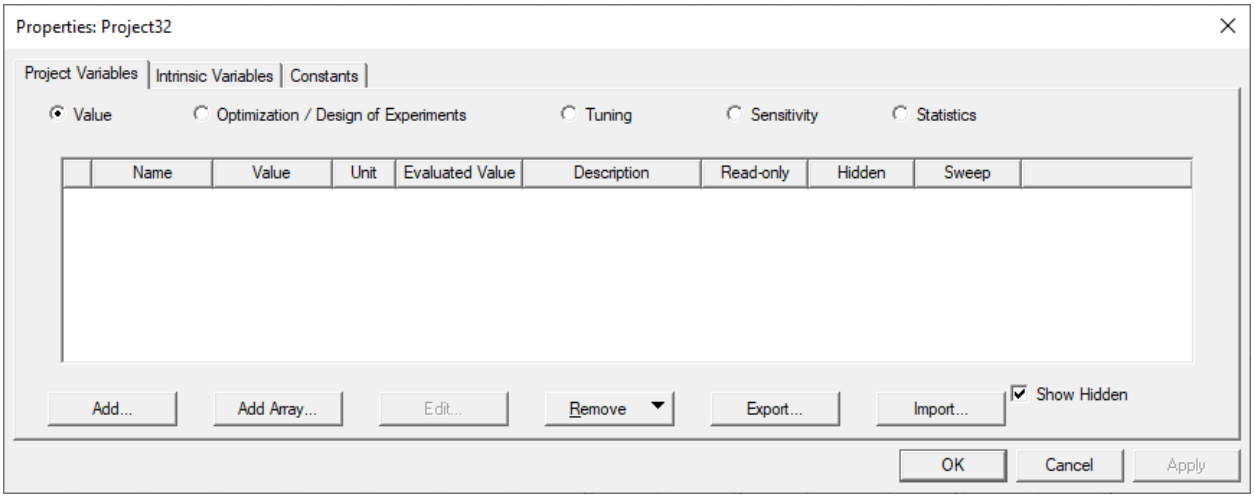
Adding a Project Variable

To define a Project Variable:

1. Access **Project Variables** one of three ways:
   - Click **Project** > **Project Variables**.
   - Right-click the project name in the Project Tree and select **Project Variables**.
   - From a menu in the lower left corner of the following Optimization dialogs: **Parametric**, **Optimization**, **Sensitivity**, **Statistical**, **Design of Experiments**, and **Design Xplorer Setup**. Click **Edit Variables** and from the menu select **Edit Project Variables**.

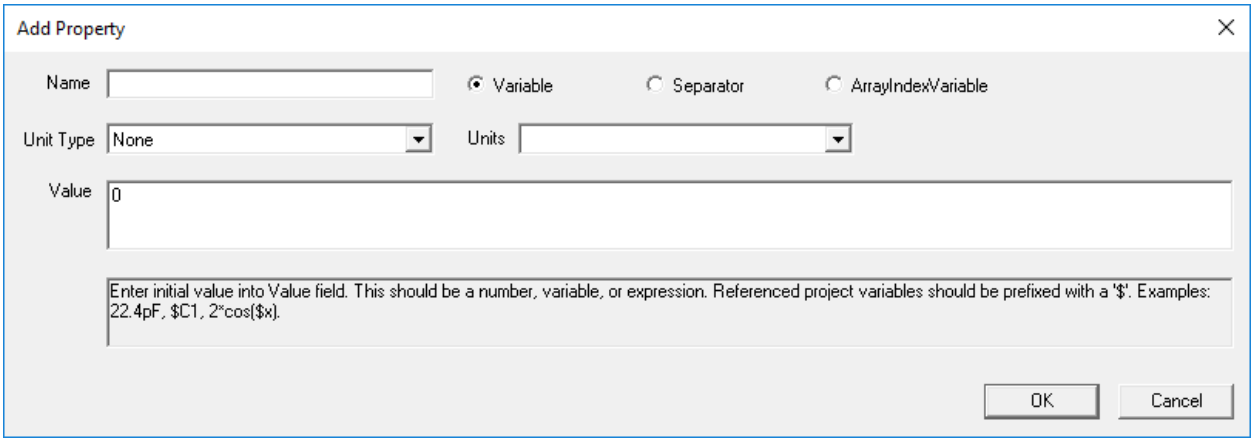
   The **Properties** window appears, on the **Project Variables** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. From the **Project Variables** tab, click **Add**.

   The **Add Property** dialog box appears.



3. Enter information for the variable, as applicable:
   - **Name** – project variable names must start with the dollar sign symbol ($), followed by a letter. The name can contain alphanumeric characters and underscores (_). You cannot use the names of Intrinsic Variables or pre-defined Constants.
   - **Variable Type** – use the **Variable**, **Separator**, and **ArrayIndexVariable** radio buttons to select the variable type.

     Your selection impacts which properties you can edit:

Variable Type	Editable Properties
Variable	Unit Type, Units, Value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Variable Type	Editable Properties
Separator	Name. A separator variable provides a bolded name for a blank line to facilitate grouping variables in variable lists.
Array Index Variable	Associate Array Variable, Value

- **Unit Type** – for Variables, use the drop-down menu to select a type from the list (for example, Charge, Density, Energy, ...). “None” is the default.

  When you select a Unit Type, the choices in drop-down menu for the Units text box adapt to that unit type. For example, selecting Length as the Unit Type causes the Unit menu to show a range of metric and english units for length. Similarly, if you select the Unit Type as Resistance, the Units drop down lists a range of standard Ohm units.

- **Units** – for Variables, use the drop-down menu to select a unit of measure.
- **Value** – for Variables and ArrayIndexVariables, enter a number, variable, or mathematical expression. The quantity entered will be the current (or default) value for the variable. If the mathematical expression includes a reference to an existing variable, this variable is treated as a dependent variable. The units for a dependent variable will automatically change to those of the independent variable on which the value depends. Additionally, dependent variables, though useful in many situations, cannot be the direct subject of optimization, sensitivity analysis, tuning, or statistical analysis.

  For ArrayIndexVariables, the index reference can be a constant (for example, 1), an index (for example, ii) or an expression (for example, ii + 1). This allows you to sweep the index and simulate for different values that are stored in the array variable itself. In particular, it also enables you to sweep different text strings. This allows you to set a property to different string values as the index is swept.

  > **Warning:**
  >
  > If you include the variable's units in its definition (in the **Value** text box), do not include the variable's units when you enter the variable name for a parameter value.

- **Associate Array Variable** – for ArrayIndexVariables, select an associated array from the drop-down list. You must have previously created an array.

4. Click **OK**.

The new variable appears in the list. You can sort project variables by clicking the Name column header. Clicking once sorts them in ascending order, noted by a triangle pointing

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

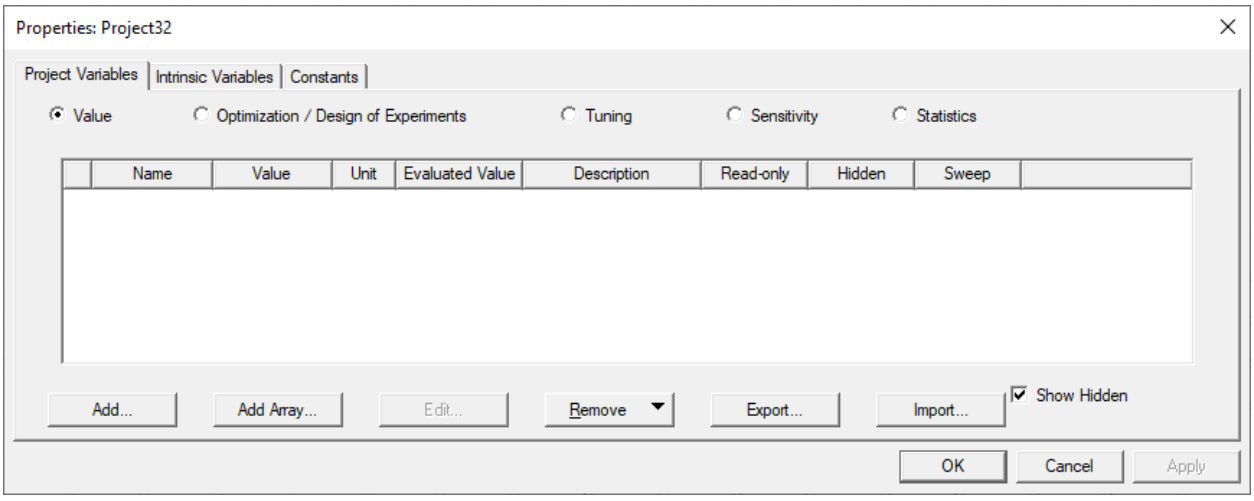
up. Clicking against sorts in descending order, noted by a triangle pointing down. Clicking a third time sorts in original order, with no triangle.

5. If desired, use the check boxes to designate a variable as **Read-only**, **Hidden**, or **Sweep**.
   - **Read-only** – when selected, the variable's name, value, unit, and description cannot be modified.
   - **Hidden** – hidden variables do not appear in the **Properties** window unless **Show Hidden** is selected.
   - **Sweep** – allows you to designate variables to include in solution indexing as a way to permit faster post-processing. Variables with the Sweep check box cleared are not used in solution indexing. If a solution exists, selecting or clearing a variable's Sweep setting produces a warning that the change will invalidate existing solutions. To continue, click OK to dismiss the warning.
6. Click **Apply** to apply changes.
7. Click **OK** to exit the window.

The new variable can be assigned to a parameter value in the project in which it was created. You can enable defined variables for Optimization/Design of Experiments, Tuning, Sensitivity, or Statistics. See: Optimetrics.

Importing and Exporting Project Variables

Once you have defined Project Variables, you can export them to a csv file, and import them to another project or design. If there are naming conflicts on import, these are flagged as errors. The **Properties** dialogs for and Project variables include **Import...** and **Export...** buttons.



When you click **Export....** a dialog lets you name the file and select a location. When you **Save**, if the file exists, an overwrite prompt should pop up.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Deleting a Project Variable

To remove a Project Variable:

1. Access **Project Variables** as described above.

   The **Properties** window appears, on the **Project Variables** tab.

2. Remove one or more variables:
   - To remove a specific variable, select it and click **Remove** > **Remove Selected**.
   - To remove all unused variables, click **Remove** > **Remove All Unused**.
   - To force the removal of all unused variables, including those in the project's undo/redo history, click **Remove** > **Force Remove All Unused**.

Editing a Project Variable

To edit a Project Variable:

1. Access **Project Variables** as described above.

   The **Properties** window appears, on the **Project Variables** tab.

2. Select the variable you want to edit and click **Edit**.
3. Change the properties as desired and click **OK**.

Adding a Project Variable Array

You can define array variables that contain numbers or strings. Number array variables can be used in component property expressions, while string array variables can be used in certain component property values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Text array variables can be used in certain component property values, but not all, and cannot be combined with operators to form more complex expressions.

The index for a text array variable reference can either be a constant (1) or can be an index (ii) or even an expression (ii + 1). This allows you to sweep the index and simulate for different values that are stored in the array variable itself. In particular, it also enables you to sweep different text strings. This allows you to set a property to different string values as the index is swept. The following are properties that currently allow text array variables:

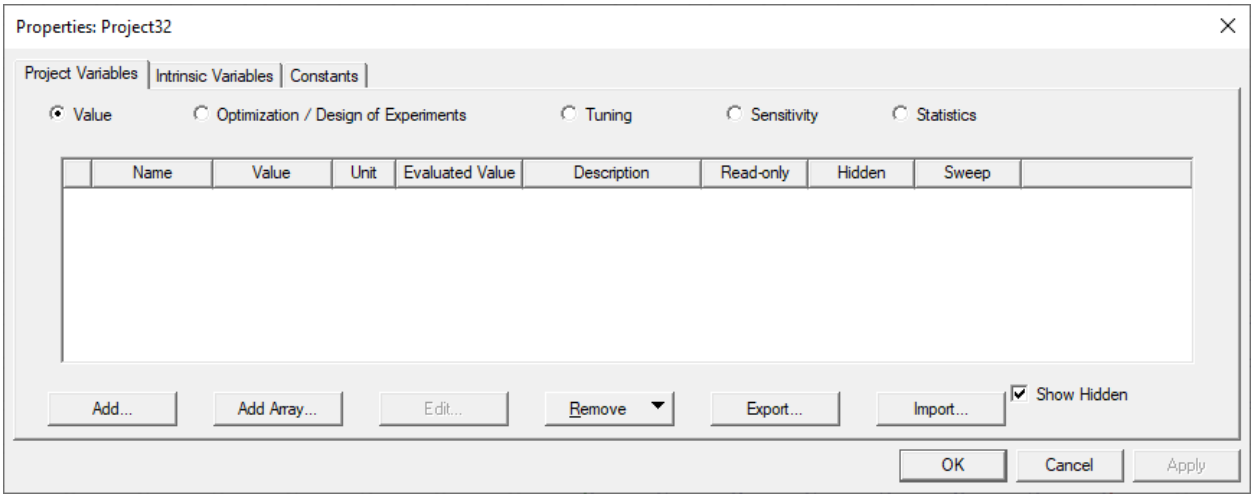
- V_PRBS
- V_PRBS_JITTER
- V_PRBSD
- V_PRBSG_JITTER
- V_PSK
- V_QAM
- V_CPM

To define an array variable, you must first define a Project Variable Array:

1. Access **Project Variables** one of three ways:
    - Click **Project** > **Project Variables**.
    - Right-click the project name in the Project Tree and select **Project Variables**.
    - From a menu in the lower left corner of the following Optimization dialogs: **Parametric**, **Optimization**, **Sensitivity**, **Statistical**, **Design of Experiments**, and **Design Xplorer Setup**. Click **Edit Variables** and from the menu select **Edit Project Variables**

    The **Properties** window appears, on the **Project Variables** tab.

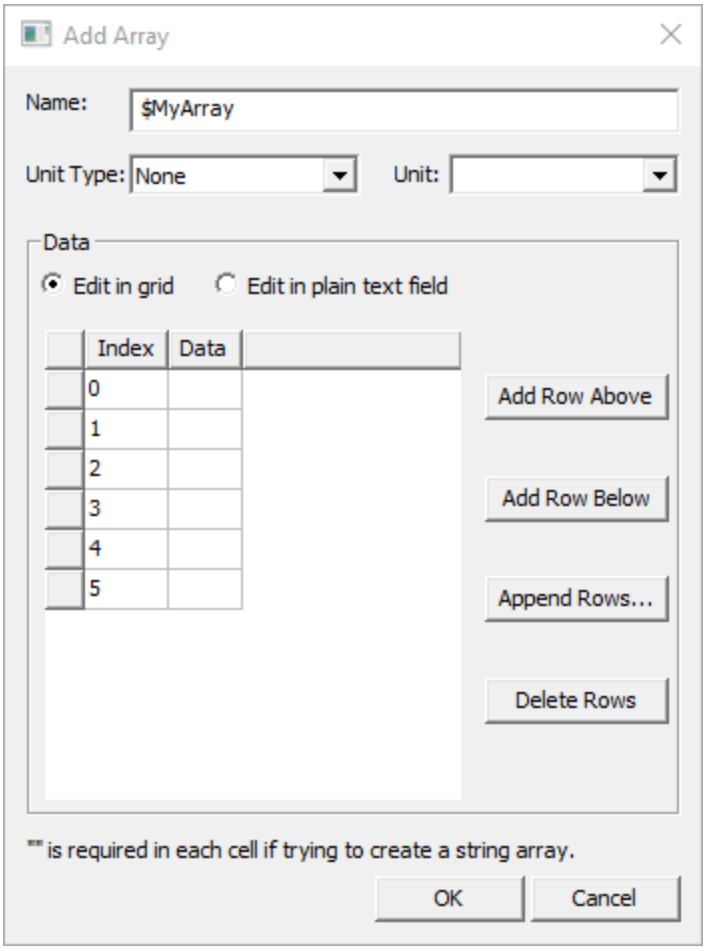
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **Add Array**.

   The **Add Array** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The value grid displays each array item's **Index** number, and the **Data** associated with that index number.

3. Give the variable array a **Name**, and select a **Unit Type** and **Unit** from the drop-down menus.
4. Use the control buttons to add, delete, and reposition rows in the value grid at left. The default is **Edit in grid**, but you may select **Edit in plain text field**.

> **Note:**
>
> Quotation marks (“ ”) are *required* as delimeters when array values are entered in either the grid or text field.

5. When you have finished entering values, click **OK**.

   The array appears in the **Project Variables** list.

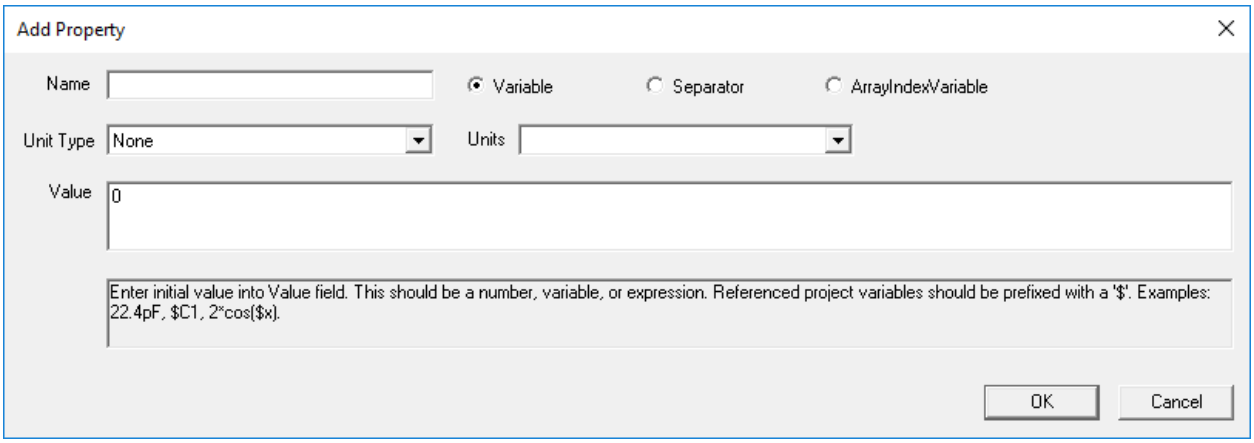
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can add a variable to the array by adding an ArrayIndexVariable on the **Project Variables** tab.

1. Access **Project Variables** one of three ways:
   - Click **Project** > **Project Variables**.
   - Right-click the project name in the Project Tree and select **Project Variables**.
   - From the **Setup Optimization** dialog box, click **Edit Variables** > **Edit Project Variables**.

   The **Properties** window appears, on the **Project Variables** tab.
2. From the **Project Variables** tab, click **Add**.

   The **Add Property** dialog box appears.



3. Enter information for the variable, as applicable:
   - **Name** – project variable names must start with the dollar sign symbol ($), followed by a letter. The name can contain alphanumeric characters and underscores (_). You cannot use the names of Intrinsic Variables or pre-defined Constants.
   - **Variable Type** – select ArrayIndexVariable.
   - **Value** – enter a number, variable, or mathematical expression. The quantity entered will be the current (or default) value for the variable. If the mathematical expression includes a reference to an existing variable, this variable is treated as a dependent variable. The units for a dependent variable will automatically change to those of the independent variable on which the value depends. Additionally, dependent variables, though useful in many situations, cannot be the direct subject of optimization, sensitivity analysis, tuning, or statistical analysis.

     For ArrayIndexVariables, the index reference can be a constant (for example, 1), an index (for example, ii) or an expression (for example, ii + 1). This allows you to sweep the index and simulate for different values that are stored in the array

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

variable itself. In particular, it also enables you to sweep different text strings. This allows you to set a property to different string values as the index is swept.

> **Warning:**
>
> If you include the variable's units in its definition (in the **Value** text box), do not include the variable's units when you enter the variable name for a parameter value.

- **Associate Array Variable** – for ArrayIndexVariables, select an associated array from the drop-down list. You must have previously created an array.

4. Click **OK**.

List of Intrinsic Variables

AEDT recognizes a set of intrinsic variables that can be used to define expressions. Intrinsic variable names are reserved and may not be used as user-defined variable names.

To view the list of Intrinsic Variables, click **Project** > **Project Variables** and select the **Intrinsic Variables** tab.

Name	Unit	Description
_Empty		Empty value, taken to be model default by simulator (Twin Builder)
_I1	mA	Armature current; **this variable is only used in RMxprt**.
_I2	mA	Exciting current; **this variable is only used in RMxprt**.
_I3	mA	Current; **this variable is only used in RMxprt**.
_I4 through _I9		Not used
_t,**_u**,**_v**		Variables to define parametric equation-based curve.
_V1 to **_V9**	mV	Port Voltage in user-defined model (V).
Ang	Ang	Angle. Postprocessing variable
Budget Index		Postprocessing variables; cannot be set by the user
Distance	mm	
Electrical Degree	deg	Electric degree of the rotating machine; cannot be set by the user.
F	GHz	Frequency of the circuit/system analysis
F1, **F2**, **F3**	GHz	Frequency tones 1, 2, 3 in harmonic balance analysis. (Hz).
FNoi	GHz	Offset noise frequency in harmonic balance noise analysis.
Freq	Hz	Frequency for analysis and postprocessing; cannot be set by the user

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Unit	Description
Hmax	ns	
Hmin	ns	
Ia, **Ib**	mA	Postprocessing variables; cannot be set by the user
Index		Identifier for a data point; cannot be set by the user
IWave Phi, **IWave Theta**	deg	Incident wave spherical coordinate variables. Phi is the angle from the origin in the z direction and Theta the angle from the x-axis.
Normalized Deformation, **Normalized Distance**		Postprocessing variables; cannot be set by the user
OP	mW	Postprocessing variables; cannot be set by the user
Pass		Post processing variables, not settable by the user.
Phase	deg	Angle of a complex number.
Phi	deg	The angle measured from the x-axis and can be from 0 to 360 degrees.
R	mm	R is the cylindrical coordinate system variable.
Rho		Rho is the spherical coordinate system variable.
Rspeed	rpm	Speed of the machine.
Spectrum	GHz	Postprocessing variable.
Temp	cel/deg	Analysis temperature (deg)
Tend	ns	
Theta	deg	The angle measured from the z-axis, which is the axis perpendicular to the plane of the work space, and must be from 0 to 180 degrees.
Time	ns	Time point in transient analysis
Time0	ns	Time 0 point in a transient analysis
Vac, **Vbe**, **Vce**, **Vds**, **Vgs**	mV	Post processing variables; cannot be set by the user
X,**Y**,**Z**	mm	Point coordinate variables in the modeler.
ZAng, **XRho**	deg	A spherical coordinate system; cannot be set by the user

List of Constants

To view the list of constants, click **Project** > **Project Variables** and select the **Constants** tab.

Name	Value	Description
abs0	-273.15	Absolute zero (C)
boltz	1.3806503e-23	Boltzmann constant (J/K)
c0	299792458	Speed of light in a vacuum (m/s)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Value	Description
e0	8.854187817e-12	Permittivity of vacuum (F/m)
elecq	1.602176462e-19	Electron charge (C)
eta	376.730313461	Impedance of vacuum (Ohm)
false	0	Boolean False
g0	9.80665	Acceleration due to gravity of Earth (m/s^2)
mathE	2.718281828	Euler's number (Napier's constant)
pi	3.1415926535898	Ratio of circumference to diameter
planck	6.6260755e-34	Planck's constant (m^2*kg/s)
true	1	Boolean True
u0	1.2566370614359e-06	Permeability of vacuum (H/m)

Working with Design Variables

Design Variables (Local Variables) are only available within the HFSS design for which they are defined. A design variable can be assigned to a parameter value in the design. You can also add a variable defined with an array of values.

The **Design Properties** menu item allows you to access up to three tabs, depending on your design:

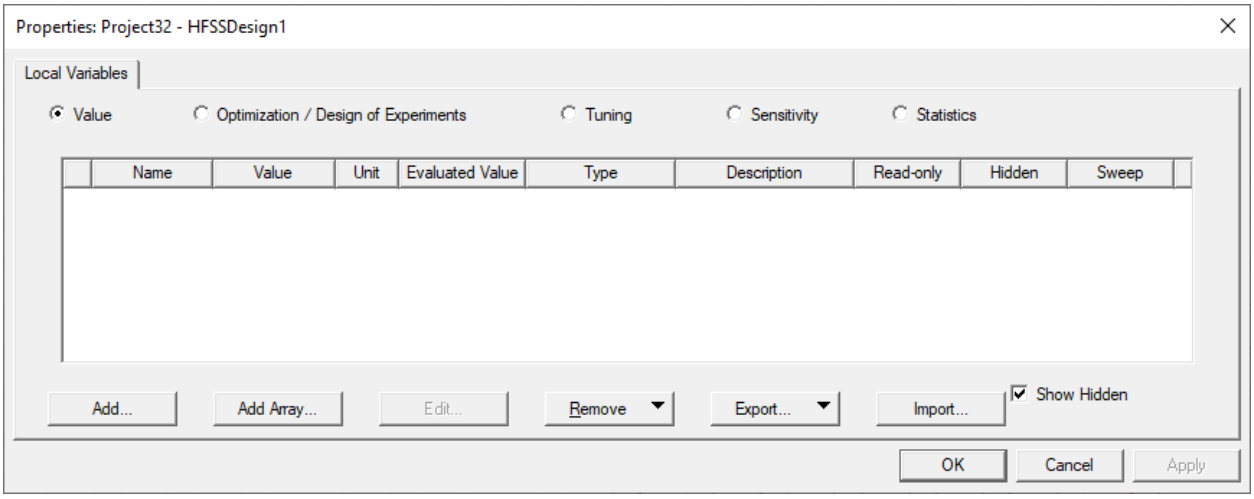
- **Parameter Defaults** – local variables with default values that can be overridden in instances of a design. For example, if three subcircuit instances contain a parameter default C1 that is defined as equal to 11.3pF, C1 may be overridden as 11.8pF in the first instance, overridden as 10.9pF in the second, and left at its default value of 11.3pF in the third. A property value that has been set by means of a parameter default is called a *passed parameter*. Parameter defaults can be defined at the design level.
- **Local Variables** – variables, which you define, that apply to the current design only. Local variables can be defined at the design level.
- **General** – pre-defined design variables that cannot be changed. You cannot use the names of these variables when creating a new variable.

> **Note:**
>
> Not all tabs appear for all designs.

Importing and Exporting Variables

Once you have defined Design Variables, you can export them to a csv file, and import them to another project or design. If there are naming conflicts on import, these are flagged as errors. The Properties dialogs for Design Variables include **Import...** and **Export...** buttons.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Defining Local Variables at the Design Level

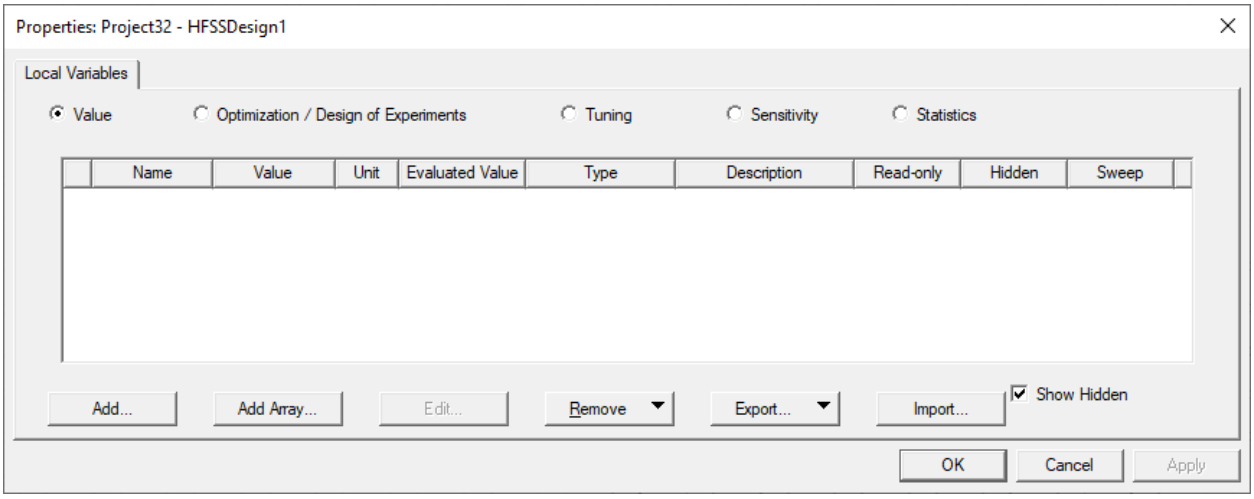
Adding a Local Variable

To define a Local Variable:

1. Access **Design Properties** one of three ways:
   - Click **HFSS** > **Design Properties**.
   - Right-click the design name in the Project Tree and select **Design Properties**.
   - From a menu in the lower left corner of the following Optimization dialogs: **Parametric**, **Optimization**, **Sensitivity**, **Statistical**, **Design of Experiments**, and **Design Xplorer Setup**. Click **Edit Variables** and from the menu select **Edit Design Variables**

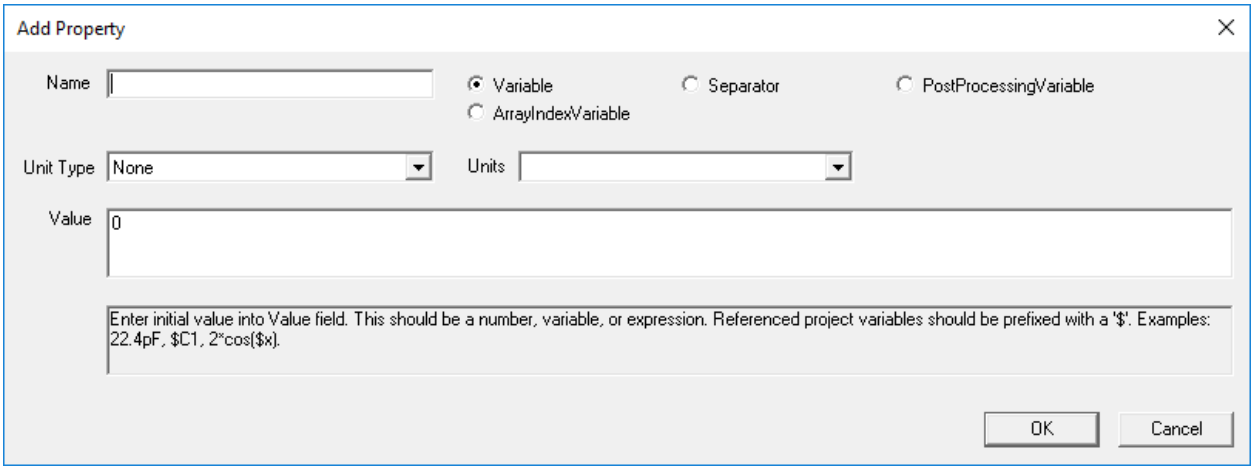
   The **Properties** window appears with the **Local Variables** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **Add**.

   The **Add Property** window appears.



3. Enter information for the variable, as applicable:
   - **Name** – the name must start with a letter and can contain alphanumeric characters and underscores (_). You cannot use any names already defined on the **General** tab.
   - **Variable Type** – use the **Variable**, **Separator**, **ArrayIndexVariable**, and **PostProcessingVariable** radio buttons to select the variable type.

     > **Note:**
     >
     > Not all variable types are available for all designs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Your selection impacts which properties you can edit:

Variable Type	Editable Properties
Variable	Unit Type, Units, Value.
Separator	Name. A separator variable provides a bolded name for a blank line to facilitate grouping variables in variable lists.
Array Index Variable	Associate Array Variable, Value
PostProcessingVariable	Unit Type, Units, Value.

- **Unit Type** – for Variables and PostProcessingVariables, use the drop-down menu to select a type from the list (for example, Charge, Density, Energy, ...).
- **Units** – for Variables and PostProcessingVariables, use the drop-down menu to select a unit of measure.
- **Value** – for Variables, ArrayIndexVariables, and PostProcessingVariables, enter a number, variable, or mathematical expression. The quantity entered will be the current (or default) value for the variable. If the mathematical expression includes a reference to an existing variable, this variable is treated as a dependent variable. The units for a dependent variable will automatically change to those of the independent variable on which the value depends. Additionally, dependent variables, though useful in many situations, cannot be the direct subject of optimization, sensitivity analysis, tuning, or statistical analysis.

  For ArrayIndexVariables, the index reference can be a constant (for example, 1), an index (for example, ii) or an expression (for example, ii + 1). This allows you to sweep the index and simulate for different values that are stored in the array variable itself. In particular, it also enables you to sweep different text strings. This allows you to set a property to different string values as the index is swept.

  > **Warning:**
  >
  > If you include the variable's units in its definition (in the **Value** text box), do not include the variable's units when you enter the variable name for a parameter value.

- **Associate Array Variable** – for ArrayIndexVariables, select an associated array from the drop-down list. You must have previously created an array.

4. Click **OK**.

The new variable appears in the list. You can sort local variables by clicking the Name column header. Clicking once sorts them in ascending order, noted by a triangle pointing up. Clicking against sorts in descending order, noted by a triangle pointing down. Clicking a third time sorts in original order, with no triangle.

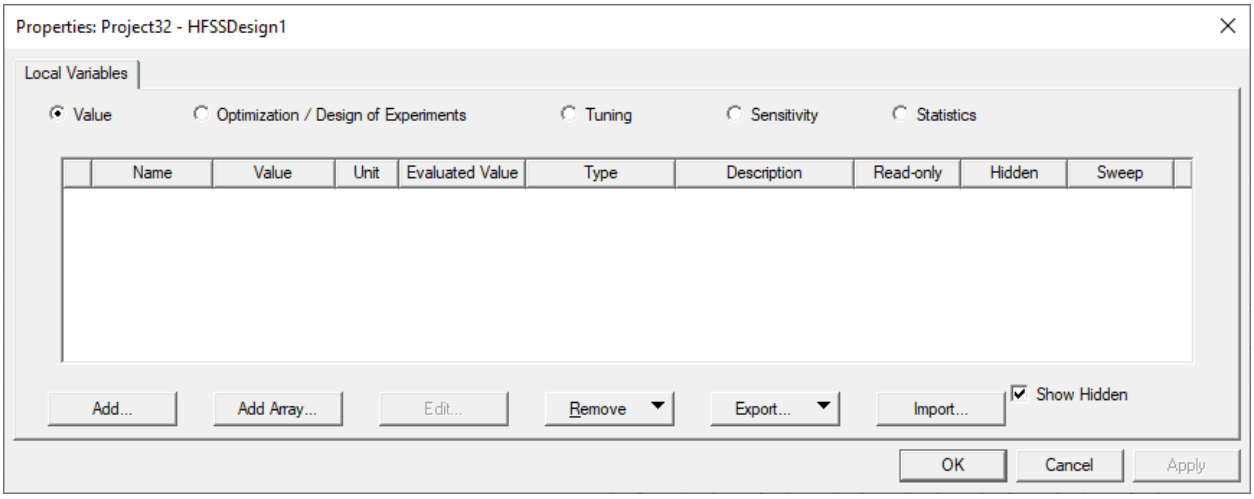
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. If desired, use the check boxes to designate a variable as **Read-only**, **Hidden**, or **Sweep**.
    - **Read-only** – when selected, the variable's name, value, unit, and description cannot be modified.
    - **Hidden** – hidden variables do not appear in the **Properties** window unless **Show Hidden** is selected.
    - **Sweep** – allows you to designate variables to include in solution indexing as a way to permit faster post-processing. Variables with the Sweep check box cleared are not used in solution indexing. If a solution exists, selecting or clearing a variable's Sweep setting produces a warning that the change will invalidate existing solutions. To continue, click OK to dismiss the warning.
6. Click **Apply** to apply changes.
7. Click **OK** to exit the window.

The new variable can be assigned to a parameter value in the project in which it was created. You can enable defined variables for Optimization/Design of Experiments, Tuning, Sensitivity, or Statistics. See: Optimetrics.

Importing and Exporting Variables

Once you have defined Design Variables, you can export them to a csv file, and import them to another project or design. If there are naming conflicts on import, these are flagged as errors. The Properties dialogs for Design Variables include **Import...** and **Export...** buttons.



Deleting a Local Variable

To remove a Local Variable:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Access **Local Variables** as described above.
2. Remove one or more variables:
    - To remove a specific variable, select it and click **Remove** > **Remove Selected**.
    - To remove all unused variables, click **Remove** > **Remove All Unused**.
    - To force the removal of all unused variables, including those in the project's undo/redo history, click **Remove** > **Force Remove All Unused**.

Editing a Local Variable

To edit a Local Variable:

1. Access **Local Variables** as described above.
2. Select the variable you want to edit and click **Edit**.
3. Change the properties as desired and click **OK**.

Overriding a Local Variable

You can override a variable value from an analysis setup's **General** tab.

Select the **Override** check box next to the value you wish to override, and enter a new value in the **Value** field.

Design Variable	Override	Value	Units
padrad	☐	0.6	mm
viarad	☐	0.2	mm

Defining Parameter Defaults at the Design Level

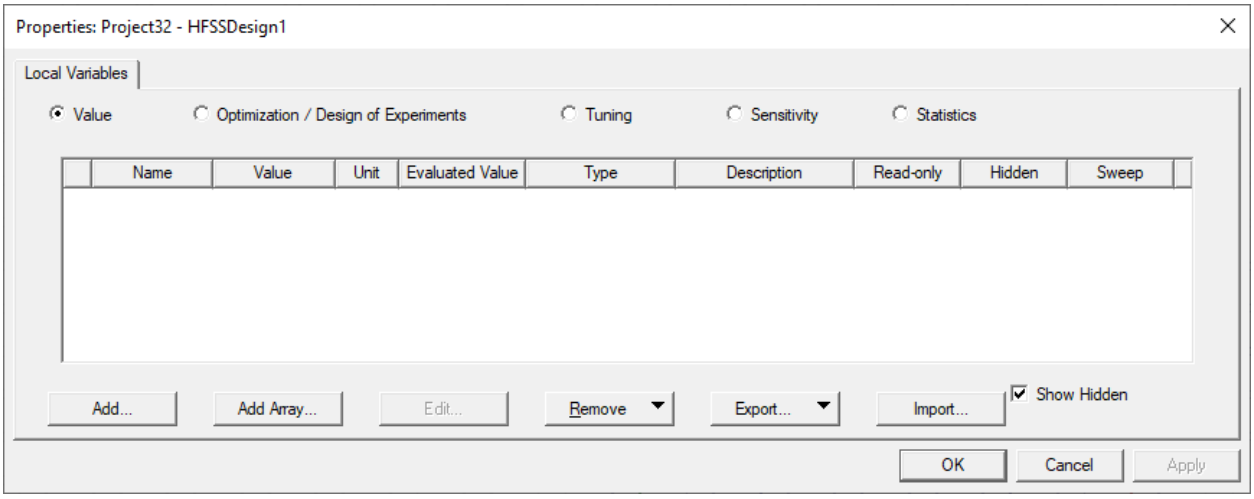
Adding a Parameter Default

To define a new parameter default at the design level:

1. Access **Design Properties** one of three ways:
    - Click **HFSS** > **Design Properties**.
    - Right-click the design name in the Project Tree and select **Design Properties**.
    - From a menu in the lower left corner of the following Optimization dialogs: **Parametric**, **Optimization**, **Sensitivity**, **Statistical**, **Design of Experiments**, and **Design Xplorer Setup**. Click **Edit Variables** and from the menu select **Edit Design Variables**

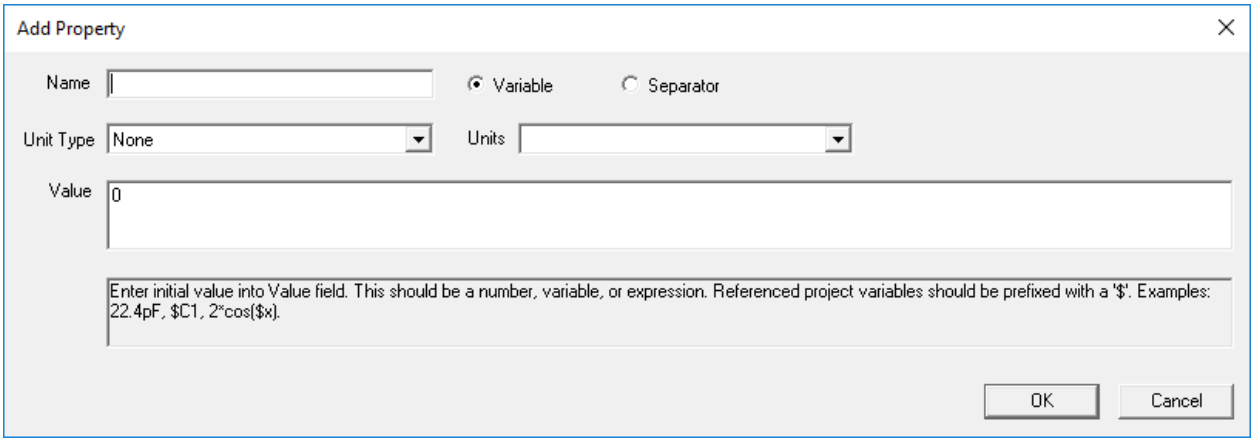
    The **Properties** window for design variables appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **Add...**.

   The **Add Property** window appears.



3. Enter information for the parameter default, as applicable:
   - **Name** – the name can contain alphanumeric characters and underscores (_). You cannot use any names already defined on the **General** tab.
   - **Variable Type** – select **Variable** to enter a value, or select **Separator** to provide a bolded name for a blank line.

     Your selection impacts which properties you can edit:

Variable Type	Editable Properties
Variable	Unit Type, Units, Value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Variable Type	Editable Properties
Separator	Name. A separator variable provides a bolded name for a blank line to facilitate grouping variables in variable lists.

- **Unit Type** – for Variables, use the drop-down menu to select a type from the list (for example, Charge, Density, Energy, ...).
- **Units** – for Variables, use the drop-down menu to select a unit of measure.
- **Value** – for Variables, enter a valid numeric quantity. Alternately, you can assign a variable to a parameter by typing a variable name or mathematical expression in place of a parameter value.

4. Click **OK**.

   The new variable appears in the list. You can sort local variables by clicking the Name column header. Clicking once sorts them in ascending order, noted by a triangle pointing up. Clicking against sorts in descending order, noted by a triangle pointing down. Clicking a third time sorts in original order, with no triangle.

5. If desired, use the check boxes to designate a variable as **Read-only**, **Hidden**, or **Sweep**.
   - **Read-only** – when selected, the variable's name, value, unit, and description cannot be modified.
   - **Hidden** – hidden variables do not appear in the **Properties** window unless **Show Hidden** is selected.
6. Click **Apply** to apply changes.
7. Click **OK** to exit the window.

Deleting a Parameter Default

To remove a Parameter Default:

1. Access **Parameter Defaults** as described above.
2. Select a parameter from the list and click **Remove**.

Editing a Parameter Default

To edit a Parameter Default:

1. Access **Parameter Defaults** as described above.
2. Select the parameter you want to edit and click **Edit**.
3. Change the properties as desired and click **OK**.

Overriding a Parameter Default

You can override a parameter default from an analysis setup's **General** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select the **Override** check box next to the value you wish to override, and enter a new value in the **Value** field.

Design Variable	Override	Value	Units
padrad	☐	0.6	mm
viarad	☐	0.2	mm

Adding a Design Variable Array

You can define array variables that contain numbers or strings. Number array variables can be used in component property expressions, while string array variables can be used in certain component property values. Design variable arrays are only available within the HFSS design for which they are defined.

> **Note:**
>
> Text array variables can be used in certain component property values, but not all, and cannot be combined with operators to form more complex expressions.
>
> The index for a text array variable reference can either be a constant (1) or can be an index (ii) or even an expression (ii + 1). This allows you to sweep the index and simulate for different values that are stored in the array variable itself. In particular, it also enables you to sweep different text strings. This allows you to set a property to different string values as the index is swept. The following are properties that currently allow text array variables:
>
> - V_PRBS
> - V_PRBS_JITTER
> - V_PRBSD
> - V_PRBSG_JITTER
> - V_PSK
> - V_QAM
> - V_CPM

To define an array variable, you must first define a Local Variable Array:

1. Access **Local Variables** one of three ways:
    - Click **HFSS** > **Design Properties**.
    - Right-click the design name in the Project Tree and select **Design Properties**.

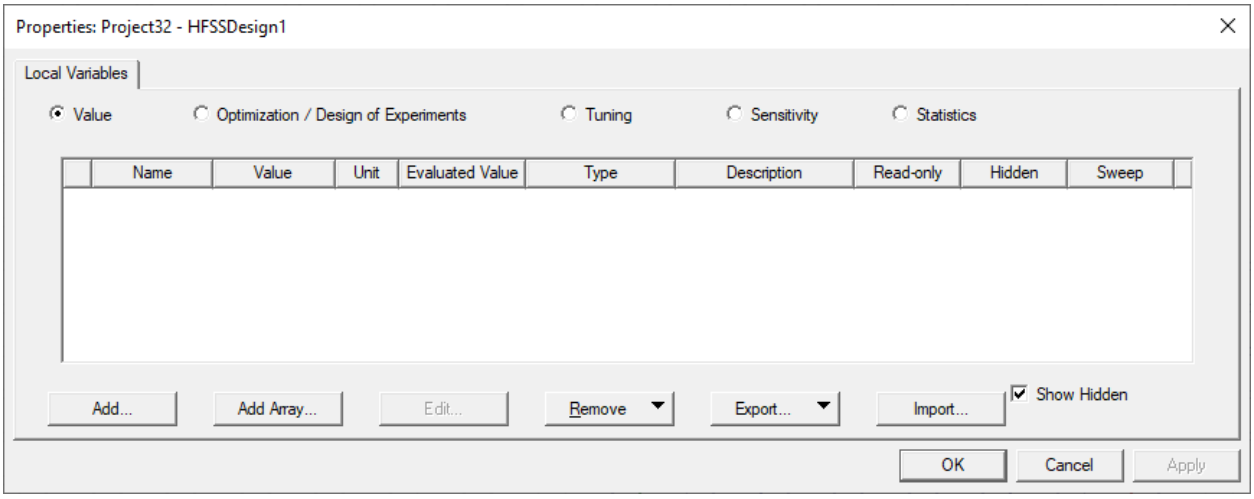
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From a menu in the lower left corner of the following Optimization dialogs: **Parametric**, **Optimization**, **Sensitivity**, **Statistical**, **Design of Experiments**, and **Design Xplorer Setup**. Click **Edit Variables** and from the menu select **Edit Design Variables**

The **Properties** window appears.

2. Select the **Local Variables** tab.

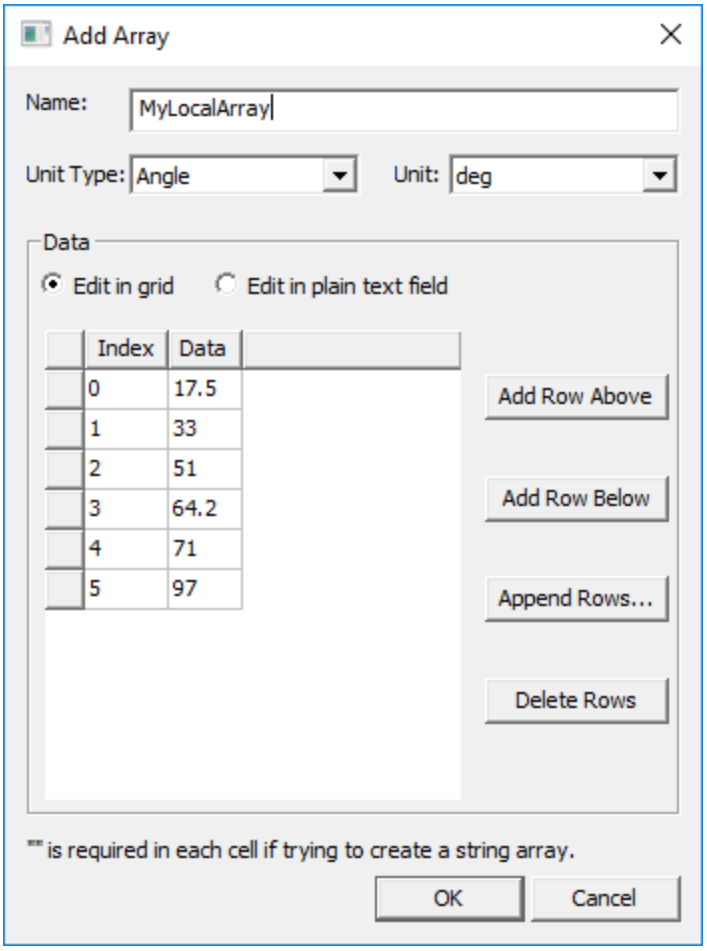
   The **Local Variables** tab displays.



3. Click **Add Array**.

   The **Add Array** window appears.

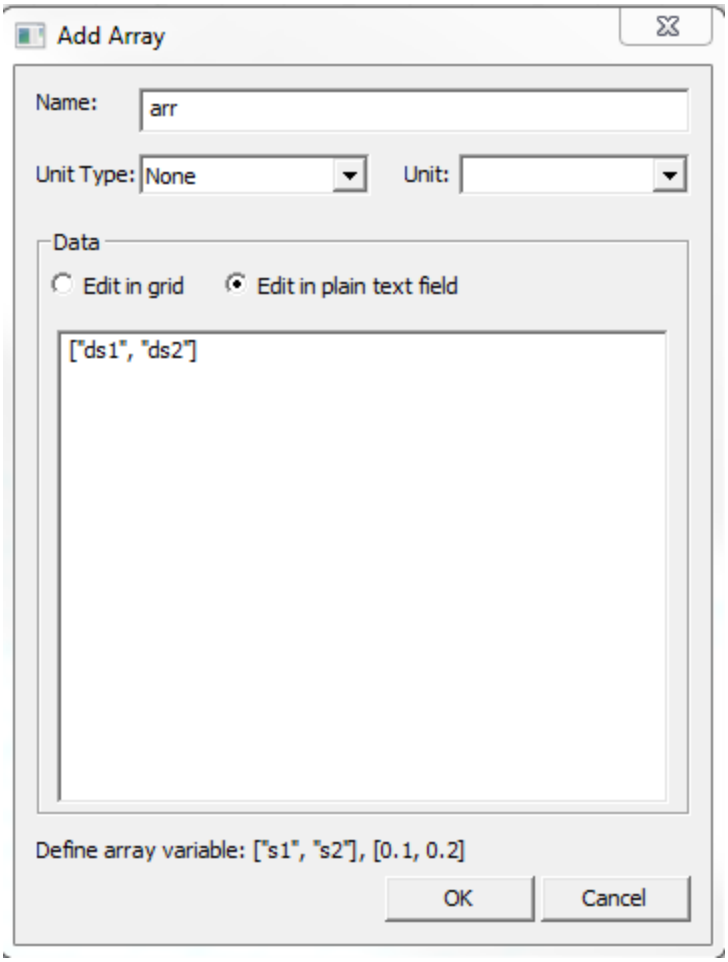
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The value grid displays each array item's **Index** number, and the **Data** associated with that index number.

If you elected to **Edit in plain text field** in the Add Array dialog box, the bracketed and comma delimited format is used.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Give the variable array a **Name**, and select a **Unit Type** and **Unit** from the drop-down menus.
5. Use the control buttons to add, delete, and reposition rows in the value grid at left. The default is **Edit in grid**, but you may select **Edit in plain text field**.

   > **Note:**
   >
   > Quotation marks ("") are *required* as delimeters when array values are entered in either the grid or text field.

6. When you have finished entering values, click **OK**.

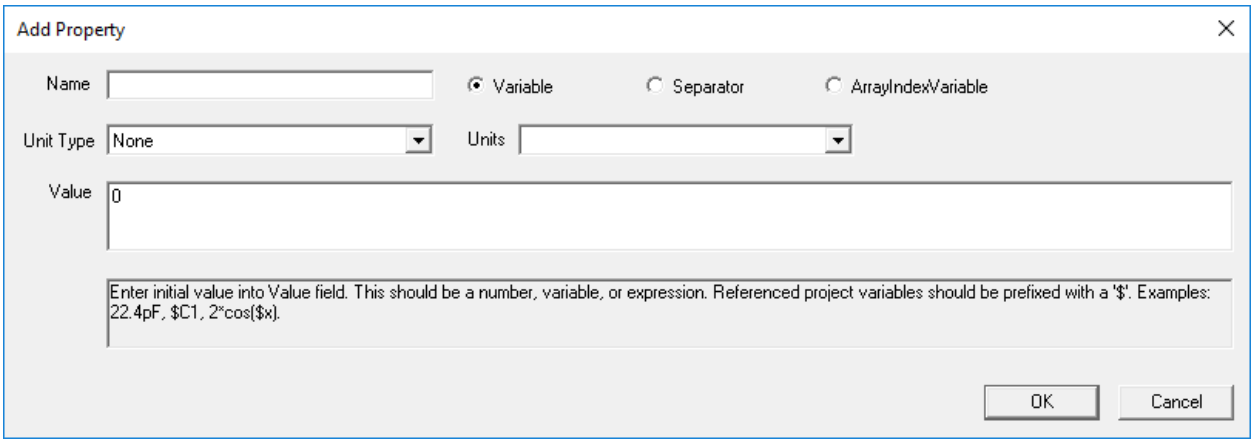
   The array appears in the **Local Variables** list.

You can add a variable to the array by adding an ArrayIndexVariable on the **Local Variables** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Access **Local Variables** one of three ways:
   - Click **HFSS** > **Design Properties**.
   - Right-click the design name in the Project Tree and select **Design Properties**.
   - From the **Setup Optimization** dialog box, click **Edit Variables** > **Edit Design Variables**.
2. From the **Local Variables** tab, click **Add**.

   The **Add Property** dialog box appears.



3. Enter information for the variable, as applicable:
   - **Name** – the name can contain alphanumeric characters and underscores (_). You cannot use the names of Intrinsic Variables or pre-defined Constants.
   - **Variable Type** – select ArrayIndexVariable.
   - **Value** – enter a number, variable, or mathematical expression. The quantity entered will be the current (or default) value for the variable. If the mathematical expression includes a reference to an existing variable, this variable is treated as a dependent variable. The units for a dependent variable will automatically change to those of the independent variable on which the value depends. Additionally, dependent variables, though useful in many situations, cannot be the direct subject of optimization, sensitivity analysis, tuning, or statistical analysis.

     For ArrayIndexVariables, the index reference can be a constant (for example, 1), an index (for example, ii) or an expression (for example, ii + 1). This allows you to sweep the index and simulate for different values that are stored in the array variable itself. In particular, it also enables you to sweep different text strings. This allows you to set a property to different string values as the index is swept.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Warning:**
>
> If you include the variable's units in its definition (in the **Value** text box), do not include the variable's units when you enter the variable name for a parameter value.

- **Associate Array Variable** – for ArrayIndexVariables, select an associated array from the drop-down list. You must have previously created an array.

4. Click **OK**.

Converting Variables and Parameter Defaults

Design (Local) variables can be converted to parameter defaults, and vice versa.

To convert a parameter default or local variable:

1. Access **Design Properties** one of three ways:
   - Click **HFSS** > **Design Properties**.
   - Right-click the design name in the Project Tree and select **Design Properties**.
   - From the **Setup Optimization** dialog box, click **Edit Variables** > **Edit Design Variables**.

   The **Properties** window appears.
2. Select either the **Parameter Defaults** tab or the **Local Variables** tab, depending on which item you want to convert.
3. Select the variable/parameter and click **Convert to Parameter** or **Convert to Variable**.

   The item will move to the list on the appropriate tab.

Some special cases to consider:

- If a local variable references another local variable, it cannot be converted to a parameter default. You will see a pop-up message if you attempt to convert. This is because local variables can reference parameter defaults, but parameter defaults cannot reference local variables.
- Parameter defaults can be overridden and can have different values in each subcircuit, while a Local Variable cannot. If a parameter default has been overridden and you convert it to a variable, it will lose all overrides. Attempting to do so will display a pop-up message asking if you really want to do the conversion and lose the override values.

Adding Variables and Parameter Defaults from Components

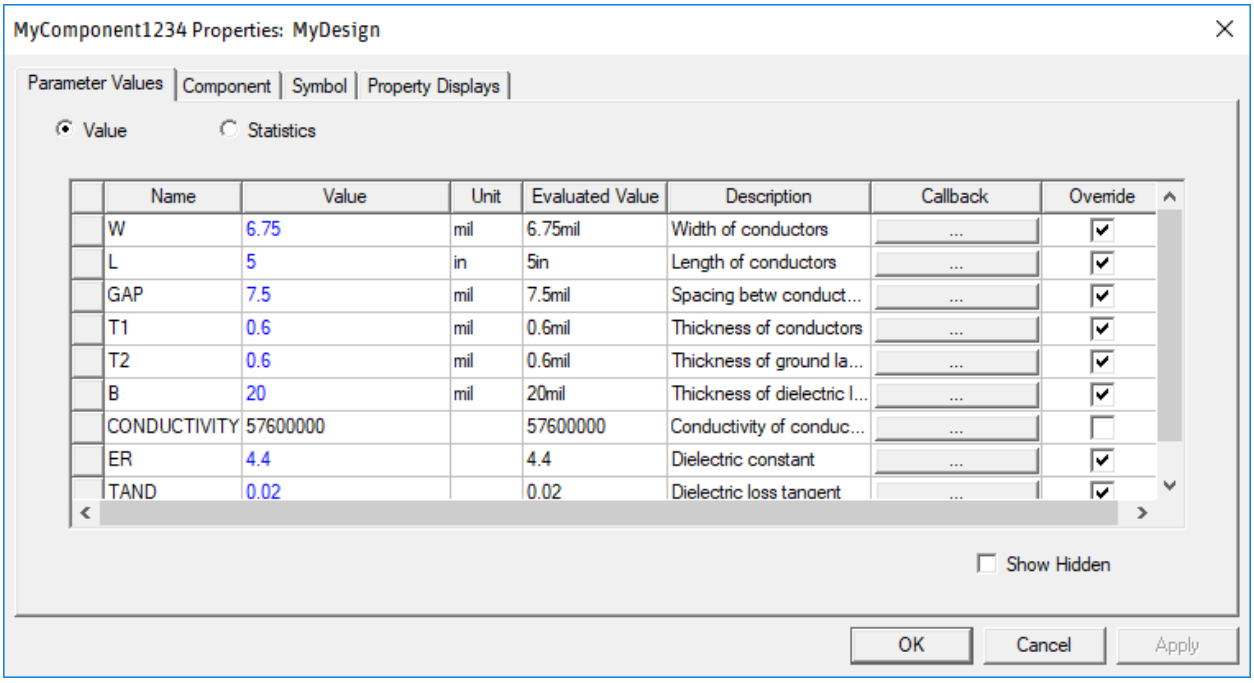
To define a new variable or parameter default from a component:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Display a component's **Properties** window one of three ways:
   - Double-click the component.
   - Select the component and click **Edit** > **Properties**.
   - Select the component, right-click and select **Properties**.

   The **Properties** window appears.
2. Select the **Parameter Values** tab.

   The **Parameter Values** tab displays.



3. Select the **Value** radio button.
4. Select the **Value** field for the parameter you want to set equal to a local variable.
5. Type the variable name. The name may not be the same as an intrinsic variable or constant.

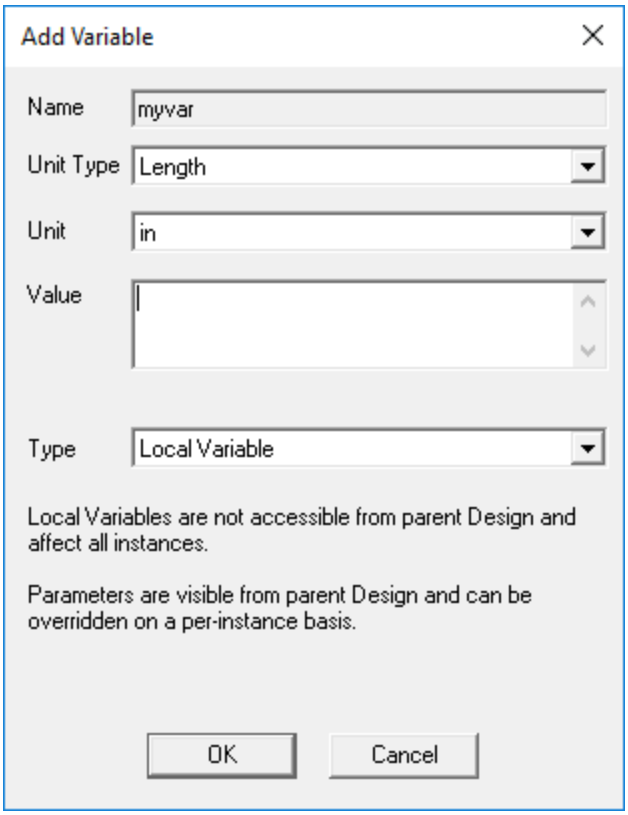
   Variable names may include alphanumeric characters and underscores (_).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Important:**
>
> If you wish to define a Project Variable, you must begin the name with a dollar sign ($).
>
> When you enter a name beginning with $, you will only be allowed to define it as a Project Variable.

6. Press **Enter**.

   The **Add Variable** window appears.



7. Select a **Unit Type** and **Unit** using the drop-down menus.
8. Enter a **Value**.

   The Value can be a number, variable, or mathematical expression. The quantity entered will be the current (or default) value for the variable. If the mathematical expression includes a reference to an existing variable, this variable is treated as a dependent variable. The units for a dependent variable will automatically change to those of the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

independent variable on which the value depends. Additionally, dependent variables, though useful in many situations, cannot be the direct subject of optimization, sensitivity analysis, tuning, or statistical analysis.

> **Warning:**
>
> If you include the variable's units in its definition in the **Value** text box, do not include the variable's units when you enter the variable name for a parameter value.

9. From the **Type** drop-down menu, select **Local Variable** or **Parameter Default**.

> **Note:**
>
> If you wish to define a Project Variable, your variable name must begin with a dollar sign ($).

10. Click **OK** to exit the **Add Variable** window.
11. Click **Apply** to save your changes.
12. Click **OK** to exit the **Properties** window.

Fixed vs. Non-Fixed Variables

By default, both Project variables and Design variables are included in the list of variables that index solution data. As a result, they are visible when selecting data to be displayed in plots.

However, if you define a variable as *fixed*, it is not swept and will not index solution data. Consequently, defining variables as fixed can speed up simulation—particularly if there are many variables to simulate.

To define a variable as fixed:

1. Open the **Properties** window for either Project or Design variables.
2. Clear the **Sweep** check box next to the variable you want to be fixed.

	Name	Value	Unit	Evaluated Value	Type	Description	Read-only	Hidden	Sweep
	wg_width	7.1	mm	7.1mm	Design		☐	☐	☑
	fillet_radius	0	mm	0mm	Design		☐	☐	☐

- If a **Sweep** check box is selected (default) the corresponding variable is NOT fixed.
- If a **Sweep** check box is cleared, the variable is fixed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

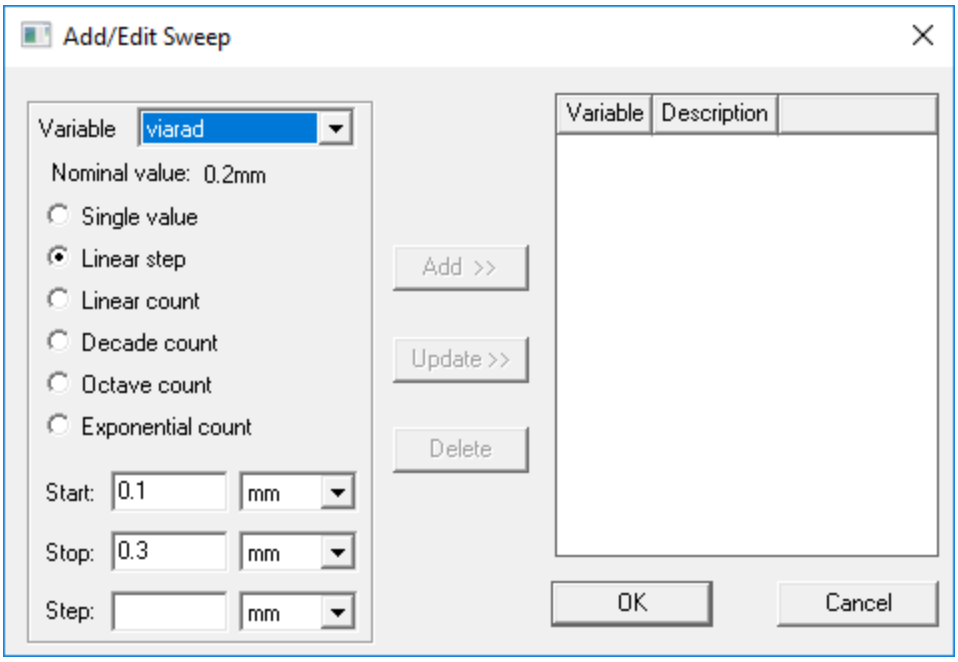
Non-Fixed Variable Sweeps

When setting up a sweep, the **Setup** window contains a list of variables being swept.

To add a non-fixed variable sweep:

1. Click **Add**.

   The **Add/Edit Sweep** window appears.



2. Use the **Variable** drop-down menu to select a non-fixed variable.
3. Select the desired settings and click **Add**.
4. Click **OK** to close the **Add/Edit Sweep** window.
5. Click **OK** to exit the **Setup**.

Orphaned Sweeps

If you create a simulation setup that contains a sweep of a variable, and then subsequently clear the variable's **Sweep** check box, the variable becomes fixed and, as a result, its sweep is "orphaned." The sweep is removed from the setup and the simulation runs as if the sweep did not exist.

- When you close the **Properties** window, a warning message is added to the **Message Window** for each orphaned sweep.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you edit an orphaned variable in the **Properties** window and reselect its **Sweep** box, the orphaned sweep will be restored to the setup list.
- If you double-click a setup that contains orphaned sweeps, a pop-up dialog box asks if you want to delete the orphaned sweeps. If you respond **Yes**, all orphaned sweeps are deleted when the **Properties** window is closed. Even if you reselect a **Sweep** check box, its orphaned sweep will NOT be restored. This action is undoable.
- Orphaned sweeps are not written to disk when the project is saved, so once you save a project and close it, any orphaned sweeps are permanently lost.

Choosing a Variable to Optimize

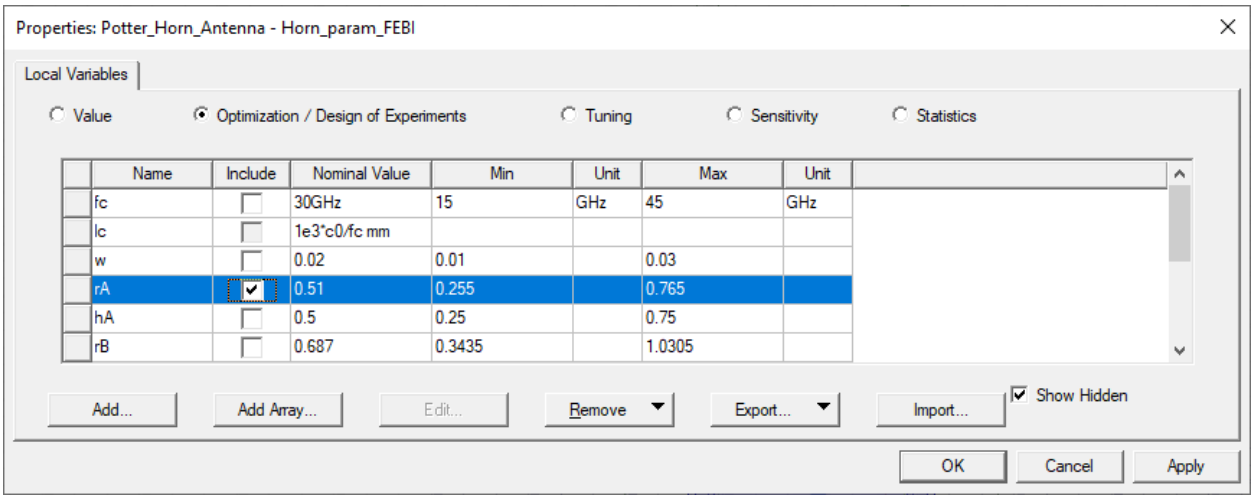
Before a variable can be optimized, you must specify that you intend for it to be used during an optimization analysis in the **Properties** window.

To specify that a variable is to be used during optimization:

1. Navigate either to the Project Variables **Properties** window (**Project** > **Project Variables**), or to the Design Variables **Properties** window (**HFSS** > **Design Properties**).

   The **Properties** window appears.

2. Use the tabs to navigate to the variable you want to optimize.
3. Select the **Optimization / Design of Experiments** radio button.
4. Use the **Include** check boxes to select variables for optimization.

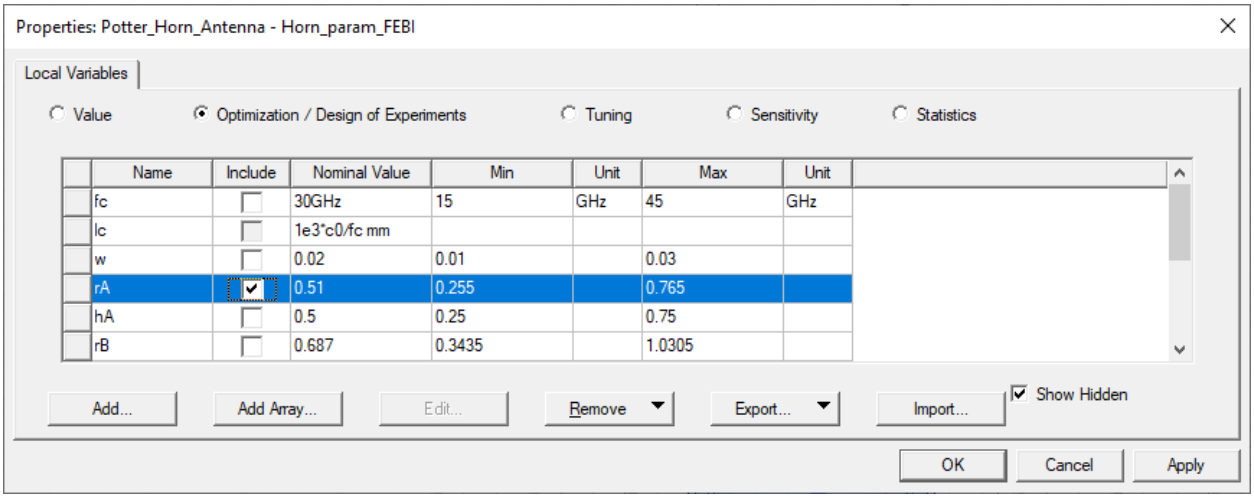


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

Dependent variables cannot be optimized.

Variables containing complex numbers cannot be used in an Optimetrics sweep or for optimization, statistical, sensitivity, and tuning setups.



The selected variable(s) will be available for optimization in an Optimetrics setup defined in the current design or project.

5. If desired, use the **Min** and **Max** fields to override the default minimum and maximum values that Optimetrics will use for the variable in every optimization analysis. During optimization, the optimizer will not consider variable values that lie outside of this range.

Choosing a Variable to Tune

Before a variable can be tuned, you must specify that you intend for it to be tuned in the **Properties** window.

To specify that a variable is to be tuned:

1. Navigate either to the Project Variables **Properties** window (**Project** > **Project Variables**), or to the Design Variables **Properties** window (**HFSS** > **Design Properties**).

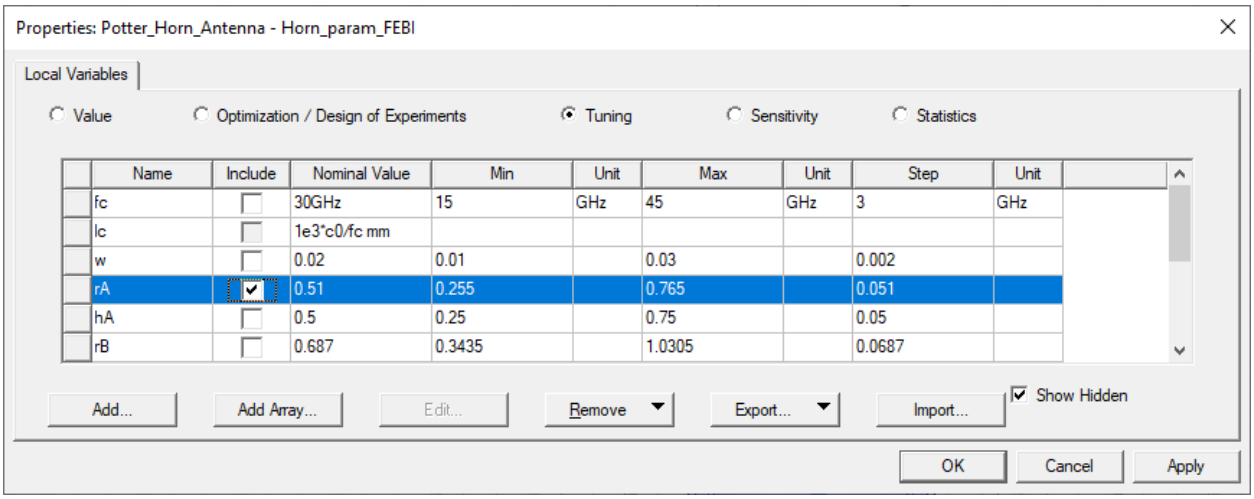
   The **Properties** window appears.

2. Use the tabs to navigate to the variable you want to optimize.
3. Select the **Tuning** radio button.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Use the **Include** check boxes to select variables for tuning.

> **Important:**
>
> Dependent variables cannot be tuned.
>
> Variables containing complex numbers cannot be used in an Optimetrics sweep or for optimization, statistical, sensitivity, and tuning setups.



The selected variable(s) will be available for tuning.

Including a Variable in a Sensitivity Analysis

Before a variable can be included in a sensitivity analysis, you must specify that you intend for it to be used during a sensitivity analysis in the **Properties** window.

To specify that a variable is to be used during sensitivity analysis:

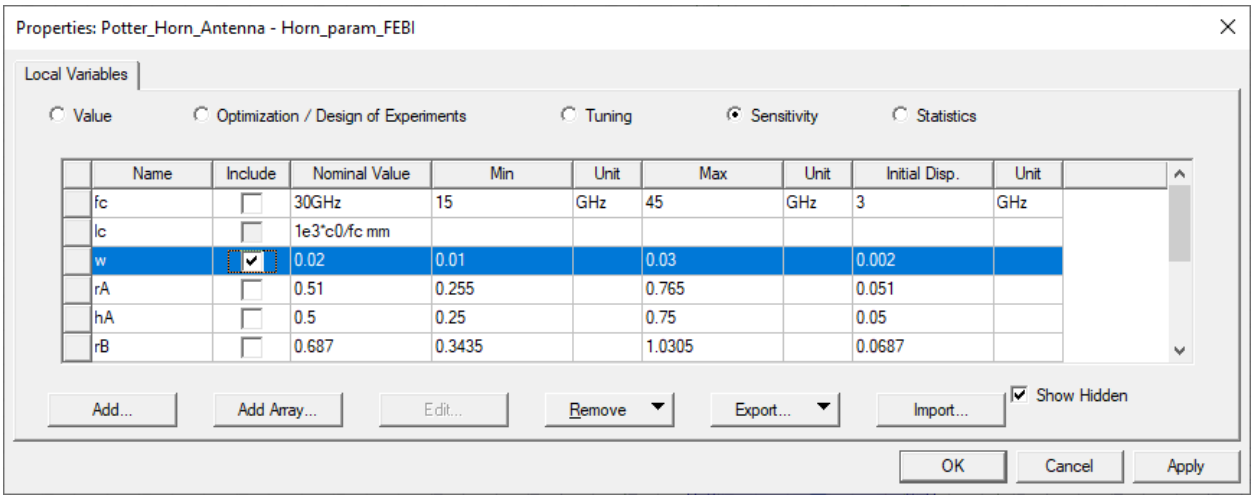
1. Navigate either to the Project Variables **Properties** window (**Project** > **Project Variables**), or to the Design Variables **Properties** window (**HFSS** > **Design Properties**).

   The **Properties** window appears.

2. Use the tabs to navigate to the variable you want to use.
3. Select the **Sensitivity** radio button.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Use the **Include** check boxes to select variables for sensitivity analysis.



> **Important:**
>
> Dependent variables cannot be included in a sensitivity analysis.
>
> Variables containing complex numbers cannot be used in an Optimetrics sweep or for optimization, statistical, sensitivity, and tuning setups.

The selected variable(s) will be available for sensitivity analysis in the current design or project.

5. If desired, use the **Min** and **Max** fields to override the default minimum and maximum values that Optimetrics will use for the variable in every sensitivity analysis. During analysis, the optimizer will not consider variable values that lie outside of this range.
6. If desired, use the **Initial Disp.** field to override the default initial displacement value that Optimetrics will use for the variable in every sensitivity analysis. During analysis, Optimetrics will not consider a variable value for the first design variation that is greater than this step size away from the starting variable value.

Including a Variable in a Statistical Analysis

Before a variable can be included in a statistical analysis, you must specify that you intend for it to be used during a statistical analysis in the **Properties** window.

To specify that a variable is to be used during statistical analysis:

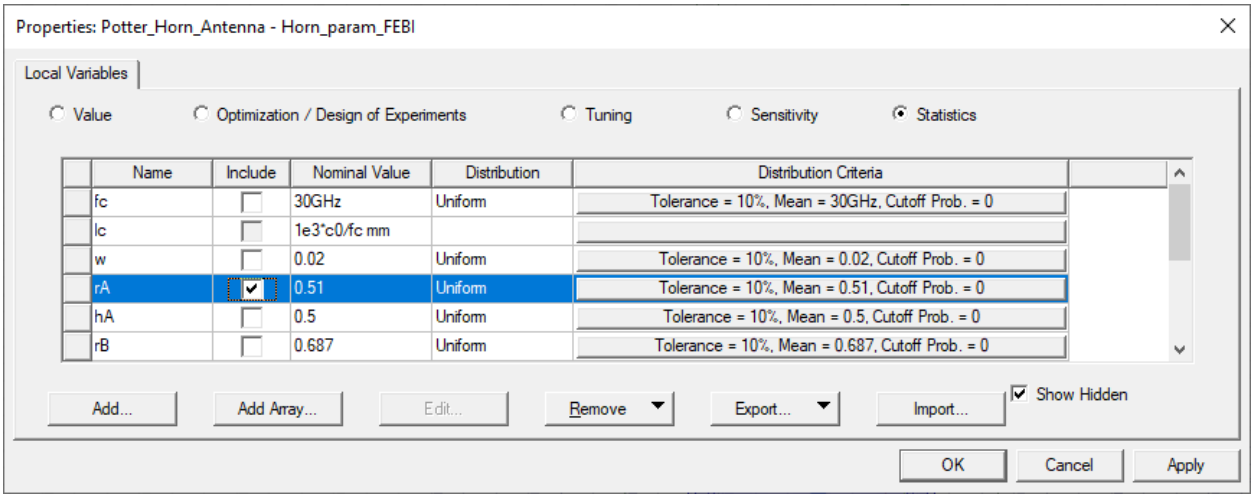
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Navigate either to the Project Variables **Properties** window (**Project** > **Project Variables**), or to the Design Variables **Properties** window (**HFSS** > **Design Properties**).

   The **Properties** window appears.

2. Use the tabs to navigate to the variable you want to use.
3. Select the **Statistics** radio button.
4. Use the **Include** check boxes to select variables for statistical analysis.

> **Important:**
>
> Dependent variables cannot be included in a statistical analysis.
>
> Variables containing complex numbers cannot be used in an Optimetrics sweep or for optimization, statistical, sensitivity, and tuning setups.



   The selected variable(s) will be available for statistical analysis in the current design or project.

5. If desired, click the **Distribution Criteria** button to override the distribution criteria that Optimetrics will use for the variable in every statistical analysis.

Copying and Pasting a Variables List

From either the Project or Design variables window, you can copy a list of variables and their values to the clipboard. You can then paste them in a plain text, tab-separated format. This is useful for creating documentation.

To copy and paste the variables list:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Navigate to the Project Variables or Design Variables **Properties** window.
2. Right-click and select **Copy to Clipboard**.
3. Open a text editor and **paste**.

The pasted variables appear in a tab separated column format. Fields that do not contain values are left blank.

Name	Value	Unit	"Evaluated Value	"Description	Read-only	Hidden
$width	14.8570192	mm	14.8570192mm		false	false
$length	7.824547736	mm	7.824547736mm		false	false
$height	0.45*$width		6.68565864mm		false	false

Working with Datasets

Datasets are collections of plotted data points that can be extrapolated into an equation based on the piecewise linear makeup of the plot. Each plot consists of straight line segments whose vertices represent their end points. A curve is fitted to the segments of the plot and an expression is derived from the curve that best fits the segmented plot. The created expression is then used in piecewise linear Intrinsic Functions. You can add datasets at either the project level or the design level. They can be for various purposes, including to define frequency-dependent port impedances or frequency-dependent global variables.

Project-level datasets are typically used for defining material properties at the project level (applicable to all designs in the project).

Design-level datasets can be used in geometry entities like part commands, coordinate systems, points, and planes. Design-level datasets do not work with equation-based surfaces or curves. Design-level datasets can be used directly with piecewise linear functions in expressions or indirectly through variables that can refer to the dataset.

Design-level datasets can also be used in the following operations:

- **Creating or editing geometry** – when a geometry uses a dataset directly, edit dataset invalidates the solution. When a geometry uses a variable that is defined by dataset, editing the dataset does not invalidate the solution.
- **Creating an Animation** – based on a variable which can index in datasets.
- **Copying and Pasting Geometry** – if a part refers to a design dataset, it will be pasted to destination design.

Datasets are provided for Circuit, Circuit Netlist, and HFSS 3D Layout designs.

The **Datasets** dialog box provides a browsable listing of all datasets currently defined for the project or design. A preview window displays a plot of the currently selected dataset. Controls

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

allow you to **Add**, **Edit**, **Remove**, and **Clone** datasets; to **Import** and **Export** characteristic data; and to launch the **SheetScan** tool to extract data from graphics such as data sheets.

To access the **Datasets** window:

- For project-level datasets, including 3D datasets, click **Project** > **Datasets**.
- For design-level datasets, click **HFSS** > **Design Datasets**.

Adding Datasets

The following procedure describes how to add a dataset manually.

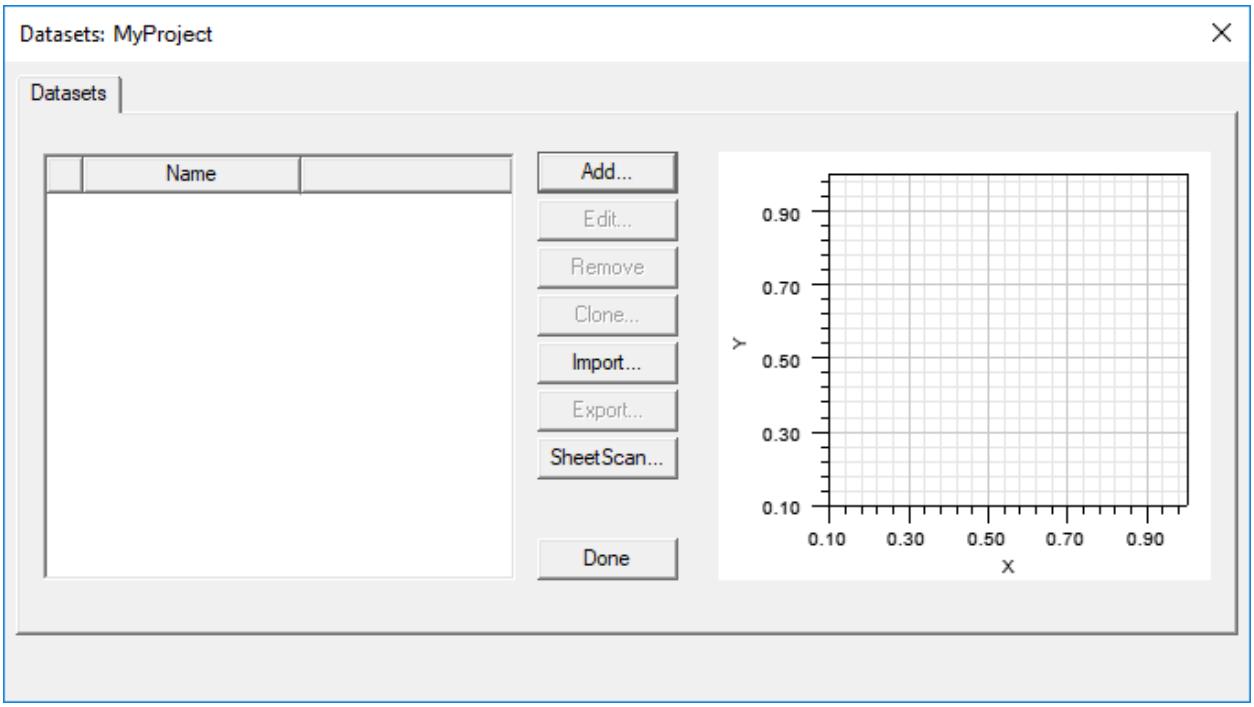
You can also add datasets:

- By importing data from an external tab delimited file using the **Import...** button.
- Using the Sheetscan tool.

To add a dataset manually:

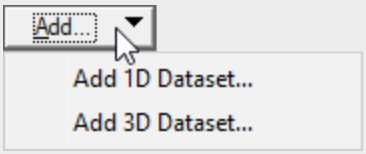
1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets** from the menu bar, or click **Datasets** on the **Desktop** ribbon tab.
   - For design-level datasets, click **HFSS** > **Design Datasets** from the menu bar.

   The **Datasets** window appears, listing any project or design-level datasets.

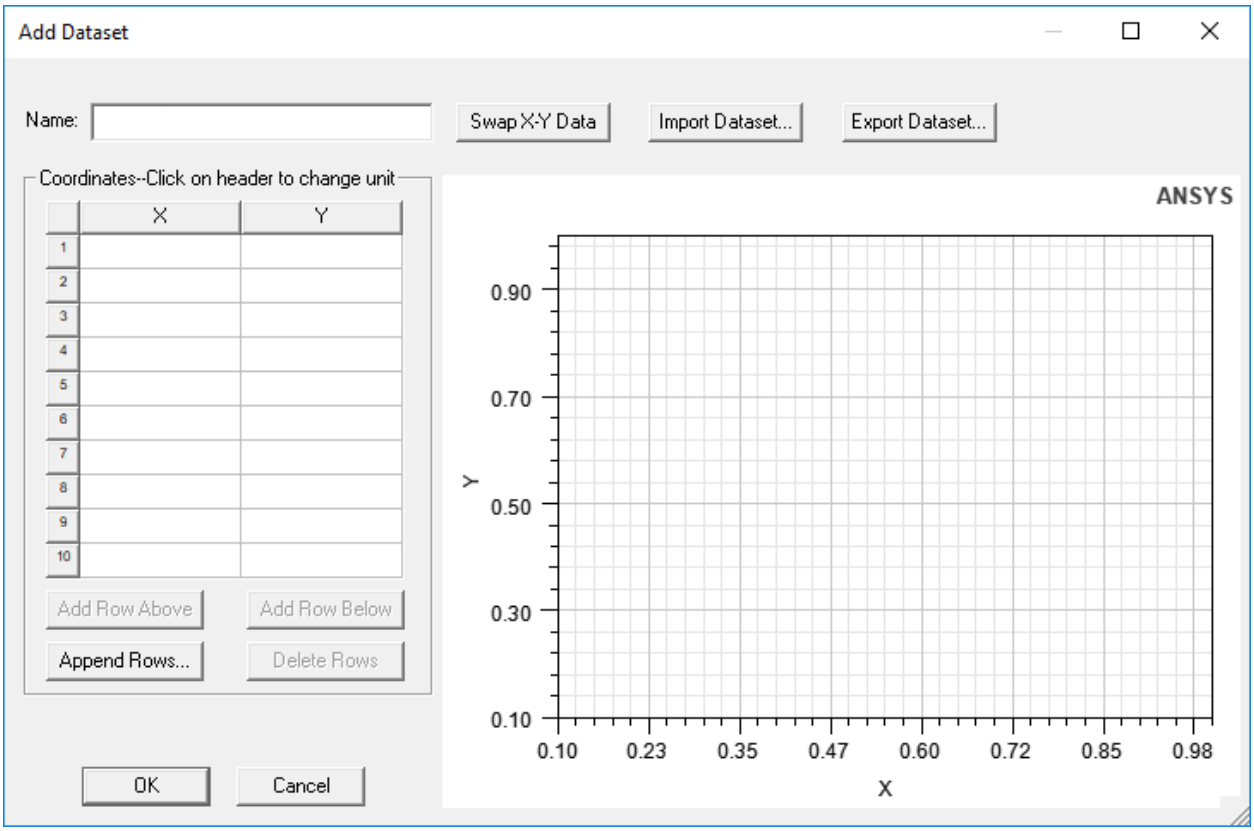


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Add**.

> **Note:**
>
> If you are adding a project-level dataset, and the project includes an HFSS design, you will be presented the option to add either a **1D Dataset** or **3D Dataset**.
>
> 
>
>
> See: Adding a 3D Dataset.

The **Add Dataset** window appears.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The left side of the **Add Dataset** window contains fields for the dataset name and a table for X- and Y- coordinates. The right side contains a graphic display that draws a line for the coordinates you add. It also includes buttons for the following functions:

- **Swap X-Y Data** – this swaps the X- and Y- coordinates and adjusts the graphical display.
- **Import Dataset** – this provides a way to import data sets from an external source. The format is a tab-separated points file. Clicking the button opens a file browser window.
- **Export Dataset** – this provides a way to export the current dataset to a tab separated points file. Clicking the button opens a file browser window.
- **Add Row Above** – adds a new row to the table above the selected row.
- **Add Row Below** – adds a new row to the table below the selected row.
- **Append Row** – opens a dialog box that lets you specify a number of rows to add to the table.
- **Delete Row** – deletes the selected row or rows.

3. Enter a **Name** for the dataset.

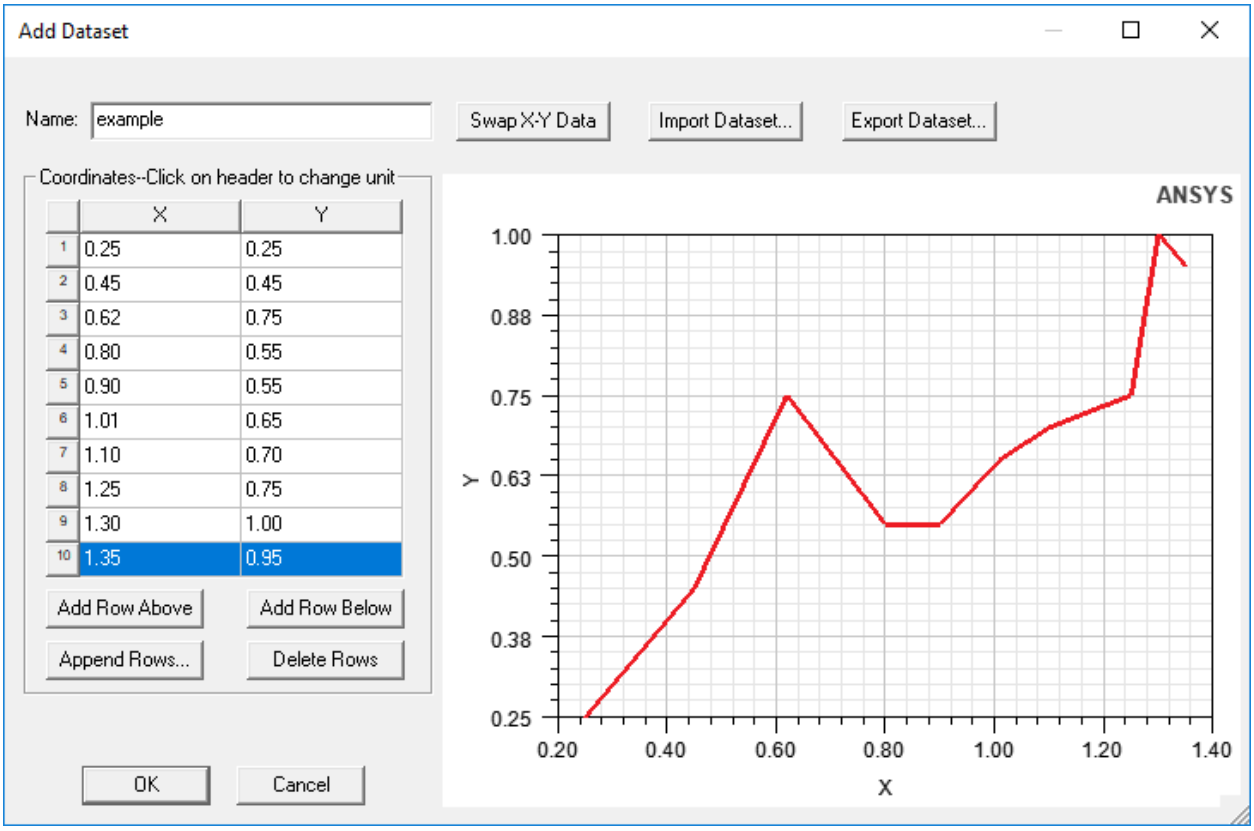
> **Note:**
>
> Project-level datasets must begin with a dollar sign character ($). If you do not type one, it will be added for you.

4. Enter X/Y coordinates, either by manually typing or by clicking **Import Dataset**.

   If desired, click the **X** and **Y** column headers to change the unit.

   The display updates to show your coordinates.

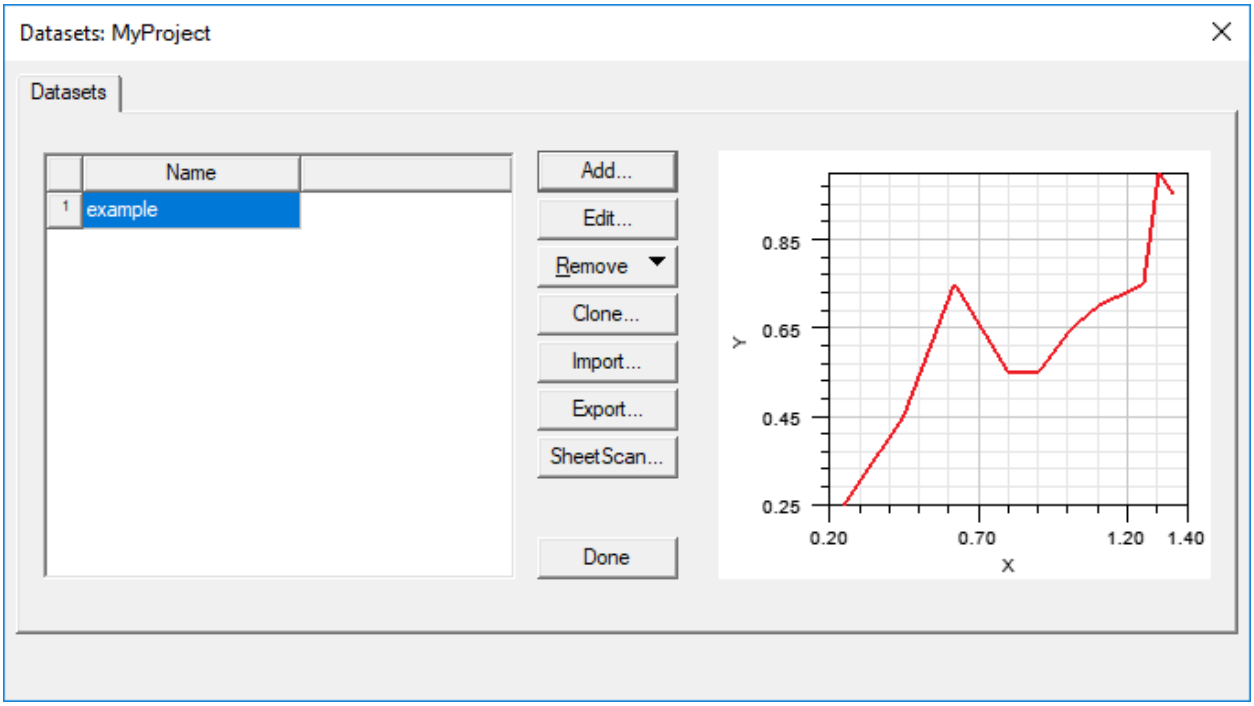
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. When you are finished, click **OK** to exit the **Add Datasets** window.

   The new dataset appears in the **Datasets** list.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The dataset plot is extrapolated into an expression that can be used in parametric analyses, boundary definitions, or assigned to a material property value.

You can now edit, remove, clone, or export the dataset. You can also change the plot display properties.

6. Click **Done** to exit the **Datasets** window.

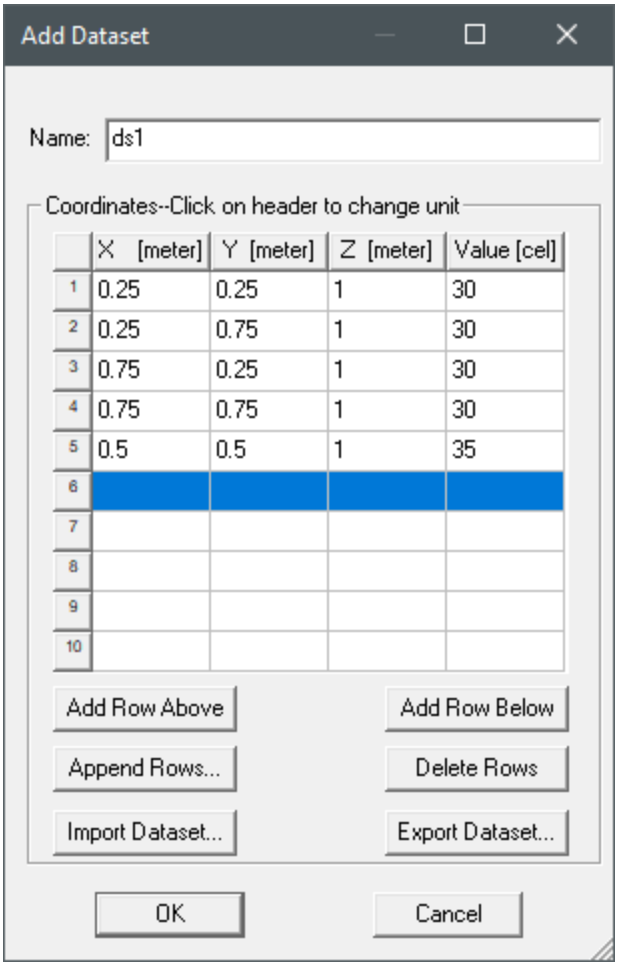
Adding 3D Datasets

HFSS, Maxwell, and Icepak designs support 3D datasets (representing a spatial or volumetric profile). Depending on which of these three design types you are using, you will be able to create only design-level 3D datasets, only project-level 3D datasets, or both.

In HFSS designs, you can define only project-level 3D datasets (used for the definition of material properties that vary according to the spatial coordinates).

To define a 3D dataset, specify a list of **X**, **Y**, and **Z** coordinates and the corresponding **Values** in the **Add Dataset** window. For example, the first line in the image below specifies a temperature of 30° C at (0.25, 0.25, 1). The data specifies a variation of temperature on the plane $0.25 \le X \le 0.75$, $0.25 \le Y \le 0.75$, $Z = 1$. In this example, the values in the **Values** column are temperature, but can be set to a different unit depending on the property you wish to assign. To change the unit, click the **X**, **Y**, **Z**, or **Value** header row.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

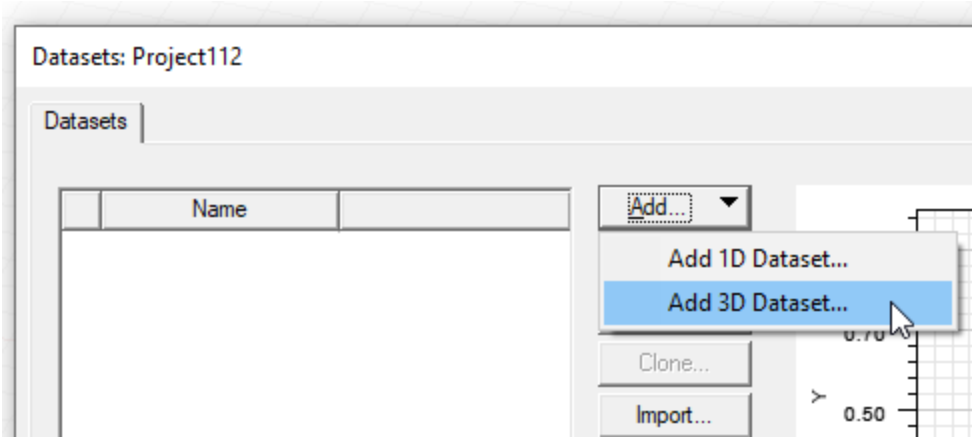


1. Using the menu bar, click **Project** > **Datasets** or, on the **Desktop** ribbon tab, click **Datasets**.

   The **Datasets** dialog box appears. This lists any existing datasets.

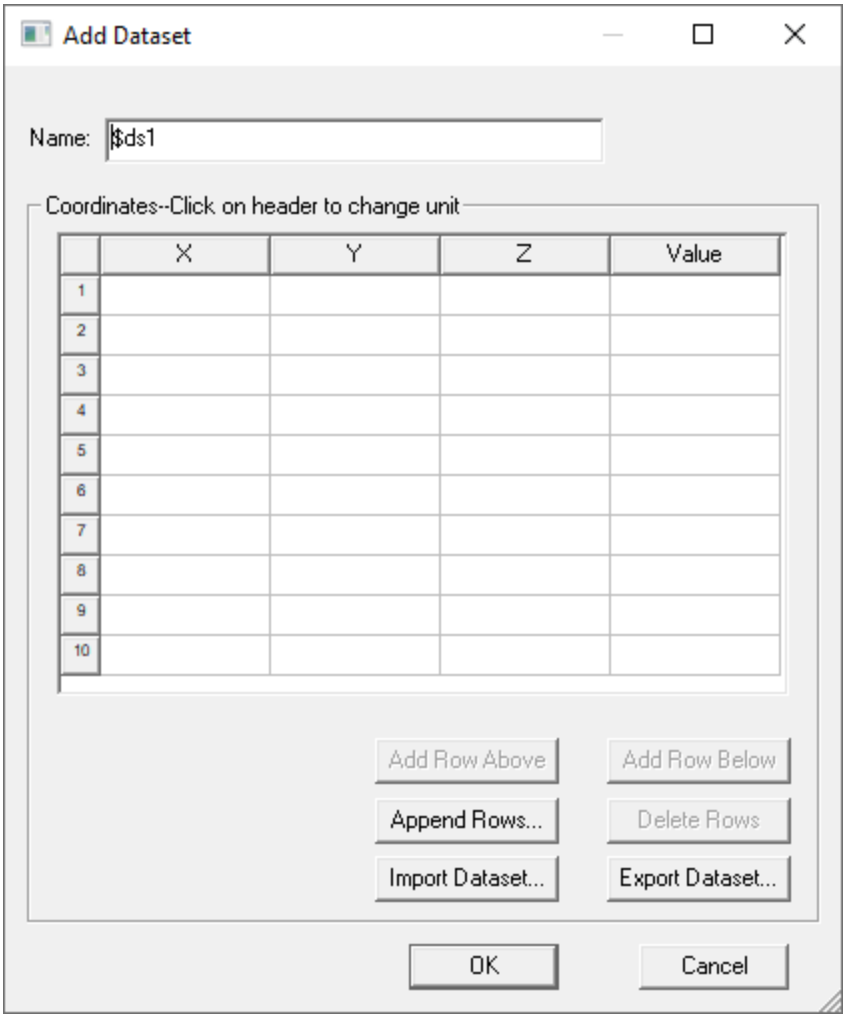
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Add** > **Add 3D Dataset**.



The **Add Dataset** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

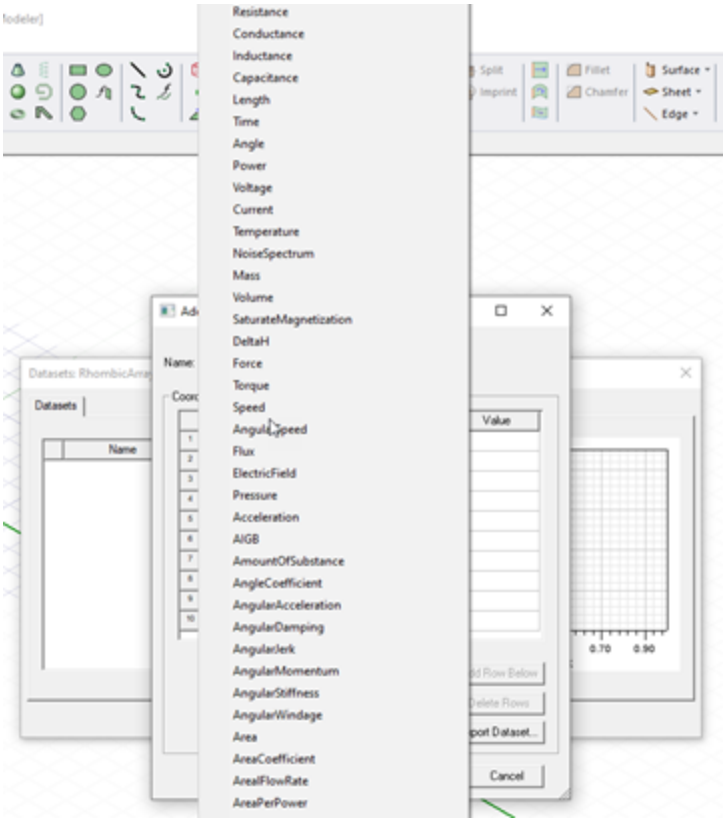


The window contains fields for the dataset name and a table for X, Y, and Z coordinates and corresponding values. It can be resized to display additional rows and columns, and includes buttons for the following functions:

- **Add Row Above** – adds a new row to the table above the selected row.
- **Add Row Below** – adds a new row to the table below the selected row.
- **Append Row** – opens a dialog box that lets you specify a number of rows to add to the table.
- **Delete Row** – deletes the selected row or rows.
- **Import Dataset** – this provides a way to import data sets from an external source. The format is a tab separated points file. Clicking the button opens a file browser window.
- **Export Dataset** – this provides a way to export the current dataset to a tab separated points file. Clicking the button opens a file browser window.

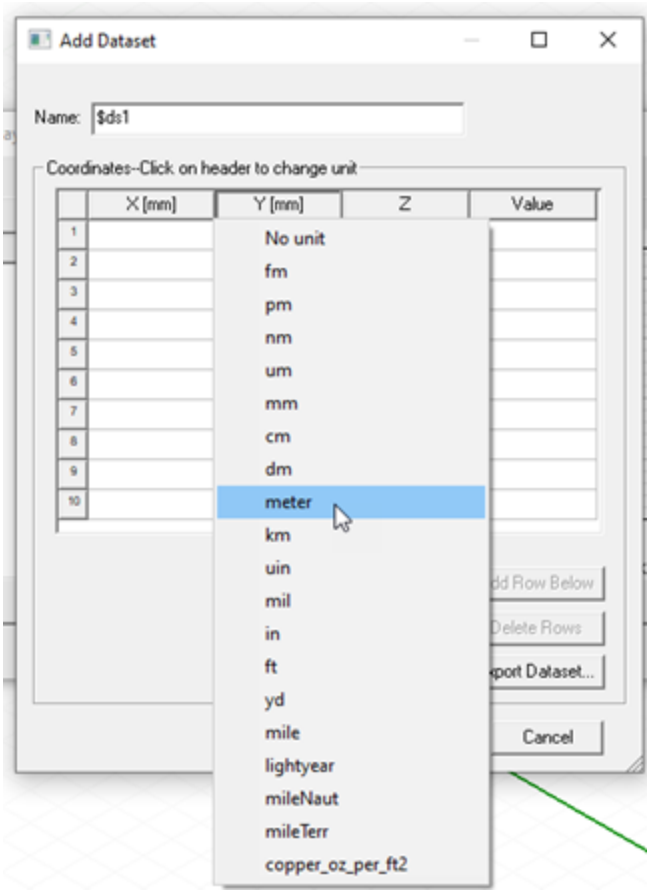
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Optionally, type a name other than the default for the dataset in the **Name** field.
4. Click the **X**, **Y**, **Z** and **Value** column headers to select a unit category. The default is No Unit.



Click the header again to select a unit in that category. You can select No unit.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selected units are displayed for each column in the header cell.

5. Enter the X, Y, and Z coordinates and the corresponding values by one of the following methods:
   - **Import Dataset**
   - Type coordinates and associated value in the **X**, **Y**, **Z**, and **Value** columns.
6. When you are finished entering the data point coordinates and values, click **OK**.

> **Note:**
>
> When used with a clp function (Closest point interpolation) in an expression, performance will degrade with large 3D dataset.

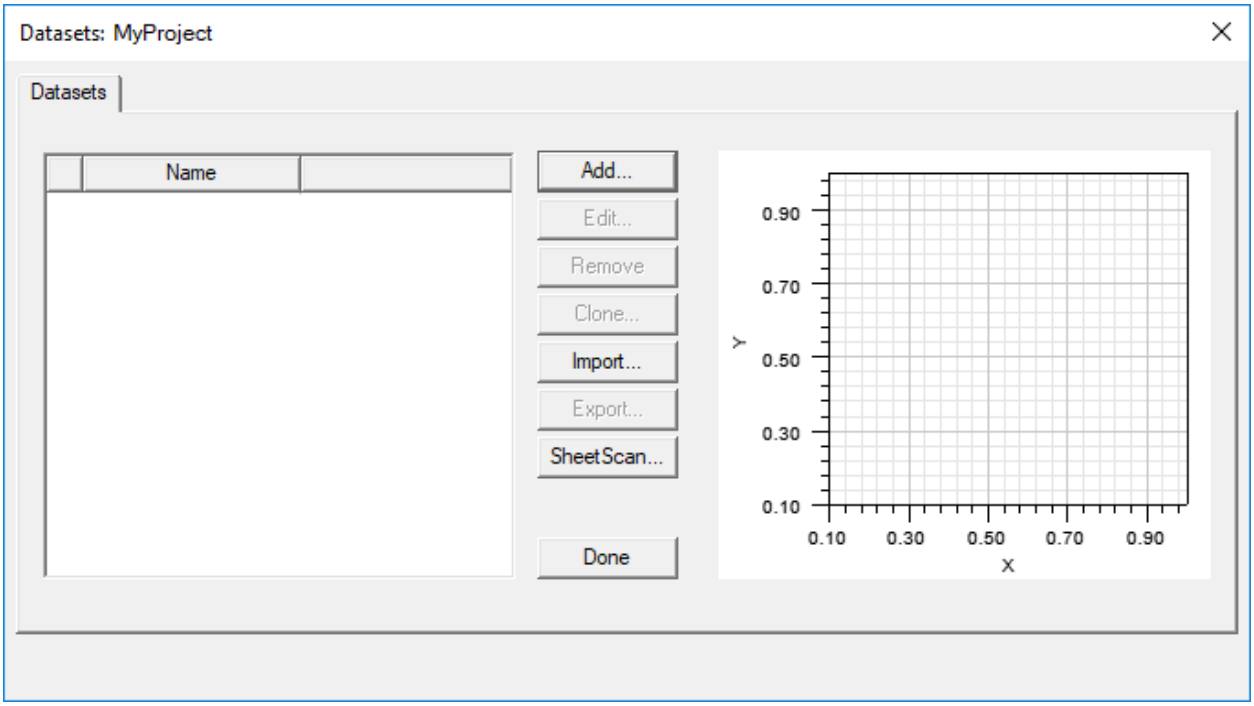
Importing Datasets

To import a dataset from a file:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **HFSS** > **Design Datasets**.

   The **Datasets** window appears.



2. Click **Import**.
3. In the file browser that appears, use the drop-down menu to select the file format of the file you wish to import.

> **Note:**
>
> The table below lists the file types supported for *Project level datasets*. For *Design level datasets*, only **.tab** files are supported.

.mdx, .mda	Twin Builder Characteristic format
.xls, .xlsx	Microsoft Excel
.txt	text file
.csv	Comma-separated value
.out	Maxwell SPICE (read-only - reads data inside the KW_DATA section)
.cfg	Comtrade (IEEE Std C37-111-1999)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

.dat	TEK Oscilloscope
.tab	tab delimited data files

a. If you select an **.mdx**, **.mda**, **.out**, **.cfg**, or **.tab** file, the data is imported immediately into **Datasets** window.

b. If you select an **.xls** or **.xlsx** file containing multiple sheets, a **Table Properties** dialog box appears, where you can choose the desired sheet from a drop-down menu. Otherwise, selecting an .xls or .xlsx file imports the data immediately.

> **Note:**
>
> Microsoft Excel is required to import .xls or .xlsx files.
>
> Only the first two columns of data are imported, the left-most column containing the X-coordinate values. The x-coordinate values for successive data points must increase within ten significant digits. Non-numeric entries are assigned a value of zero.
>
> The first row of data is assumed to contain column headings and is ignored.

c. Selecting a .**txt** or .**csv** file opens an **Import** dialog box where you can specify settings for reading the data in the file for import. You can choose the separator(s) and decimal symbol, as well as the line at which to begin the import. The dialog box shows both the original text and the text as it would appear when imported based on the current import settings.

4. When satisfied with the import settings, click **OK** to import the data.

   Imported datasets take the name of the imported file. If a dataset of the same name already exists, a number will be appended to the end of the new dataset (for example, a duplicate 'dataset' becomes 'dataset1').

   Once the data has been imported, you can edit, remove, clone, or export the dataset. You can also change plot display properties.

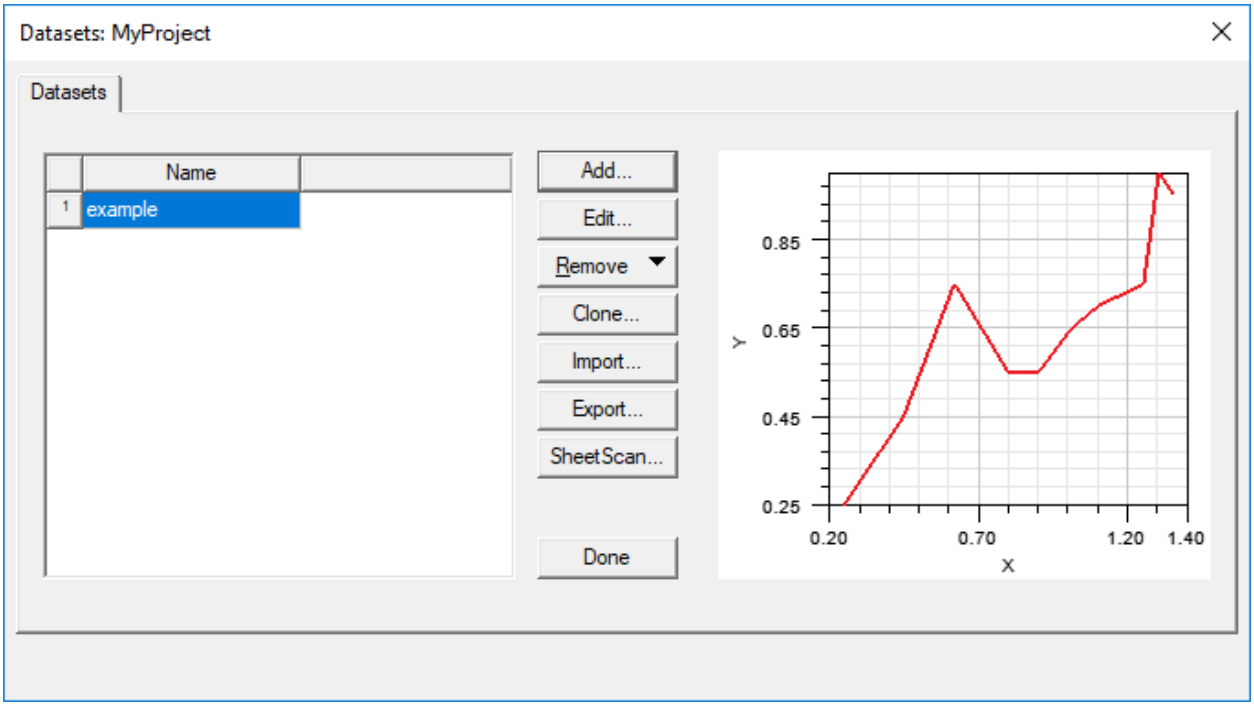
Editing Datasets

To modify an existing dataset:

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **HFSS** > **Design Datasets**.

   The **Datasets** window appears, listing any project or design-level datasets.

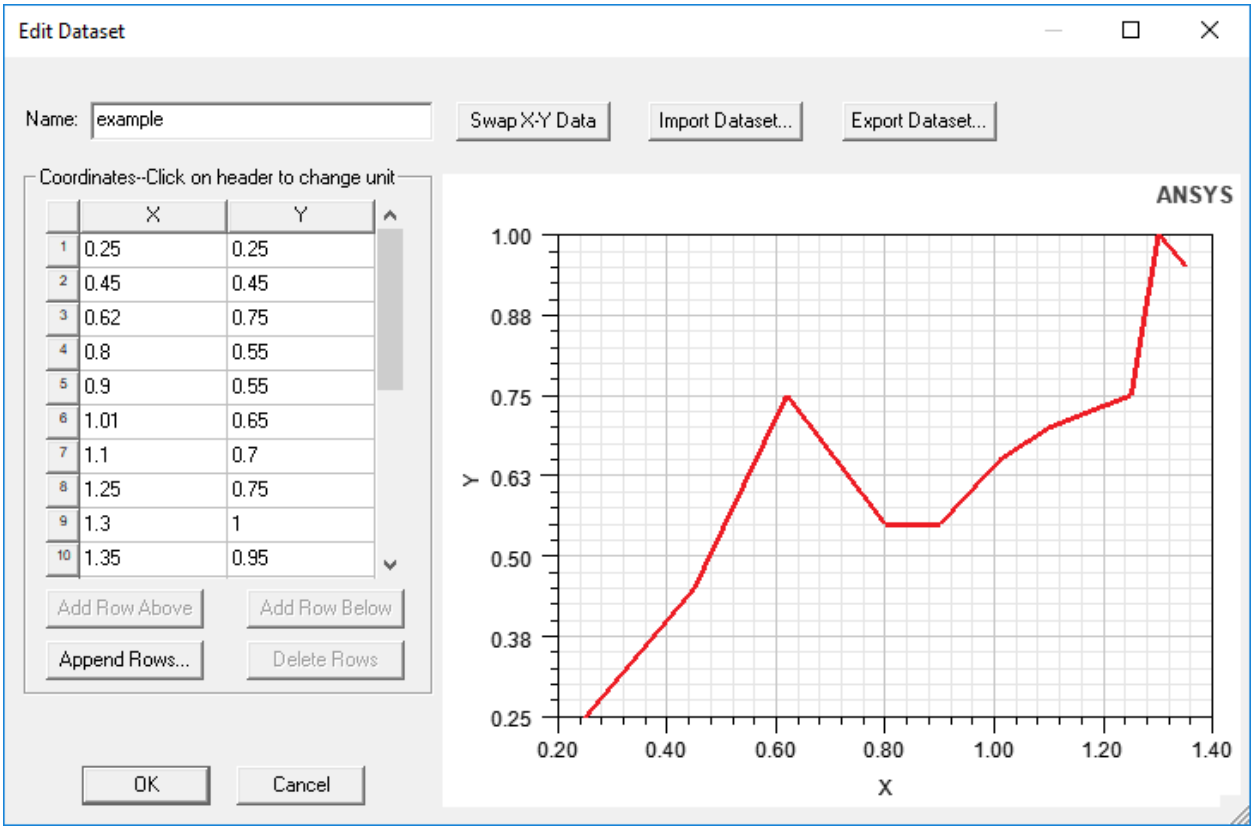
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **Edit**.

   The **Edit Dataset** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Change the **Name** and **X/Y** coordinates as desired.
4. When you are finished, click **OK** to exit the **Edit Dataset** window.
5. Click **Done** to exit the **Datasets** window.

Cloning Datasets

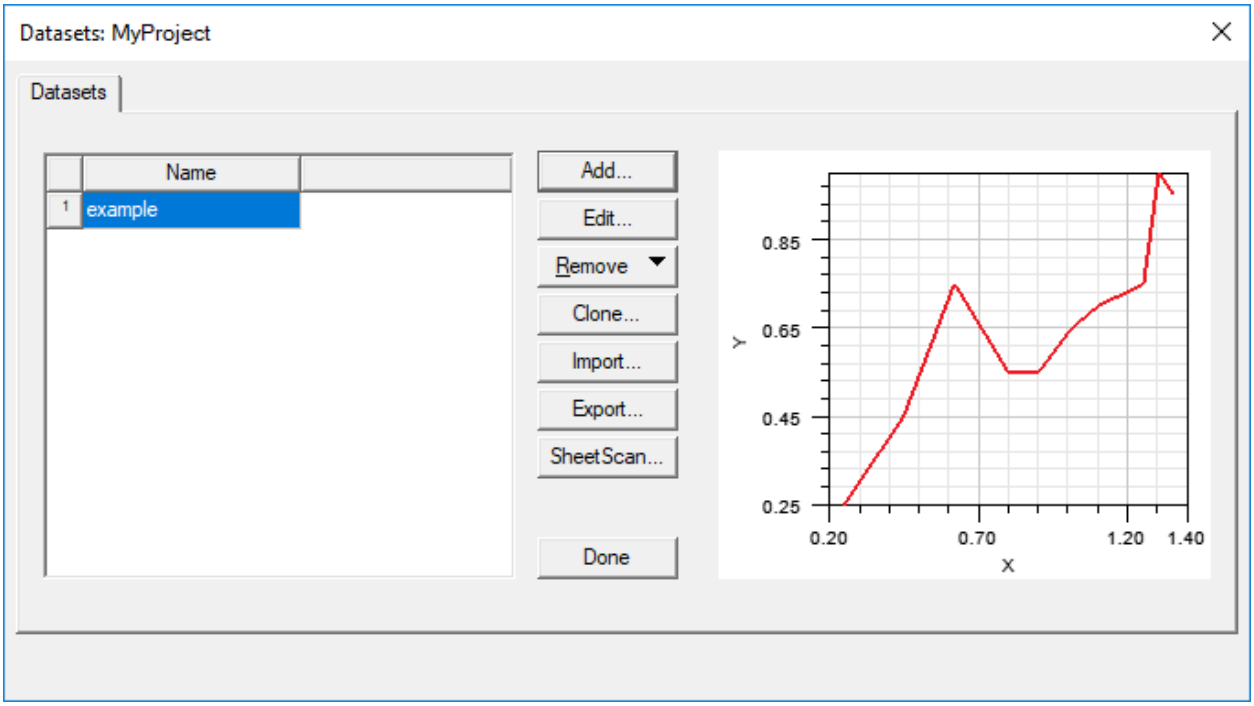
Cloning a dataset creates an identical copy of an existing dataset. The clone can then be modified as necessary.

To clone a dataset:

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **HFSS** > **Design Datasets**.

   The **Datasets** window appears, listing any project or design-level datasets.

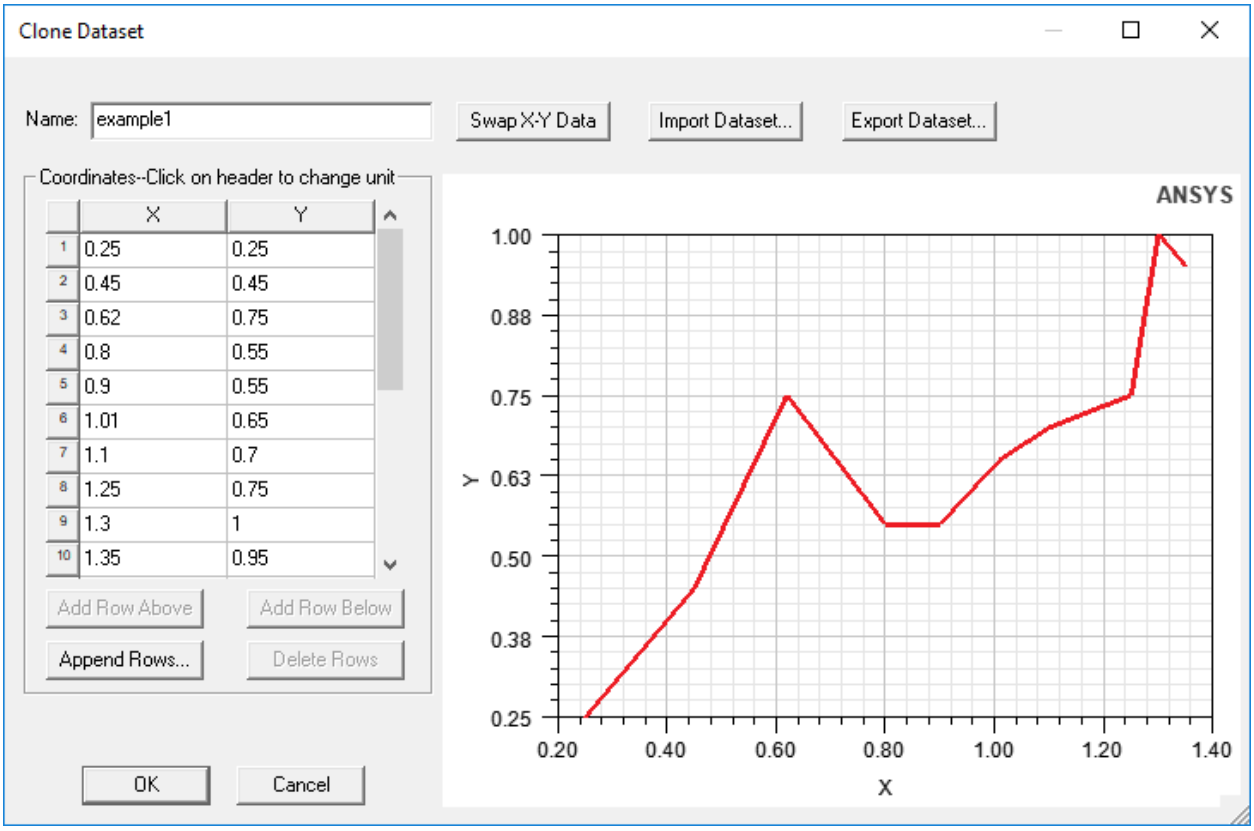
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **Clone**.

   The **Clone Dataset** window appears. The default name is the original dataset name with a number appended to the end.

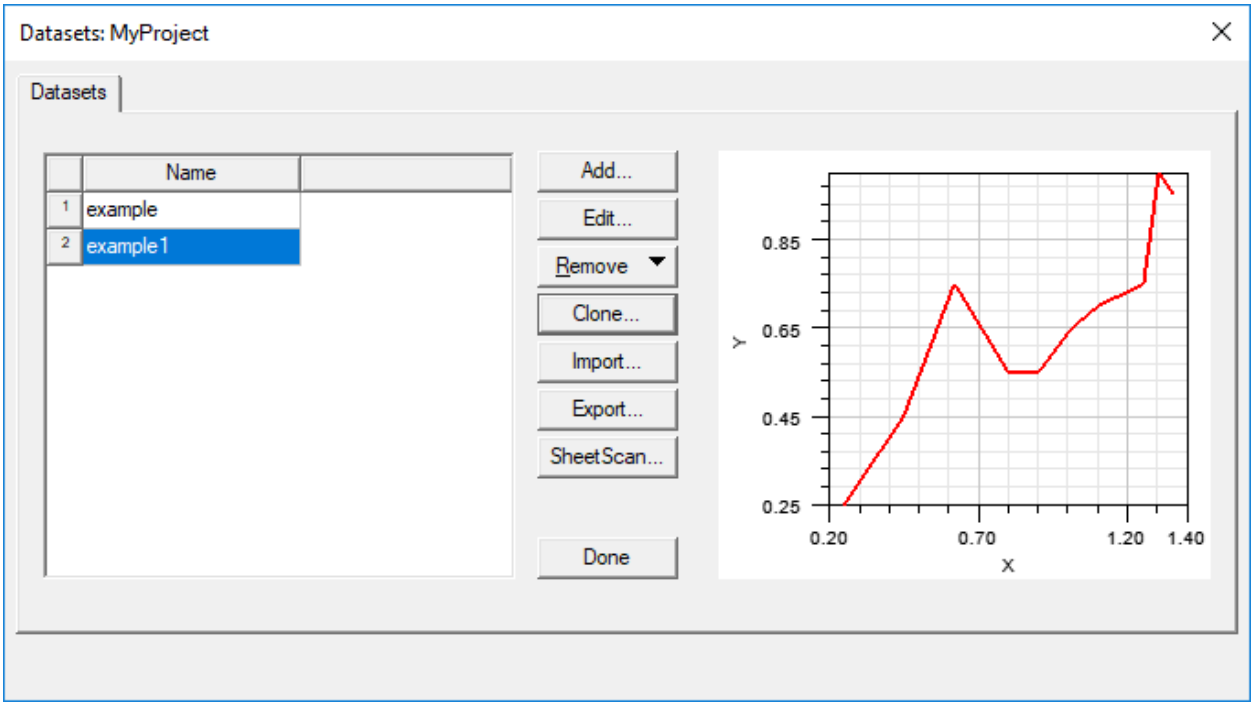
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Change the **Name** and **X/Y** coordinates as desired.
4. When you are finished, click **OK** to exit the **Clone Dataset** window.

   The cloned dataset appears in the list.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Click **Done** to exit the **Datasets** window.

Exporting Datasets

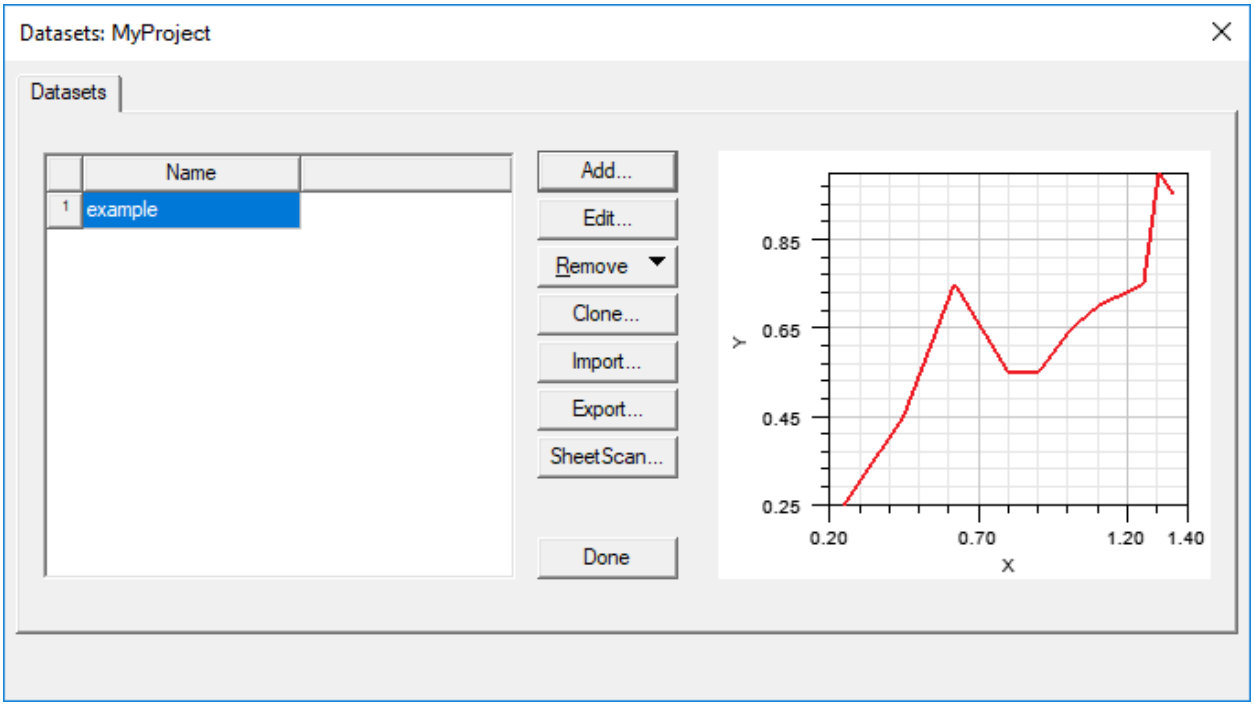
You can export datasets in **.tab** format.

To export a dataset:

1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **HFSS** > **Design Datasets**.

   The **Datasets** window appears, listing any existing datasets.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select a dataset and click **Export**.
3. Browse to the location where you would like to save the *.tab file.
4. Enter a name for the file.
5. Click **Save**.

   The dataset file is saved to the specified location for later import.

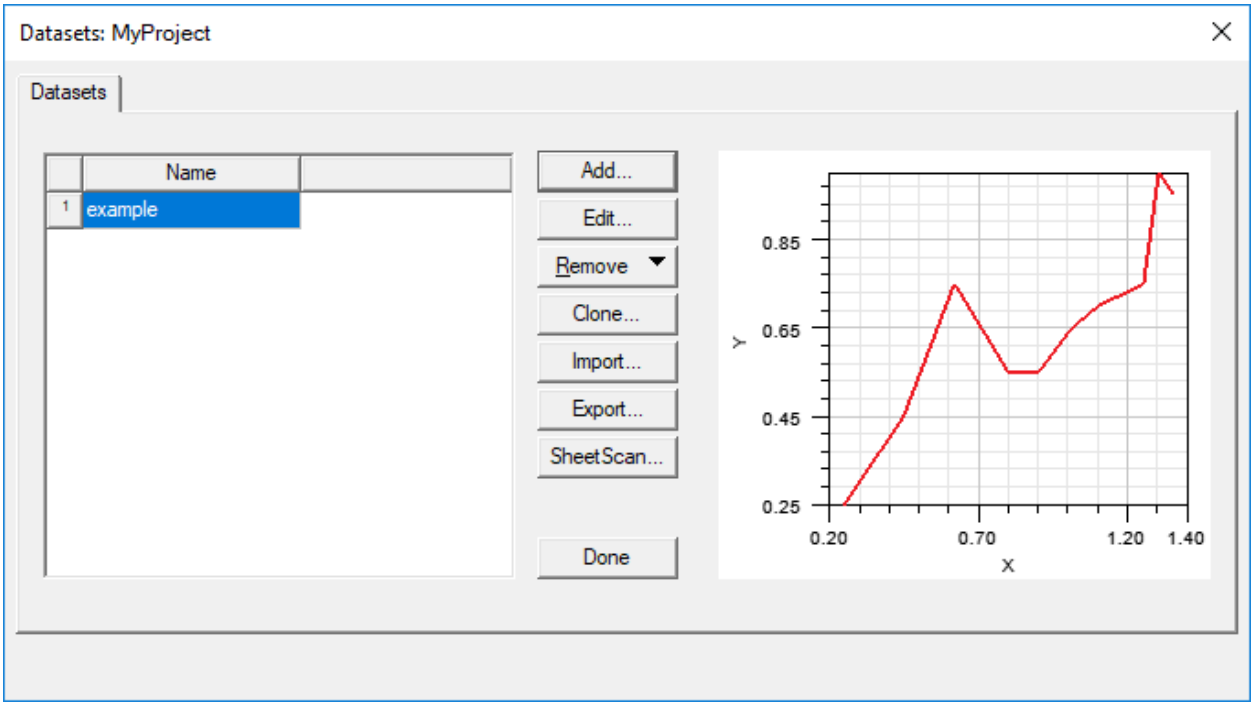
Removing Datasets

To remove a previously added dataset:

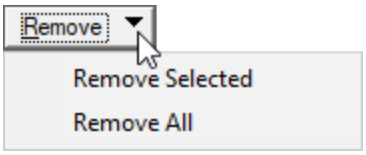
1. Access the **Datasets** window:
   - For project-level datasets, click **Project** > **Datasets**.
   - For design-level datasets, click **HFSS** > **Design Datasets**.

   The **Datasets** window appears, listing any existing datasets.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select a dataset and click **Remove**.
3. Select either **Remove Selected** to remove the selected dataset or **Remove All** to remove all project or design-level datasets.



The selected dataset(s) are deleted from the list.

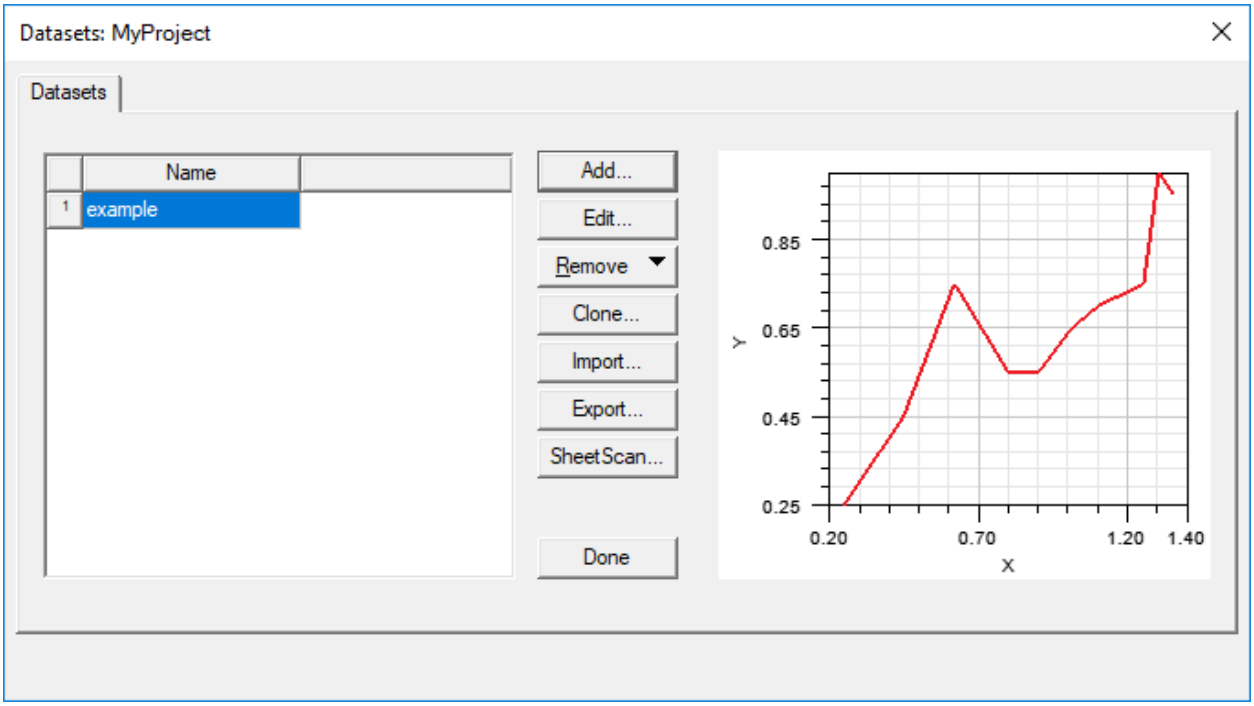
> **Note:**
>
> The **Remove All** option does not work across dataset types. That is, selecting **Remove All** from the **Project Datasets** window does not remove design-level datasets, and vice versa.

4. Click **Done** to exit the **Datasets** window.

Changing Dataset Plot Properties

From the **Datasets** window, you can change the appearance of a dataset's plot.

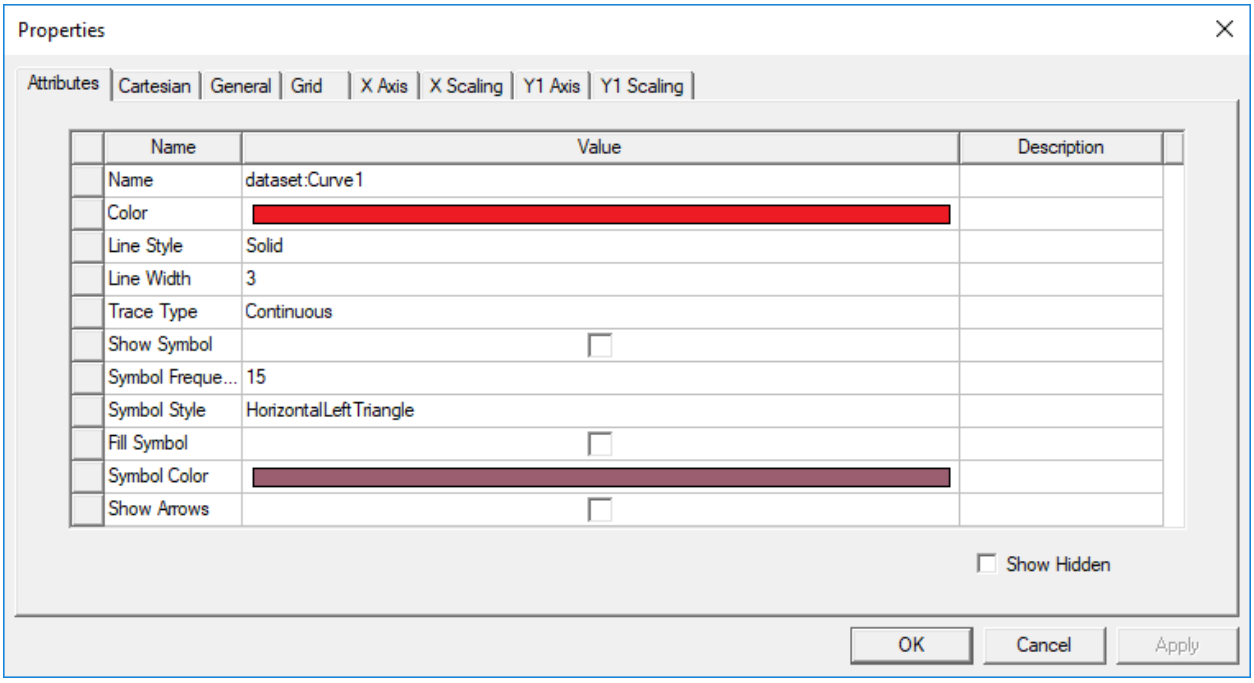
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To change a plot's appearance:

1. Double-click a plot element (for example, a line, axis, or title) to open that element's **Properties** window.

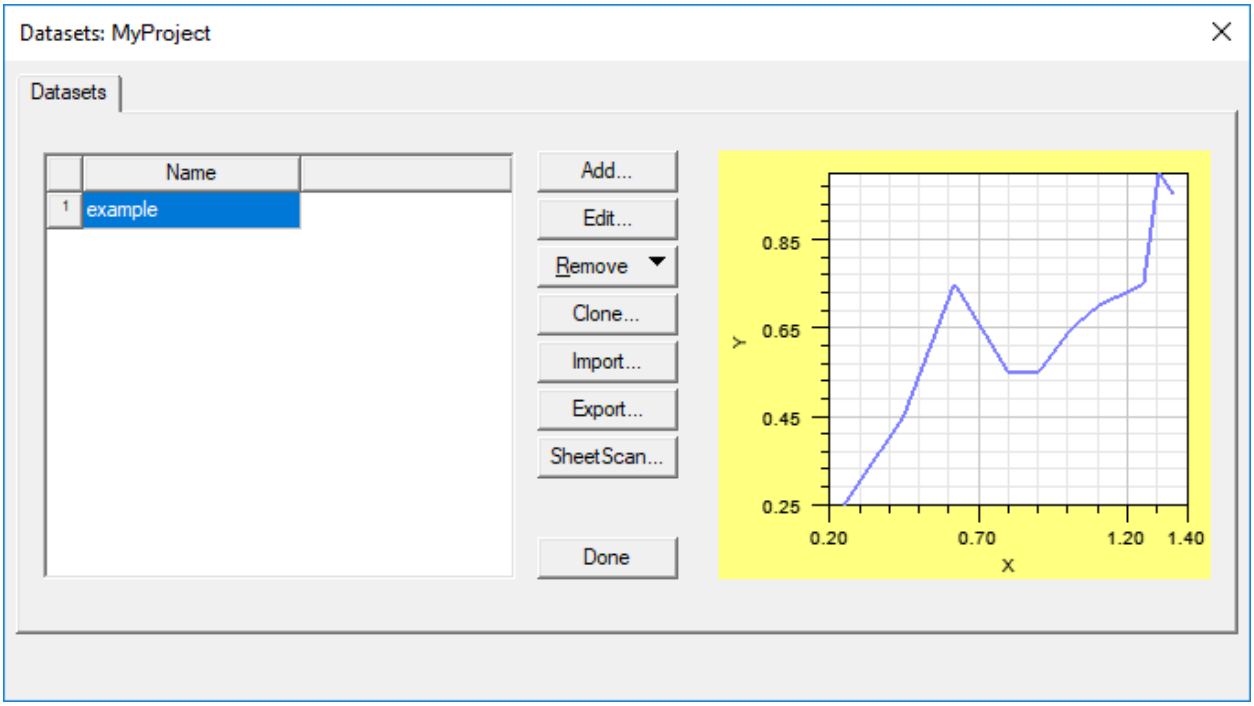
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Set the colors, fonts, and styles as applicable and as desired.
3. Click **Apply**.
4. Click **OK** to exit the **Properties** window.

   The plot now displays as specified.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Using SheetScan

SheetScan allows you to extract characteristics data from graphics such as data sheets which have been scanned and saved in any of the following formats: .bmp, .dib, .jpg, .gif, .tif, .tga, .pcx, .htm, or .html.

Access SheetScan from the **Datasets** window:

- For project-level datasets, click **Project** > **Datasets**.
- For design-level datasets, click **HFSS** > **Design Datasets**.

In addition to importing graphic files directly, SheetScan also can be used to browse the Internet for datasheet information and transfer a snapshot of the web page to the SheetScan editor where you can map axes on the image as an overlay. You can then manually add datapoints to approximate the characteristic curve(s) on the datasheet.The sampled data can then be converted to Ansys Electronics Desktop format, and the extracted data exported to an Ansys Electronics Desktop dataset or saved to a tab-delimited file.

The process for creating a dataset using SheetScan involves four basic operations:

- Loading a datasheet into SheetScan.
- Defining a coordinate system for the imported datasheet picture.
- Defining a characteristic curve using the datasheet picture as reference.
- Exporting the characteristic curve data to a file or to a dataset.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SheetScan Menus and Settings

In the SheetScan window, various menus and toolbars allow you to access SheetScan functionality.

The top menu bar contains the following submenus:

- **File** – allows you to open, close, save, and print files.
- **Edit** – provides access to cut, copy, and paste functions.
- **View** – allows you to select which toolbars appear in the SheetScan window (see below).
- **Options** – provides access to document settings.
- **Picture** – provides options for loading a picture or HTML file into SheetScan, and for deleting a picture.

> **Note:**
>
> The **Delete** action cannot be undone. If you delete a picture from the SheetScan editor, you must reload it from the source file or Internet web page.

- **Coordinate System** – provides options for creating and editing a coordinate system.
- **Curve** – provides options that allow you to define a curve.
- **Window** – provides options for rearranging the currently open SheetScan windows (for example, overlapping and tiling).
- **Help** – accesses the Electronics Desktop online help.

SheetScan View Options

You can toggle the following toolbars via **View** > **Toolbar**:

- **Standard** – provides access to basic functions such as file Open and Save, Cut, Copy, Paste, Print, and Help.

- **Curve** – contains a drop-down menu allowing you to select a curve and tools for working with curve values.

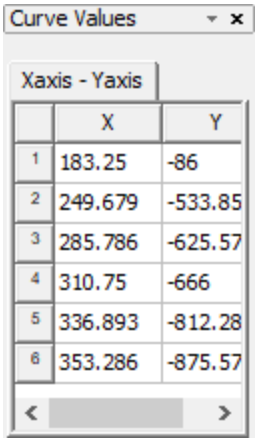
- **Zoom** – provides tools for scaling the current view.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From the **View** menu, you can also toggle the following:

- **Status Bar** – displays, at the bottom of the screen, the current cursor coordinates.

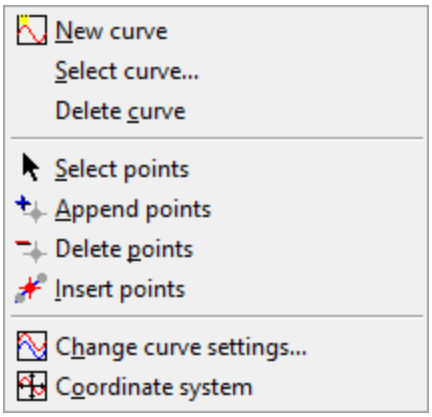
- **Curve Values** – dockable window that displays the data points used while creating a characteristic curve.

- **Grid** – displays the coordinate grid you created over the sheet.

SheetScan Right-click Menu

Once you have loaded a sheet, you can right-click in the SheetScan window to access curve and coordinate system options:



SheetScan Settings

Default settings can be changed from **Options** > **Settings**.

The **Settings** dialog box contains three tabs:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Document** – allows you to set the width and height of the sheet created when a picture is imported. You can either enter the dimensions manually, or allow SheetScan to adapt the dimensions to the picture being loaded.
- **Axis** – allows you to set the name, units, scaling, and offset for the X and Y axis. You can also select **Monotonicity in X** to prevent adding consecutive data points whose X-values are not increasing.
- **Representation** – allows you to choose whether to connect points on the characteristic curve and to choose the connecting line color. You can also decide whether to display markers and choose marker colors.

> **Note:**
>
> You can also override the default settings on the **Axis** and **Representation** tabs for individual curves (See: Defining a Characteristic Curve in SheetScan).

Loading a Datasheet Picture into SheetScan

By default, SheetScan opens a new, blank datasheet editing window.

There are two ways to load a datasheet picture into the editor: directly, or by using the HTML Viewer.

Loading a Datasheet Picture Directly

1. Browse directly to the datasheet picture file by choosing **Picture** > **Load Picture** to open a file browser window.
2. When you have located the desired file, click **OK** to load the image into the SheetScan editor. Supported file types include: .bmp, .dib, .jpg, .gif, .tif, .tga, .pcx, .htm, and .html.

Loading a Datasheet Picture Using the HTML Viewer

1. Choose **Picture** > **Internet** to open the HTML Viewer.
2. Browse the Internet for the desired datasheet.
3. Resize the HTML Viewer window and adjust its scrollbars until the desired portion of the datasheet is in view.
4. Click the **To SheetScan** button ( ) to copy the visible contents into the SheetScan editor window.

> **Note:**
>
> To hide the datasheet picture, select **View** > **Picture**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

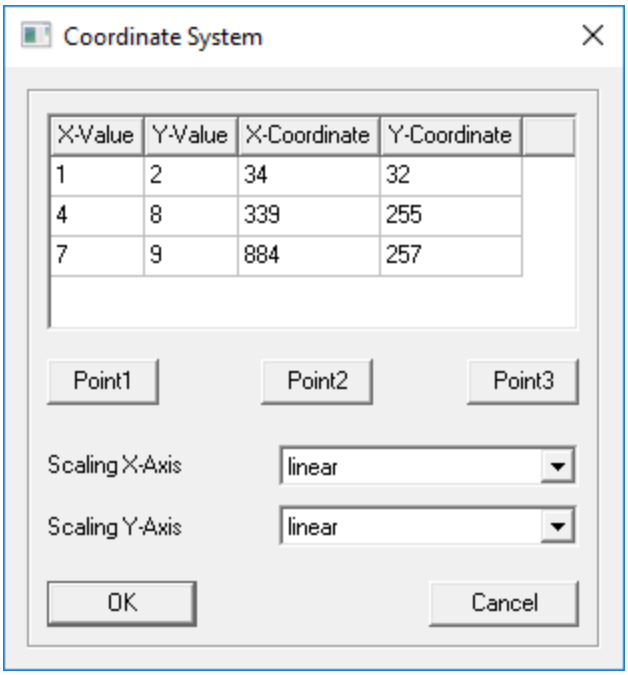
After loading a datasheet picture into the SheetScan editor, the next step is defining a coordinate system for the imported picture.

Defining a SheetScan Coordinate System

You can define a coordinate system for a graph on a datasheet picture that you have previously loaded into the SheetScan editor.

To define the coordinate system:

1. Select **Coordinate System** > **New** to open the **Coordinate System** dialog box.



   You can enter points manually or select them using a crosshair.

2. To activate the crosshair cursor, click **Point1**.

   The cursor transforms into a crosshair.

3. Position the cursor over a corner of the datasheet graph and click the left mouse button.

   The **Coordinate System** dialog box reappears, displaying the X- and Y-Coordinate values for the chosen point.

4. Enter the X-Value and Y-Value for this point. Typically, these values will correspond to the values taken from the axis scale values on the datasheet.
5. Select the desired scaling (linear, logarithmic, or decibel) for both the X and Y axes.
6. Repeat the above steps for **Point2** and **Point3**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Click **OK**.

   The grid is placed over the graphic.

   > **Note:**
   >
   > - You can edit the grid after placement by selecting **Coordinate System** > **Properties**, by clicking the coordinate system icon on the Curve toolbar ( ), or by right-clicking in the SheetScan editing window and selecting **Coordinate System**.
   > - You can hide the grid by selecting **View** > **Grid**.

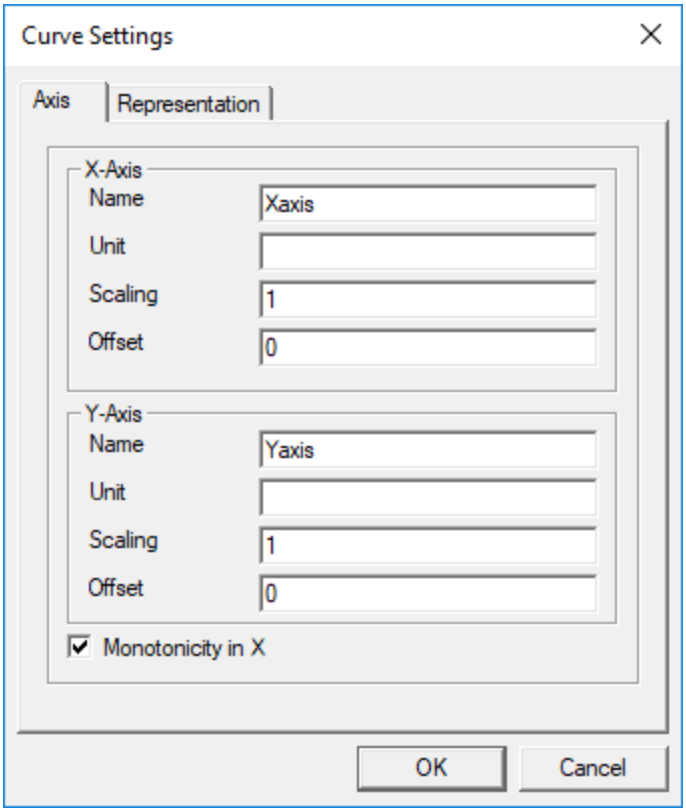
8. You can now define a Characteristic Curve.

Defining a Characteristic Curve in SheetScan

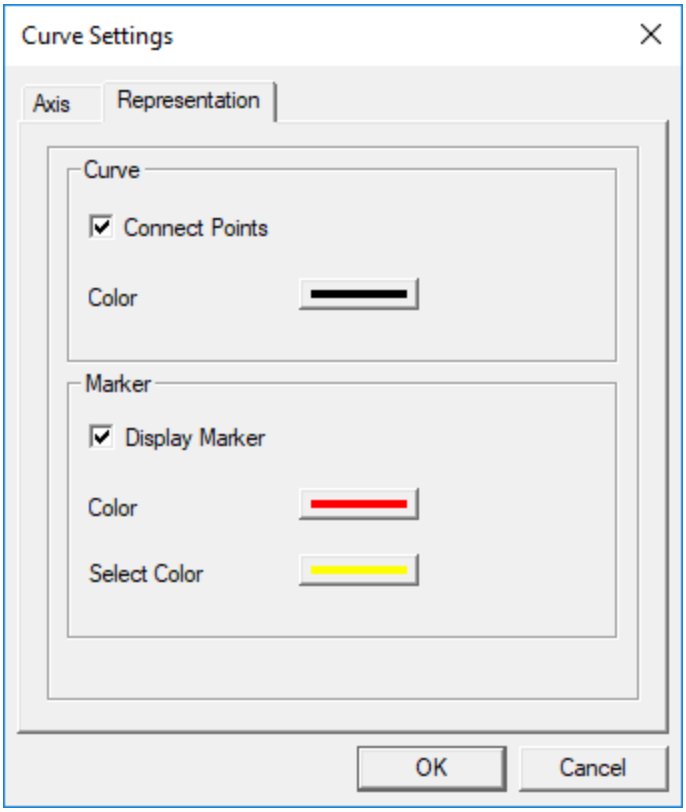
Once you have loaded a datasheet picture in the editor and have defined a coordinate system, you can define one or more characteristic curves as follows:

1. Choose **Curve** > **New**.

   The **Curve Settings** dialog box opens, containing the **Axis** and **Representation** tabs.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Use the fields on the **Axis** tab to define the curve's properties. Use the buttons on the **Representation** tab to change the curve's appearance.
3. Click **OK**.

   The cursor changes to a crosshair.

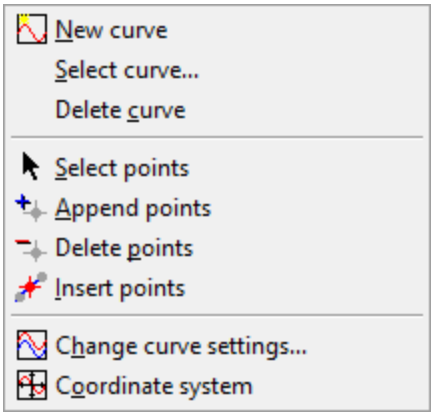
4. Click the points of the characteristic which you want to capture for the dataset. The points are connected automatically.
5. Repeat steps 1 through 4 for each additional characteristic curve you wish to define.

After characteristic curves have been defined, you can perform various operations on them. See: *Performing Operations on SheetScan Curves.*

Performing Operations on SheetScan Curves

Use the **Curve** menu (from the toolbar or by right-clicking in the sheet) to perform actions on SheetScan curves.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can perform the following actions:

- **New Curve** – allows you to define a new characteristic curve.
- **Select Curve** – opens the **Select Curve** dialog box, listing all active curves. Select the desired curve and click **OK**.
- **Delete Curve** – deletes the selected curve.

  > **Warning:**
  >
  > You cannot undo this action. If you delete a curve and its data points from the SheetScan editor, you must reconstruct it manually.

- **Select Points** – select this option, then click a point to select it. **Ctrl+click** allows you to select multiple points.
- **Append Points** – select this option, then click to add data points to the end of a curve.
- **Delete Points** – select this option, then click a data point to remove it from the curve.
- **Insert Points** – select this option, then click to insert new data points between existing data points.
- **Change Curve Settings** – opens the **Curve Settings** dialog box, where you can edit the initial curve settings.
- **Coordinate System** – opens the Coordinate System dialog box, where you can edit the coordinate system settings.

Checking for Monotonicity in X

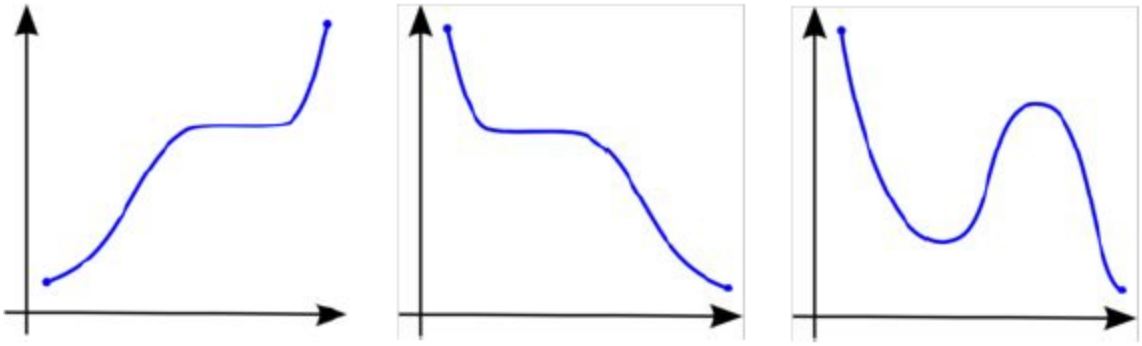
HFSS requires that characteristics be monotonically increasing along the X-axis. In other words, successive data points must have increasing X-values, while Y-values may both increase and decrease.

To check for monotonicity in X from SheetScan:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select **Curve** > **Check Monotonicity**.

   If the characteristic curve is monotonically increasing in X-value, the check completes without notice. Typical examples of curves that meet monotonicity criteria are shown below.



2. If the characteristic curve is not monotonically increasing in X-value, a dialog box displays informing you that errors were found. Click **Yes** to have SheetScan automatically correct the errors.

Importing Characteristic Data into SheetScan

SheetScan supports data import from the following file types:

- Twin Builder Characteristic (*.mdx, *.mda)
- Microsoft Excel (*.xls, *.xlsx)
- Text (*.txt)
- Comma Separated Value (*.csv)
- Spice (*.out)
- Comtrade (*.cfg)
- TEK Oscilloscope (*.dat)

> **Note:**
>
> Before importing characteristic data, you must define a coordinate system.

To import characteristic curve data into SheetScan:

1. Click **File** > **Import**.
2. In the file **Open** dialog box, select the desired data file and click **Open**.
   a. Selecting an **.xls** or **.xlsx** file containing multiple sheets opens a **Table Properties** dialog box where you can choose the desired sheet from a drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

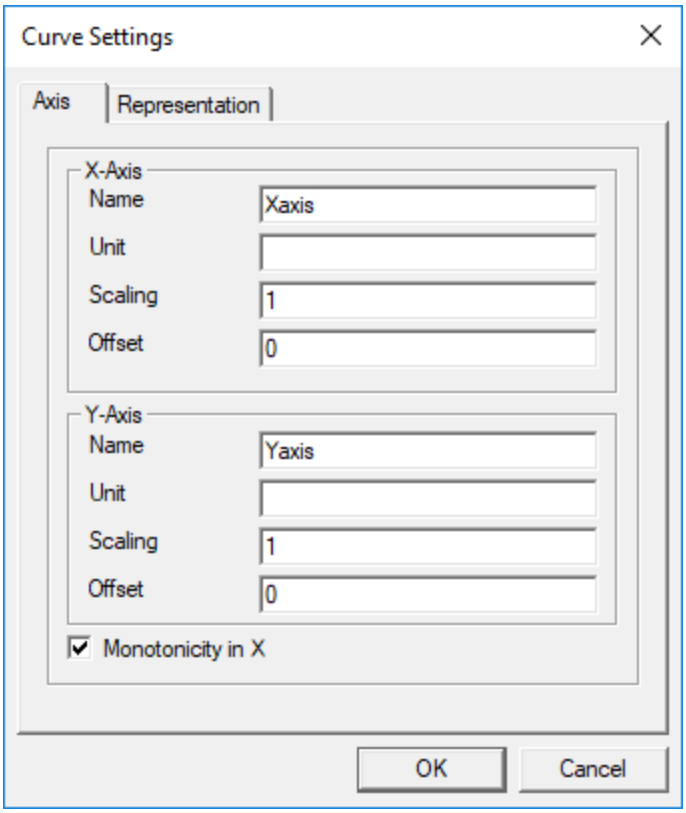
Otherwise, selecting an **.xls** or **.xlsx** file imports the data immediately into the **Add Dataset** dialog box.

> **Note:**
>
> - Only the first two columns of data are imported, the left-most column containing the X-coordinate values. The x-coordinate values for successive data points must increase within ten significant digits. Non-numeric entries are assigned a value of zero.
> - The first row of data is assumed to contain column headings and is ignored.

b. Selecting a **.txt** or **.csv** file opens an **Import** dialog box in which you can specify how to settings for reading the data in the file for import. You can choose the separator(s) and decimal symbol, as well as the line at which to begin the import. The dialog box shows both the original text and the text as it would appear when imported based on the current import settings.

When satisfied with the import settings, click **OK** to import the data.

The **Curve Settings** dialog box opens.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Change **Curve Settings** as needed and click **OK** to complete the data import.

   The new characteristic curve is added to the current SheetScan sheet.

Exporting SheetScan Data

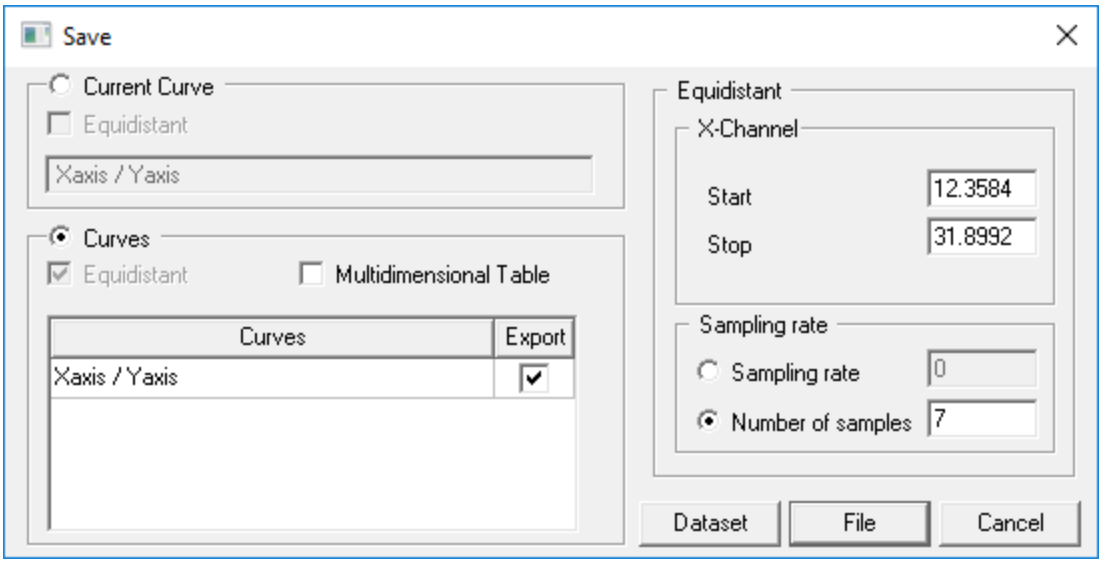
After defining a coordinate system and adding a curve, you can export SheetScan curve data directly to:

- Electronics Desktop Dataset
- Twin Builder Characteristic (*.mdx) file
- Comma Separated Value (*.csv) file
- Comtrade (*.cfg) file

To export curve data:

1. Click **File** > **Export**.

   The **Save** dialog box appears.



2. In the **Save** dialog box, choose **Current Curve** (default) to export current curve data, or **Curves** to choose the curve(s) whose data you wish to export. Choosing **Curves** reveals a list box showing all of the curves available for export.

> **Note:**
>
> The **Multidimensional Table** option is not currently supported.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Choose **Equidistant** if you want to set the **Start** and **Stop X-Channel** values and a **Sample Rate** or **Number of samples** for the exported dataset(s).
4. Click **Dataset** to export curve data directly to the project's dataset file, or click **File** to export as one of the supported file types.

Defining an Expression

Expressions are mathematical descriptions that typically contain Intrinsic Functions, such as sin (x), and arithmetic operators, such as +, -, *, and /, as well as defined variables. For example, you could define: x_size = 1mm, y_size = x_size + sin(x_size). Defining one variable in terms of another makes it a dependent variable. Dependent variables, though useful in many situations, cannot be the subject of optimization, sensitivity analysis, tuning, or statistical analysis.

The Constants tab of the **Project Variables** window lists all available pre-defined constants. These may not be reassigned a new value.

Numerical values may be entered in Ansys shorthand for scientific notation. For example, $5x10^7$ could be entered as **5e7**.

Valid Operators for Expressions

The operators that can be used to define an expression or function are performed in a sequence.

The following table lists valid operators and the sequence in which they are accepted (listed in decreasing precedence):

()	Parenthesis	1
!	Not	2
^ (or **)	Exponentiation (If you use "**" for exponentiation, as in previous software versions, it is automatically changed to "^".)	3
-	Unary minus	4
*****	Multiplication	5
/	Division	5
+	Addition	6
-	Subtraction	6
==	Equals	7
!=	Not equals	7
>	Greater than	7
<	Less than	7
>=	Greater than or equal to	7
<=	Less than or equal to	7
&&	Logical and	8

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

||	Logical or	8

Using Intrinsic Functions in Expressions

HFSS recognizes a set of intrinsic trigonometric and mathematical functions that can be used to define expressions. Intrinsic function names are reserved and may not be used as variable names.

> **Note:**
>
> If units are not specified, all trigonometric functions interpret their arguments as radians. Likewise, inverse trigonometric functions' return values are in given in radians. When the argument to a trigonometric expression is a variable, the units are assumed to be radians. To have values interpreted in degrees, supply the argument with the unit name **deg**.

The following intrinsic functions may be used to define expressions:

Function	Description	Syntax
abs	Absolute value (|x|)	abs(x)
sin	Sine	sin(x)
cos	Cosine	cos(x)
tan	Tangent	tan(x)
asin	Arcsine	asin(x)
acos	Arccosine	acos(x)
atan	Arctangent (in range of -90 to 90 degrees)	atan(x)
atan2	Arctangent (in range of -180 to 180 degrees)	atan2(y,x)
asinh	Hyperbolic Arcsine	asinh(x)
atanh	Hyperbolic Arctangent	atanh(x)
sinh	Hyperbolic Sine	sinh(x)
cosh	Hyperbolic Cosine	cosh(x)
tanh	Hyperbolic Tangent	tanh(x)
even	Returns 1 if integer part of the number is even; returns 0 otherwise.	even(x)
odd	Returns 1 if integer part of the number is odd; returns 0 otherwise.	odd(x)
sgn	Sign extraction	sgn(x)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

exp	Exponential (e^x)	exp(x)
pow	Raise to power (x^y)	pow(x,y)
if	If	if(cond_exp,true_exp,false_exp)
pwl	Piecewise Linear. (**pwl** can be used with datasets for Design Variables but not for Project variables).	pwl(dataset_exp,variable)
pwl_periodic	Piecewise Linear for periodic extrapolation on x.	pwl_periodic (dataset_exp,variable)
sqrt	Square Root	sqrt(x)
ln	Natural Logarithm (The "log" function has been discontinued. If you use "log(x)" in an expression, the software automatically changes it to "ln (x)".)	ln(x)
log10	Logarithm base 10	log10(x)
int	Truncated integer function	int(x)
nint	Nearest integer	nint(x)
max	Maximum value of two parameters	max(x,y)
min	Minimum value of two parameters	min(x,y)
mod	Modulus	mod(x,y)
rem	Returns the fractional part of a decimal number such that rem (x) = x-int(x)	rem(x)
clp	Implements smooth interpolation employing weighted impact of all points of the dataset (not just the closest one). See formula below. **Note:** If used with a large 3D dataset, clp function will degrade.	clp (*datasetName* , X,Y,Z)

clp Formula

$$clp\left(Dataset,X,Y,Z\right)=\begin{cases} DatasetValue_i \text{ if } r_i\left(X,Y,Z\right) < r_{rounding} \\ \Sigma_i \frac{DatasetValue_i}{r_i^4} \Big/ \Sigma_i \frac{1}{r_i^4} \quad otherwise \end{cases}$$

$$r_i\left(X,Y,Z\right)=\sqrt{\left(X-DatasetX_i\right)^2+\left(Y-DatasetY_i\right)^2+\left(Z-DatasetZ_i\right)^2}$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using Piecewise Linear Functions in Expressions

The following piecewise linear intrinsic functions are accepted in expressions:

- **`pwl (dataset_expression, variable)`** – interpolates along the X-axis and returns a corresponding Y value.
- **`pwlx (dataset_expression, variable)`** – interpolates along the X-axis and returns a corresponding Y value.
- **`pwl_periodic (dataset_expression, variable)`** – also interpolates along the X-axis, but periodically.

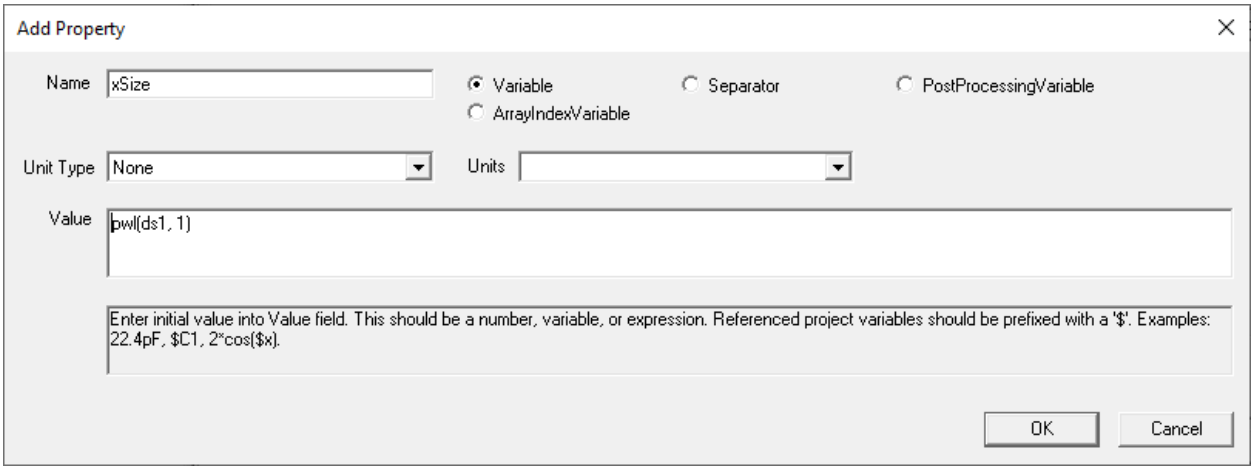
You can use **pwl** in an expression that uses datasets for such things as a frequency dependent material property (See: Adding Datasets).

For example, you can specify BulkConductivity as:

```
pwlx($ds1, Freq)
```

You can create a design variable representing a dimension `xSize` as `pwl(ds1, 1)` where `ds1` is a dataset.

Doing so looks like this:

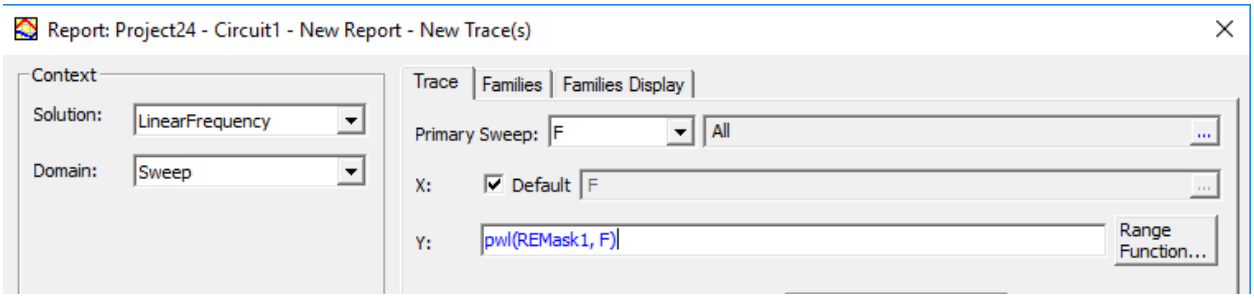


After the dataset is configured, the **Properties** window shows the new variable.

To use datasets in reports, create a report using **pwl(dataset_expression,**`variable`**)** where `variable` is the primary sweep in the report.

For example:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

pwl can be used with datasets for Design Variables but not for Project variables.

Using Dataset Expressions

A dataset is a collection of data. It can take the following form:

$ds1((x0,y0), ..., (xn,yn))

Once created, a dataset (such as $ds1) may be used as the first parameter to piecewise linear (**pwl**, **pwlx**, and **pwl_periodic**) functions, and may also be assigned to variables, in which case the variable may be used as the second parameter to **pwl**, **pwlx**, and **pwl_periodic** functions.

You can generate a dataset using a series of points in a plot on the **Datasets** dialog box (See: Adding Datasets). Each plot consists of straight line segments whose vertices represent their end points. A curve is fitted to the segments of the plot. This curve, which best fits the segmented plot, consists of the co-ordinates used in the creation of the dataset. The dataset may then be used in the piecewise linear intrinsic functions.

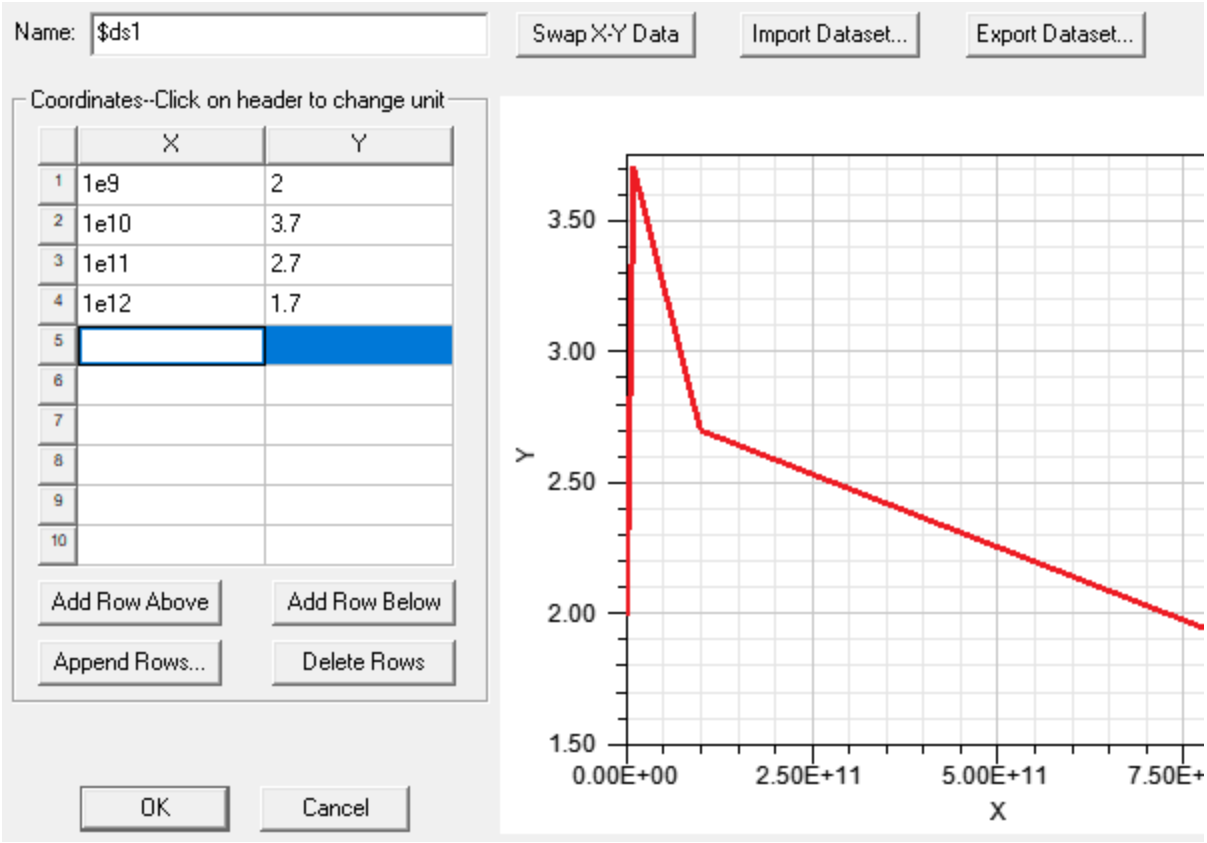
Note:

The following is an example showing how to make a material property frequency dependent using a dataset as the first parameter to a **pwl** function. The values used are arbitrary.

1. For a project-level dataset, click **Project** > **Datasets**. For a design-level dataset, click **HFSS** > **Design Datasets**.
2. Click **Add**.
3. Set the **Name** field and **Coordinates** as desired and click **OK**.

   The dataset is created.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

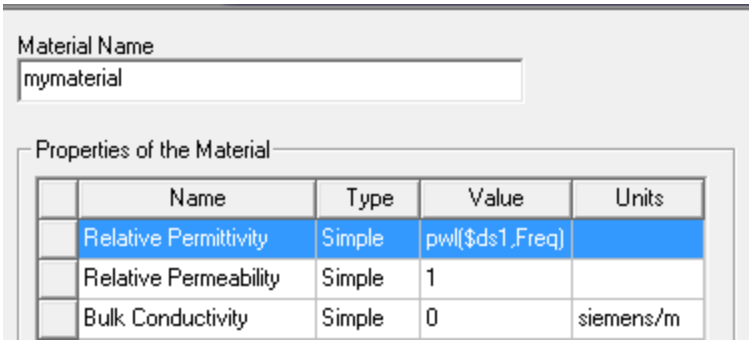


Note:

By default, the dollar sign ($) is assigned to a project dataset even if you do not use one while naming it.

4. Go to **Tools** > **Edit Libraries** > **Materials** to open the **Edit Libraries** dialog box.
5. Click **Add Material**.
6. Type in the piecewise linear function and use the dataset $ds1, as shown in the following figure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. Click **OK**.

Handling Delta Temperature Units in Expressions

When Temperature and Delta Temperature quantities are used as operands to plus or minus operations in an expression, they are handled specially. The biggest difference is an automatic unit change of the resulting values based on the units of the operands.

- When two Temperature quantities are being subtracted (either the first or both operands is/are temperature quantities), the result value has Delta Temperature units.
- When a Delta Temperature quantity is added or subtracted from a Temperature quantity, the result value has Temperature units.
- Adding or subtracting two Delta Temperature quantities results in a quantity with Delta Temperature units.
- Subtracting two temperature quantities is an observable behavior change when compared to earlier releases of Ansys Electronics Desktop.

Temperature Units

Celsius Family

- cel, delta_cel (Legacy name for delta_cel, celdiff continues to be supported)

Kelvin Family

- mkel, delta_mkel (milli Kelvin) (Legacy name for delta_mkel, mkeldiff continues to be supported)
- ckel, delta_ckel (centi Kelvin)
- dkel, delta_dkel (deci Kelvin)
- kel, delta_kel (Legacy name for delta_kel, keldiff continues to be supported)

Farenheit Family

- fah, delta_fah

Temperature-related Use Cases

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The high-level legal use cases (in unit types) are:

- Temperature - Temperature = DeltaTemperature
- Temperature +/- DeltaTemperature = Temperature
- DeltaTemperature +/- DeltaTemperature = DeltaTemperature

The minor variations in these use cases specify how the actual units (Celsius is a unit of the Temperature unit type) are handled.

Temperature - Temperature

If they have the same unit, then the Delta Temperature unit will be a matching one.

For example:

- 10 cel - 5 cel = 5 delta_cel
- 100 kel - 90kel = 10 delta_kel
- 10 fah - 1 fah = 9 delta_fah

If they have different units, they are converted to the default units for temperature difference.

For example:

- 10cel - 1 kel = 9 delta_kel

Temperature +/- DeltaTemperature

The resulting temperature quantity will retain the units of the Temperature quantity (first operand)

DeltaTemperature +/- DeltaTemperature

- If they have the same unit, the resulting Delta Temperature quantity will retain that.
- If they have different units, the resulting Delta Temperature quantity will have the default Delta Temperature units
- 5 delta_cel + 10 delta_cel = 15 delta_cel
- 10 delta_cel + 1 delta_kel = 11 delta_kel

All other use cases in a plus or minus arithmetic operation are physically meaningless and revert to previous bahavior, where Electronics Desktop converts quantities to their SI values and then operates on the plain numbers.

- Temperature + Temperature
- Temperature +/- non-Temperature/non-Delta Temperature
- Delta Temperature +/- non-Temperature/non-Delta Temperature
- non-Temperature/non-Delta Temperature +/- Delta Temperature
- non-Temperature/non-Delta Temperature +/- Temperature

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining Mathematical Functions

A mathematical function is an expression that references another defined variable. A function's definition can include both expressions and variables.

The following mathematical functions may be used to define expressions:

Basic Functions	**/**, **+**, **-**, *, mod (modulus), ** (exponentiation), **-** (Unary minus), **==** (equals), **!** (not), **!=** (not equals), **>** (greater than), **<** (less than), **>=** (greater than equals), **<=** (less than equals), **&&** (logical and), **||** (logical or)
Intrinsic Functions	**if**, **abs**, **exp**, **pow**, **ln** (natural log), **log10** (log to the base 10), **sqrt**
Trigonometric Expressions	**sin**, **cos**, **tan**, **asin**, **acos**, **atan**, **sinh**, **cosh**, **tanh**

The predefined variables X, Y, Z, Phi, Theta, R, and Rho must be entered as such. X, Y, and Z are the rectangular (Cartesian) coordinates. Phi, Theta, and Rho are the spherical coordinates. R is the cylindrical radius, and Rho is the spherical radius.

If you do not specify units, all trigonometric expressions expect their arguments to be in radians, and the inverse trigonometric functions' return values are in radians. If you want to use degrees, you must supply the unit name **deg**. When the argument to a trigonometric expression is a variable, the units are assumed to be radians. These function names are reserved and may not be used as variable names.

As far as expression evaluation is concerned: units are conversion factors (that is, from the given unit to SI). Note also that the evaluated value of an expression) is always interpreted as in SI units.

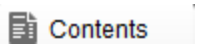
Meshing Quick Links

Use the following links for quick information on the following topics.

Defining Mesh Operations	Plotting Finite Element Mesh
Detecting and Addressing Model Problems to Improve Meshing	Handling Complicated Models
Specifying Initial Mesh Settings	

The help system provides different ways to find information and navigate quickly:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

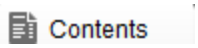
- Press **F1** on any open dialog box to open the relevant help topic.
- *A hierarchical table of contents* –  – Browse through the table of contents, expand entries, and close entries. Click an entry to see it in the content area.
- *A full text search* – To locate occurrences of a word or phrase that may be contained in the help, use the search function.

Modeling Quick Links

Use the following links for quick information on the following topics.

Setting Units of Measurement
Assigning Transparency to an Object
Subtracting Objects
Measuring Objects
Drawing Bondwires
Keyboard Shortcuts
Modifying the Model View
Working with Variables
Setting Modeler Drawing Options
Selecting Items in the Modeler Window
Drawing Objects
Choosing the Cursor Movement Mode
Importing Files
Modifying Objects
Creating a User Defined Primitive
Assigning Materials

The help system provides different ways to find information and navigate quickly:

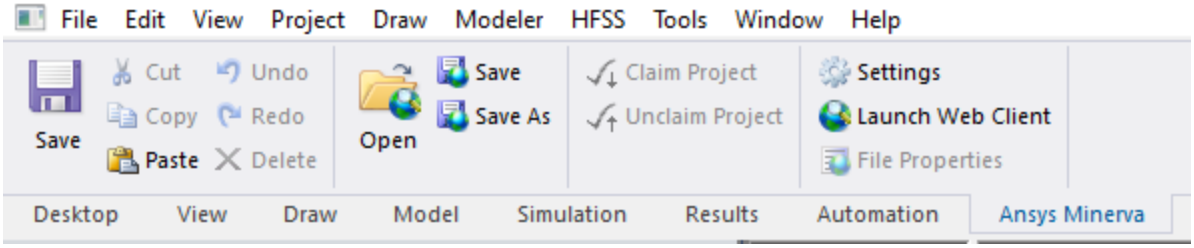
- Press **F1** on any open dialog box to open the relevant help topic.
- *A hierarchical table of contents* –  – Browse through the table of contents, expand entries, and close entries. Click an entry to see it in the content area.
- *A full text search* – To locate occurrences of a word or phrase that may be contained in the help, use the search function.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

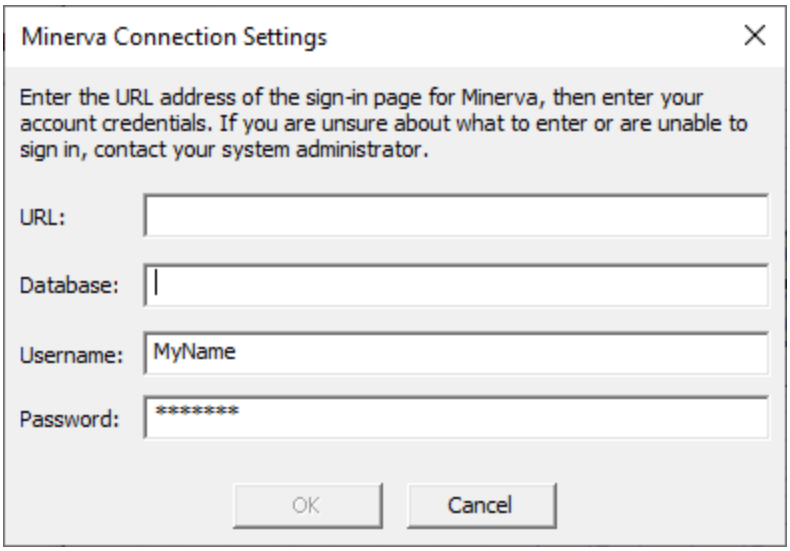
4 - Minerva Remote Storage Environment

Ansys Minerva is a remote storage environment where you can store Ansys project archive files. You can collaborate with other users by downloading a project, making changes, and then uploading the project. One of the features of Ansys Minerva is the ability to store and retrieve Ansys projects from the Minerva server. This ability has been integrated into Electronics Desktop so it is now possible to download and open a project stored on Minerva directly from the Electronics Desktop interface. In the same way, it is also possible to save a project to Minerva. The image below shows the Minerva ribbon containing Minerva commands accessible through the desktop.



Minerva has its own HTML web interface with additional features as a knowledge management application that secures critical simulation data, and provides simulation process and decision support to simulation teams across geographies and functional silos.

Minerva Settings

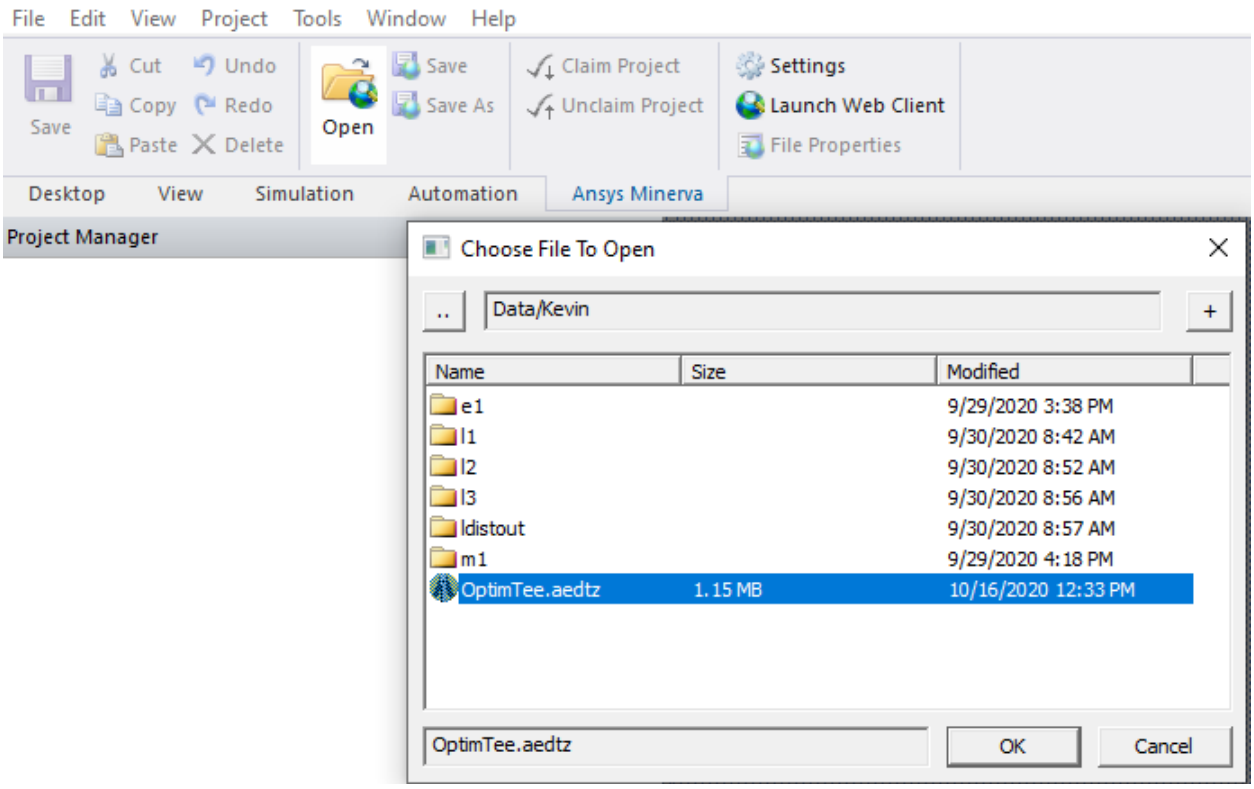


When you first login via the desktop, or to the Web Client, you must provide a Minerva URL, a database name, a user name, and a password. You will need to obtain these from your system

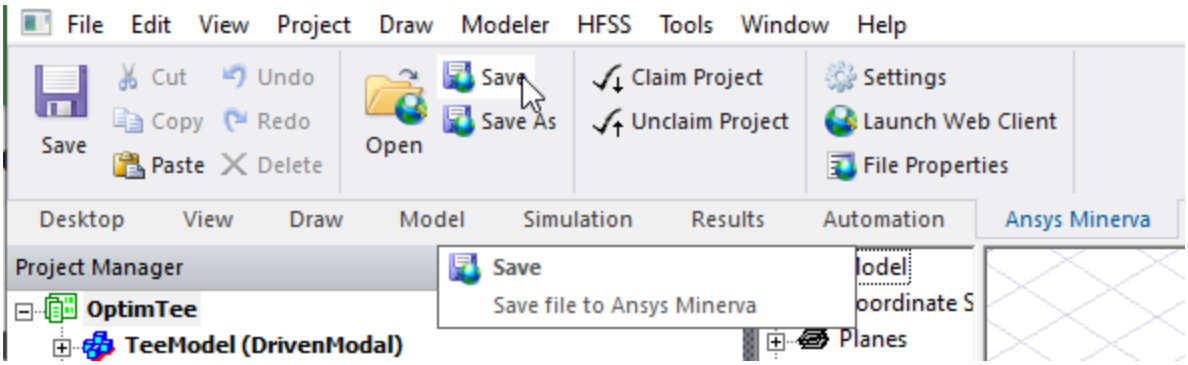
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

administrator. Once you provide the connection settings, the other Minvera commands are enabled.

Open : opens the Choose File to Open browser that lets you select and download an archive file, extract it, and open it. The [..] button moves up one level in the file hierarchy. The [+] button creates a new folder at the current level.

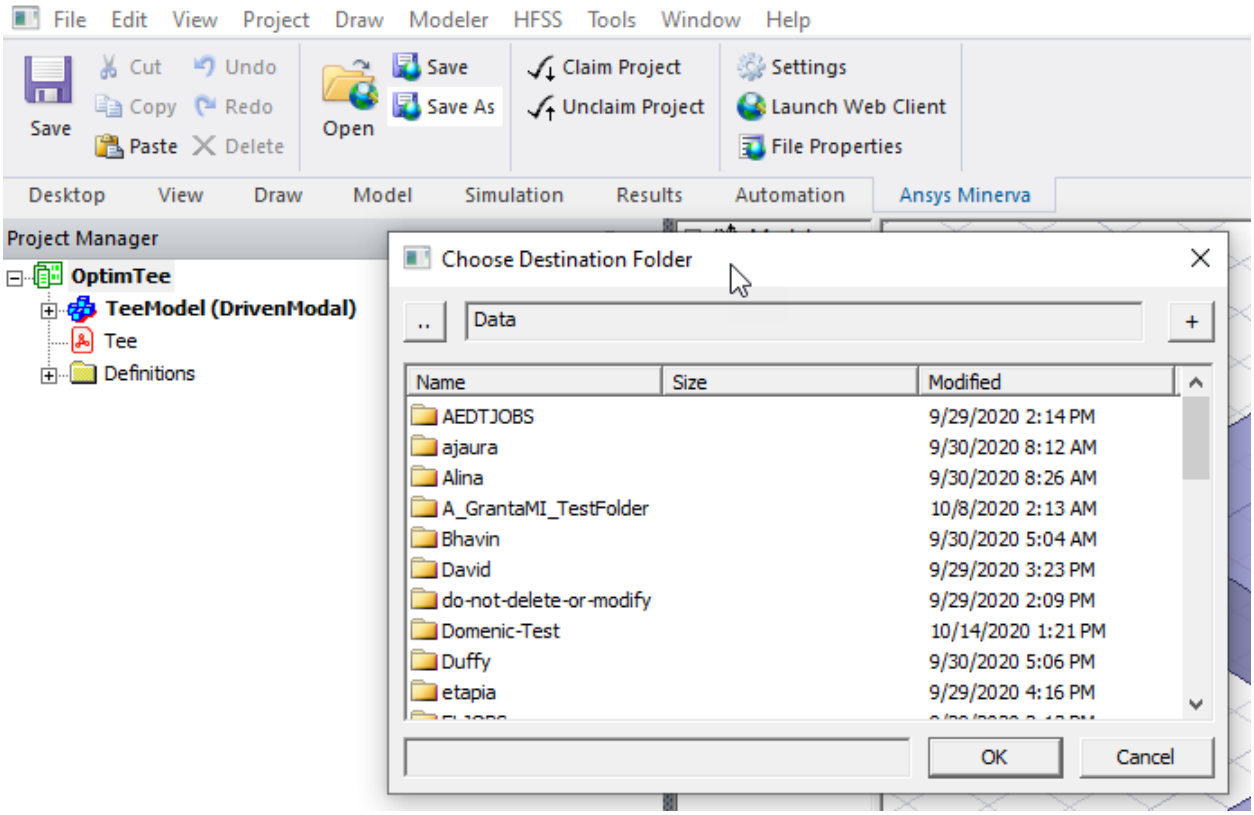


Save : archives the current project, and saves it to a remote Minerva system.



Save As... prompt for remote file, archive project, save to remote Minerva system.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In order to prevent users from overwriting each other's work, Minerva has added the concept of claiming and unclaiming files. A file claimed by one user cannot be overwritten on the Minerva server by another user. Unclaimed files can be overwritten by anyone at any time. Once a file has been claimed by one user, it cannot be claimed by a different user until it has been unclaimed by the original claimer. Note: even if a project is claimed by one user, other users are free to download, open and edit the file locally.

The recommended flow when working on a project is:

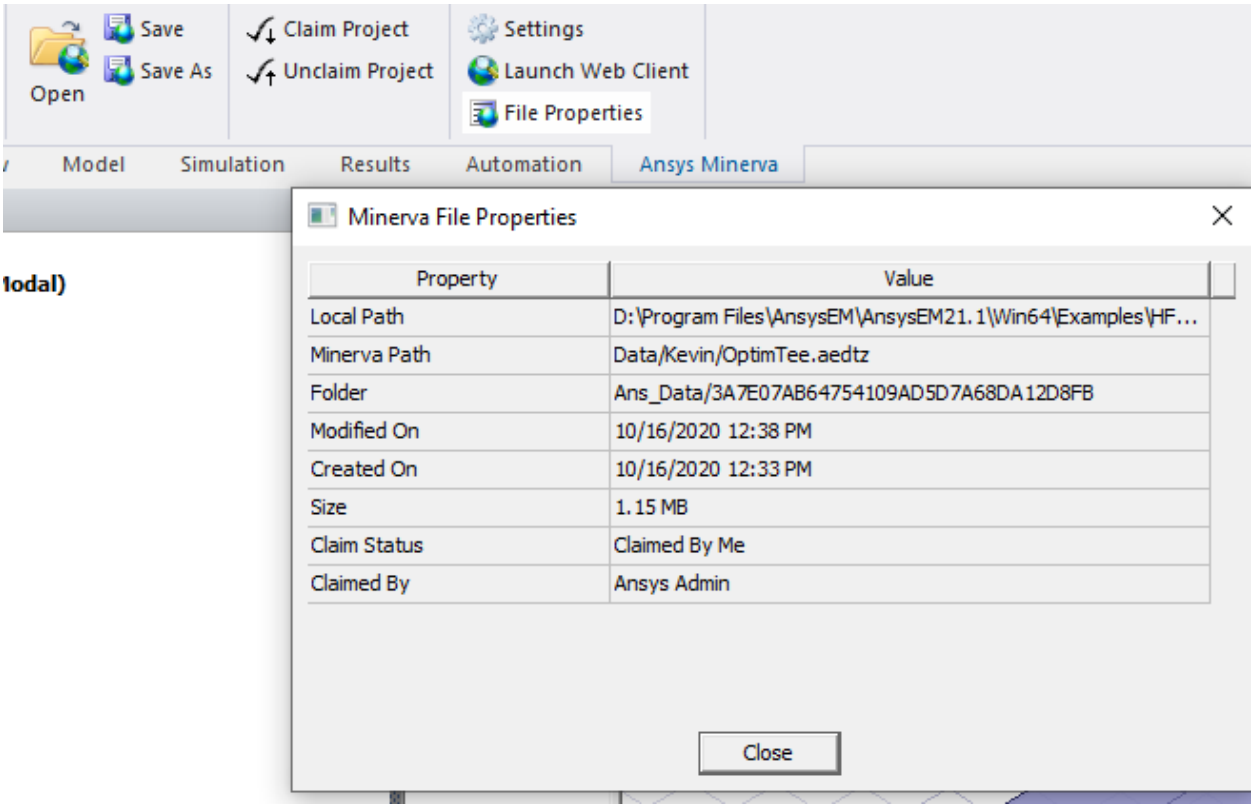
1. Claim the file, make sure you are working with the most recent version
2. Make edits
3. Save the file to Minerva
4. Unclaim the file

Claim Project: claims file so it cannot be modified by other users.

Unclaim Project: releases claimed file.

File properties: opens a dialog letting you view remote file properties. Notice the Claim Status and Claimed By properties.

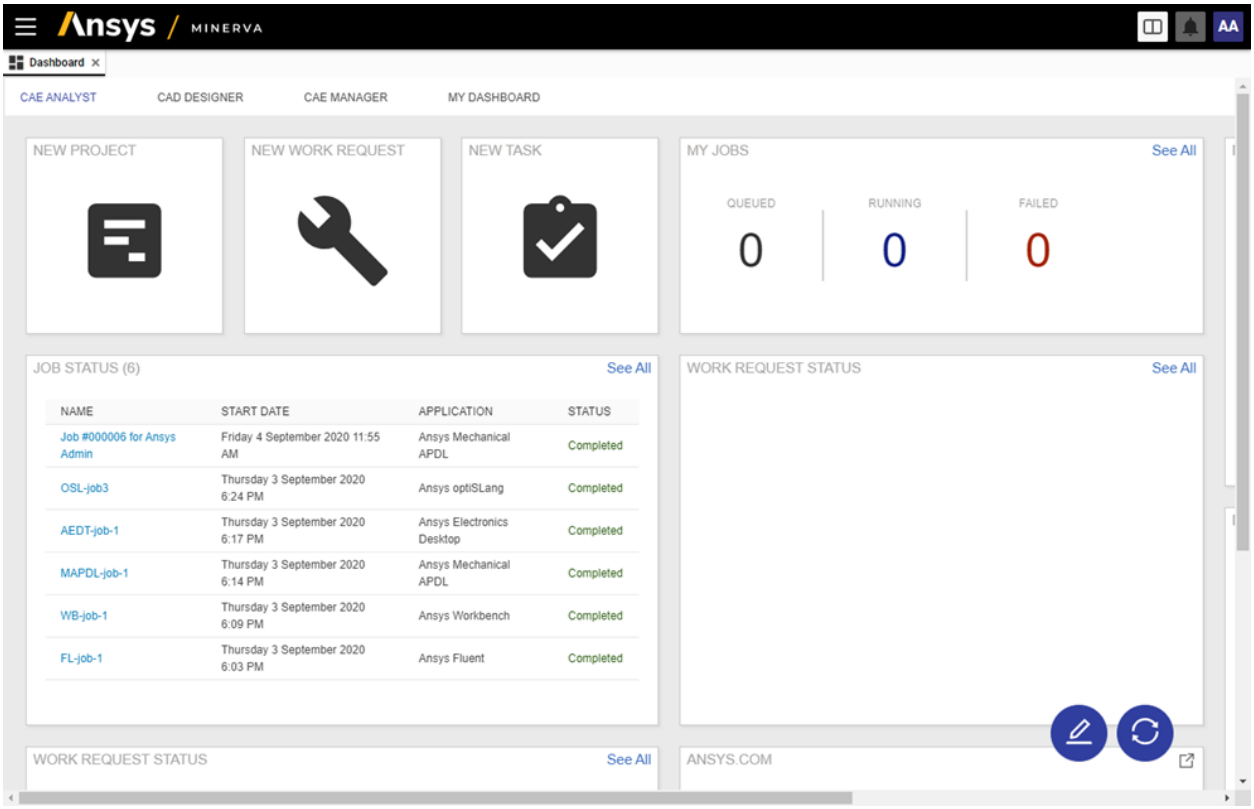
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Launch Web Client:

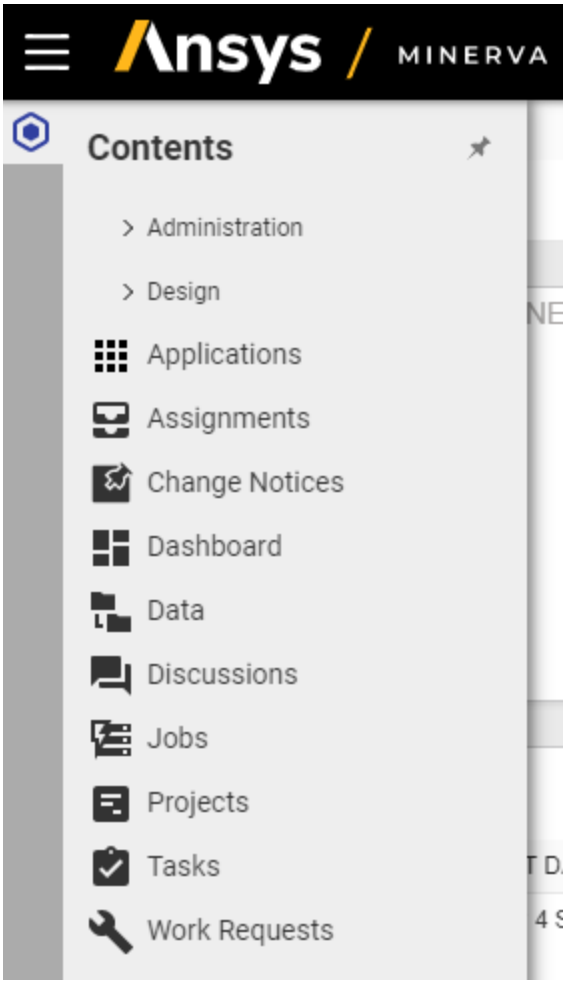
Ansys Minerva, powered by Aras, is a knowledge management application that secures critical simulation data, and provides simulation process and decision support to simulation teams across geographies and functional silos. Launching the Minerva Web Client gives access to these features. This opens a dialog calling for you to specify provide login information. It attempts to open a web browser window, so your default browser must be compliant.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



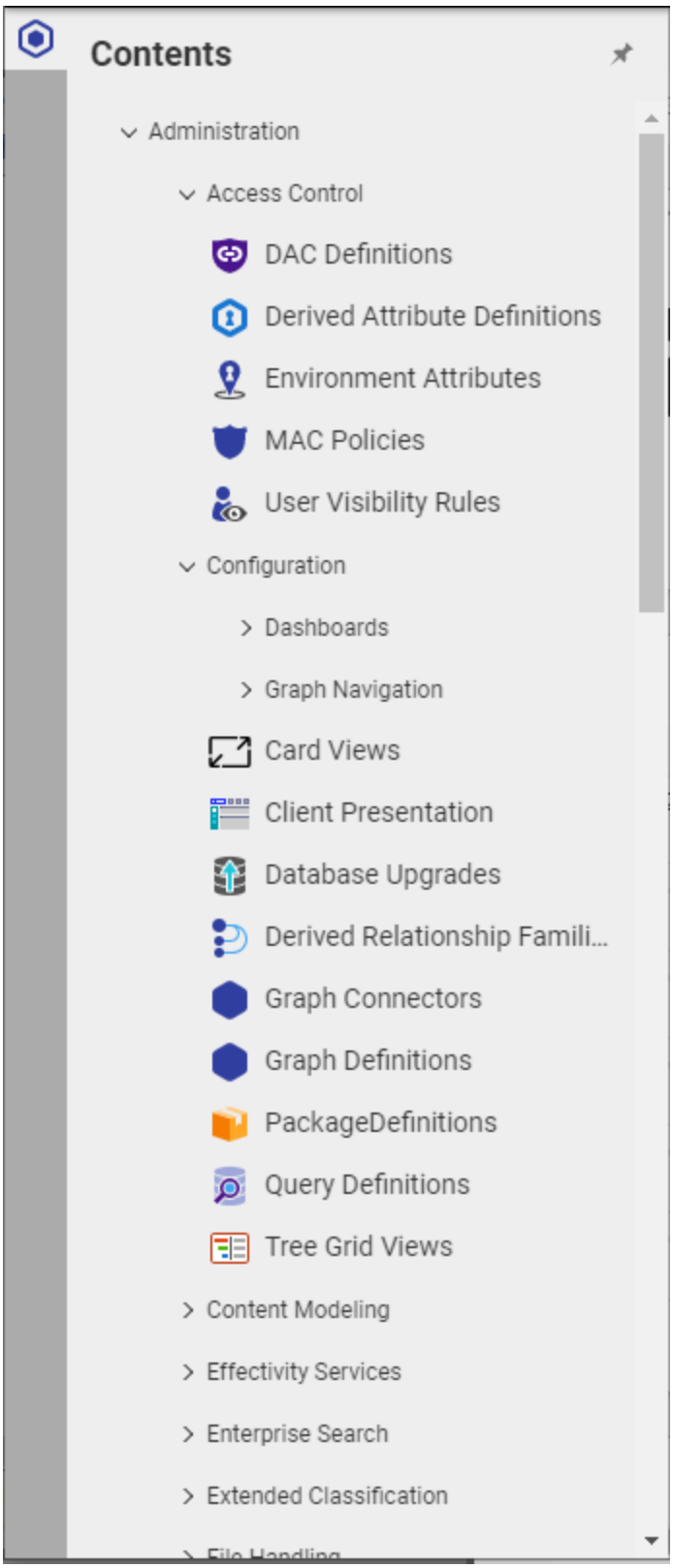
Clicking on the triple bar icon left of the Ansys logo opens up access to the more robust Minerva Web Client functionality.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The angle icons for Administration open up further functionality.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5 - Assigning Materials

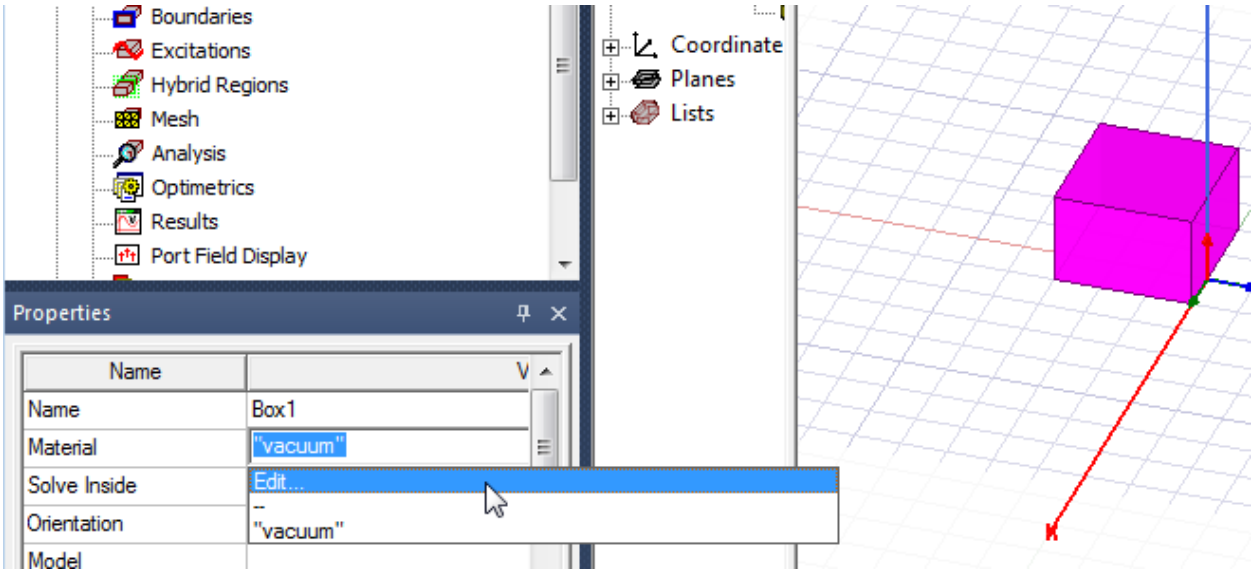
You can add, remove, and edit materials in two main ways:

- Using the **Tools** > **Edit Libraries** > **Materials** menu command.
- Under **Definitions** in the Project Manager, right-click **Materials** and select **Edit Library**.

Regardless of which of the preceding two methods you use to edit a library, any new material you create exists only in the current project. Similarly, if you edit an existing material, the edited version becomes a *Project* material and exists only within the current project. To make a new or modified material available for other projects, you must export it to a user library and choose that library (or select **Show all libraries**) within the **Select Definition** dialog box.

To assign a material to an object, follow this general procedure:

1. Select the object to which you want to assign a material.
2. Click **Modeler** > **Assign Material** or select the Material field in the **Properties** window for the selected object, and select **Edit...** from the drop-down menu:

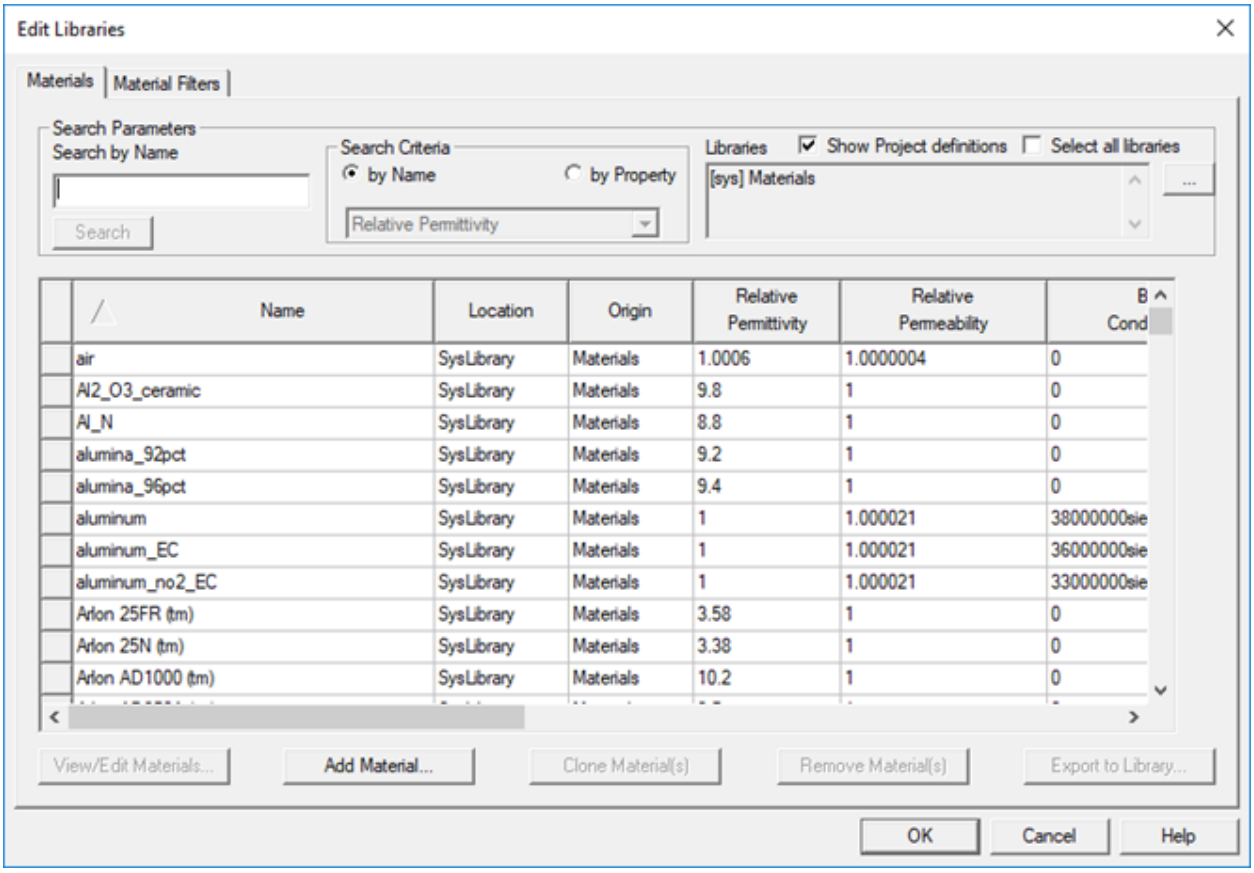


The **Select Definition** window appears. The current material is highlighted, with the Name, Location, Origin library, and parameter values shown.

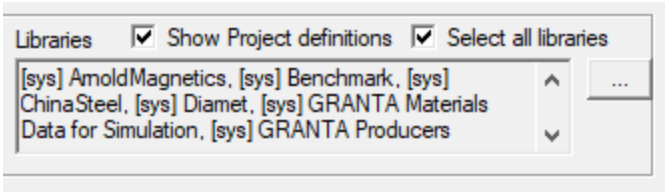
> **Note:** You can resize the dialog box to show more of the materials and/or material properties without having to use the scroll bars.

Select Definition Dialog Box - Materials

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

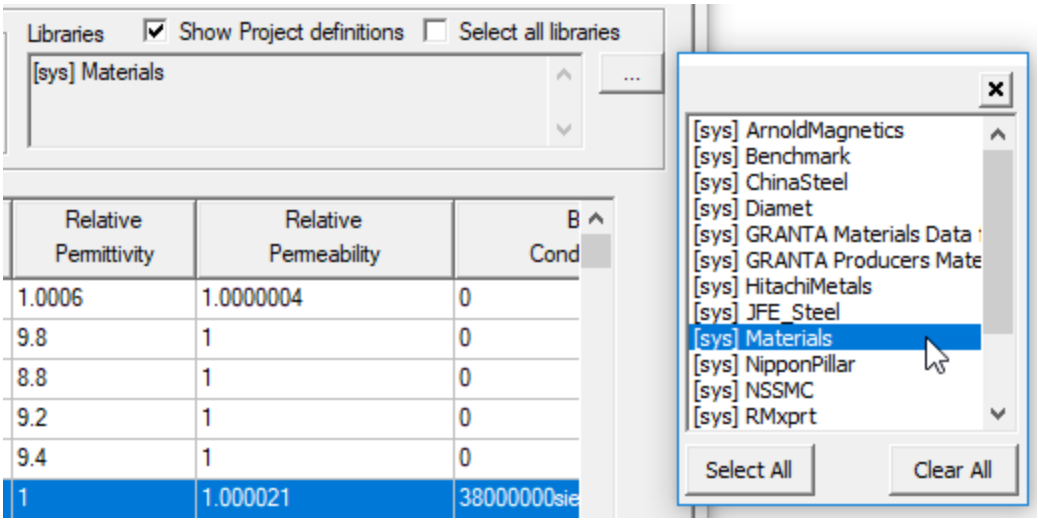


When the **Select all libraries** check box is selected, the window lists all of the materials in Ansys Electronics Desktop's global material library as well as the project's local material library

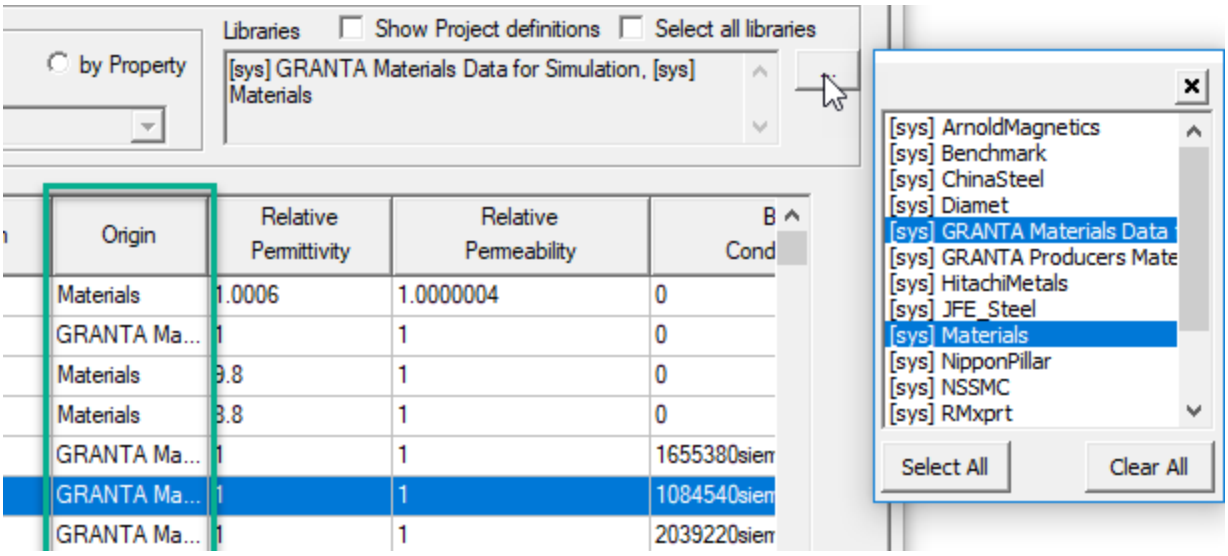


If you click the ellipsis button **[...]**, you see a list of all available libraries. Any selected libraries are highlighted. Shift-Click allows you select a range. Ctrl-Click allows you to select any libraries.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Origin** column shows the originating library for each material, whether the **sys** library, or one of the additional libraries listed in the Libraries pane.



For further information on the materials and their intended uses, you can refer to the published information on materials from those libraries. For example, the Schott materials are described in detail in the Wily Series in Materials for Electronic & Optoelectronic Applications, *Microwave Dielectric Materials and Applications*, edited by M. T. Sebastian, Rick Ubic, and Heli Jantunen, volumes 1 and 2.

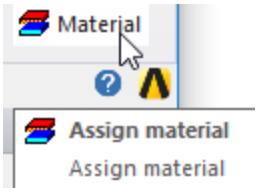
GRANTA Materials Data for Simulation (MDS) is an optional, licensed feature, including more than 700 generic materials and over 2100 producer-specific magnetic materials and PCB materials. When you use or view the GRANTA licensing libraries, the GRANTA

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

license is checked out and held for 30 minutes. Viewing the contents of a GRANTA library in the Materials dialog box or modifying an object to use a GRANTA material causes a license check out and hold. If the license is already checked out, the hold time is reset to 30 minutes.

You can also open the **Select Definition** window in one of the following ways:

- In the **Properties** dialog box for the object, click the material name under the **Attributes** tab. A drop-down menu shows an **Edit...** button that opens the **Select Definition** window.
- With an object selected in the **Modeler** window, on the **Draw** ribbon, select the **Material** icon.

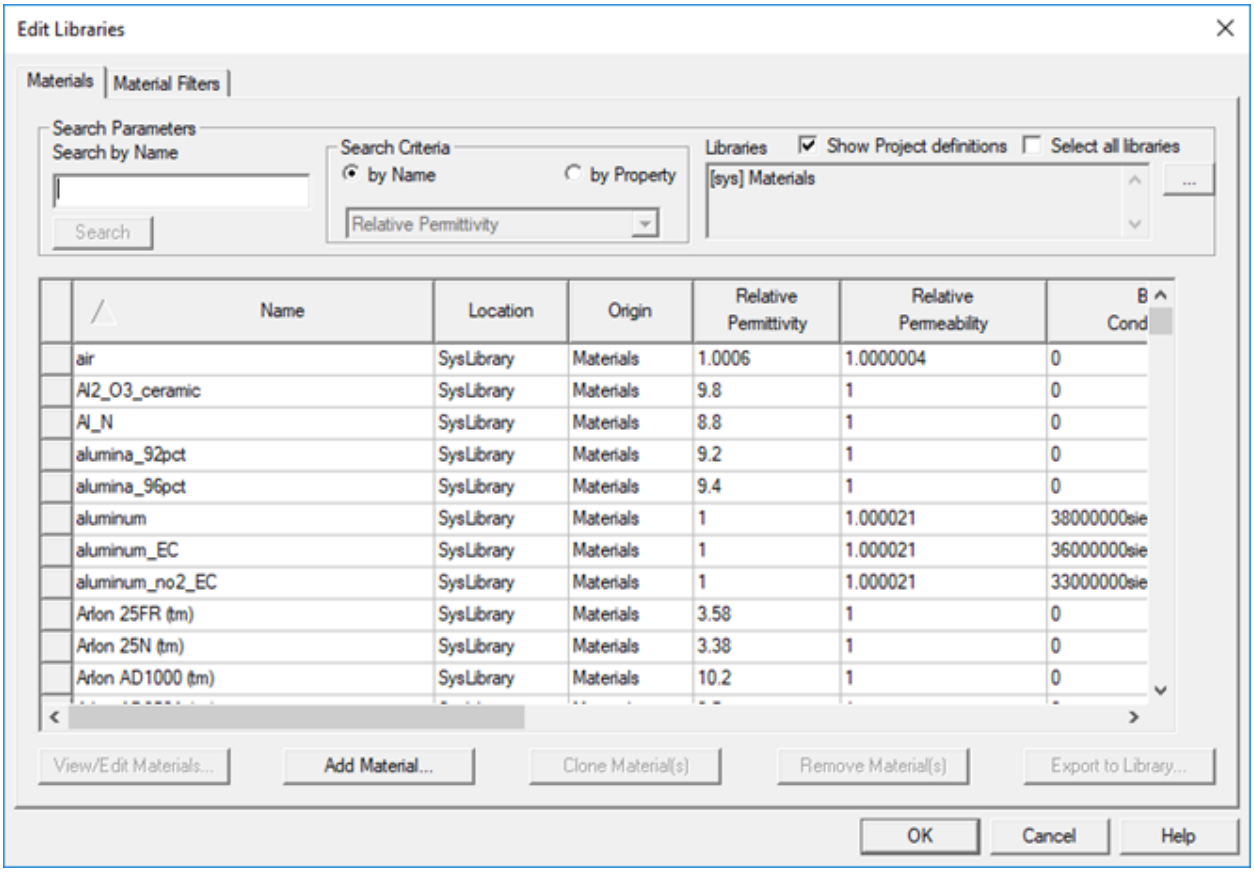


The menu also lists materials included in the current project. Selecting one of these materials provides another way to assign materials to an object.

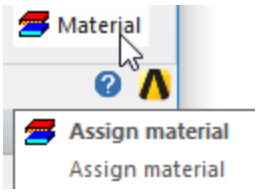
- Right-click **Model** in the project tree, and then click **Assign Material** on the shortcut menu.
- Right-click the object in the history tree, and then click **Assign Material** on the shortcut menu.

Select Definition Dialog Box - Materials

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. With an object selected in the **Modeler** window, on the **Draw** ribbon, select the **Material** icon.



4. Select a material from the list.

> **Note:**
>
> You can search the listed materials by name or property value.

If the material you want to assign is not listed, add a new material to the global or local material library, and then select it.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **OK**.

   The material you chose is assigned to the object.

Note:

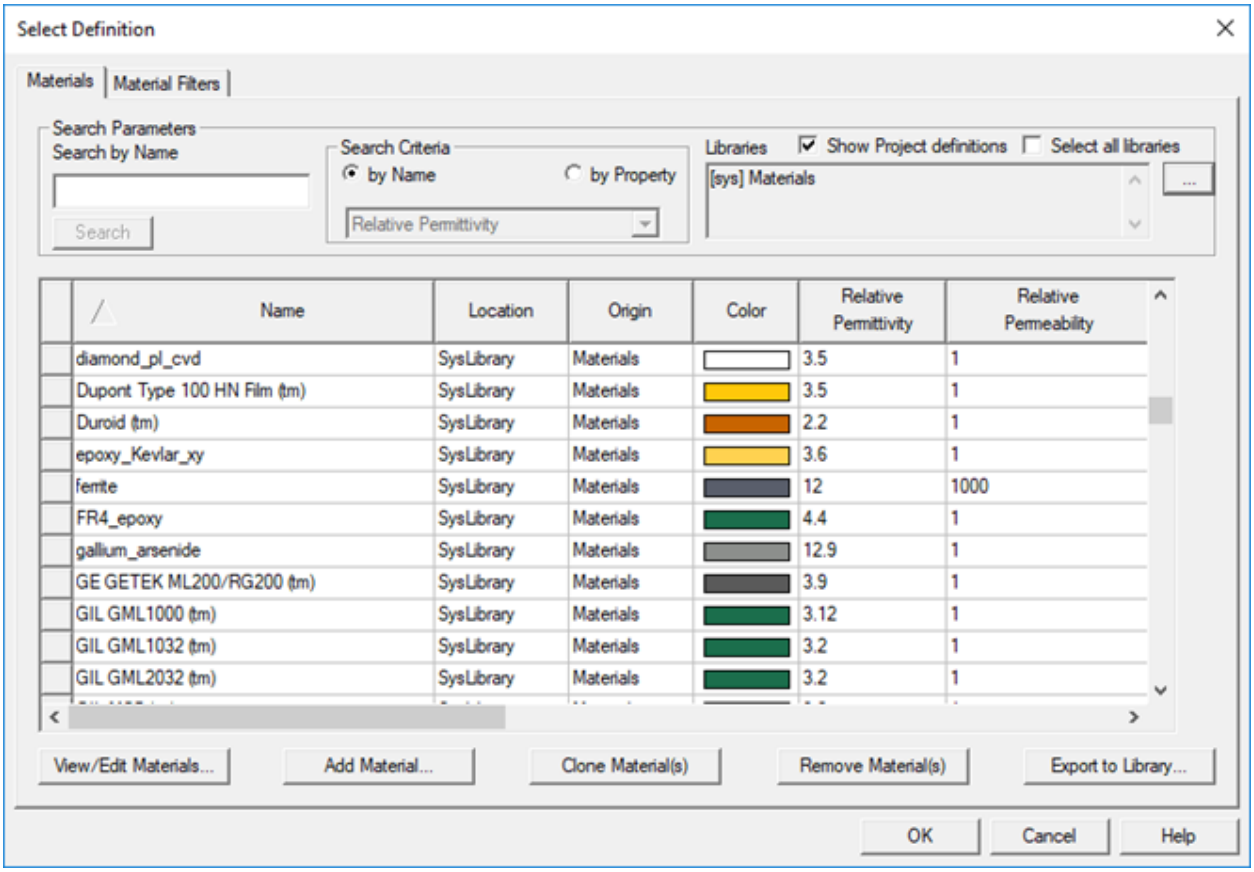
For IE Regions:

- Problems with dielectrics will generally run more slowly, so a warning is issued in this case.
- Only isotropic materials are allowed.
- Frequency-dependent materials are allowed.
- No spatial-dependent materials are allowed.
- You cannot set a background material in the solver. The solver assumes *vacuum* as the background material.

In the History Tree, by default, objects are grouped by material. You can easily change the default grouping. Using the menu bar, click **Modeler** > **History Tree Layout** and clear **Group Objects by Material** along with any other listed option you may wish to disable. You can also perform this action from the **Tree View** drop-down menu on the **View** ribbon tab or from the shortcut menu that appears when you right-click most of the items in the History Tree.

If you check **Show Material Colors** on the Material Filters tab of the **Select Definition** or **Edit Library** dialog box, the **Materials** tab will include a **Color** column showing a color swatch for each listed material.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can edit the color and transparency values for materials in the View/Edit Material dialog box.

Solving Inside or on the Surface

When you assign a material to an object, you can specify whether to generate a field solution inside the object or on the surface of the object. If you elect to generate a solution inside the object, a mesh is created inside the object and generate a solution from the mesh. If you elect to generate a solution on the surface of the object, only a surface mesh is created for the object.

If you want a solution to be generated inside an object, select **Solve Inside** in the **Properties** window. Conversely, if you want a solution to only be generated on the surface of an object, clear the **Solve Inside** option in the **Properties** window.

By default, **Solve Inside** is selected for all objects with a bulk conductivity less than 10^5 siemens/meter and for perfect insulators. By default, the **Solve Inside** option in the **Properties** window is clear for perfect conductors.

To change the threshold for solving inside objects, do the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Tools**>**Options**> **General Options**.
2. In the left pane, select **HFSS** > **Materials**.
3. Enter a new value in the **Solve Inside threshold** text box.

A finite conductivity boundary condition is placed on the surfaces of an object that should not be solved inside based on the material properties of that object. You can also incorporate a DC Thickness for the implicit boundary condition by setting an appropriate thickness value as described in Assigning DC Thickness.

Assigning DC Thickness

The **Assign DC Thickness** option allows you to more accurately compute the DC resistance of a thin conducting object for which Solve Inside is not selected. Skin impedance of the object will be calculated using the defined finite thickness, which can be Automatic (the default), Manually assigned per object, or Infinite per object. This option also exists for finite conductivity boundaries.

The **Assign DC Thickness** option on the **Boundaries** menu is enabled if at least one object contains a good conducting isotropic material (such as copper), and the **Solve Inside property** is not selected. If the object meets these conditions, you can assign a DC thickness, either by enabling the automatic default, or by specifying a value for a selected object.

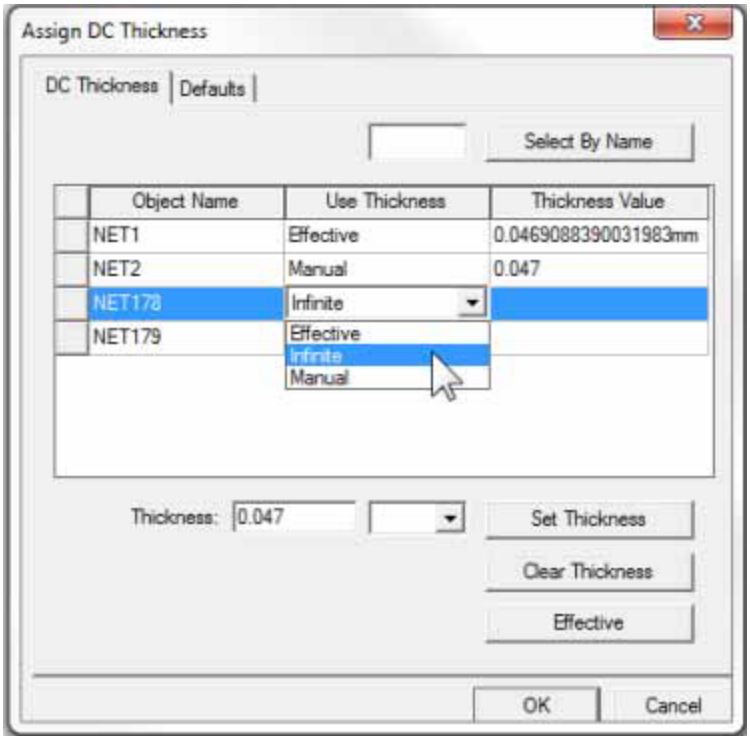
To see the **Assign DC Thickness** dialog box:

1. Select **HFSS** > **Boundaries** > **Assign DC Thickness**.

   This displays the **Thickness of Objects for DC Resistance** dialog box with the DC Thickness tab selected. Objects to which the thickness can be applied are listed in the **Object**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name column.



2. Select the objects to assign a value. You can select objects either by:

- Clicking on the **Object Name** to highlight it.
- Use the **Select By Name** field to type the object name, and click **Search**.

  The first object to match the name is highlighted.

Selecting an object highlights the **Thickness** field and the **Set Thickness** button.

3. In the Use Thickness column, you can specify that the value the object uses is Automatic, Infinite, or Manual.

You can disable automatic assignment on the **Defaults** tab of the **Assign DC Thickness** dialog box. The Automatic value is calculated as Thickness ~ 2* Volume / Surface Area

It should be noted that this is a calculation for an "effective" DC thickness to be used by the correction calculation. For a "thin" object this will work well. For example, a rectangular microstrip trace described by a box with dimensions 100 by 10 by 1 the volume is 1000 and the surface area is 2220 resulting in an apparent thickness of .9009, close to the geometric thickness of "1". For arbitrary shapes of "thicker" objects, this calculation will not work as well. For example, a cube with sides of 1 will have volume of 1 and surface area of 6, and a resulting apparent thickness of .3333. Another example is the case of a cylindrical wire (e.g., bond wire).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this case, the automatic effective DC thickness will be about R_o, which gives us the best approximation of the DC resistance of a cylindrical wire.

The intention is that the auto-thickness will provide an accurate representation the majority of the time and is superior to not using any DC thickness setting. When the automatic value is not appropriate, you can override it using the manual technique.

DC thickness impedance is an approximation. It is accurate just for TE/TM waves when the widths are infinite which clearly never occurs in a "real" design. The fact that the object is finite causes an increase of the effective impedance due to current crowding/edge effects. Thus entering the exact geometrical thickness would actually underestimates the impedance. By returning a DC thickness smaller than the geometric thickness the automatic DC thickness compensates for this underestimation resulting in a slightly higher impedance as desired.

4. To manually apply a value, enter a **Thickness** value, select the units and click the **Set Thickness** button.

This applies the value to the selected object and changes the **Use Thickness** selection for that object to **Manual**.

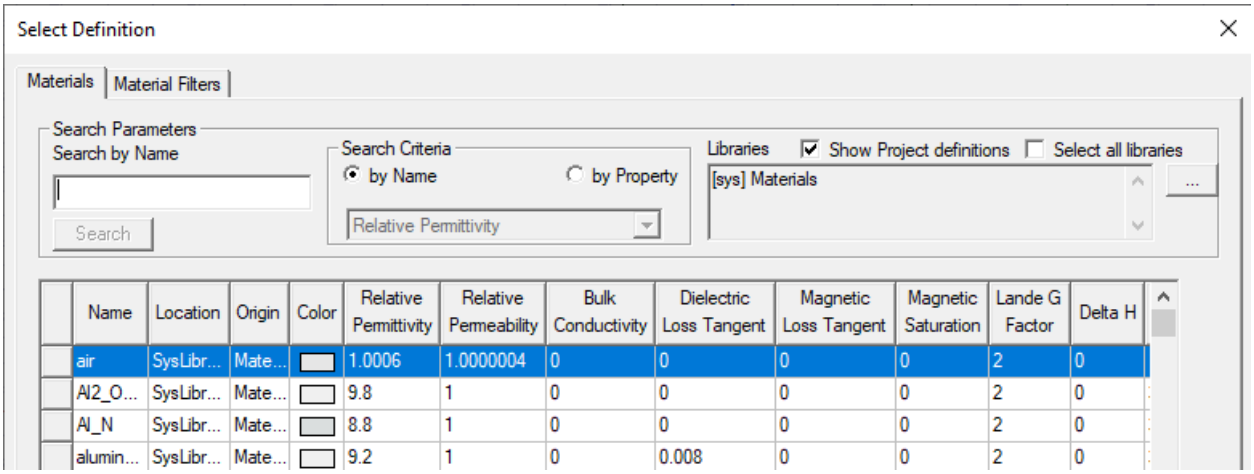
> **Note:**
>
> If you enter a “0” for the thickness, HFSS gives a warning that this will cause infinite impedance that causes isolation.

5. To change the value, select the **Clear Thickness** button and then enter a different value. You can also manually select or deselect the box and manually enter or delete a thickness value in the table.
6. When you have assigned the values you need, click **OK** to close the dialog box.

Searching for Materials

You can search for materials in the **Select Definition** dialog box. The default Search Criteria is **by Name**, which is shown in the following example. Alternatively, you can choose to search by Property.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

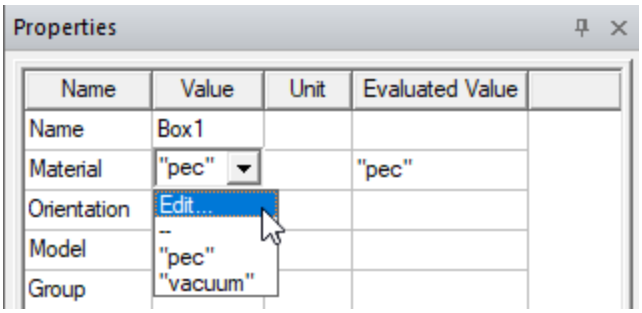


To search for a material **by Name**:

1. Access the **Select Definition** dialog box using one of the following methods:

   With one or more objects selected (that is, to assign a material to selected objects):

   - Click **Modeler** > **Assign Material**
   - Click the **Material** *value* in the docked **Properties** window, and select **Edit...** from the drop-down menu.

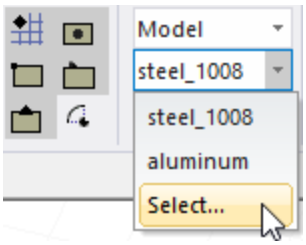


   - On the **Draw** ribbon tab, click **Assign material**.

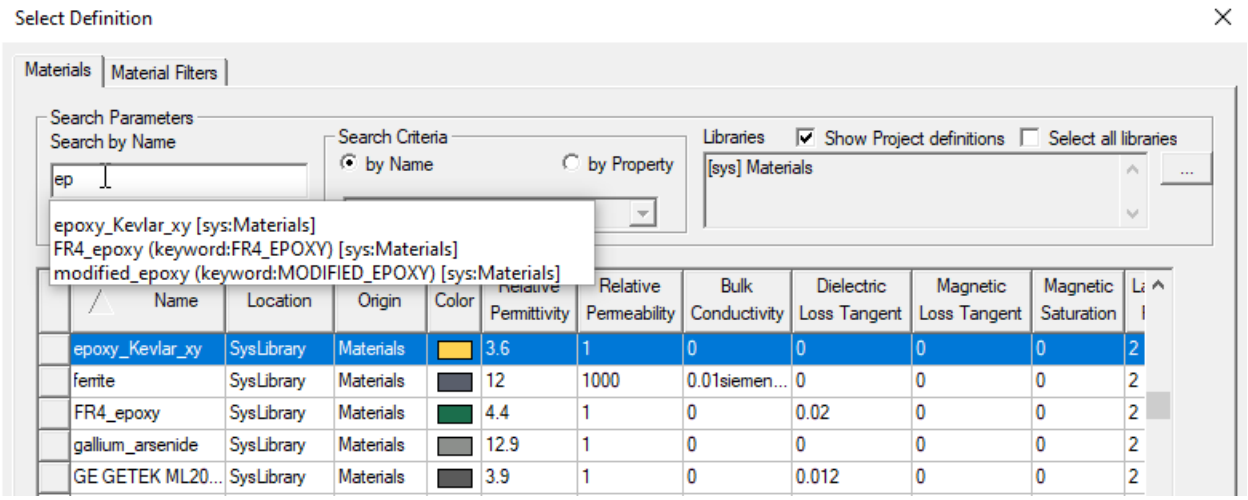
   With nothing selected (that is, to set the default material):

   - On the **Draw** ribbon tab, choose **Select** from the **Default material** drop-down menu:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. In the **Search Criteria** section, ensure that **by Name** is selected, and in Libraries, specify the Libraries that you want to search. Only loaded libraries participate in text and keyword matching.
3. In the **Search by Name** text box, type a portion of the desired material name. The search text is case-insensitive.



A drop down under the Search Parameters field will show the a list of the materials from selected libraries matching any part of the current text, and, if the Granta Materials library is selected, matching keywords in the material definitions. The first row containing the material name most similar to the characters you typed will be selected.

If the selected material is not the one you are searching for, do one of the following:

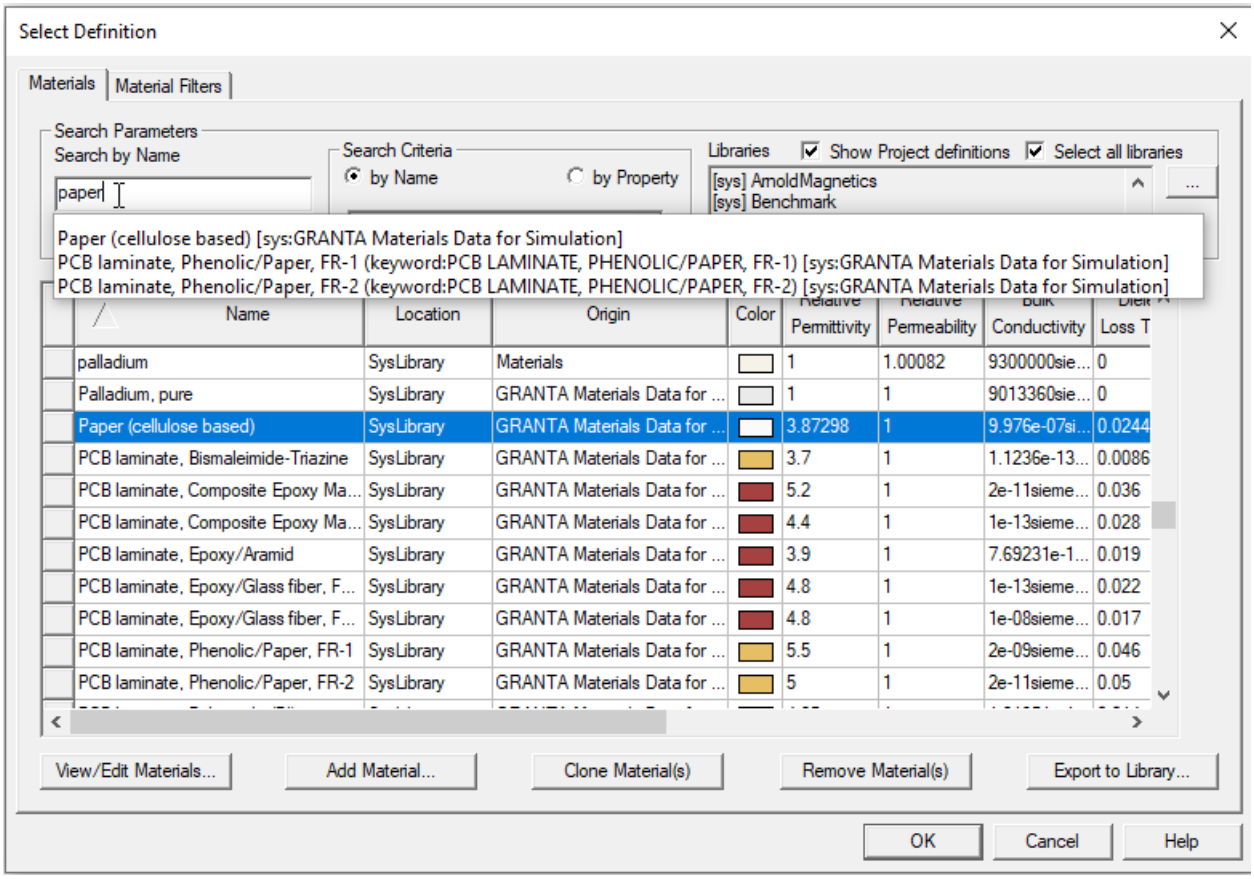
- Use the keyboard's arrow keys to select the material above or below the currently selected row.
- Use the scroll bar to scroll the listed materials upward or downward and click the desired material when it is visible.
- Type different characters in the **Search by Name** text box.

.

When the **Select all libraries** option is selected, the window lists all of the materials in Ansys Electronics Desktop's global material libraries that are applicable to the current

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

design type as well as those in the project's local materials library. The Granta Materials Data for Simulation library now includes keywords that are used in the auto-complete matching. For example, entering "glass" will generate an auto-complete list of all materials with a name containing "glass" or with a keyword that contains "glass", all case-insensitive, for materials showing in the grid. The drop down of potential matches under the search field applies to all libraries. What shows in the grid is defined by which libraries are selected, what material filters are selected, project/design-specific validation, and whether project materials are selected for display.



Searching by Material Property

1. Access the **Select Definition** dialog box using one of the following methods:

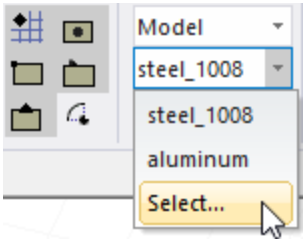
   With one or more objects selected (that is, to assign a material to selected objects):

   - Click **Modeler** > **Assign Material**
   - Click the **Material** *Value* in the docked *Properties* window, and select **Edit...** from the drop-down menu.
   - On the **Draw** ribbon tab, click **Assign material**.

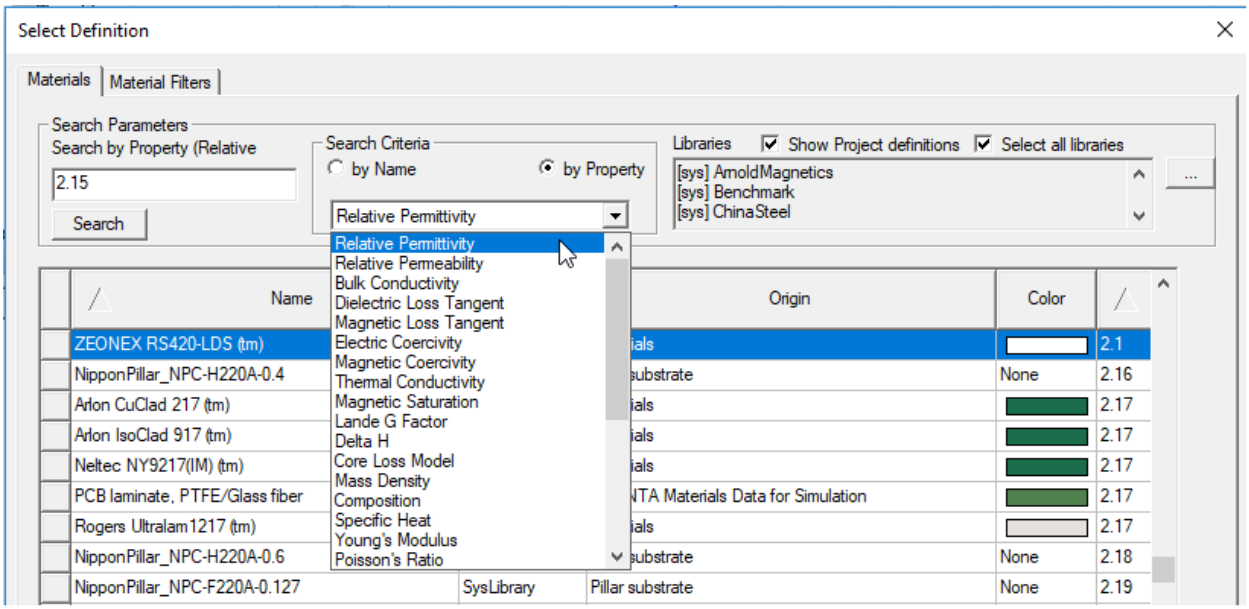
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

With nothing selected (that is, to set the default material):

- On the **Draw** ribbon tab, choose **Select** from the **Default material** drop-down menu:



2. In the **Search Criteria** section, select **by Property**.
3. Select a material property from the pull-down list:



> **Note:**
>
> By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed in the table, though all properties are available in this drop-down menu. To view the complete table of properties, see Filtering Materials.

4. In the **Search Parameters** area, type a numerical value in the **Search by Property** text box and then click **Search**.

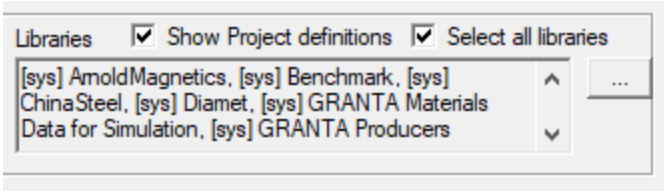
   The materials are sorted according to the property you selected. Additionally, the material with the property value closest to the one you typed, but without exceeding it, is selected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the selected material is not the one you are searching for, do one of the following:

- Use the keyboard's arrow keys to select the material above or below the currently selected row.
- Use the scroll bar to scroll the listed materials upward or downward and click the desired material when it is visible.
- Type a different numerical in the **Search by Property** text box and click **Search** again.

When the **Select all libraries** option is selected, the window lists all of the materials in Ansys Electronics Desktop's global material libraries that are applicable to the current design type as well as those in the project's local materials library.



Adding New Materials

You can add a new material to a project or global user-defined material library. To make the new project material available to all projects, you must export the material to a global user-defined material library.

Materials are added using the **View/Edit Material** dialog box, which can be opened from either the **Select Definition** window or the **Edit Libraries** window.

To open the **Select Definition** window:

- Click **Modeler** > **Assign Material**.

  The **Select Definition** window appears.

To open the **Edit Libraries** window:

- Click **Tools** > **Edit Libraries** > **Materials**. In the project tree, you can also right-click **Materials**, and select **Edit Library**.

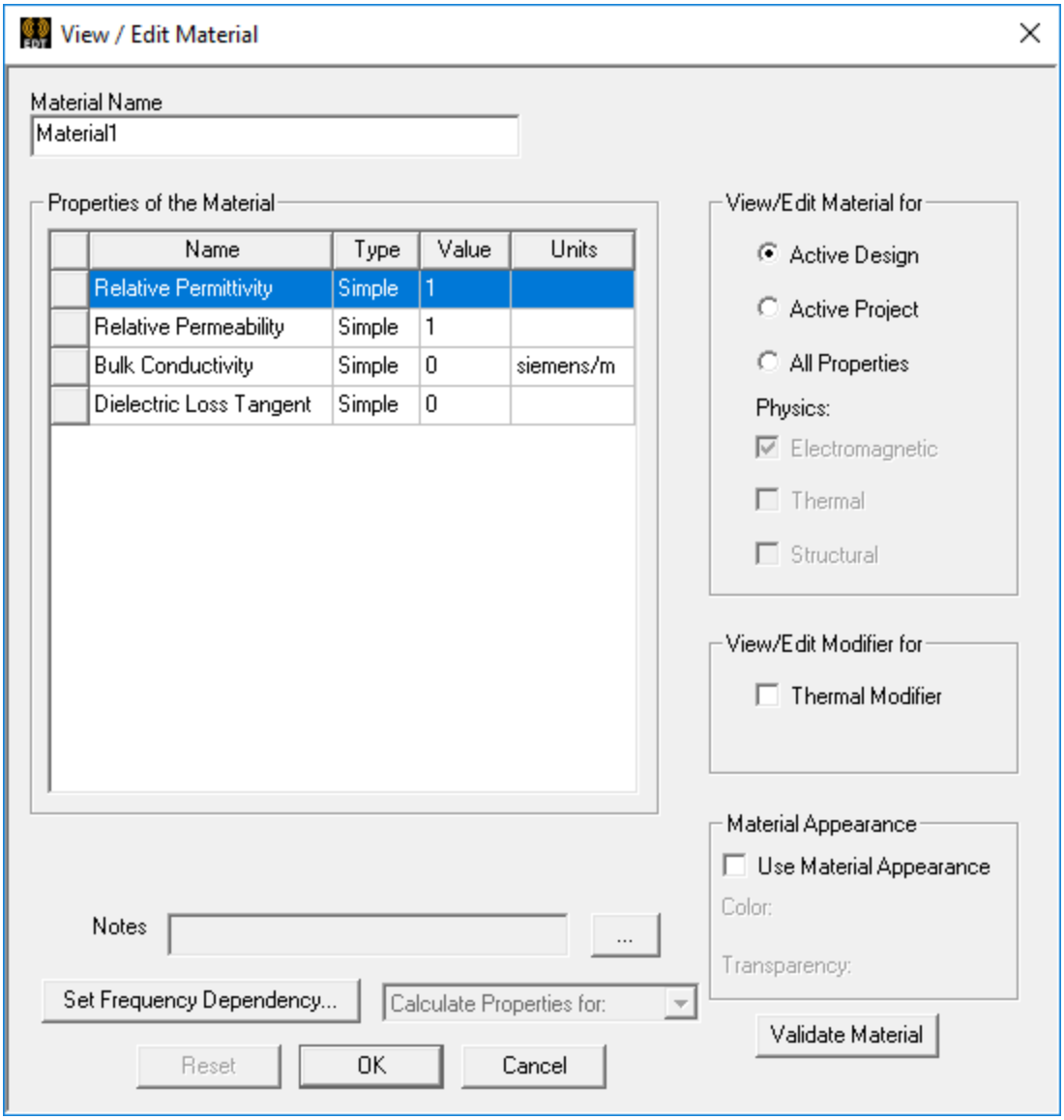
  The **Edit Libraries** window appears.

To add a new material:

1. From either the **Select Definition** window or the **Edit Libraries** window, click **Add Material**.

   The **View/Edit Material** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



By default, only properties commonly used by the selected product are displayed. To view the complete table of properties, see Filtering Materials.

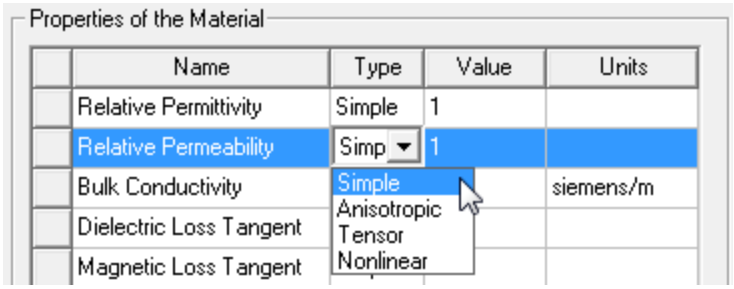
2. Type a name for the material in the **Material Name** text box, or accept the default.
3. Use the radio buttons in the **View/Edit Material for** section to specify whether the new materials apply to Active Design, Active Project, or All Properties. When **All Properties** is selected, **Physics** classification options are enabled to show or hide properties based on simulation type (Electromagnetic, Thermal, or Structural).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> If a material is edited in a design type for which the **Physics** type has not been set (e.g., an HFSS design but Electromagnetic physics type was not set), the **Physics** type will be automatically set in the material.

You can also enable the View/ Edit Modifier check box for Thermal Modifier. Checking this box causes the Thermal Column to display at the right side of the Properties of the Material table. Selecting **Edit** rather than None causes display of the Edit Thermal Modifier dialog.

4. Enter a **Material Name**, or accept the default.
5. For each available material property, select a **Type** from the drop-down menu. Only applicable types appear for each material property. Some properties only use the Simple type, while others offer four or more types.



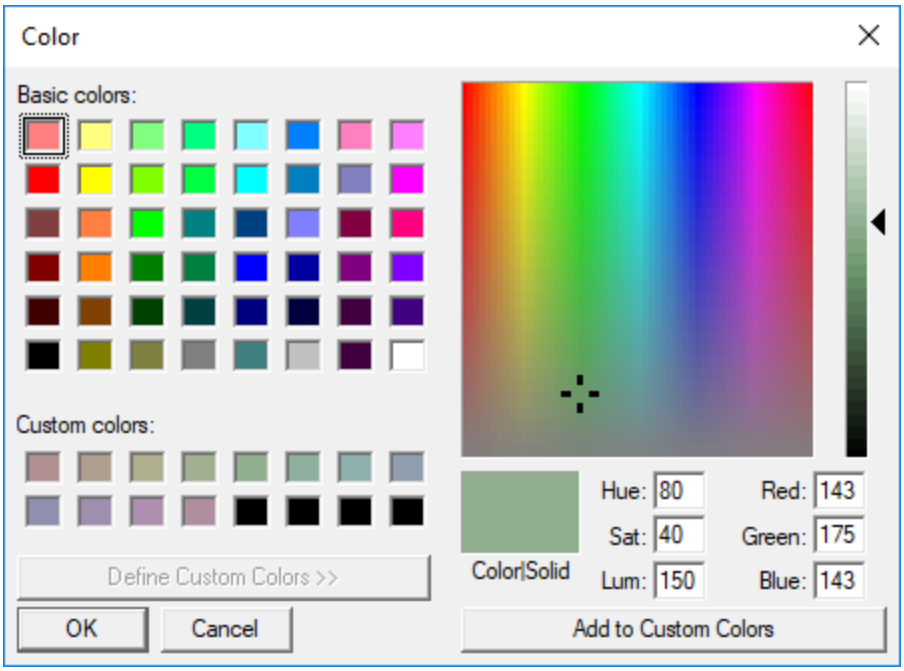
6. If the material is *linear*, enter a **Value** for the following material properties:
   - Relative Permeability
   - Relative Permittivity
   - Bulk Conductivity
   - Dielectric Loss Tangent
   - Magnetic Loss Tangent

If the material is a *ferrite*, enter a value greater than 0 in the Magnetic Saturation **Value** box. You may also choose to enter values in the Lande G Factor and Delta H **Value** boxes. Because Delta H values are measured at specific frequencies, you should also enter a - Measured Frequency value (default 9.4 GHz).

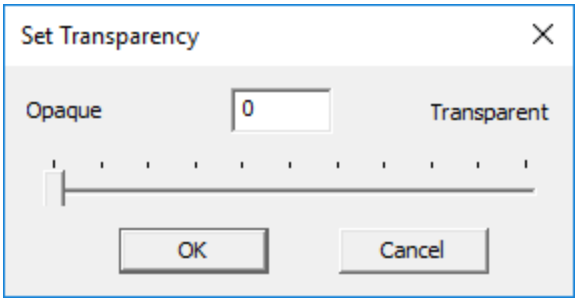
> **Note:**
>
> You may enter a variable name or mathematical expression in the **Value** box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. If one or more of the material properties are dependent on frequency, click **Set Frequency Dependency**, and then follow the directions for defining frequency dependent materials.
8. To modify the units for a material property, double-click the **Units** box and select a new unit system.
9. For Material Appearance, you can check the box to enable the fields for you to specify a color and transparency. Clicking the color bar opens a color selection window:



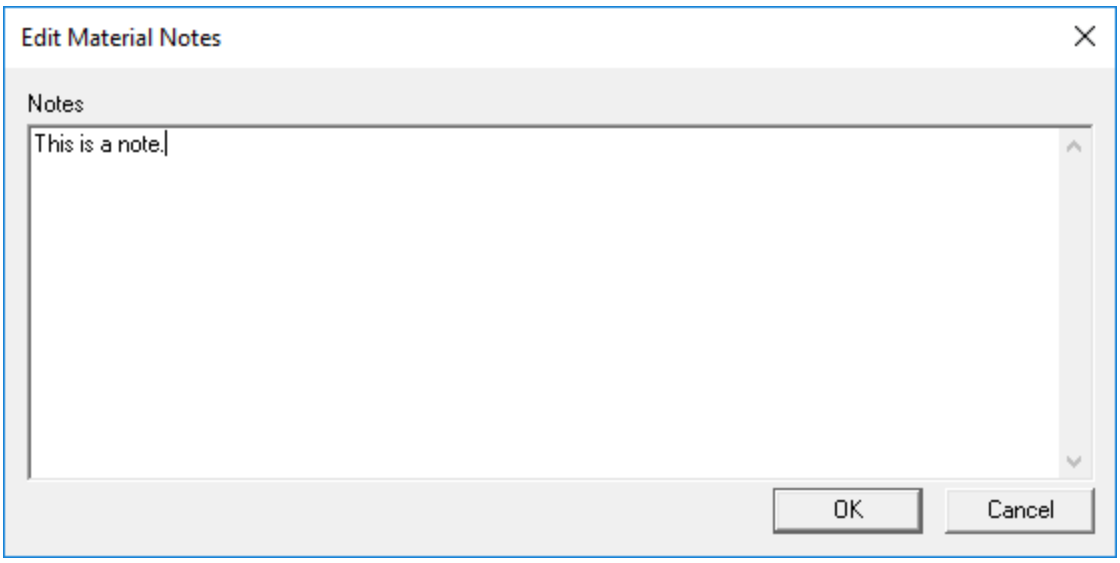
10. Clicking the Transparency box opens a Transparency dialog box with a text field and slider bar for selection.



11. Click **OK**.
12. To modify the units for a material property, double-click the **Units** box and select a new unit system.
13. Click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14. If you want to add descriptive notes for the new material, click the ellipsis button [...] next to the **Notes** field. This opens a dialog box in which you can enter text.



15. Click **OK** to add the Notes.
16. Click **OK** on the **View/Edit Materials** dialog to add the new material to the material library.

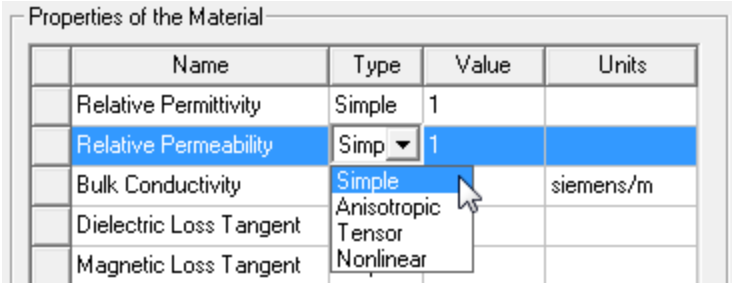
Assigning Material Property Types

Material properties can be assigned using the **View/Edit Material** dialog box, using one of the following material property types, based on the applicability to the material property:

Simple	The material is homogeneous and linear.
Anisotropic	The material's characteristics vary with direction. This models only the diagonal.
Tensor	A full 3x3 matrix general tensor, which may or may not be symmetrical.
Nonlinear	A BH Curve.

Select a material property type for each property from the **Type** drop-down menu. Of the four possibilities, only those applicable to the named material will be listed on the pull-down for type. Some properties only use the Simple type. Others include three or four potential types.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the material property is anisotropic, its characteristics are defined by its anisotropy tensor. You must define three diagonals for anisotropic permittivity, electric loss tangent, conductivity, permeability, and magnetic loss tangent. Each diagonal represents a tensor of your model along an axis.

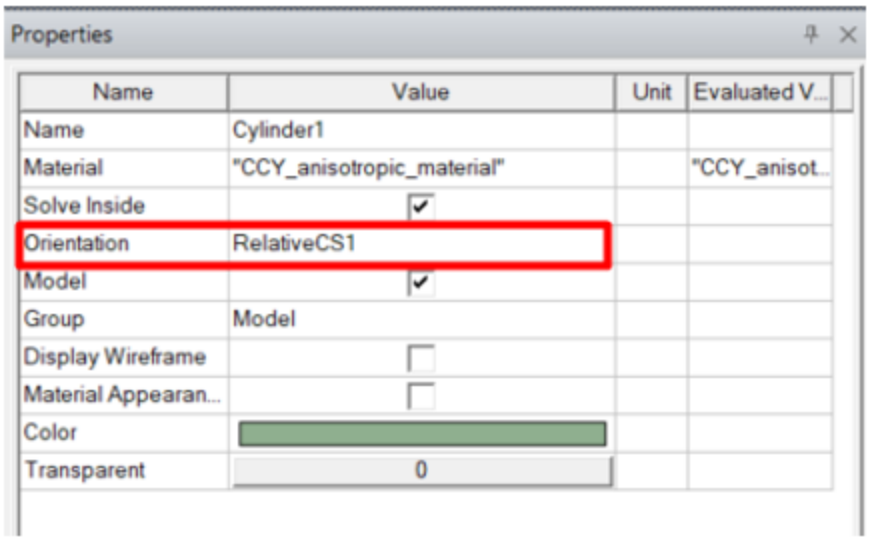
These tensors are relative to the coordinate system specified as the object's Orientation property. By specifying different orientations, several objects can share the same anisotropic material but be oriented differently.

Defining Anisotropic Tensors

If a material property is anisotropic, its characteristics are defined by its anisotropy tensor. Each diagonal represents a tensor of your model along an axis. These tensors are relative to the coordinate system specified as the object's *Orientation* property (when supported). Otherwise, the tensors conform to the global Cartesian coordinate system. By specifying different orientations, several objects can share the same anisotropic material but be oriented differently. Spherical and Cylindrical tensors must be converted to Cartesian coordinates, as described below. In such cases you must specify the coordinate system carefully.

> **Note:** Spatial-dependent material properties are calculated according to coordinates in the 'Orientation CS'.This CS should be set carefully.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Important:

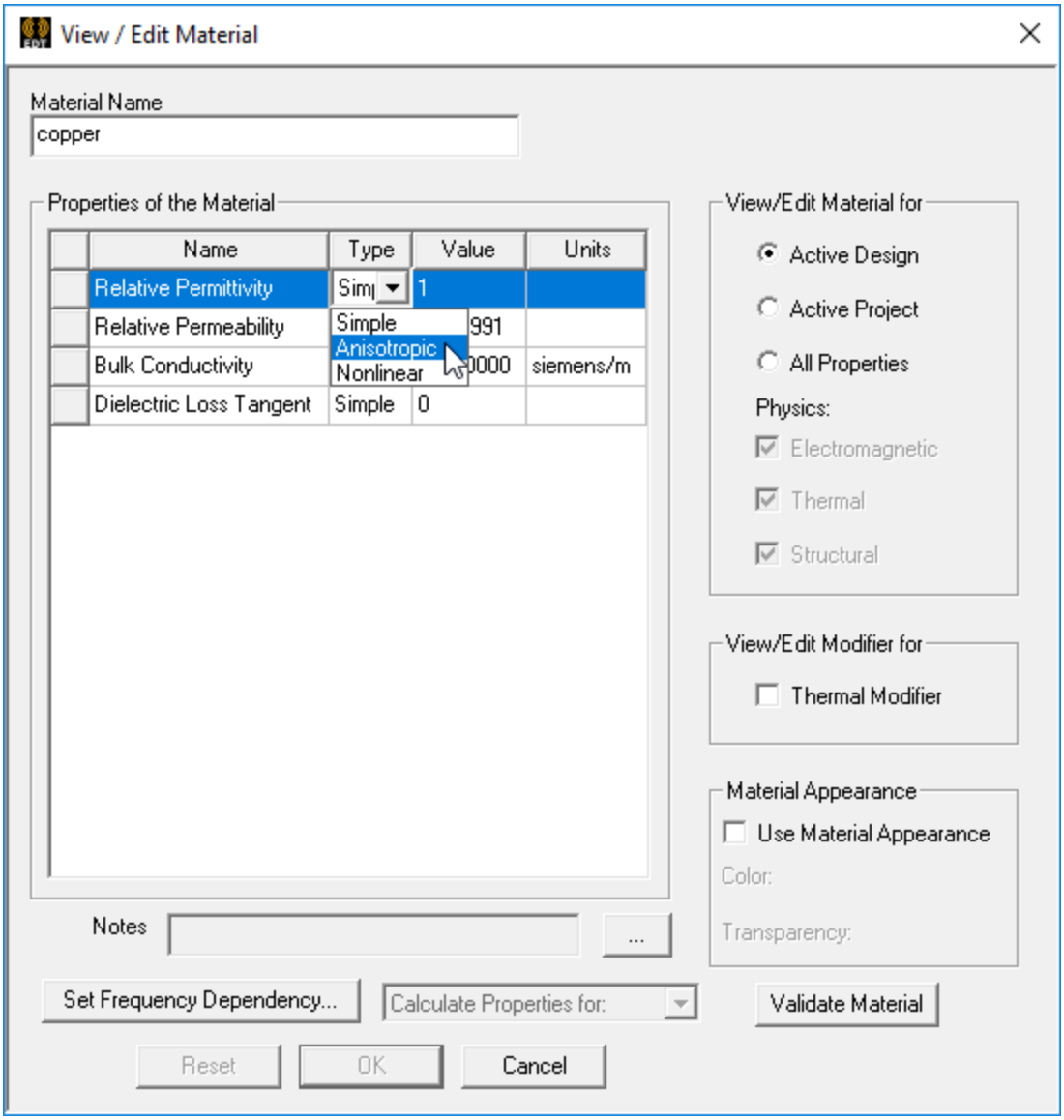
Anisotropic materials are supported in 2D models but are not available for Q3D.

Assigning Anisotropic Tensors

To assign anisotropic tensors to permittivity, electric loss tangent, conductivity, permeability, and magnetic loss tangent:

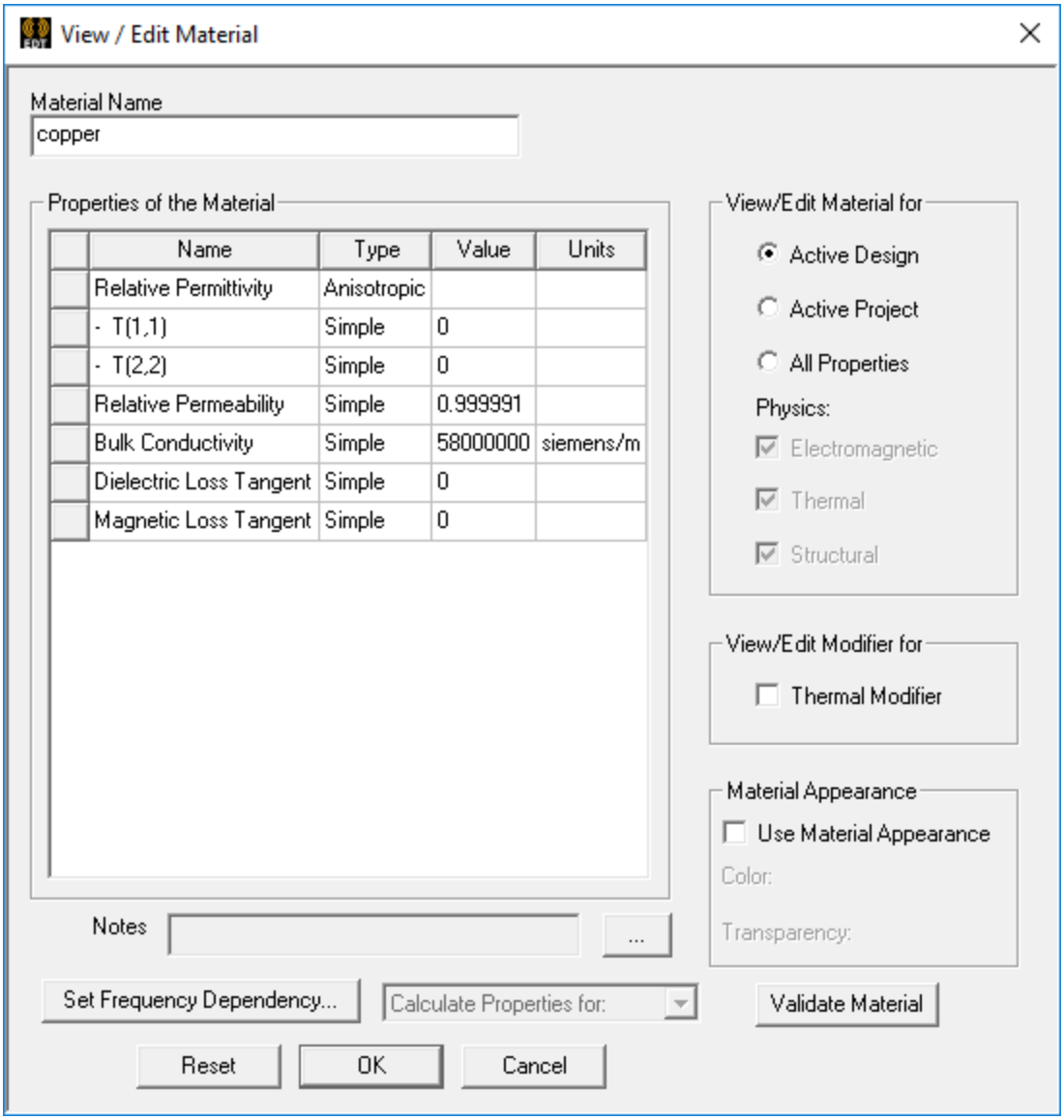
1. From the **View/Edit Material** window, use the **Type** drop-down menu to select **Anisotropic**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Two rows, labeled **T(1,1)** and **T(2,2)** appear, as shown below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. For each of the new anisotropic property rows, use the **Type** drop-down menu to select either **Simple** or **Nonlinear** (when supported). This setting determines the type of **Value** you can enter.
3. In the **Value** field, enter the permittivity, electric loss tangent, conductivity, permeability, and magnetic loss tangent. along each axis of the material's tensor.
    - For **Relative Permeability** – enter the relative permeability along each axis of the material's permeability tensor. This can be a simple value, a variable, a constant, or a Nonlinear BH Curve.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For **Relative Permittivity** – enter the material's relative permittivity along each tensor axis. This can be a simple value or a variable. If the relative permittivity is the same in all directions, use the same Simple values for each axis.
- For **Conductivity** – enter the material's conductivity along each tensor axis. This can be a simple value or a variable.
- For **Dielectric Loss Tangent** – enter the ratio of the imaginary relative permittivity to the real relative permittivity in one direction. This can be a simple value or a variable.
- For **Magnetic Loss Tangent** – enter the ratio of the imaginary relative permeability to the real relative permeability in one direction. This can be a simple value or a variable.

4. Click **OK** to save the values and return to the **Select Definition** window.

Cylindrical Anisotropic Material Properties: Tensor Conversion from Cylindrical to Cartesian CS

HFSS accepts definitions for spatial-dependent martial properties only using definitions in a Cartesian coordinate system. Cylindrical and Cartesian bases are related as

$$\begin{bmatrix}\hat{e}_x\\ \hat{e}_y\\ \hat{e}_z\end{bmatrix} = \begin{bmatrix}cos\varphi & -sin\varphi & 0\\ sin\varphi & cos\varphi & 0\\ 0 & 0 & 1\end{bmatrix}\begin{bmatrix}\hat{e}_\rho\\ \hat{e}_\varphi\\ \hat{e}_z\end{bmatrix} = \boldsymbol{M}_{Cyl\to Cart}\begin{bmatrix}\hat{e}_\rho\\ \hat{e}_\varphi\\ \hat{e}_z\end{bmatrix}$$

where M is a transformation matrix.

Tensors, describing material properties, could be transformed accordingly as

$$\bar{\bar{A}}_{xyz} = \boldsymbol{M}_{Cyl\to Cart}\bar{\bar{A}}_{\rho\varphi z}\boldsymbol{M}^T_{Cyl\to Cart} = \begin{bmatrix}cos\varphi & -sin\varphi & 0\\ sin\varphi & cos\varphi & 0\\ 0 & 0 & 1\end{bmatrix}\bar{\bar{A}}_{\rho\varphi z}\begin{bmatrix}cos\varphi & sin\varphi & 0\\ -sin\varphi & cos\varphi & 0\\ 0 & 0 & 1\end{bmatrix}$$

If we assume only the diagonal components as non-zeros for the tensor in the Cylindrical basis

$$\bar{\bar{A}}_{\rho\varphi z} = \begin{bmatrix}A_{\rho\rho} & 0 & 0\\ 0 & A_{\varphi\varphi} & 0\\ 0 & 0 & A_{zz}\end{bmatrix}$$

the resulting tensor in the Cartesian basis could be derived as

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\bar{\bar{A}}_{xyz} = \begin{bmatrix} A_{\rho\rho}cos^2\varphi + A_{\varphi\varphi}sin^2\varphi & (A_{\rho\rho} - A_{\varphi\varphi})sin\varphi cos\varphi & 0 \\ (A_{\rho\rho} - A_{\varphi\varphi})sin\varphi cos\varphi & A_{\rho\rho}sin^2\varphi + A_{\varphi\varphi}cos^2\varphi & 0 \\ 0 & 0 & A_{zz} \end{bmatrix}$$

We should define and using X, Y and Z:

$$\rho = \sqrt{X^2 + Y^2}$$

$$\varphi = \begin{cases} \text{atan}\left(\frac{Y}{X}\right) + \begin{cases} 180° \text{ sign}(Y) & if \quad X < 0 \\ 0 \quad if \; X \geq 0 \end{cases} & if \quad \rho > 0 \\ \\ 0 \quad if \quad \rho = 0 \end{cases} \qquad -180° \leq \varphi \leq 180°$$

Related HFSS Project variables:

$rho	sqrt(X^2+Y^2)
$phi	if($rho>0, atan(Y/X)+180deg*if(X<0,sgn(Y),0),0)

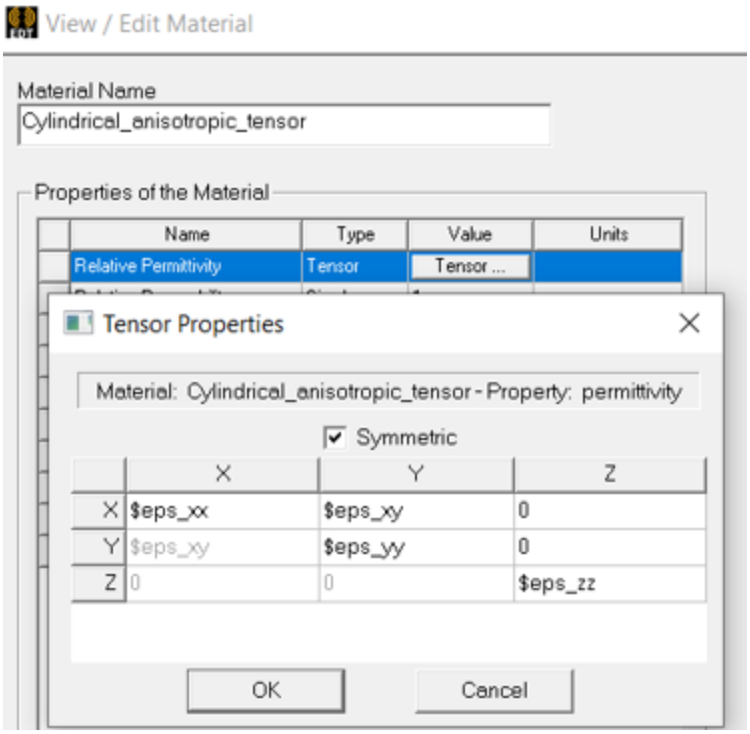
Note: The material characteristics close to Z-axis could be slightly incorrect because of uncertainty of in the case of =0.

First you create all required Project variables.

Name	Value	Unit	Evaluated V...
$eps_rr	4		4
$eps_pp	8		8
$eps_zz	12		12
$rho	sqrt(X^2+Y^2)		*****
$phi	if($rho>0, atan(Y/X)+180deg*if(X<0,sgn(Y),0),0)		*****
$eps_xx	$eps_rr*cos($phi)^2+$eps_pp*sin($phi)^2		*****
$eps_xy	(eps_rr-eps_pp)*sin($phi)*cos($phi)		*****
$eps_yy	$eps_rr*sin($phi)^2+$eps_pp*cos($phi)^2		*****

You can then use the variables to create the material.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

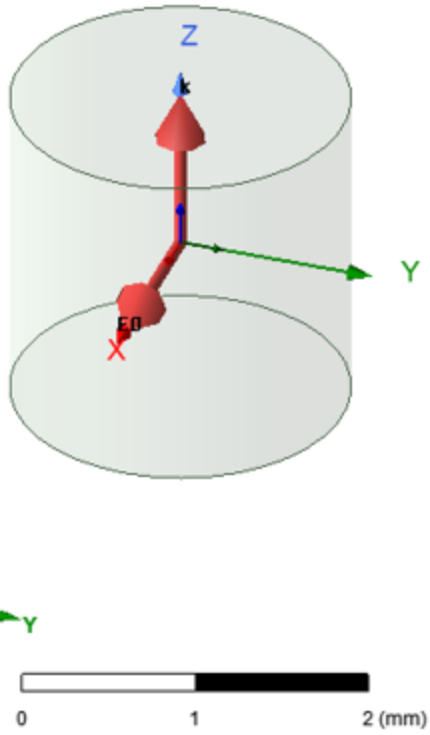


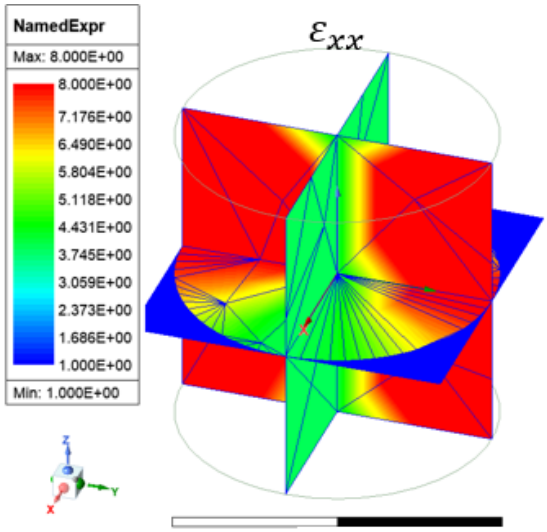
For the test case for cylindrical anisotropic material definition, using converted tensors based on variables:

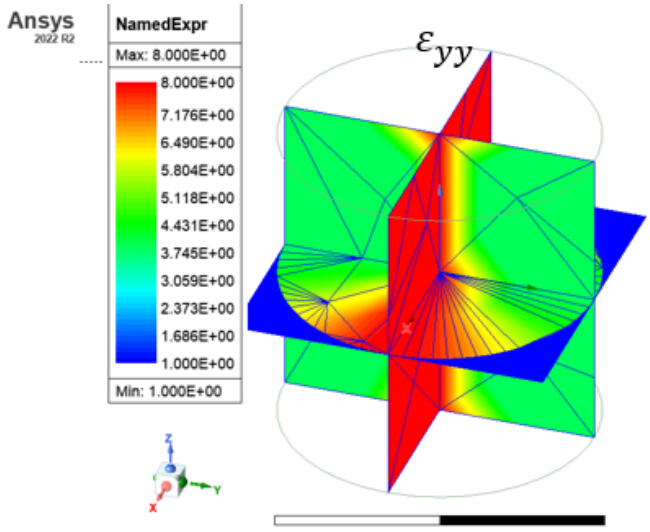
$$\varepsilon_{\rho\rho} = 4 \;\; \varepsilon_{\varphi\varphi} = 8 \;\; \varepsilon_{zz} = 12$$

For this problem type, consider that scattering on a cylinder where diameter and height are equal to 0.2 wavelength and a plane wave excitation along Z-axis.

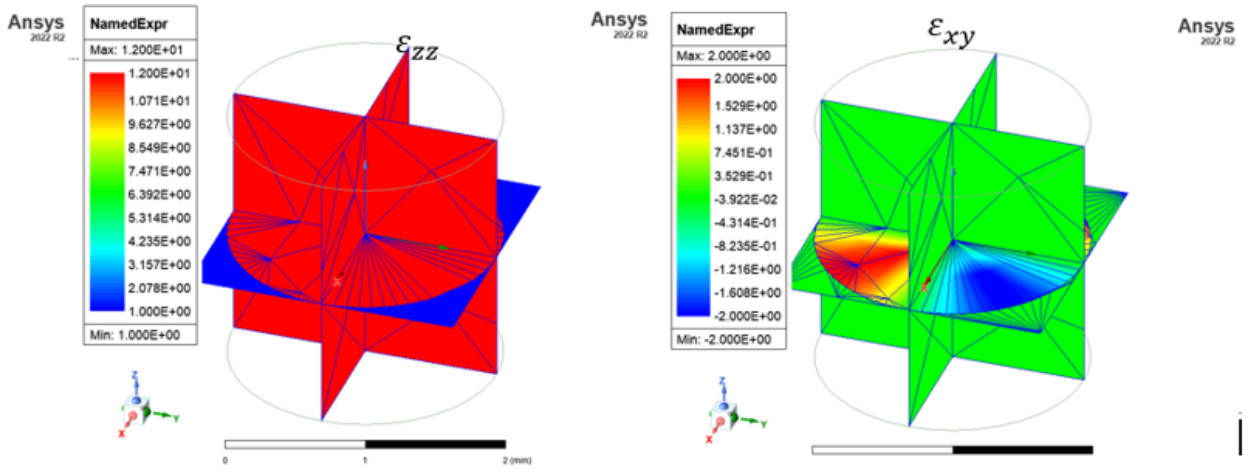
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.







Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Spherical Anisotropic Material Properties: Tensor Conversion from Spherical to Cartesian CS

HFSS accepts definitions for spatial-dependent martial properties only using definitions in a Cartesian coordinate system. Spherical and Cartesian bases are related as

$$\begin{bmatrix}\hat{e}_x\\ \hat{e}_y\\ \hat{e}_z\end{bmatrix} = \begin{bmatrix}sin\theta cos\varphi & cos\theta sin\varphi & -sin\varphi\\ sin\theta sin\varphi & cos\theta cos\varphi & cos\varphi\\ cos\theta & -sin\theta & 1\end{bmatrix}\begin{bmatrix}\hat{e}_\rho\\ \hat{e}_\theta\\ \hat{e}_\varphi\end{bmatrix} = \boldsymbol{M}_{Sph\to Cart}\begin{bmatrix}\hat{e}_\rho\\ \hat{e}_\theta\\ \hat{e}_\varphi\end{bmatrix}$$

where **M** is a transformation matrix

Tensors, describing material properties, could be transformed accordingly as

$$\bar{\bar{A}}_{xyz} = \boldsymbol{M}_{Sph\to Cart}\bar{\bar{A}}_{\rho\theta\varphi}\boldsymbol{M}^T_{Sph\to Cart}$$

We should define ρ,θ , and φ using X, Y and Z

$$\rho = \sqrt{X^2+Y^2+Z^2} \qquad \rho_{XY} = \sqrt{X^2+Y^2}$$

$$\varphi = \begin{cases}\text{atan}\left(\frac{Y}{X}\right) + \begin{cases}180°\,\text{sign}(Y) & if\ \ X<0\\ 0 & if\ X\geq 0\end{cases} & if \quad \rho_{XY}>0\\ \\ 0 & if \quad \rho_{XY}=0\end{cases}$$

$$\theta = \begin{cases}\text{acos}\left(\frac{Z}{\rho}\right) & if \quad \rho>0\\ \\ 0 & if \quad \rho=0\end{cases}$$

$$0 \leq \theta \leq 180°$$
$$-180° \leq \varphi \leq 180°$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Related HFSS Project variables

$rho	sqrt(X^2+Y^2+Z^2)
$rho_xy	sqrt(X^2+Y^2)
$theta	if($rho>0, acos(Z/$rho), 0)
$phi	if($rho_xy>0, atan(Y/X)+180deg*if(X<0,sgn(Y),0),0)

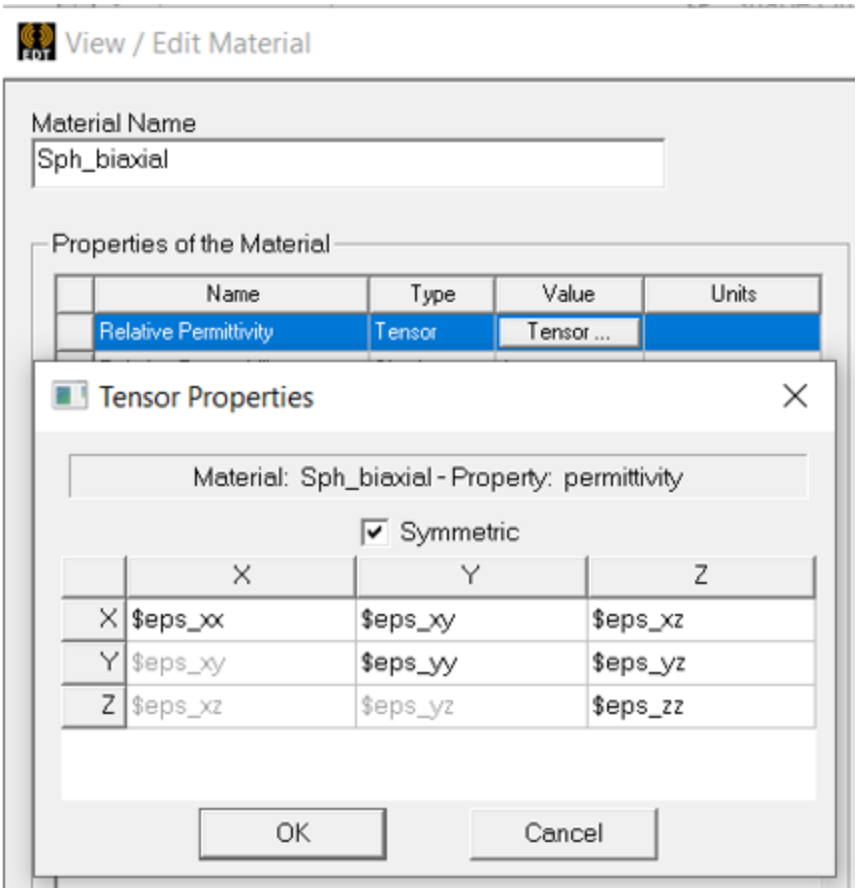
Note: the material characteristics close to the Z-axis could be slightly incorrect because of uncertainty of φ in the case of ρ_{xy}=0.

First you must create the necessary project variables.

Name	Value
$eps_rr	4
$eps_tt	8
$eps_pp	12
$rho	sqrt(X^2+Y^2+Z^2)
$rho_xy	sqrt(X^2+Y^2)
$theta	if($rho>0, acos(Z/$rho), 0)
$phi	if($rho_xy>0, atan(Y/X)+180deg*if(X<0,sgn(Y),0),0)
$eps_xx	($eps_rr*sin($theta)^2+$eps_tt*cos($theta)^2)*cos($phi)^2+$eps_pp*sin($phi)^2
$eps_xy	($eps_rr*sin($theta)^2+$eps_tt*cos($theta)^2-$eps_pp)*sin($phi)*cos($phi)
$eps_xz	(eps_rr-eps_tt)*sin($theta)*cos($theta)*cos($phi)
$eps_yy	($eps_rr*sin($theta)^2+$eps_tt*cos($theta)^2)*sin($phi)^2+$eps_pp*cos($phi)^2
$eps_yz	(eps_rr-eps_tt)*sin($theta)*cos($theta)*sin($phi)
$eps_zz	$eps_rr*cos($theta)^2+$eps_tt*sin($theta)^2

You can then use the variables to create the material.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



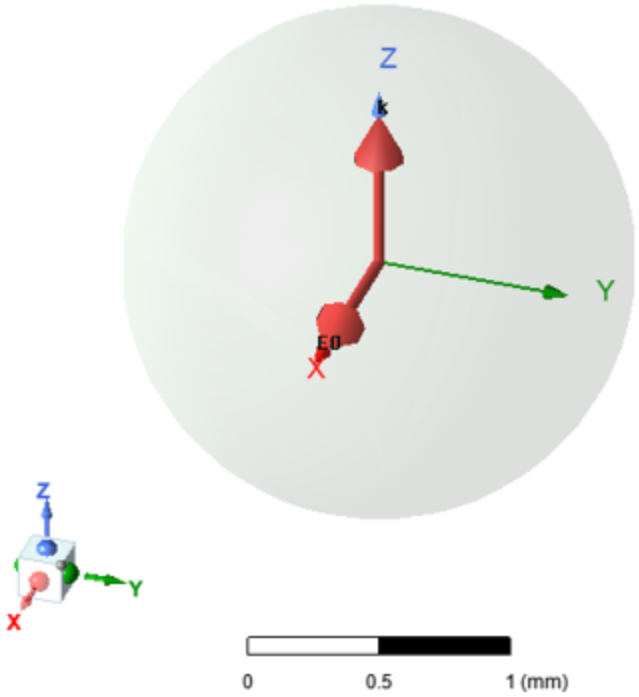
For the test case:

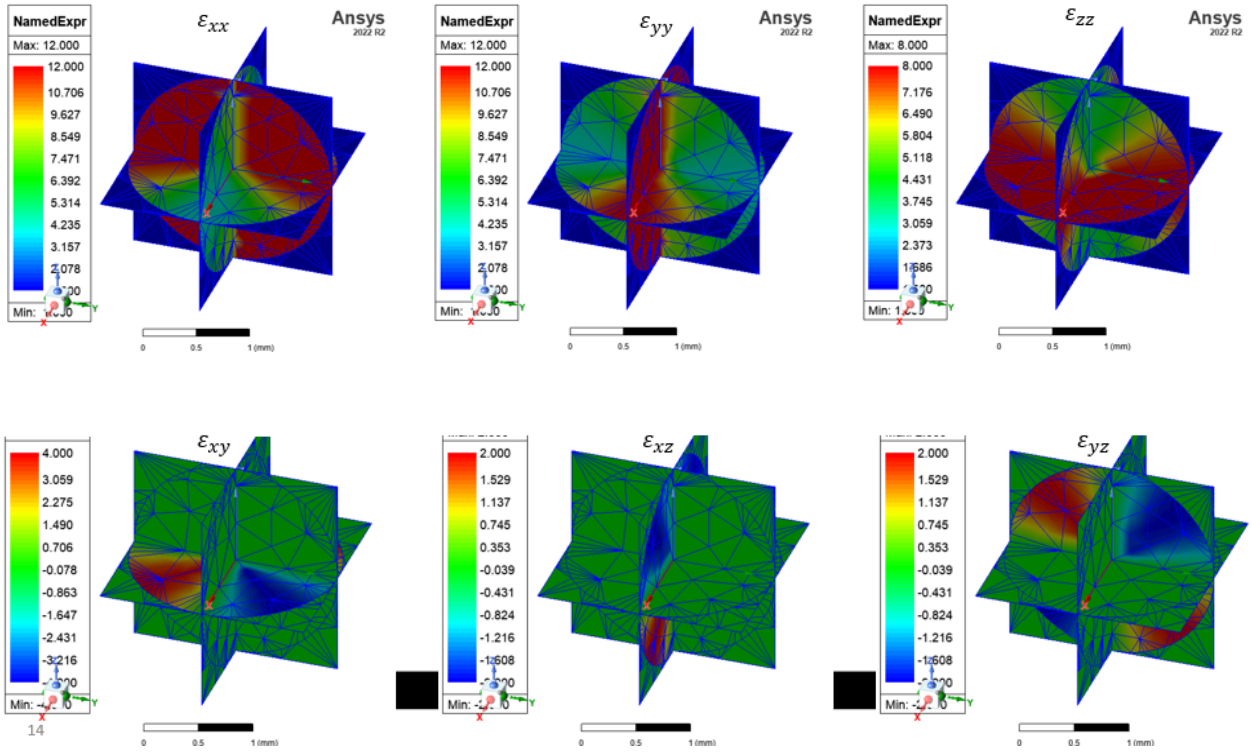
$$\varepsilon_{\rho\rho} = 4 \;\; \varepsilon_{\theta\theta} = 8 \;\; \varepsilon_{\varphi\varphi} = 12$$

Problem type: scattering on a sphere(diameter is equal to 0.2 wavelength).

Plane wave excitation along Z-axis

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defining Variable Material Properties

When defining or modifying a material's properties, each material property value in the **View/Edit Material** dialog box can be assigned a project variable. Simply type the project variable's name in the appropriate **Value** box. Project variables are used for material properties because materials are stored at the project level.

For example, define a project variable with the name **MyPermittivity** and define its value as **4**. To assign this property value to a material, type **$MyPermittivity** in the **Relative Permittivity Value** box for the material. Be sure to include the prefix **$** before the project variable name, which notifies the software that the variable is a project variable.

> **Note:**
>
> By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed. To view the complete table of properties, see Filtering Materials.

Defining Frequency-Dependent Material Properties

HFSS provides several frequency-dependent material models. The Piecewise Linear and Frequency Dependent Data Points models apply to both the electric and magnetic properties of the material. However, they do not guarantee that the material satisfies causality conditions, and so they should only be used for frequency-domain applications.

The Debye and Djordjevic-Sarkar models apply only to the electrical properties of dielectric materials. These models satisfy the Kramers-Kronig conditions for causality, and so are preferred for applications (such as TDR or Full-Wave Spice) where time-domain results are needed. The Design Settings also include an automatic Djordjevic-Sarkar model to ensure causal solutions when solving frequency sweeps for simple constant material properties.

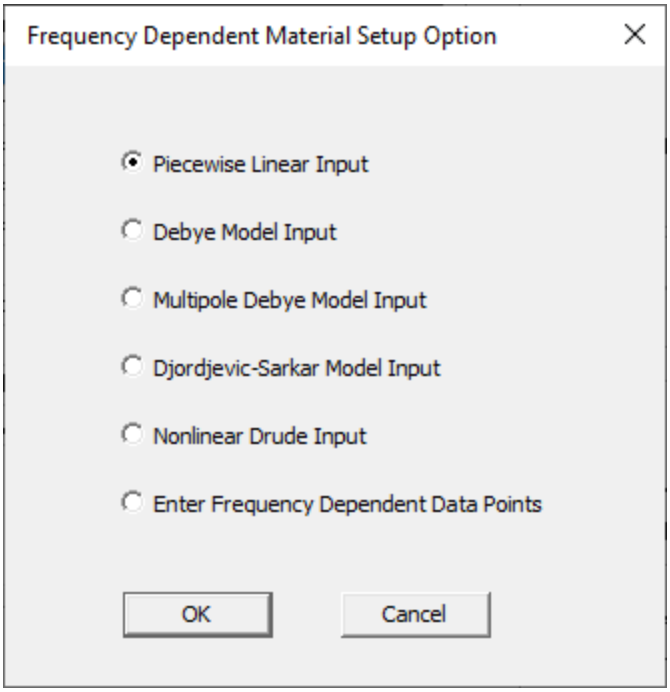
In HFSS, you can assign conductivity either directly as bulk conductivity, or as a loss tangent. This provides flexibility, but you should only provide the loss once. The solver uses the loss values just as they are entered.

Assigning Frequency-Dependent Properties

1. From the **Select Definition** window, select a material and click **View/Edit Material**.
2. Click **Set Frequency Dependency**.

   The **Frequency Dependent Material Setup Option** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The options are:

- **Piecewise Linear Input** – defines the material property values as a restricted form of piecewise linear model with exactly 3 segments (flat, linear, flat). You specify the property's values at an upper and lower corner frequency. Between these corner frequencies, HFSS linearly interpolates the material properties; above and below the corner frequencies, HFSS extrapolates the property values. This dataset can be modified with additional points if desired.
- **Debye Model Input** – a single-pole model for the frequency response of a lossy dielectric material. In some materials, up to about a 10-GHz limit, ion and dipole polarization dominate and a single pole Debye model is adequate. HFSS allows you to specify an upper and lower measurement frequency, and the loss tangent and relative permittivity values at these frequencies. You may optionally enter the permittivity at optical frequency, the DC conductivity, and a constant relative permeability.
- **Multipole Debye Model Input** – allows you to provide the data of relative permittivity and loss tangent versus frequency. Based on this data, the software dynamically generates frequency-dependent expressions for relative permittivity and loss tangent through the Multipole Debye Model. The input dialog plots these expressions together with your input data through the linear interpolations. The generated expressions provide the new value for the material properties of relative permittivity and loss tangent. Both the expressions and data triples can be saved and reloaded.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Djordjevic-Sarkar Model Input** – for low-loss dielectric materials (particularly FR-4) commonly used in printed circuit boards and packages. In effect, it uses an infinite distribution of poles to model the frequency response, and in particular the nearly constant loss tangent, of these materials. This allows you to enter the relative permittivity and loss tangent at a single measurement frequency. You may optionally enter the relative permittivity and conductivity at DC.
- **Nonlinear Drude Input** – Drude materials are now supported by the transient solvers, producing plasma density data within the material domains. This means that plasma density plots can be produced.
- **Enter Frequency Dependent Data Points** – allows you to enter, import or edit frequency dependent data sets for each material property. Any number of data points may be entered. This is an arbitrary piecewise linear model.

3. Select the input type and click **OK**.

   An input type-specific dialog box appears, based on your selection.

   Piecewise Linear Frequency Dependent Material Input, Debye Model Input, and Djordjevic-Sarkar Model Input remember the values previously used and also include plots to show the property curves in real time as changes to the input are made. Input values for each are saved as material attached data for the material being edited. These data items are saved with materials when they are written into a project file or exported to a material library. Note that when a frequency-dependent setup method is used and the values are pre-populated with saved data, the dialog box title will have "(Update)" appended.

After you have entered the data for your selection, you return to the **View/Edit Material** window. New default function names appear in the material property text boxes. HFSS automatically creates a dataset for each material property. Based on a varying property's dataset, HFSS can interpolate the property's values at the desired frequencies during solution generation.

To modify the dataset with additional points, see *Editing Datasets*.

> **Note:**
>
> Neither the piecewise nor the loss model asks for frequency-dependent conductivity because the constant sigma represents DC loss and the frequency-dependent loss tangent represents polarization losses.

Frequency Dependence Visualization

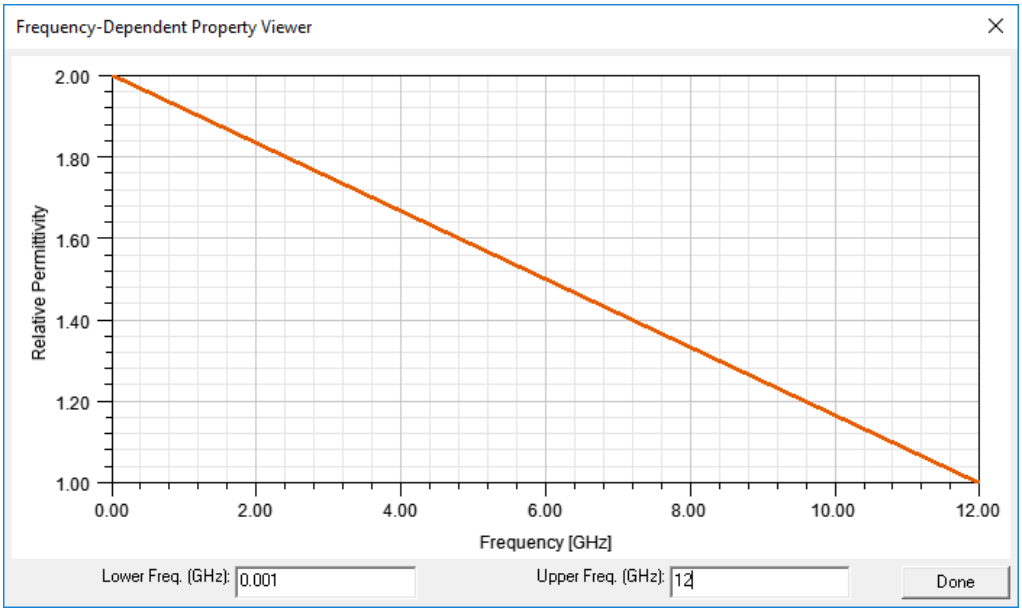
When you view or edit material properties, it is important to have a sense of how properties may vary with frequency. Frequency-dependent properties come in a variety of forms, ultimately resulting in some value expression or dataset. Plots of properties as a function of frequency are available through the **View/Edit Materials** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To view the plot:

1. From the **Select Definition** window, select a material and click **View/Edit Material**.
2. Right-click a property (for example, relative permittivity or dielectric loss tangent), and select **View Property vs. Frequency**.

   The **Frequency-Dependent Property Viewer** appears, displaying the plot.



The window contains boxes where you can edit the frequency range for the plot. If the property was set by one of the nonlinear frequency setup methods, the frequency range will be derived from the data for that method, and edits to the lower/upper frequencies are not saved. Otherwise, the frequency range lower/upper frequency limit defaults are stored in the registry, and are updated if you modify the values. If values are not yet stored in the registry, the range defaults to 1MHz-10GHz. Note that **the View Property vs. Frequency** menu option is not displayed for choice properties because frequency-dependence doesn't apply to those.

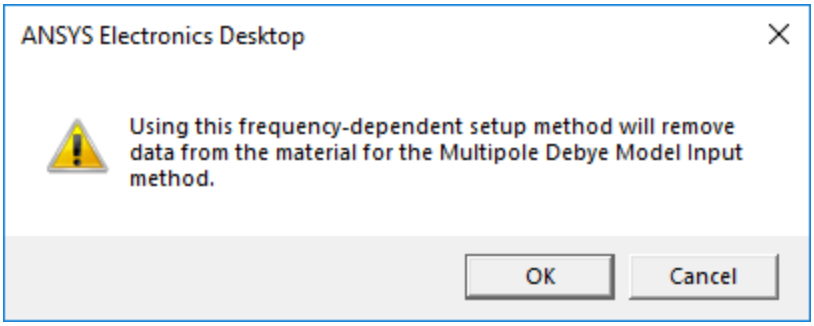
Saved Input Data Invalidation For Frequency-Dependent Setup

You will be prompted with a warning in two circumstances:

- Data associated with one of the frequency-dependent setup methods is attached to the material definition, and a property which would be set by this method is modified.
- Data associated with one of the frequency-dependent setup methods is attached to the material definition, and a different setup method is subsequently used.

You can choose **OK** to continue with the edit and remove the invalidated setup data, or choose **Cancel** to discard any changes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

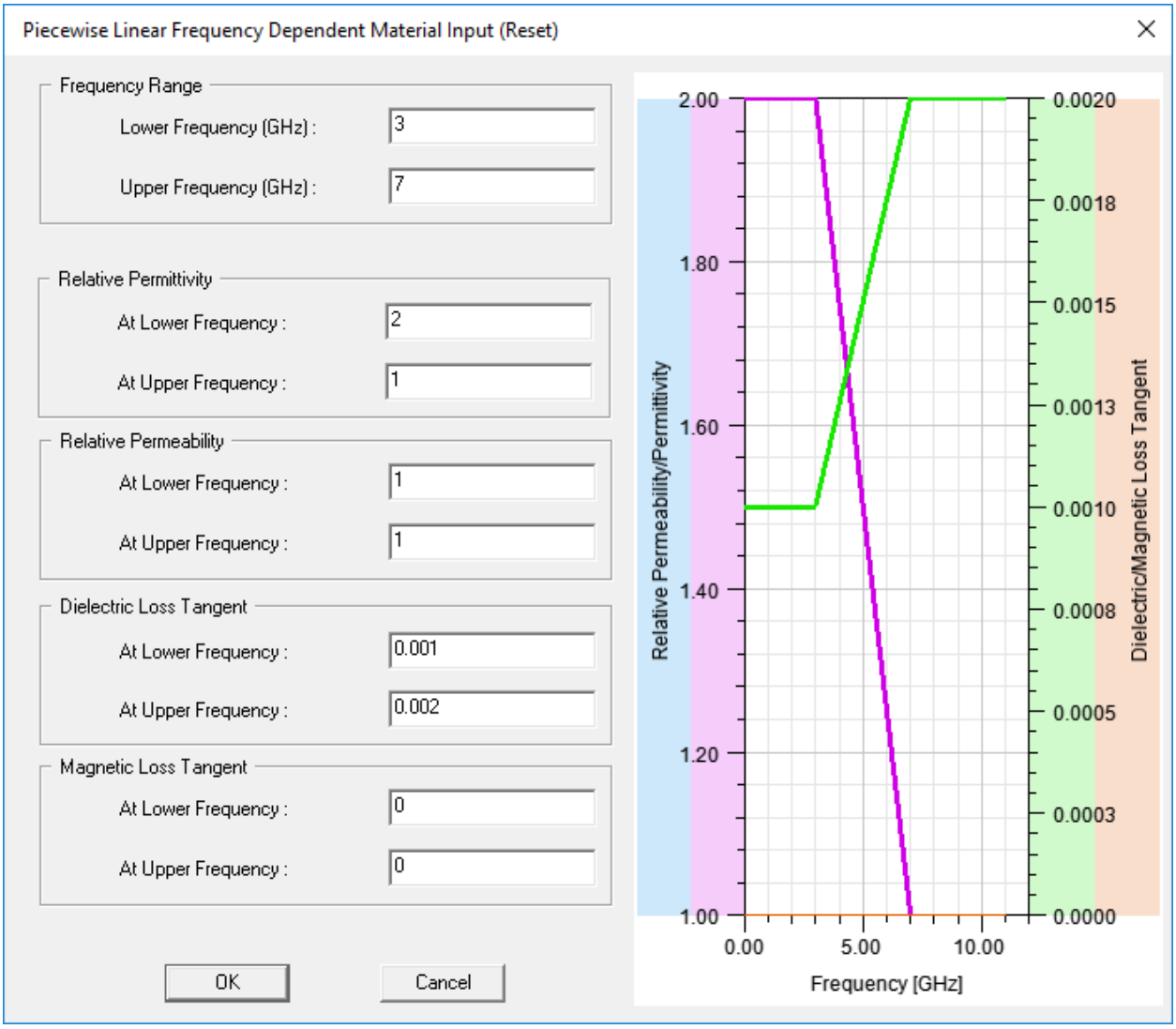


Assigning Frequency Dependent Material: Piecewise Linear Input

When you select **Piecewise Linear Input** as the model for the frequency dependent material property, the **Piecewise Linear Frequency Dependent Material Input** window appears.

From this window, you can enter material property values and view a property vs. frequency plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Use the fields to enter lower and upper frequency values for:

- **Frequency Range** – HFSS assumes that the material's property values remain constant between these frequencies.
- **Relative Permittivity** – If the permittivity of the material does not vary with frequency, enter the same value you entered for the Frequency Range Lower Frequency.
- **Relative Permeability** – If the permeability of the material does not vary with frequency, enter the same value you entered for the Frequency Range Lower Frequency.
- **Dielectric Loss Tangent** – If the dieletric loss tangent of the material does not vary with frequency, enter the same value you entered for the Frequency Range Lower Frequency.
- **Magnetic Loss Tangent** – If the magnetic loss tangent of the material does not vary with frequency, enter the same value you entered for the Frequency Range Lower Frequency.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

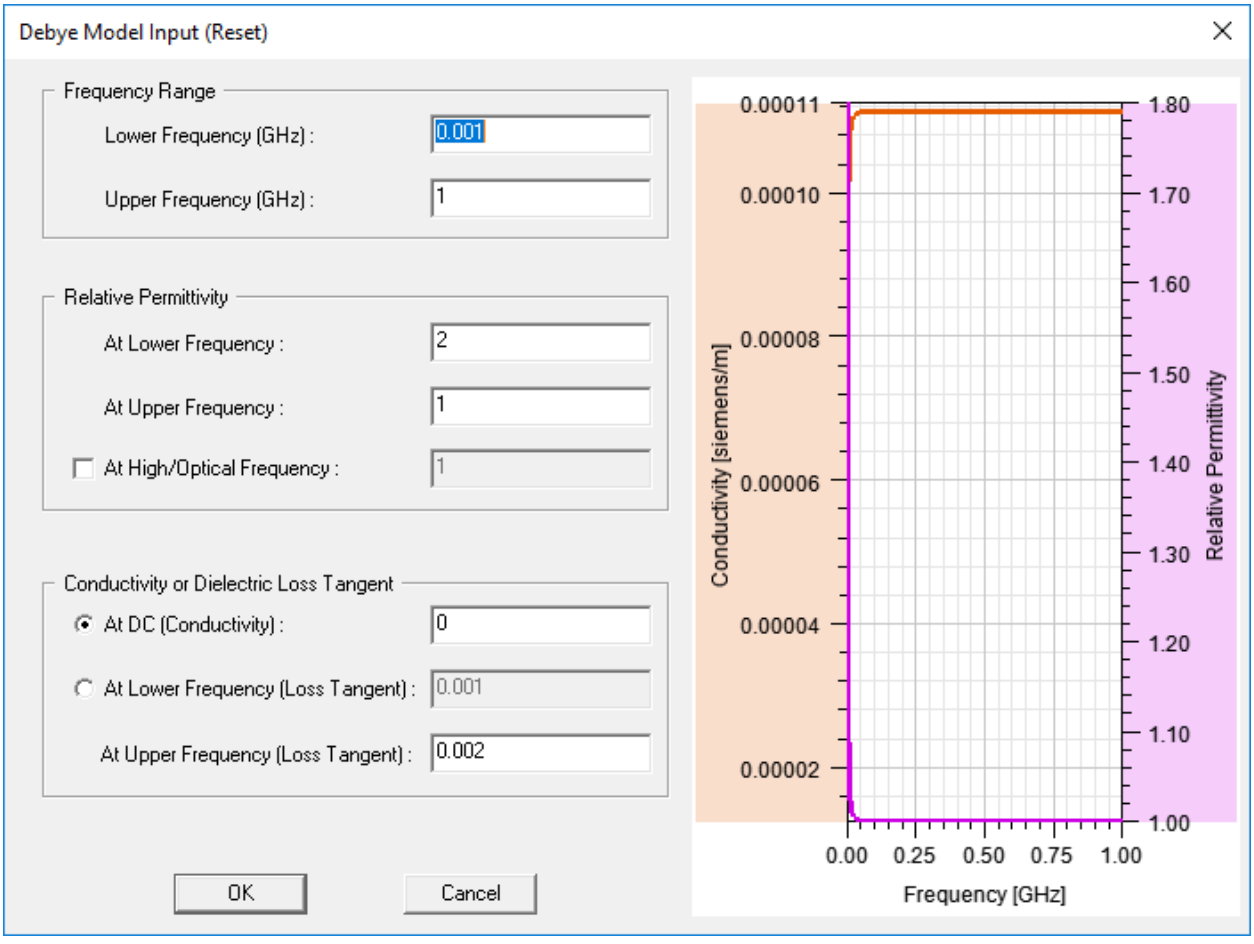
> **Note:**
>
> Neither the piecewise or the loss models ask for frequency dependent conductivity because there the constant sigma represents the DC loss and the frequency dependent loss tangent represents the polarization losses.

To modify the dataset with additional points, see Editing Datasets.

Assigning Frequency Dependent Material: Debye Model Input

When you select **Debye Model Input** as the model for the frequency dependent material property, the **Debye Model Input** window appears.

From this window, you can enter material property values and view a property vs. frequency plot.



Use the fields to enter lower and upper frequency values for:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Frequency Range** – HFSS assumes that the material's property values remain constant between these frequencies.
- **Relative Permittivity** – If the permittivity of the material does not vary with frequency, enter the same value you entered for the Frequency Range Lower Frequency. If you need to specify a value for At **High/Optical Frequency**, select the check box and enter a value.
- **Conductivity or Dielectric Loss Tangent** – Select either **At DC** to specify conductivity, or **At Lower Frequency** to specify Loss Tangent.

After you have entered the data for your selection, click **OK** to return to the **View/Edit Material** window. New default function names appear in the material property text boxes. A dataset is automatically created for each material property. Based on a varying property's dataset, HFSS can interpolate the property's values at the desired frequencies during solution generation.

To modify the dataset with additional points, see Editing Datasets.

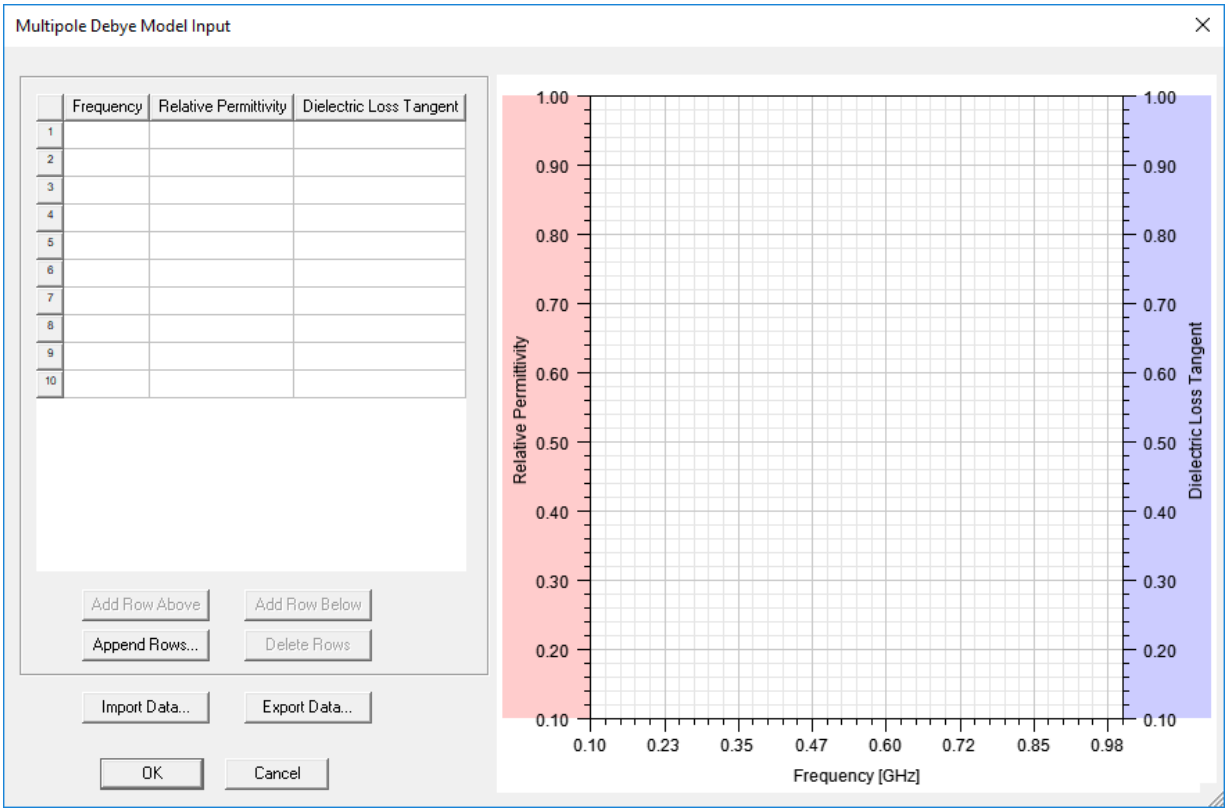
> **Note:**
>
> Neither the piecewise or the loss models ask for frequency dependent conductivity because there the constant sigma represents the DC loss and the frequency dependent loss tangent represents the polarization losses.

Assigning Frequency Dependent Material: Multipole Debye Model Input

When you select **Multipole Debye Model Input** as the model for the frequency dependent material property, the **Multipole Debye Model Input** window appears.

From this window, you can enter material property values and view a property vs. frequency plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The window contains a table that allows you to specify **Frequencies**, and the material's **Relative Permittivity** and **Dielectric Loss Tangent** at each frequency.

> **Note:**
>
> - The minimum value for Frequency is 0. There is no upper limit.
> - The minimum value for Relative Permittivity is 1. There is no upper limit.
> - The minimum value for Dielectric Loss Tangent is 0. There is no upper limit.
> - To get good results, provide at least 5 frequency points.

Enter data values manually, or click **Import Data** to import data from a *.tab file.

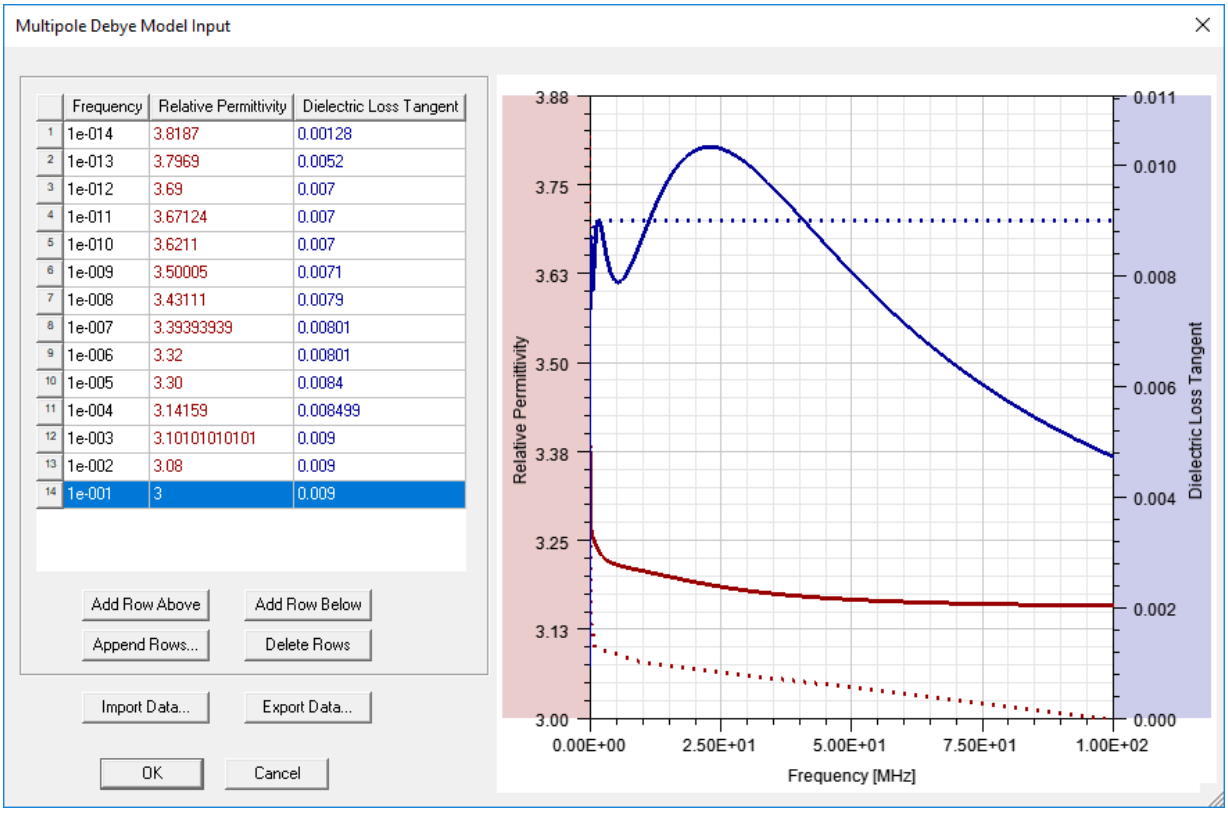
Below is an example of the file format. Each row provides Frequency (assumed to be Hz), Permittivity, and Loss Tangent.

```
0.00001 3.8136 0.00128

0.00010 3.7914 0.00520

0.00100 3.7500 0.00700
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
0.01000 3.7119 0.00700
0.10000 3.6742 0.00700
1.00000 3.6354 0.00700
2.15444 3.6346 0.00702
3.17000 3.6325 0.01073
4.64160 3.6186 0.01500
10.0000 3.5777 0.01750
21.5444 3.5458 0.01750
26.0000 3.5383 0.01750
46.4160 3.5148 0.01750
50.0000 3.5119 0.01750
```

As you enter data, the plot updates. The input data is linearly interpolated using the Multipole Debye model.



Double-click any plot element to change its properties (color, units, scaling, etc.).

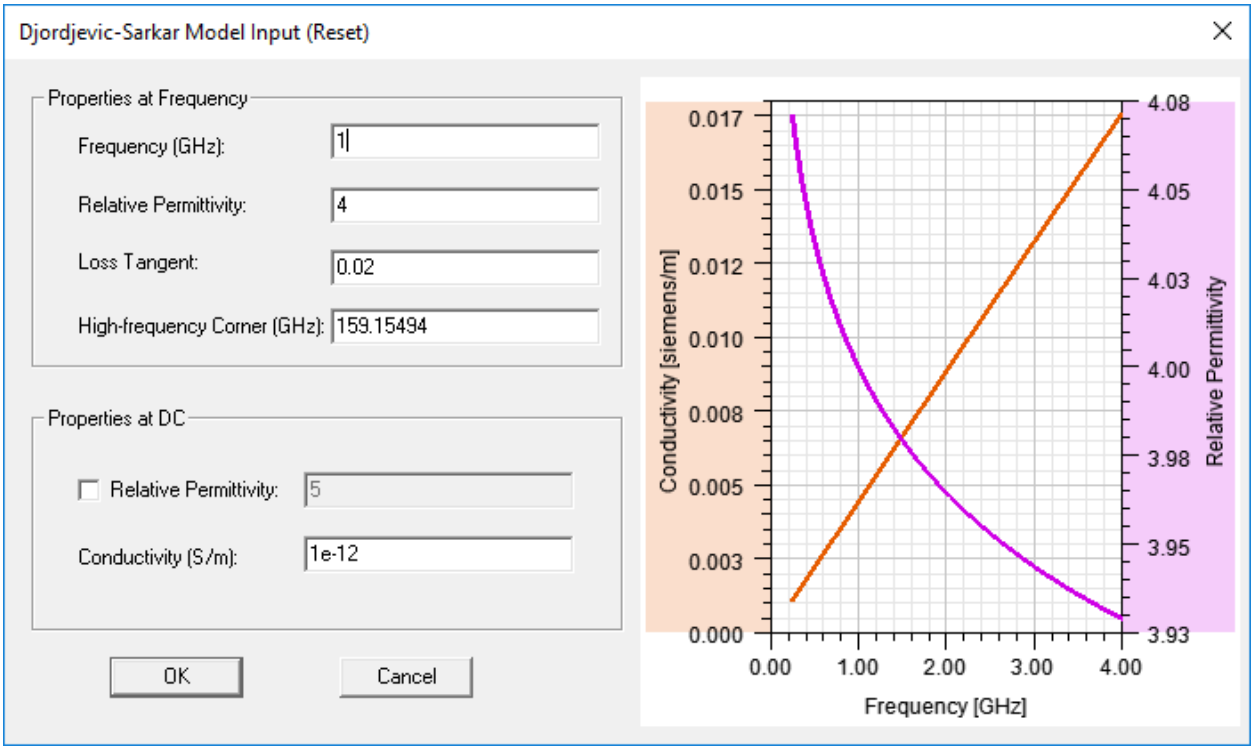
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To export the dataset in *.tab format for later use, click **Export Data**.

Assigning Frequency Dependent Material: Djordjevic-Sarkar Model Input

When you select **Djordjevic-Sarkar Model Input** as the model for the frequency dependent material property, the **Djordjevic-Sarkar Model Input** window appears.

From this window, you can enter material property values and view a property vs. frequency plot.



Use the fields to enter **Properties at Frequency** values:

- **Frequency**
- **Relative Permittivity**
- **Loss Tangent**
- **High-Frequency Corner** – the value of the High-frequency Corner should be at least 10 times higher than the Frequency. Reducing the upper corner frequency is not very attractive, because it has not been observed in experimental data. Therefore it is an upper bound on the loss tangents the Djordjevic-Sarkar model can handle. For technical details see *Technical Notes*: Djordjevic Sarkar Causal Dielectric Model.

Use the fields to enter **Properties at DC** values:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Relative Permittivity** – select the check box if you wish to specify relative permittivity.
- **Conductivity**

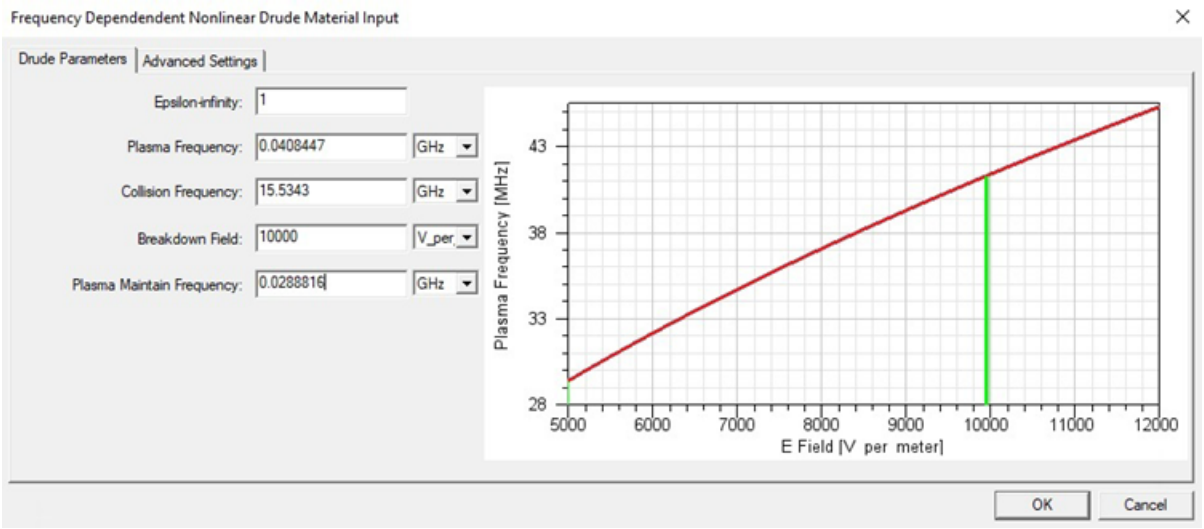
After you have entered the data for your selection, click **OK** to return to the **View/Edit Material** window. New default function names appear in the material property text boxes. A dataset is automatically created for each material property.

To modify the dataset with additional points, see Editing Datasets.

Assigning Frequency Dependent Material: Nonlinear Drude Input

When you select **Nonlinear Drude Input** as the model for the frequency dependent material property, the **Nonlinear Drude Input** window appears. Drude materials are now supported by the Transient solvers, producing plasma density data within the material domains. This means that plasma density plots can be produced.

From this window, you can enter material property values and view a plasma frequency vs. electric fields plot.

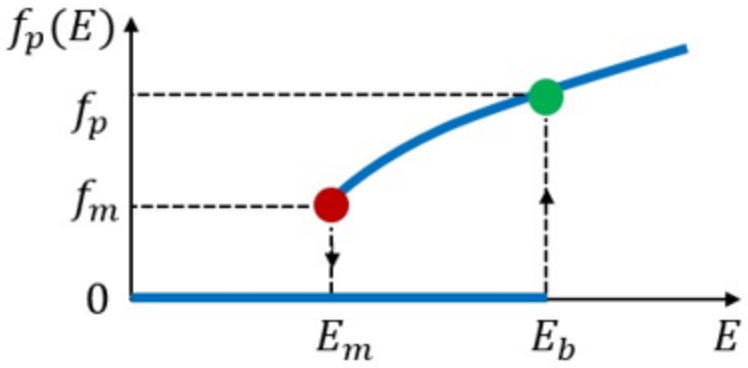


The main parameters of a nonlinear Drude material are the epsilon-infinity ε_∞, the plasma frequency f_p, the collision frequency f_c, the breakdown field E_b, and the plasma maintain frequency f_m. Given first three parameters, the relative permittivity is defined as

$$\varepsilon_r(E) = \varepsilon_\infty + \frac{\left[2\pi f_p(E)\right]^2}{s(s + 2\pi f_c)}$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where $s=j2\pi f$ is the complex frequency, ε_∞ and f_c are constants, and f_p is a function of the time-dependent field strength $E=|E(t)|$.



When the electric field strength E increases from zero then exceeds E_b, the discharge is on. The plasma frequency is uniquely determined by a parabola symmetric to the x-axis and passing through points (E_m,f_m) and (E_b,f_p). Once the discharge is on, the lowest field strength for maintaining the discharge is $E_m=E_b/2$ and the corresponding plasma maintain frequency is f_m. When E drops below E_m, the discharge is off and $f_p(E)$ becomes zero. No matter if the discharge is on or off, ε_r has a nonzero contribution from ε_∞, which is usually set to one due to the background gas being in low pressure.

Because the plasma is generated by microwaves, the radiation must be able to penetrate the plasma in order to ionize atoms and molecules. This happens when

$$f^2 \geq f_p^2 - f_c^2$$

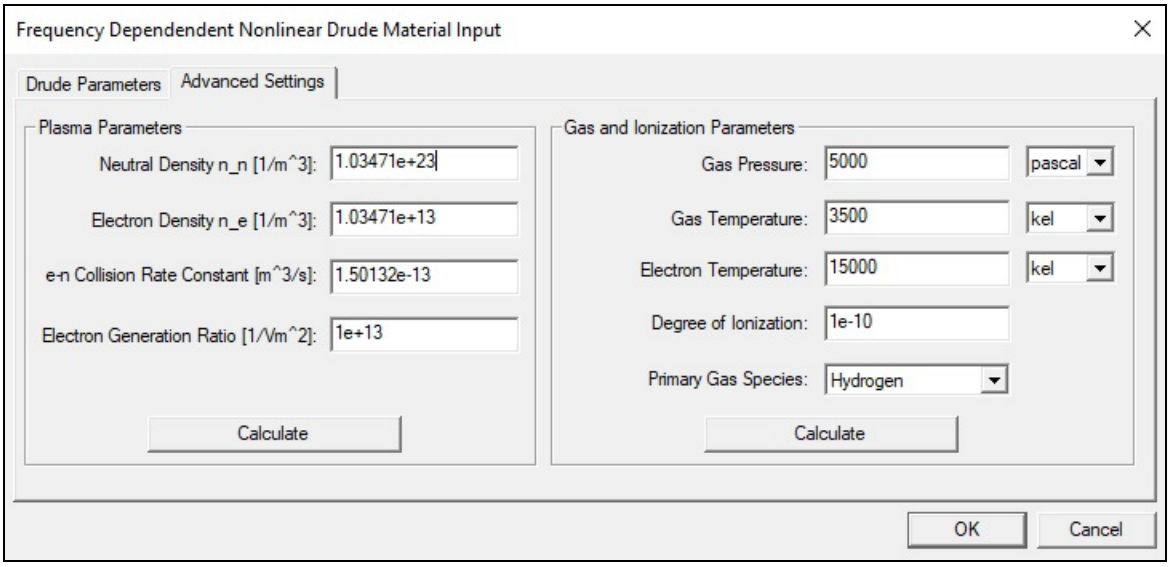
Important:

You must be cautious about selecting the frequency f of the harmonic input signal while setting up a transient excitation.

Advanced Settings

The **Advanced Settings** tab includes settings for **Plasma Parameters** and **Gas and Ionization Parameters**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Advanced Settings: Plasma Parameters

The plasma and collision frequencies can be derived from more fundamental parameters obtainable through plasma diagnostics. Given the electron density n_e, the plasma frequency f_p is equal to $1/2\pi \sqrt{((n_e e^2)/(m_e \varepsilon_0))}$, where e and m_e are the charge and mass of an electron, respectively. Given the neutral density n_n and the electron-neutral collision rate constant v_{en}, the collision frequency f_c is equal to $v_{en} n_n$.

Once the setup of n_n, n_e, and v_{en} is done, press **Calculate**. The plasma and collision frequencies on the **Drude Parameters** tab will be updated accordingly.

The electron generation ratio γ is mainly used for plasma density plot and will be explained later.

Advanced Settings: Gas and Ionization Parameters

Considering the plasma parameters are not always available without a plasma being generated and measured, there is another option to estimate plasma and collision frequencies from gas parameters. Given the pressure p and the temperature T of the discharge gas, the density of the neutral atoms can be derived from the ideal gas law

$n_n = p/(k_B T)$,

where k_B is the Boltzmann constant.

Given the degree of ionization to be α and assuming the plasma is quasi-neutral, the electron density can be calculated by

$n_e = \alpha/(1-\alpha)\, n_n$,

where $0<\alpha<1$.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

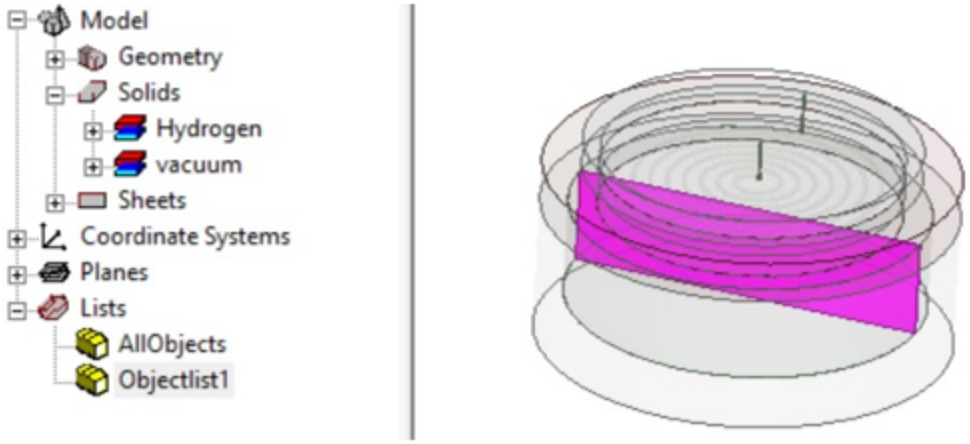
The electron-neutral collision rate depends on the primary gas species and the electron temperature.

After you edit the above parameters and click **Calculate**, the **Drude Parameters** tab updates accordingly.

Plasma Density Plot

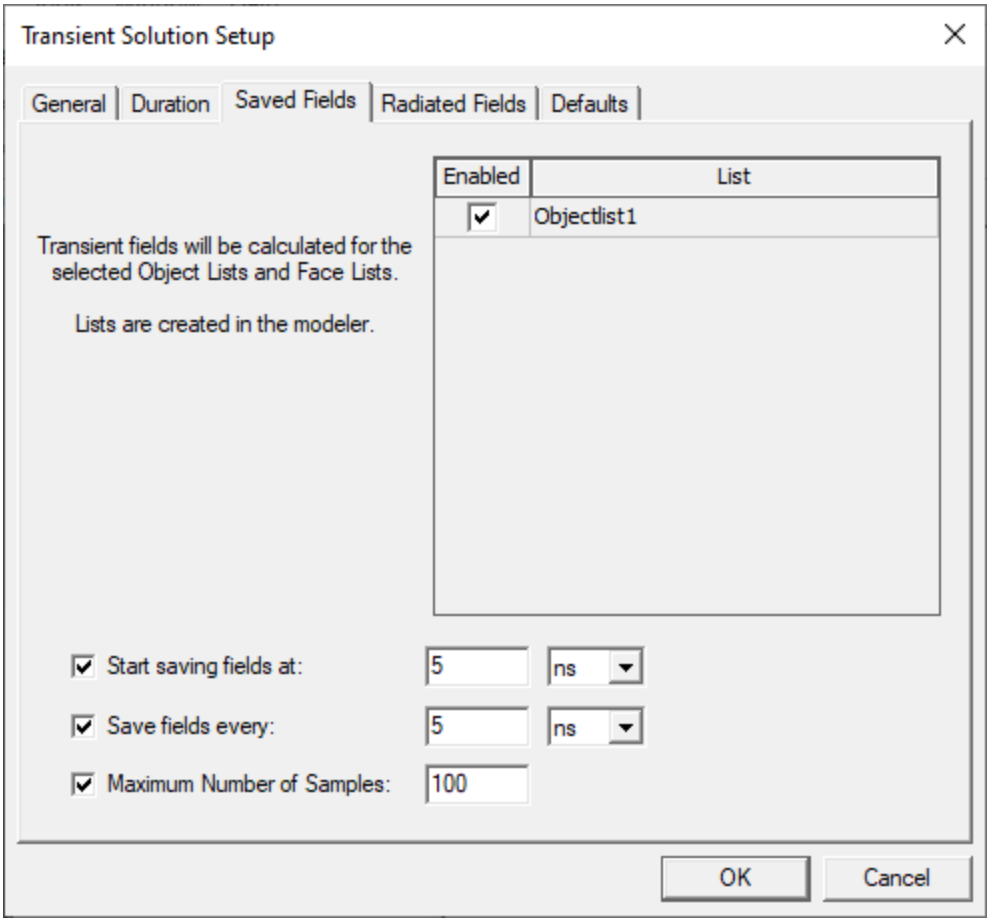
The procedure for creating a plasma density plot resembles that for a transient field plot.

First, create face or object lists for observing plasma densities and transient fields.



Next, use the **Saved Fields** tab of the **Transient Solution Setup** to select the face or object lists. By checking the boxes you can enable the Start saving fields time, and Save fields at interval. When you select a face or an object list, you can also specify the Maximum Number of Samples.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The time-varying electron density is calculated by:

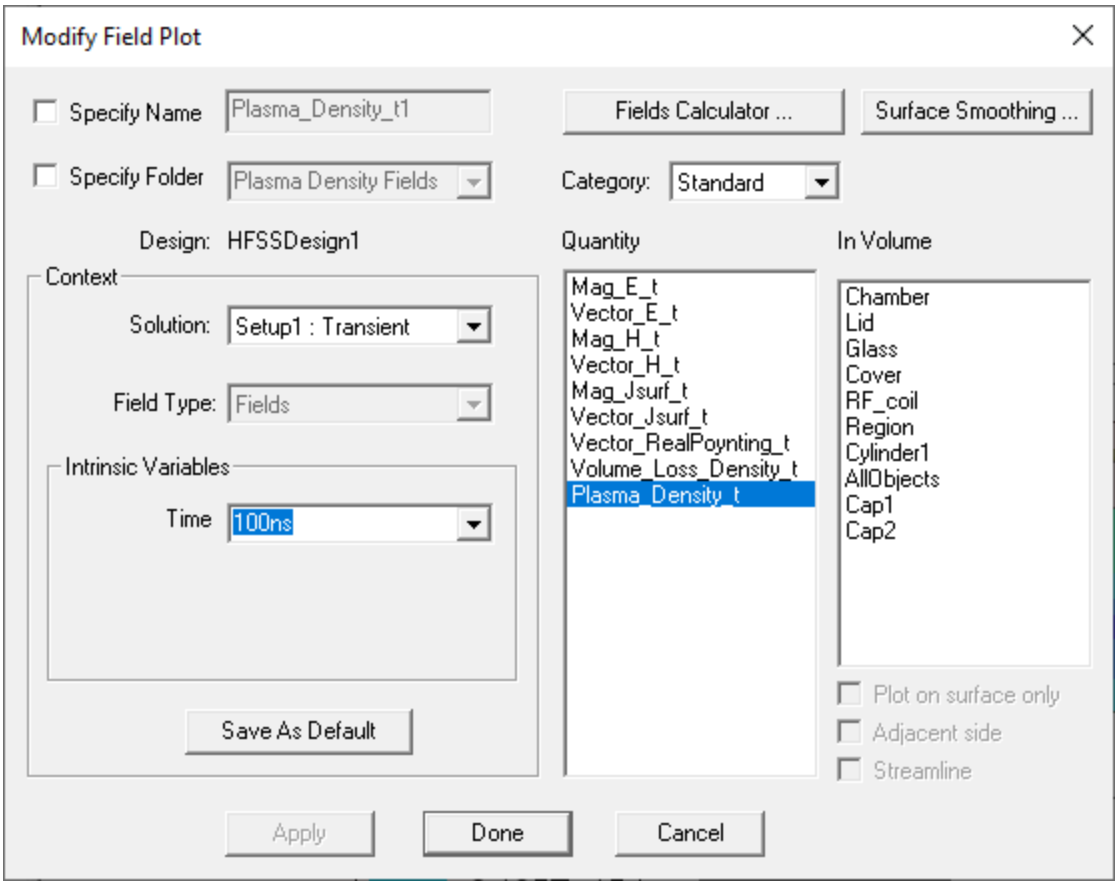
$$n_e(t)=\begin{cases} \gamma(E-E_m)+n_e(0), & E \geq E_b \\ \max\{\gamma(E-E_m)+n_e(0),\ 0\} & n_e(t-\Delta t)>0 \\ 0, & E<E_m \ \vee\ (E<E_b \ \wedge\ n_e(t-\Delta t)=0) \end{cases}$$

The initial electron density $n_e(0)$ is the electron density $m_e\varepsilon 0(2\pi f_m/e)^2$ specified by Plasma Parameters. The electron generation ratio γ characterizes the efficiency of electron generation with respect to the field variation $E(t)$-E_m. Without clicking **Calculate** in Plasma Parameters, you can set n_e and γ independently from other parameters to calibrate the plasma density plot.

Outside Drude materials, the plasma density will be zero. Besides, for a ramped sinusoidal excitation, the initial electric field is zero for the entire simulation domain. Therefore, the plasma density is also zero at the beginning of a transient simulation.

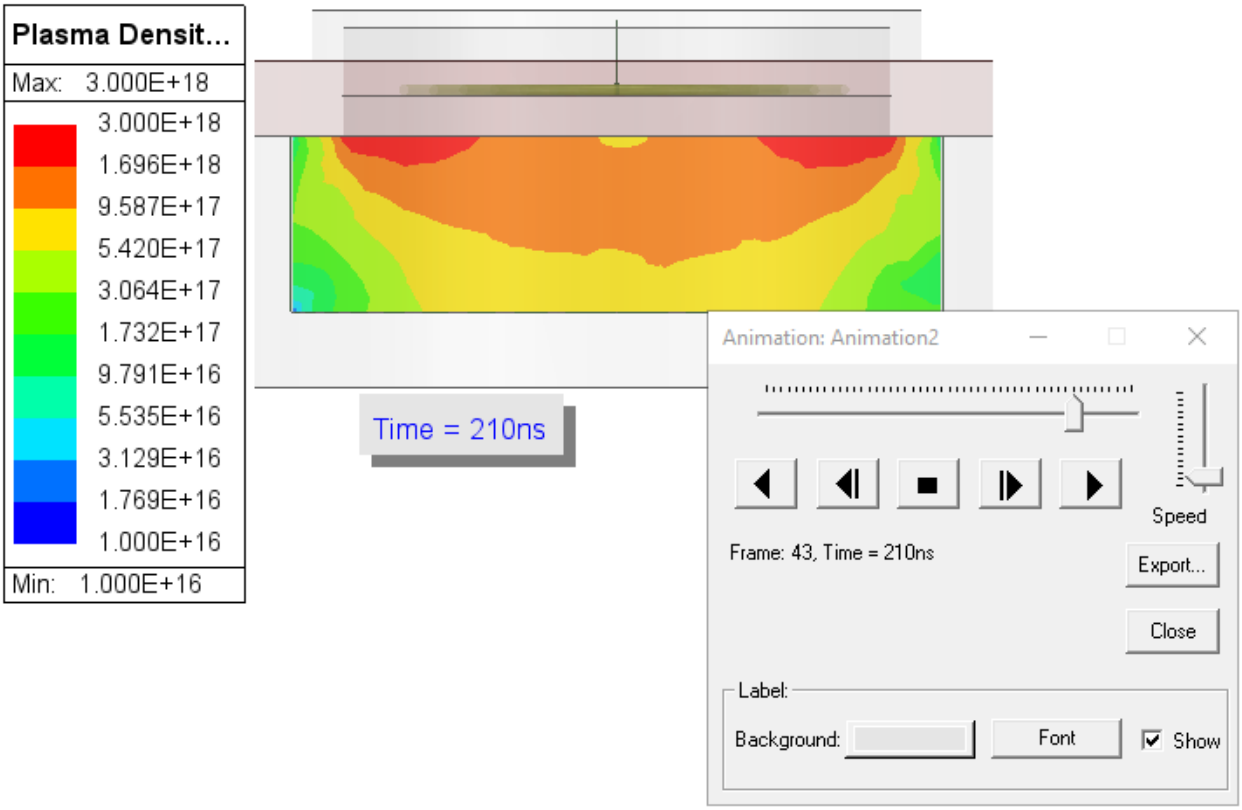
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you create a Field Plot on a transient project with a Nonlinear Drude Material involved, the Quantity menu allows you to select Plasma_Density_t.



You can animate such a Plasma_Density_t plot. The following animation shows the plasma density in an inductively coupled plasma (ICP)-CVD chamber excited by a 13.6 MHz sinusoidal signal.

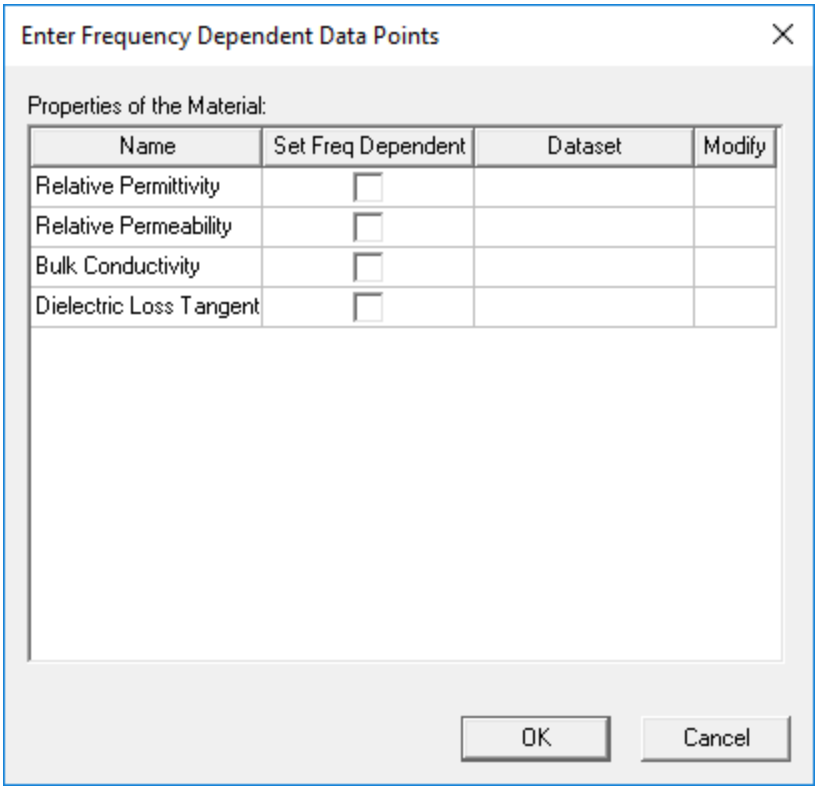
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Assigning Frequency Dependent Material: Enter Frequency Dependent Data Points

When you select **Enter Frequency Dependent Data Points** as the model for the frequency dependent material property, the **Enter Frequency Dependent Data Points** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

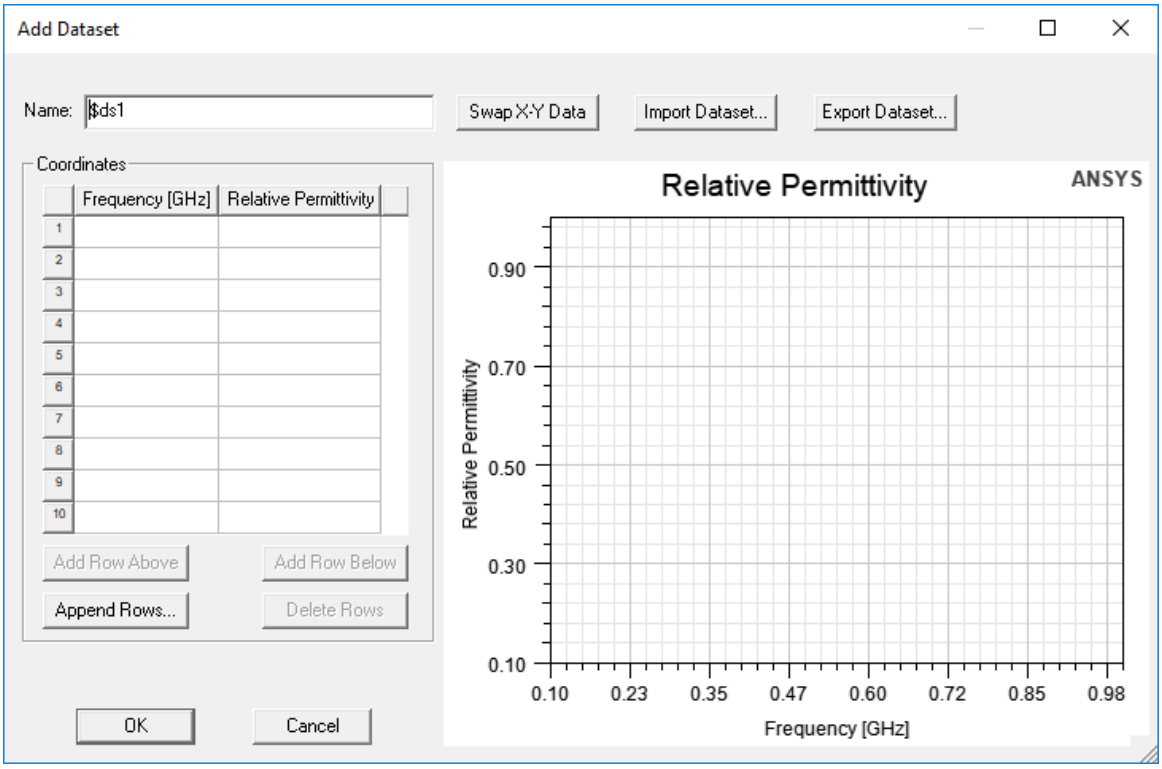


From this window, you can use datasets to set frequency-dependent material properties.

1. Select the **Set Freq Dependent** check box next to the property you wish to edit. This enables the **Dataset** field.
2. Click the property's **Dataset** field and use the drop-down menu to select **Add/Import Dataset**.

   The **Add Dataset** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



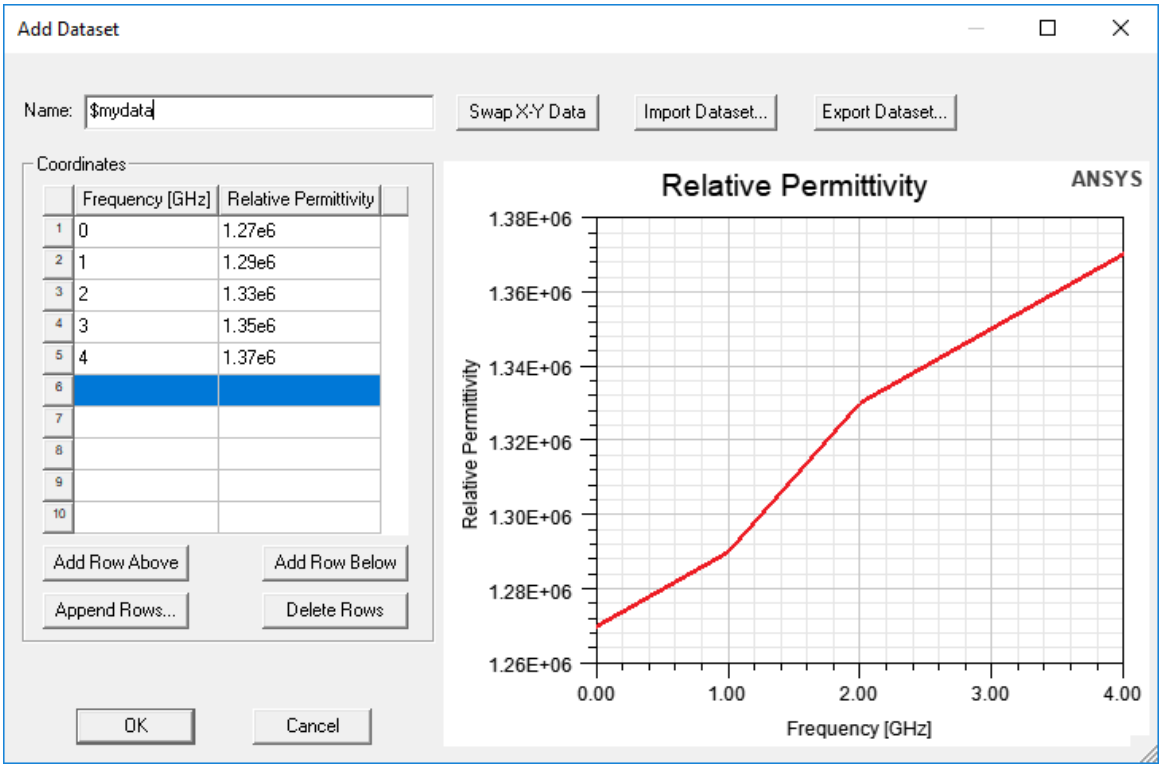
This window allows you to specify the following:

- **Name** – the default name is $ds1, but you can rename it. Note that the dollar sign ($) is automatically appended.
- **Coordinates** – contains a table that allows you to specify the Frequency (in GHz) and the specified material property's value at that frequency. Enter the data into the table manually, or click the **Import Dataset** button to populate it from a *.tab file.

You can also:

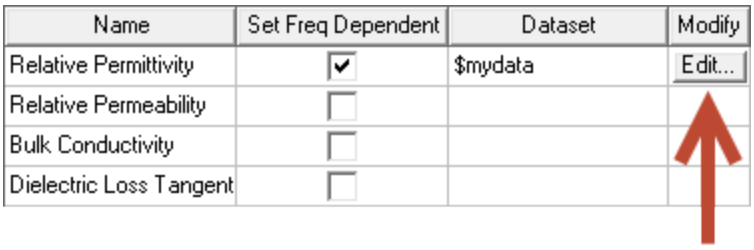
- Rearrange table items using the **Add Row Above**, **Add Row Below**, **Append Rows**, and **Delete Rows** buttons.
- **Swap X-Y Data**
- **Export Dataset** as a *.tab file for later import.
- View the table data as a plot on the right side of the window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. When you have finished adding points, click **OK**.

   You are returned to the **Enter Frequency Dependent Data Points** window, where the **Modify** field is now active. If you need to modify the dataset, click **Edit**.

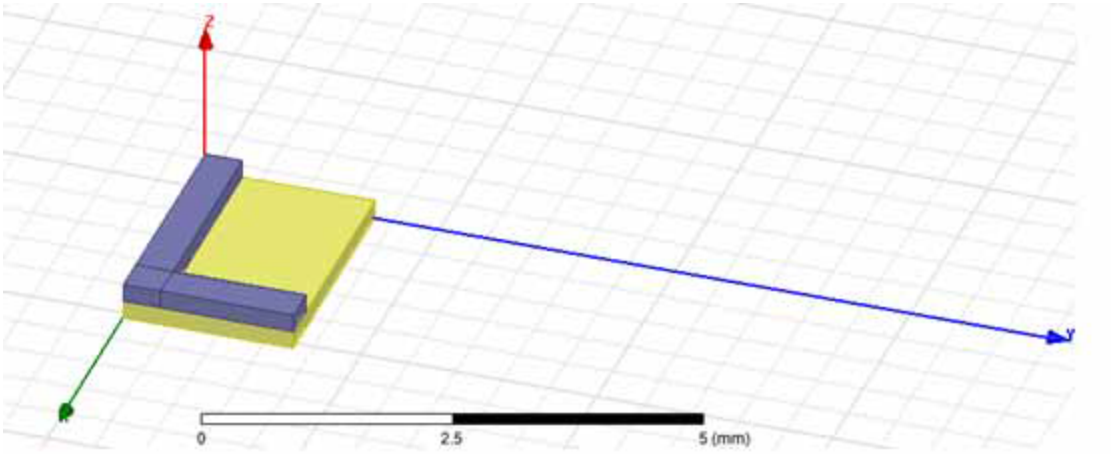


4. Click **OK** again to return to the **View/Edit Material** window.

Spatially Dependent Materials in HFSS

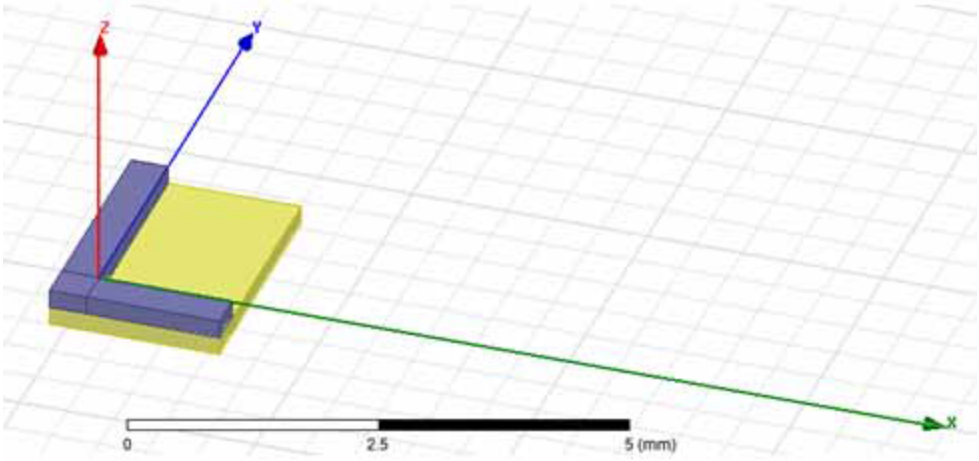
HFSS supports spatially dependent materials, that is, materials whose properties change over their length (or width or thickness). Consider the L shaped strip lines in the following figure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the conductivity of the gray striplines changes over the length of the line, then you need to express the change as a function along one of the axes. However, in this case the two striplines are normal to each other. So, you need to use a different coordinate system for each of them or a different material.

Each spatially dependent object should have its own coordinate system as that would associate each with a separate origin point. The following images show the coordinate systems for each stripline with the Y origin at the start of the object.

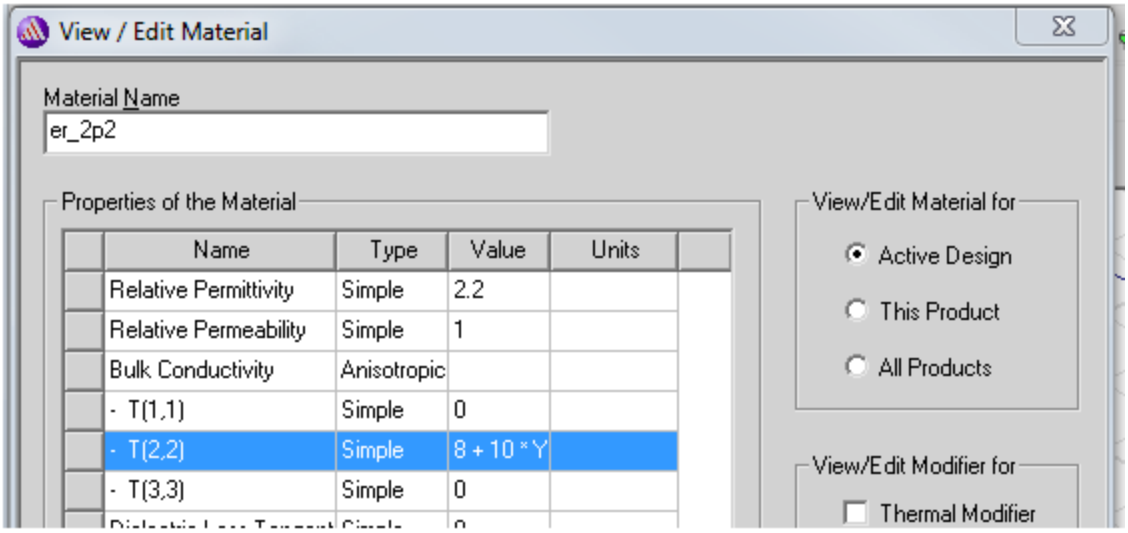


So, the Bulk Conductivity could be something like 300 + 10 * Y to show increasing conductivity over the length of the object. Of course, the entire expression language is available so more complicated functions are usable as are project variables. Design variables are not supported at this time.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Regardless of the model units defined in the design SI units (i.e. meters) are always used to compute spatially dependent material properties.

In the **View/Edit Material** dialog box, you can assign properties using the Cartesian coordinate [X, Y, Z] to assign the spatial properties for a material.



Here the value of the Y dimension of the object composed of the material, based on that object's assigned coordinate system, affects the Bulk conductivity along the T[2,2] Tensor.

> **Note:**
>
> The spatial dependency can only be expressed in Cartesian coordinates.

Spatially dependent properties can still be either isotropic or anisotropic.

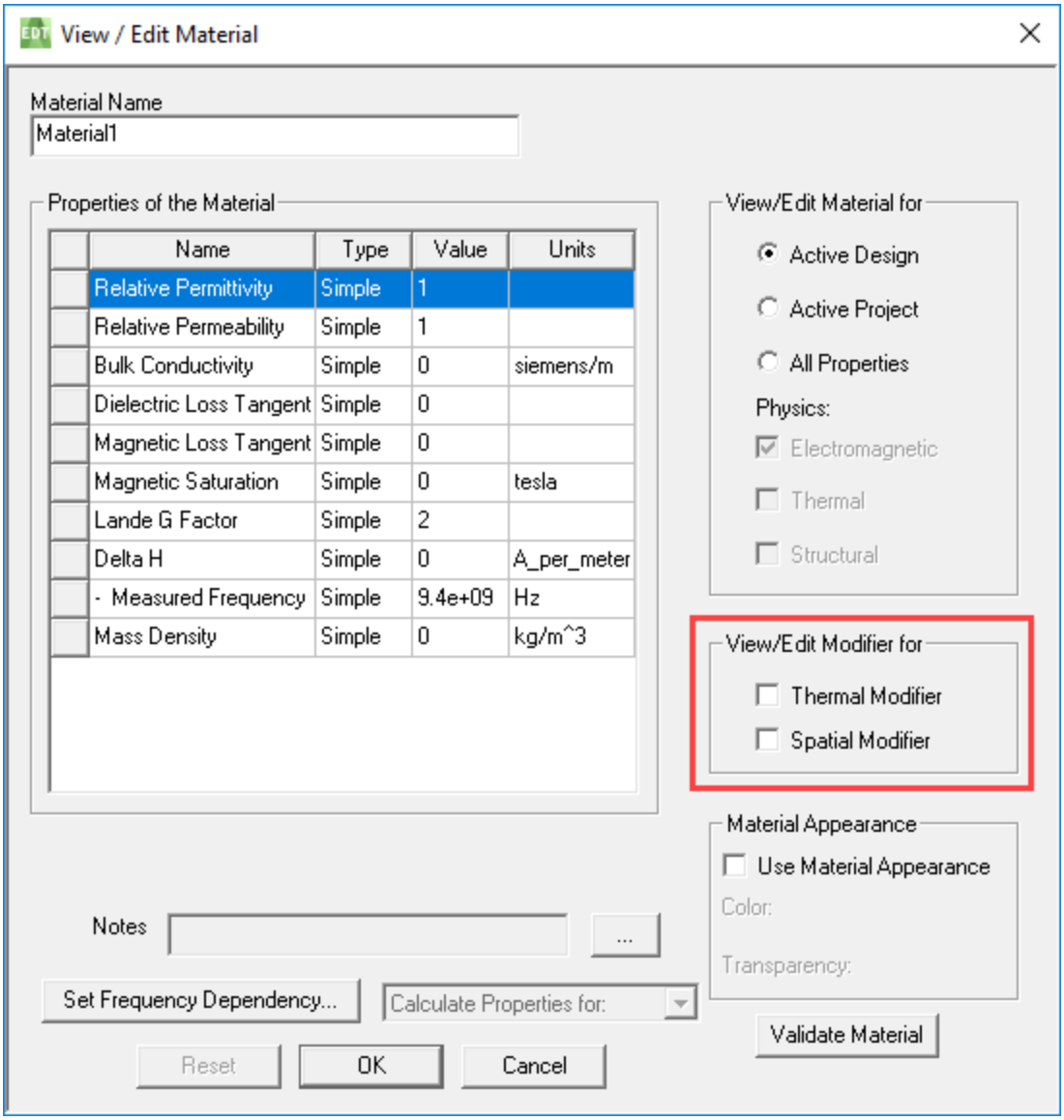
Spatially dependent properties will not be subject to solve time modifications like 'Automatic Causal Materials' transforms.

Support for 3D Spatial Modifier Dataset Material Property in HFSS

HFSS supports a 3D spatial dataset in material property. You can use expressions for HFSS materials that contain clp($ds, X,Y,Z) as a general value text, or create expressions that contain clp($ds, X, Y, Z) as a "Spatial modifier"(scalar factor).

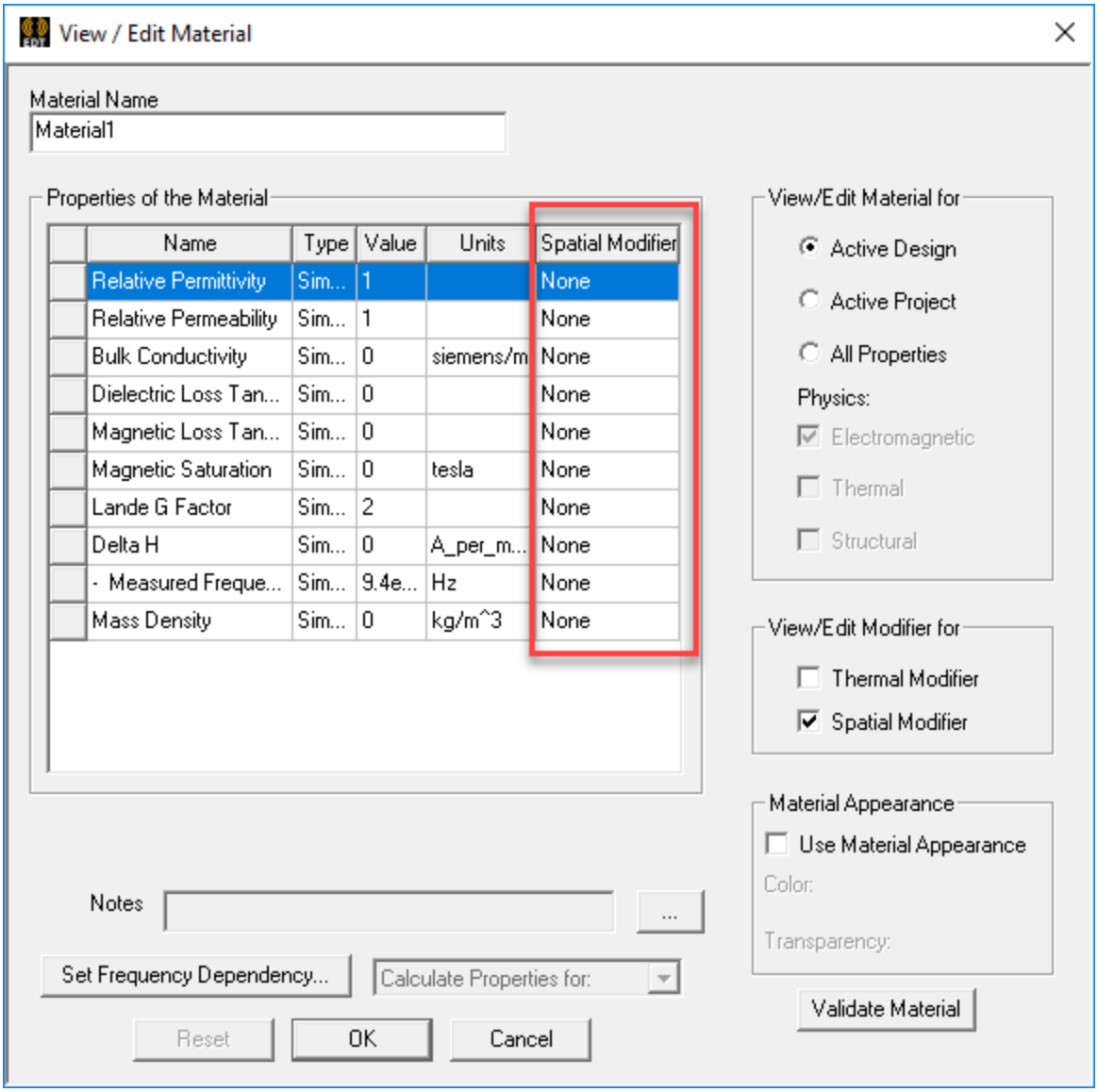
When you use the Materials window, and click either **View/Edit Materials...** or **Add Materials...**, the dialog will contain a Spatial Modifier check box in the View/Edit Modifier for pane.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

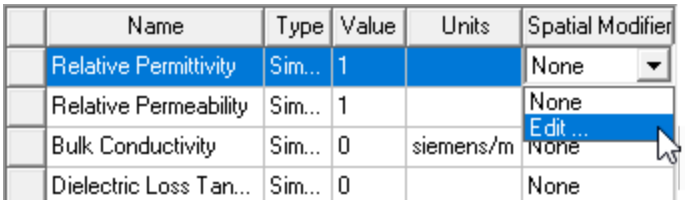


Checking **Spatial Modifier** adds another column to the **Properties of the Material** pane.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

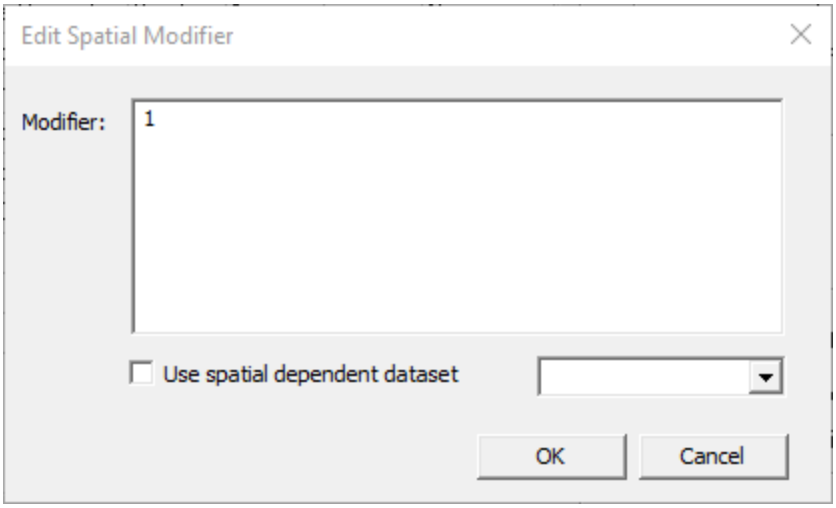


Click in the field for the any of the Properties to see the **Edit...** selection.

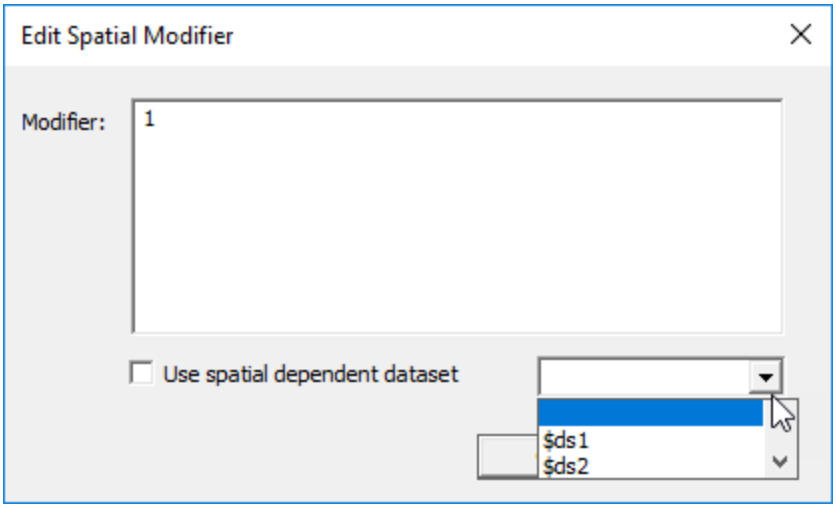


Clicking **Edit...** opens the **Edit Spatial Modifier** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

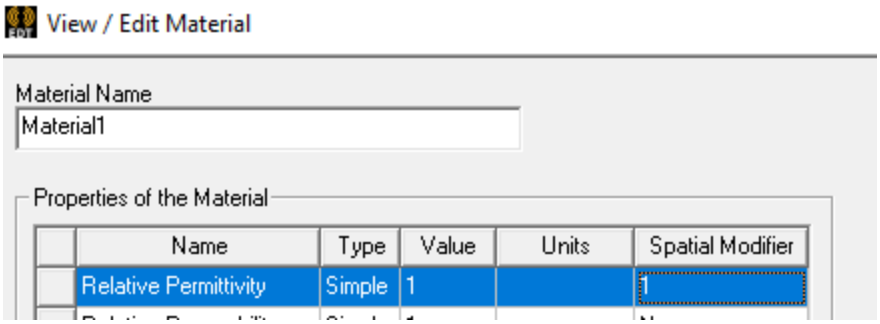


You can type in a value for the modifier, either an integer, real, variable, expression, or dataset in the form "clp($ds1,X,Y,Z)". Any existing datasets are listed in the drop-down menu.

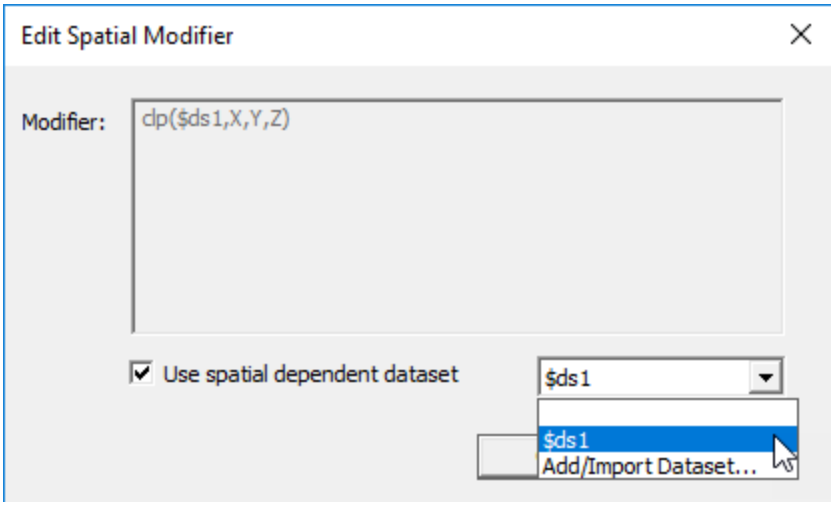


Clicking **OK** assigns the modifier you typed in or dataset you selected to the Spatial Modifier field for that Material Property.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

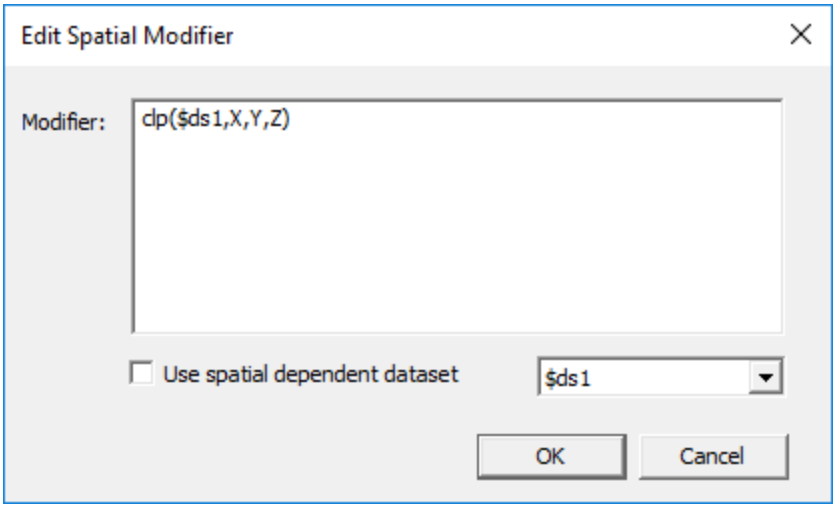


If you check, Use spatially dependent dataset, the field is grayed out and a menu lets you select $ds1 or **Add/Import Dataset...** . If you select $ds1, the grayed out field shows the modifier as clp($ds1,X,Y,Z).

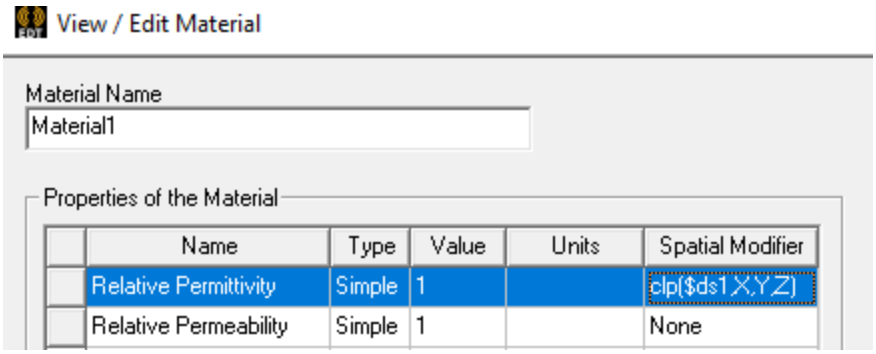


Clearing the check box leaves the modifier and enables the field.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

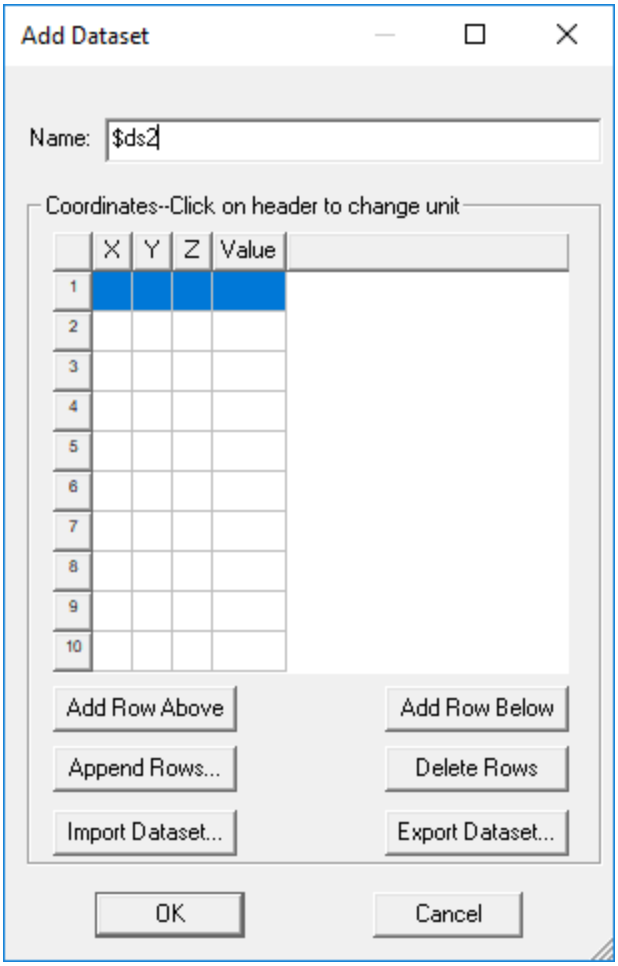


If you click OK, that Modifier is assigned to the Property.



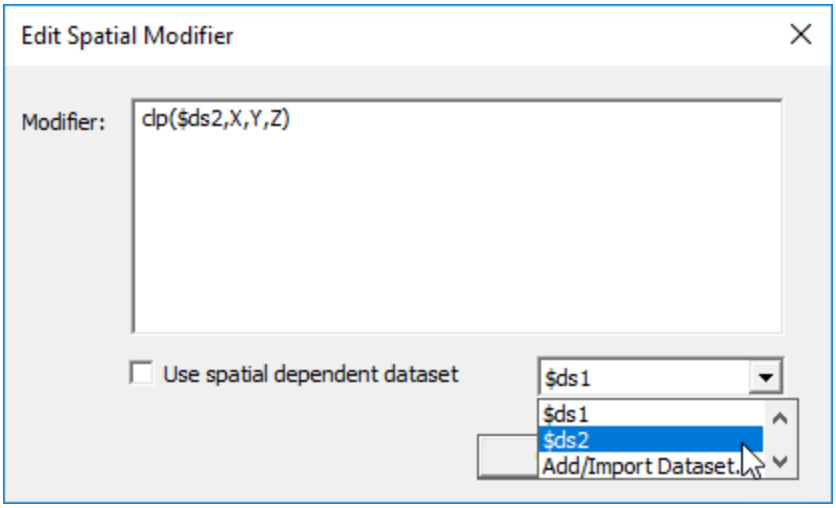
Selecting **Add/Import Dataset...** opens a dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

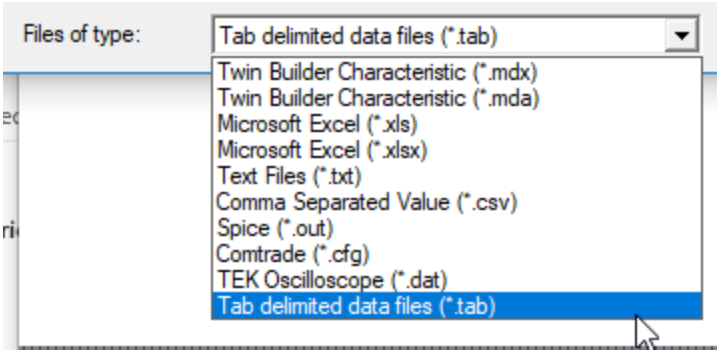


If you manually provide a dataset, you must provide at least two lines. After you **OK** a dataset, the **Add Data Set** dialog closes and the new dataset is listed as a menu choice on the **Edit Spatial Modifier** dialog menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To import a 3D dataset, only **Tab delimited data files (*.tab)** are supported (other formats do not support 3D):



Exported datasets also must be in .tab format.

Advanced Complex Material Properties and Models

This chapter shows how HFSS can incorporate complex, anisotropic, and dispersive material properties. We provide examples showing two common dispersive models to represent high frequency metal behavior and superconductive material behavior well beneath the critical temperature. The two models are the "Drude Model For Dispersive Metals" on the next page for dispersive metals at very high frequencies and the "Generalized Superconductive Model" on page 5-65 used to represent the electrical properties of superconductive materials well below the critical temperature. Both cases use frequency dependent project variables to provide the model.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Complex Material Properties

Complex material properties have been traditionally entered as either of the following: real relative permittivity, real relative permeability, electric/magnetic loss tangent, and conductivity. Though this is helpful for many industries, there are now many applications that benefit by having an entry of a real and an imaginary relative permittivity and permeability as well. In some applications, it is also helpful to have entry of a complex conductivity. HFSS now allows all of the above entries to aid in a high fidelity model of the material.

With a permittivity definition, the real part represents the phase velocity and propagation characteristics while the imaginary part represents the loss terms:

$$\tilde{\varepsilon}_r = \varepsilon_{re} - i\varepsilon_{im}$$

This expression can be expanded by adding the loss effects of metals via the conductivity:

$$\tilde{\varepsilon}_r = \varepsilon_{re} - i\varepsilon_{im} - i\frac{\sigma}{\omega\varepsilon_0} = \varepsilon_{re} - i\left(\varepsilon_{im} + \frac{\sigma}{\omega\varepsilon_0}\right)$$

The above equation seems to be the most representative for complex relative permittivity. However, on occasion it can be useful to think in terms of complex conductivity rather than complex relative permittivity. An example is the Drude Model for dispersive metals at high frequencies.

Drude Model For Dispersive Metals

The Drude Model represents the high frequency electrical behavior of metals as a function of DC conductivity and the Plasma Frequency. This model is typically represented as a complex conductivity:

$$\widetilde{\sigma(\omega)} = \sigma_{re}(\omega) - i\sigma_{im}(\omega)$$

With a conductivity definition the real term represents the classic conductive loss while the imaginary part characterizes the phase velocity and propagation. This is a bit counter intuitive, but as the Helmholtz Equation accepts a single complex quantity to represent the phase velocity and loss, it follows that how we compartmentalize the terms simply aids in understanding.

σ_0, and the mean collision time, τ

$$\sigma_{re}(\omega) = \frac{\sigma_0}{1+\omega^2\tau^2} \qquad \sigma_{im}(\omega) = \frac{\sigma_0\omega\tau}{1+\omega^2\tau^2}$$

ω_p

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\tau = \frac{\sigma_0}{\varepsilon_0 {\omega_p}^2}$$

To enter these relations into HFSS you must create frequency dependent Project Variables. All Project Variables in HFSS are scoped such that they are visible to every design in the AEDT project and are prefixed with a "$" symbol. To create the project variables, go to **Project > Project Variables...** to see a list of existing project variables. Click **Add** on the project variable list for a dialog to enter names and values for each variable you need. The values here assume that the plasma frequency and mean collision time is known. When you have added the variables, the Project variable list will look like this:

	Name	Value	Unit
	$plasma_freq	1.367e+016	
	$tau	1/(4*10^13)	
	$cond_DC	$plasma_freq^2*e0*$tau	
	$cond_re	$cond_DC/(1+(2*pi*Freq)^2*$tau^2)	
	$cond_im	$cond_DC*(2*pi*Freq)*$tau/(1+(2*pi*Freq*$tau)^2)	

You can see above that the Project variables can be made a function of frequency using the built-in variable, Freq. This frequency variable automatically adapts the variable value based upon the frequency of solution.

To show the use of the complex conductivity entry, we will provide a comparison to published literature for a wide angle InfraRed Absorber FSS[1]. In the cited paper, the material properties of the FSS are given in terms of the plasma frequency, 1.367 * 1016 radians per second, and the damping constant, Γ=4 * 1013 radians per second. It is of note that the mean collision time is equal to $1/\Gamma$ for entry. We entered these values as above to create Project variables and then created a material in HFSS that with a complex conductivity assigned.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

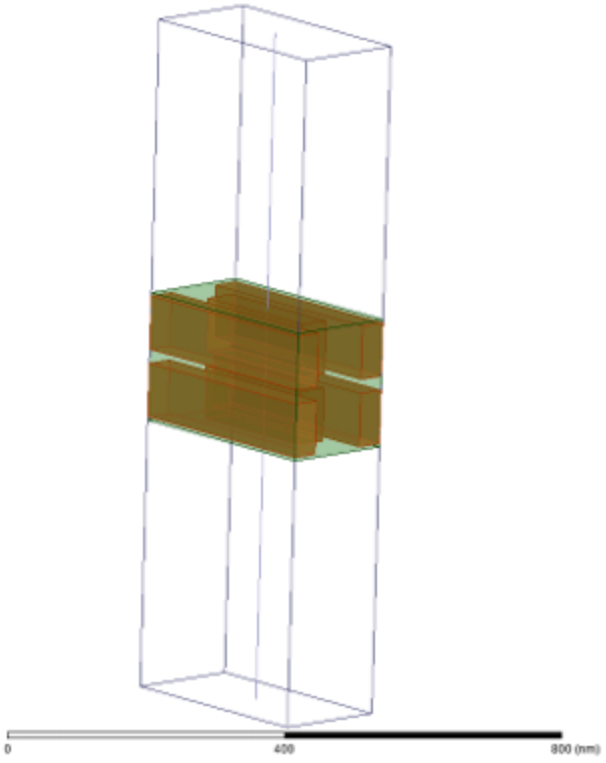
View / Edit Material

Material Name: Au_complex_cond

Properties of the Material

	Name	Type	Value
	Relative Permittivity	Simple	1
	Relative Permeability	Simple	1
	Bulk Conductivity	Simple	$cond_re-1i*$cond_im
	Dielectric Loss Tangent	Simple	0
	Magnetic Loss Tangent	Simple	0

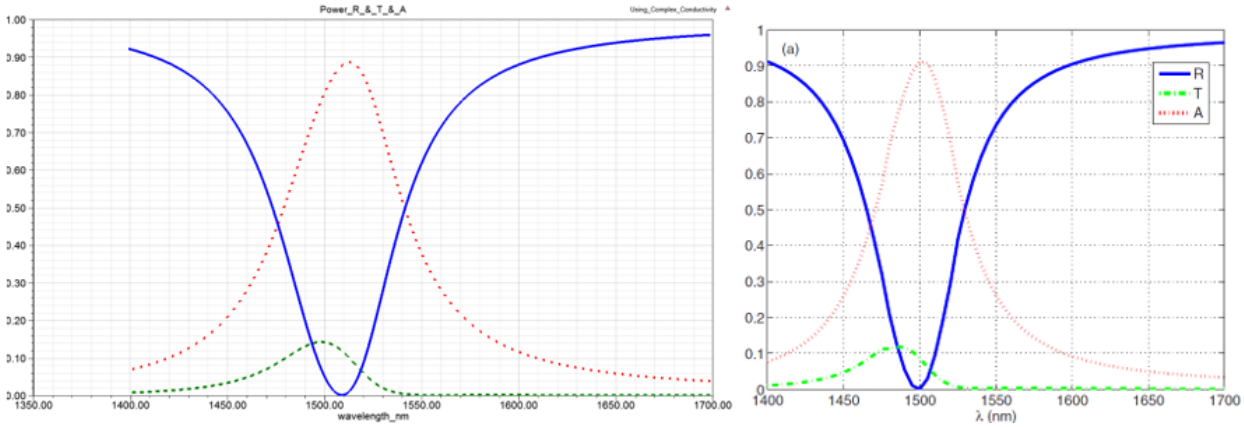
As you can see seen above, the Bulk Conductivity can be entered as a complex value with real and imaginary parts. In HFSS, to designate the imaginary part, multiply it by 1i, as shown above. The FSS looks as below with a substrate permittivity of 2.25:



When solving this FSS with the complex conductivities in the short-wave-IR, from 1.35 um to 1.7 um, we get the traces shown below: Absorption (red dash), Reflection (blue), and Transmission

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(Green dash). The plot on the left is HFSS with the complex conductivity definition and the plot on the right is from the cited paper.



Generalized Superconductive Model

The generalized superconductive model treats a superconductor as a combination of classical electrons and Cooper pair, or superconducting, electrons. The classical electrons behave as in the Drude model contributing to loss while the Superconductive electrons behave entirely differently. The common model used to describe the dispersive electrical effects of a superconductor derives itself from this '2 body fluid' model discriminating the electrons as such.

This model also represents the electrical effects of a superconductor with a complex conductivity that is a function of temperature and frequency. If the temperature is above the critical temperature, T_c, it will behave as the classical conductor, whether a semiconductor or metal. If the temperature is equal to or below the critical temperature, the behavior changes markedly to demonstrate many of the classical superconducting effects. For the 2 fluid superconductor model, the complex conductivity is as shown below:

$$\sigma_{re} = \frac{n_n}{n_s}\frac{\tau}{\Lambda}\left[\frac{1}{1+(\omega\tau)^2}\right] \qquad \sigma_{im} = \frac{1}{\Lambda\omega}\left[1+\frac{n_n}{n_s}\frac{(\omega\tau)^2}{1+(\omega\tau)^2}\right]$$

Note that the n_n/n_s ratio is the ratio of classical electrons to superconductive electrons and is a function of temperature:

$$\frac{n_n}{n_s} = \left[\frac{T}{T_c}\right]^4$$

Λ is a function of the London Penetration Depth, λ_L

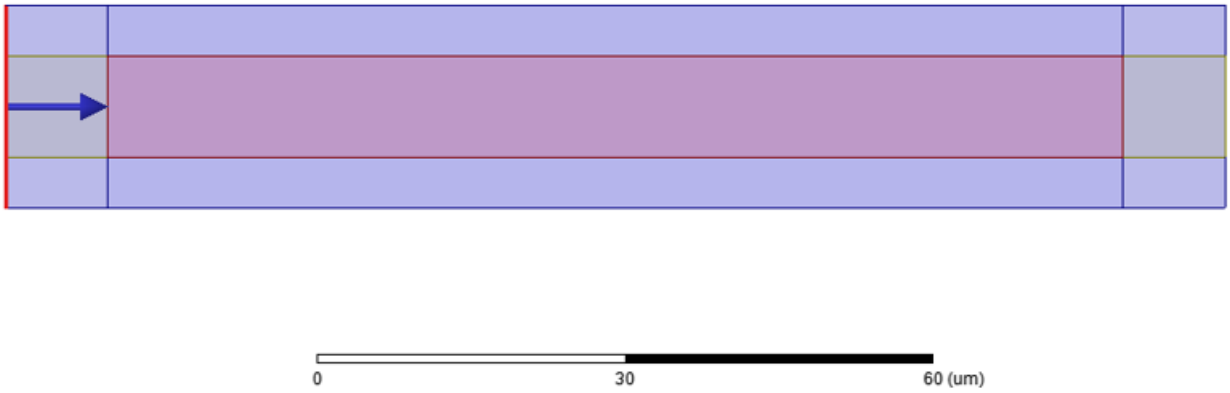
$$\Lambda = \mu_0 {\lambda_L}^2$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entry into HFSS just as we did for the Drude Model, using frequency dependent Project Variables as shown below:

Name	Value	Unit	Evaluated Value	Description
$Temp	4.2		4.2	Operating Temperature (K)
$Tc	9.3		9.3	Critical Temperature (K)
$Nn_Ns_ratio	($Temp/$Tc)^4		0.041597321555222	Ratio of Classical Electron density to Cooper Pair density
$LP_depth	5.4e-008		5.4e-008	London Penetration Depth (m)
$Lambda	u0*$LP_depth^2		3.6643536711471e-021	
$Tau	1e-012		1e-012	Mean electron collision time for Classical Conductor (sec)
$perm_n2o5	8.1834075649147		8.1834075649147	
$Re_Cond	Nn_Ns_ratio*Tau/$Lambda*(1/(1+(2*pi*Freq*$Tau)^2))		*****	Real Part of Conductivity
$Im_Cond	1/($Lambda*2*pi*Freq)*(1+$Nn_Ns_ratio*(2*pi*Freq*$Tau)^2/(1+(2*pi*Freq*$Tau)^2))		*****	Imaginary Part of Conductivity

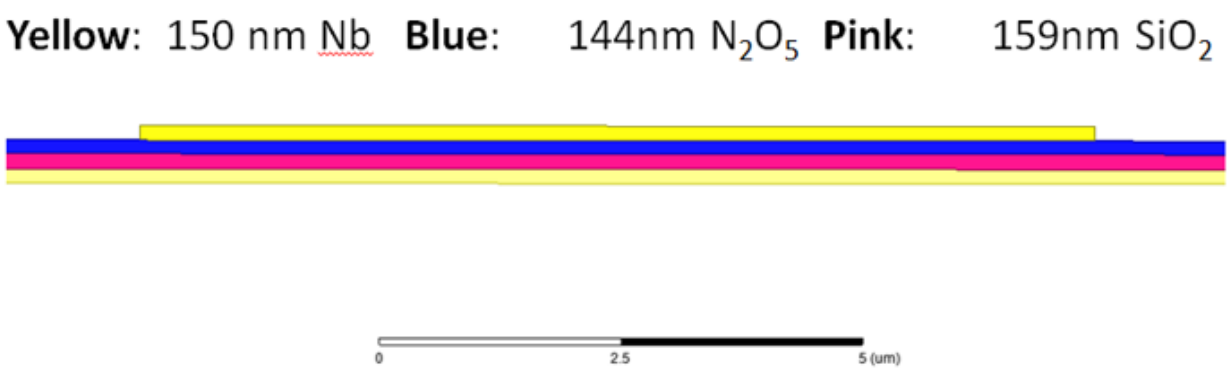
To demonstrate the accuracy of this model in HFSS, we comparison the literature for a Superconducting microstrip for the extraction of the propagation properties.[2] A material was created representing the superconductor Nb in HFSS using a complex conductivity definition similar to the Drude Model. The temperature, T, of the Nb under test was 9.3K with a critical temperature, T_c, of 4.2K. The mean collision time was given as $\tau = 1x10^{-12}$ sec.



The microstrip is simulated by having a given length of line, in this case 100 um, and extracting the propagation constant from the phase of the insertion loss and the line length at 300GHz. Note that the port solver does not yet recognize complex conductivities so a PEC is used for the ground plane and trace at the waveport and de-embedded to the Superconducting Microstrip.

λ_L = 54nm. The substrate is composed of 159 nm SIO_2 under 144nm N_2O_5. The electrical properties used were: $\varepsilon_{r,SIO2}$ =6.25 & $\varepsilon_{r,N2O5}$

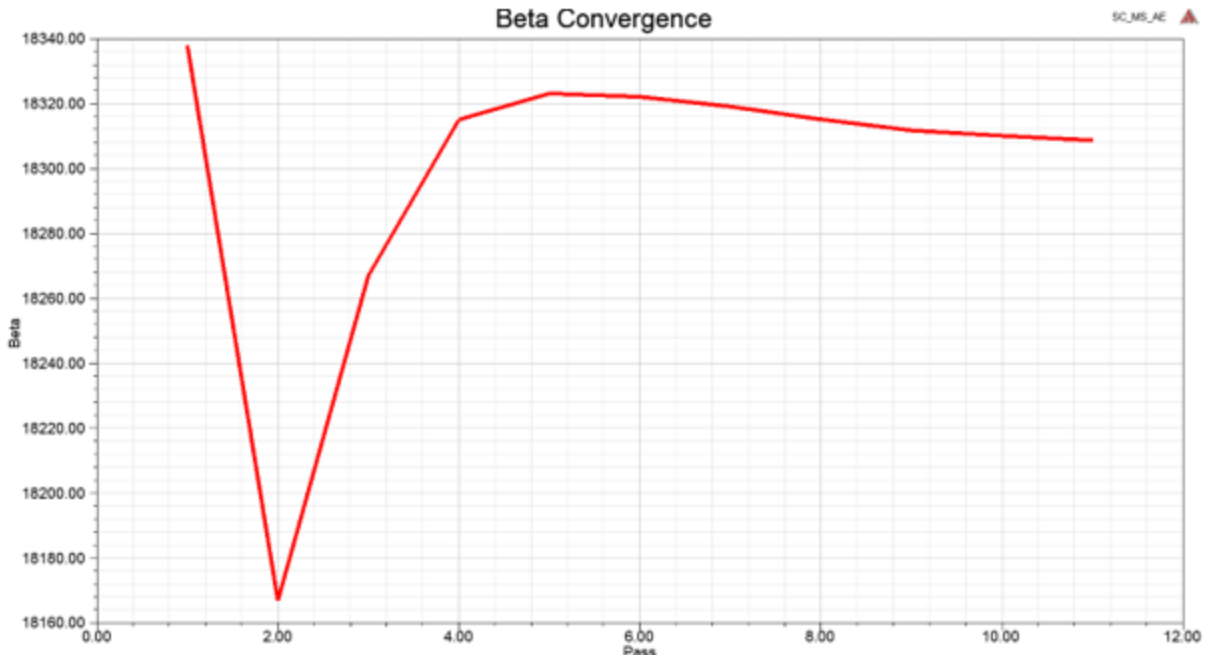
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With these dimensions the reference shows the following results for both a PEC signal and ground plane and the superconducting Nb trace and ground plane.

Model	Z_0 [Ω]	β	ε_{eff}	L_{sq} [pH]	C_{sh} [fF]
Theoretical values	4.7	18339	8.51	0.46	0.207
Perfect conductor model (HFSS)	3.82	15949	6.43	0.3232	0.215

β for an automated accurate solution. HFSS converged to 18308.5 rad/sec with a 0.01% convergence criterion on β



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

References

1) Avitzour, Yoav, et al. Wide-angle Infrared Absorber Based on a Negative-Index Plasmonic Metamaterial. *Physical Review* B 79, 045131. 2009.

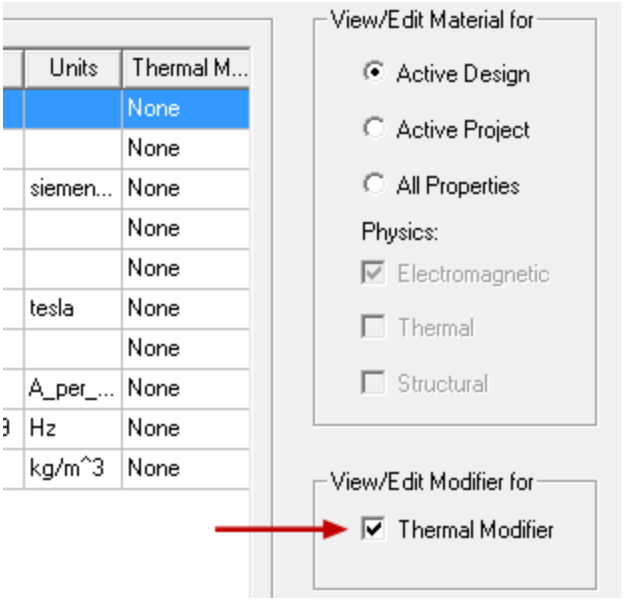
2) Rafique, Raihan. *Superconducting High Speed Passive Interconnects*. A Dissertation from Chalmers University of technology, 2006

Specifying Thermal Modifiers

Thermal modifiers are multiplicative in nature and assigning them introduces spatial dependence. Because you cannot have spatial dependence and causality, Ansys Electronics Desktop ignores Auto Causal Materials options when a thermal modifier exists.

To specify thermal modifiers for a material:

1. In the **View/Edit Material** dialog box, you must enable the **Thermal Modifier** check box:

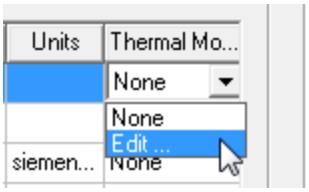


This option causes the properties table to expand to include a Thermal Modifier column.

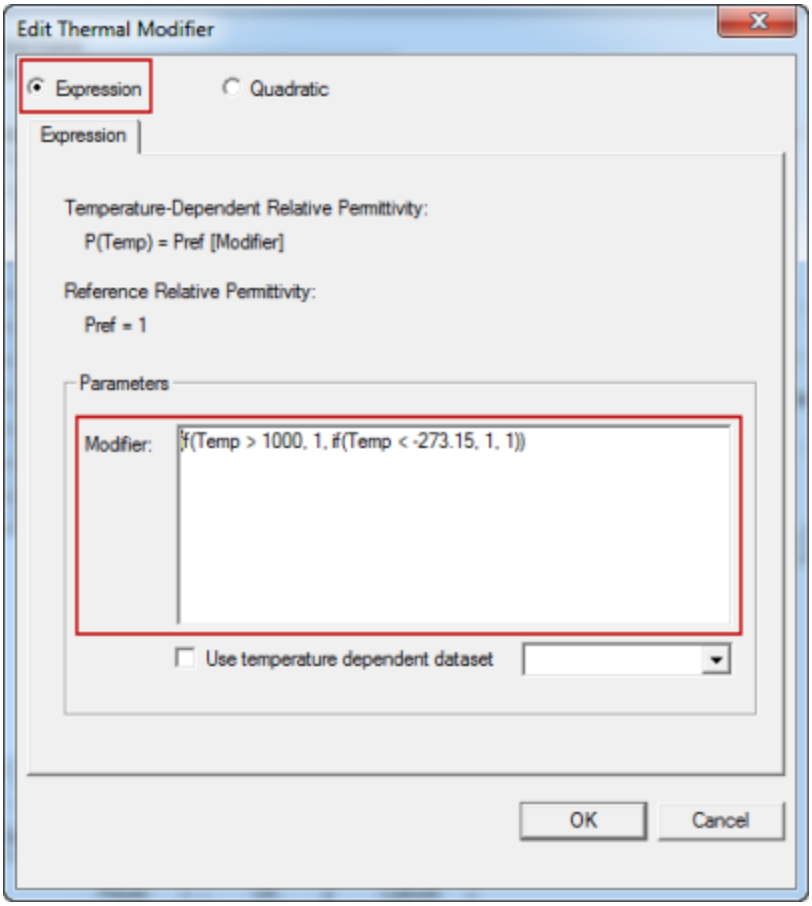
By default, the **Thermal Modifier** option is set to **None** for all material properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click a **Thermal Modifier** cell and select **Edit...** from the drop-down menu:



The **Edit Thermal Modifier** dialog box appears with the **Expression** option selected.

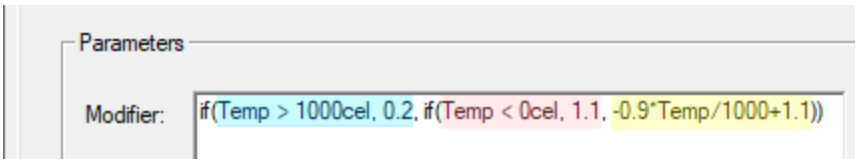


With this option selected, the **Modifier** text box is displayed under **Parameters**. This text box contains a conditional expression template, which you can use as a starting point for creating your own expression. The property for which the multiplier is being defined and its nominal value are indicated above the **Parameters** section.

- With **Expression** selected, you can write an equation for a thermal modifier in the **Modifier** text box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

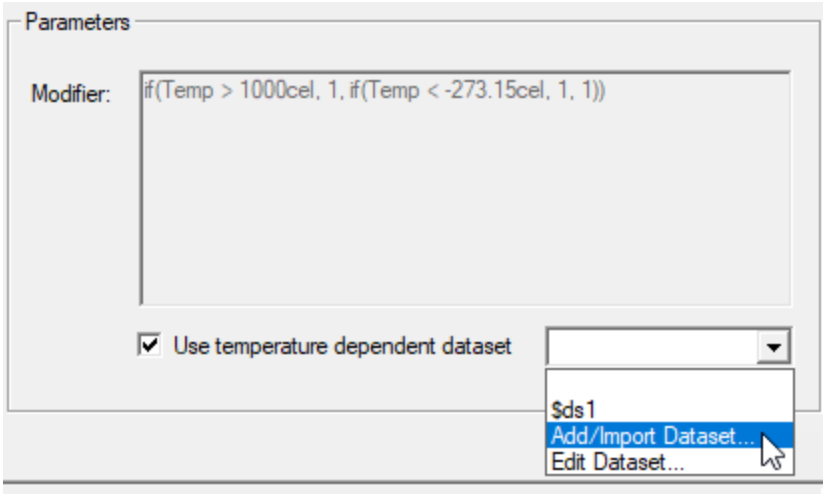
The following example defines a multiplier that varies linearly from 1.1 (at 0° C) to 0.2 (at 1000° C). The multiplier is constant at 1.1 for temperatures below 0° C and constant at 0.2 for temperature above 1000° C:



In this example, the linear equation **-0.9 * Temp / 1000 + 1.1** (highlighted in yellow) defines the multiplier for the range 0° C ≤ Temp ≤ 1000° C.

The default unit of temperature in thermal modifier expressions is Celsius (cel). However, you can override the units by appending the appropriate abbreviation to the numbers in the expression (mkel, ckel, dkel, kel, cel, rank, fah). Model temperatures, material properties, and thermal modifiers need not have consistent units; values are converted automatically as needed.

- Check **Use temperature dependent data set** to disable the Modifier text box. You can then use the drop-down menu at the bottom-right corner of the dialog box to select an existing project dataset or to create a new one by choosing **Add/Import Dataset**.



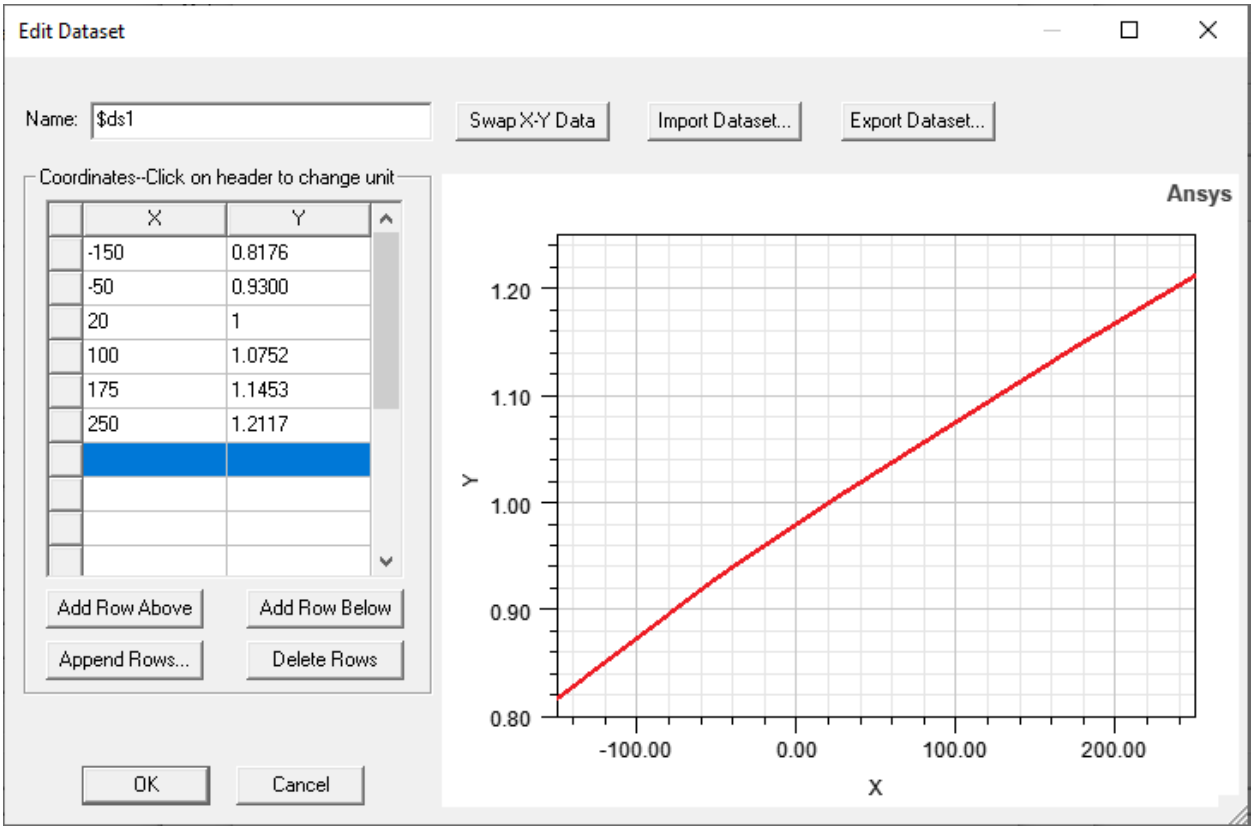
This command lets you define the thermal modifier dataset (a set of X and Y values). Alternatively, if a suitable project dataset already exists, it will be available for selection from the same drop-down menu.

When creating a new dataset, the **X** values are temperatures, and the **Y** values are the corresponding thermal modifiers. The value of the nominal material property is

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

multiplied by this thermal modifier to determine the resultant property at a given temperature. The modifier is interpolated linearly between specified data points.

The following example is a dataset defining thermal multipliers for the thermal expansion coefficient (TEC) of an aluminum-beryllium alloy. The temperature range is -150 to 250° C, and the nominal value (corresponding to 20° C) is 1.37e-5/°C:



Based on these multipliers, the resultant TEC at -150° C is approximately 1.12e-5 (= 1.37e-5 * 0.8176), and the TEC at 250° C is 1.66e-5 (= 1.37e-5 * 1.2117).

In addition to filling the table of values manually, you can also **Import** an existing tab-delimited file to create the dataset.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

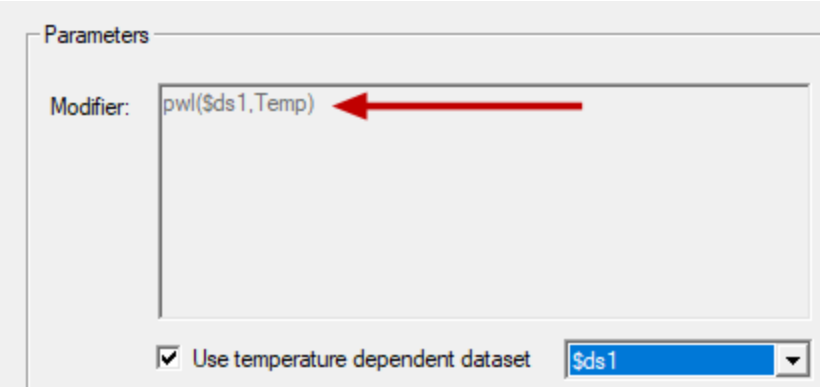
Important:

When you add a dataset from the **Edit Thermal Modifier** dialog box, you cannot specify the temperature units. The numbers in the **X** column will be interpreted as degrees Celcius and cannot be changed. Also, you cannot append a unit abbreviation (such as "fah" or "cel") to the numbers. You must enter temperature values in degrees Celcius for correct results. The values in the **Y** column are multipliers and therefore dimensionless.

Alternatively, you can create a dataset based on any desired temperature units before defining your temperature-dependent material properties. To do so, click **Project> Datasets** from the menu bar and then choose **Add**. If your project includes a design type that supports 3D datasets, a submenu appears offering a choice of a 1D or 3D dataset. If you see this choice, click **1D**. In either case, the **Add Dataset** dialog box appears.

For the dataset to be used later as a thermal modifier, you must click the **X** column header and choose **Temperature**. Then, click the **X** column header again and choose the desired temperature units. Finally, populate the table with your temperature and multiplier values and then click **OK** and **Done**. The dataset will now be available for selection from the drop-down menu in the **Edit Thermal Modifier** dialog box.

After choosing or creating a dataset, it appears in the **Modifier** field of the **Edit Thermal Modifier** dialog box, as shown in the following image:



The syntax of the dataset expression is explained as follows:

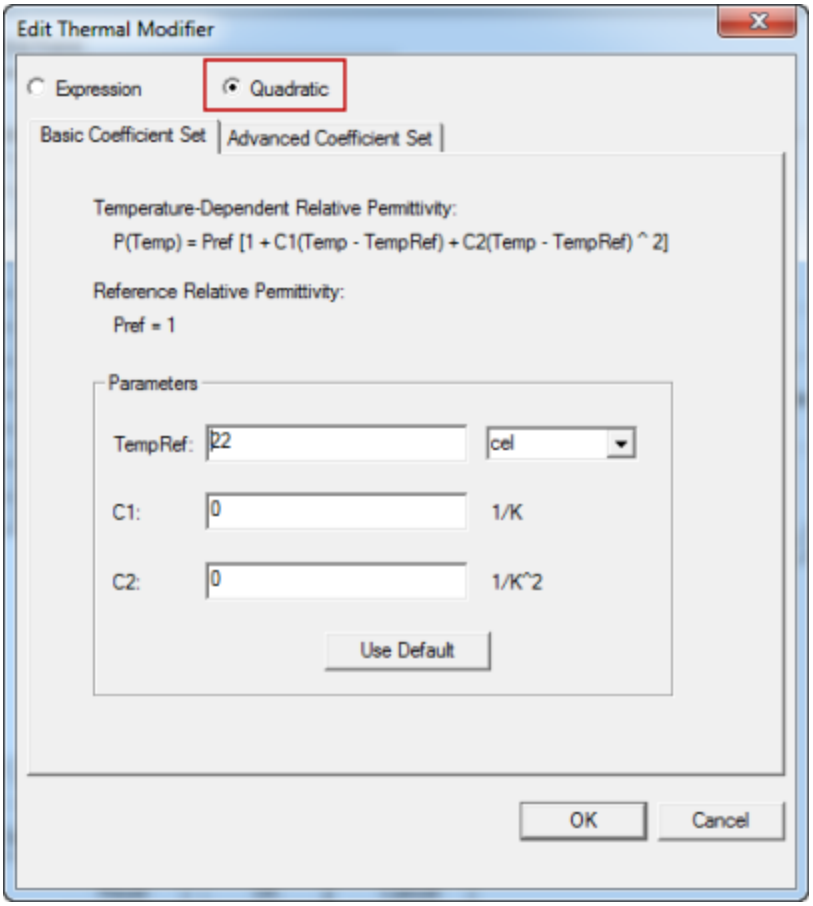
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **pwl:** Type of dataset – Piecewise linear
- **$ds1:** Name of an existing dataset – "$" indicates a *project* dataset, which is a requirement for thermal modifiers
- **Temp:** Type of X value – Temperatures

  The Y values in this case are simple multipliers and therefore unitless.

> **Note:**
>
> To edit an existing dataset, select **Edit Dataset** from the drop-down menu in the **Edit Thermal Modifier** dialog box. The **Datasets** dialog box appears. Select the dataset to modify and click **Edit**.

3. Alternatively, select the **Quadratic** option to display the tabs for **Basic Coefficient Set** and **Advanced Coefficient Set** (as shown below):

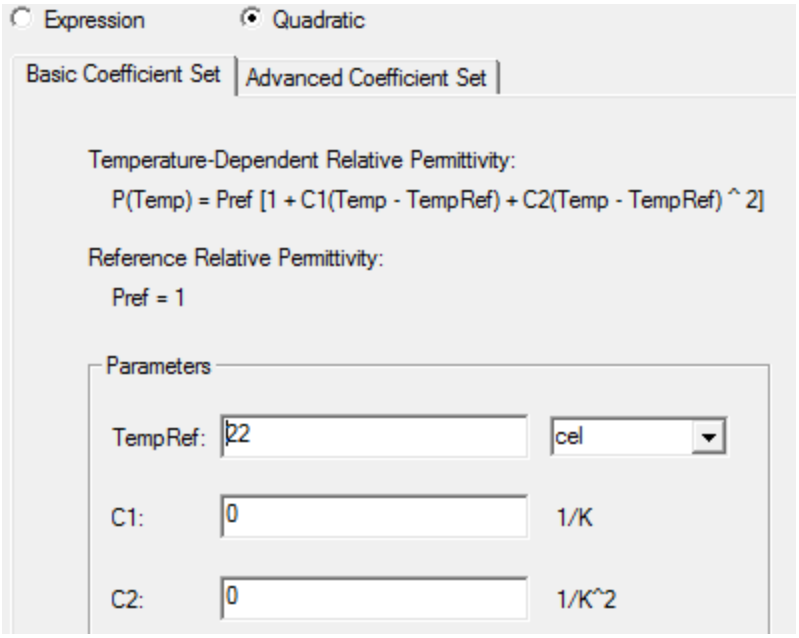


- Under the **Basic Coefficient Set** tab, you can edit fields for the TempRef and units, and fields for C1 and C2 for the following equation:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

P(Temp) = Pref[1+ C1(Temp - TempRef) + C2(Temp - TempRef)^2]

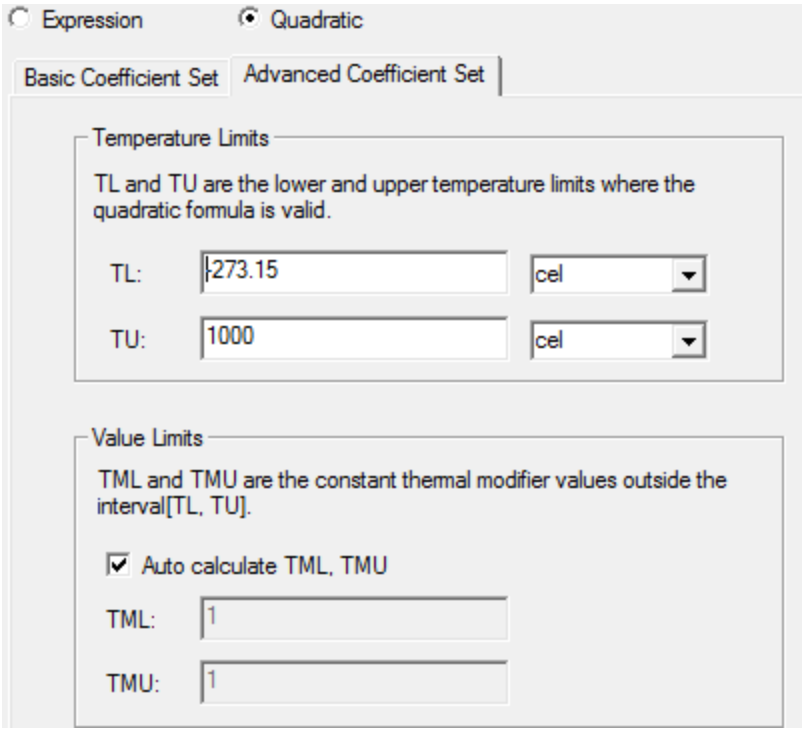
where TempRef is 22 cel by default and where the Pref is defined as the reference relative permittivity.



> **Note:**
>
> The coefficients, C1 and C2, should be negative to yield physical results.

- Under the **Advanced Coefficient Set** tab, you can edit fields for lower and upper temperature limits (TL and TU, respectively) and select their units from the drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also edit the constant value limit for the thermal modifier values outside the limits. By default, these are automatically calculated. Uncheck the Auto Calculate TML and TMU to specify new values for thermal modifier lower (TML) and thermal modifier upper (TMU).

4. Click **OK** to accept your edits and return to the View/Edit materials dialog box.

Defining Material Properties as Expressions

When defining or modifying a material's properties, each material property value in the **View/Edit Material** window can be assigned a mathematical expression. Simply type the expression in the appropriate **Value** box. Expressions typically contain intrinsic functions, such as sin(x), and arithmetic operators, such as +, -, *, and /, but do not include project variables.

> **Note:**
>
> By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed. To view the complete table of properties, see Filtering Materials.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

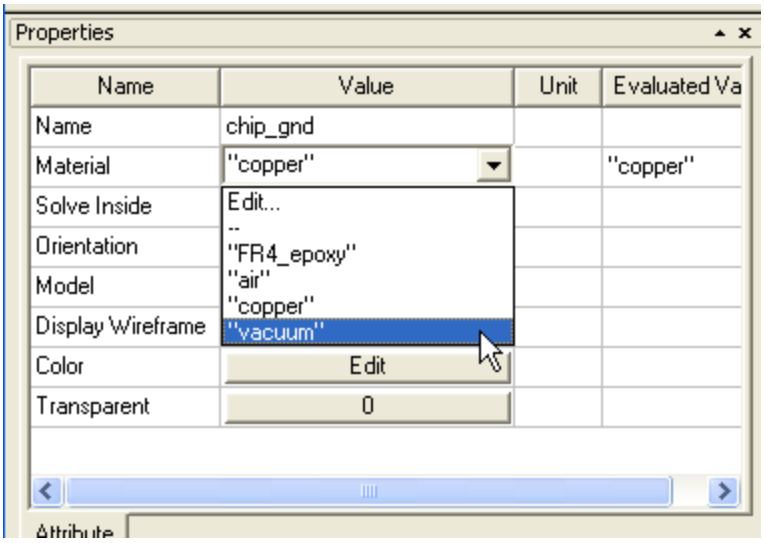
Defining Functional Material Properties

Any material property that can be specified by entering a constant can also be specified using a mathematical function. This is useful when you are defining a material property whose value is given by a mathematical relationship — for instance, one relating it to frequency or another property's value. When defining or modifying a material's properties, simply type the name of the function in the appropriate **Value** box.

> **Note:**
>
> By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed. To view the complete table of properties, see Filtering Materials.

Assigning Materials from the Object Properties Window

The Properties dialog for each object includes a materials property. If you click on the current material property you see a drop down list that includes an *Edit* command and a list of materials in the current project. You can select from the list of current materials to assign the selected material to that object.

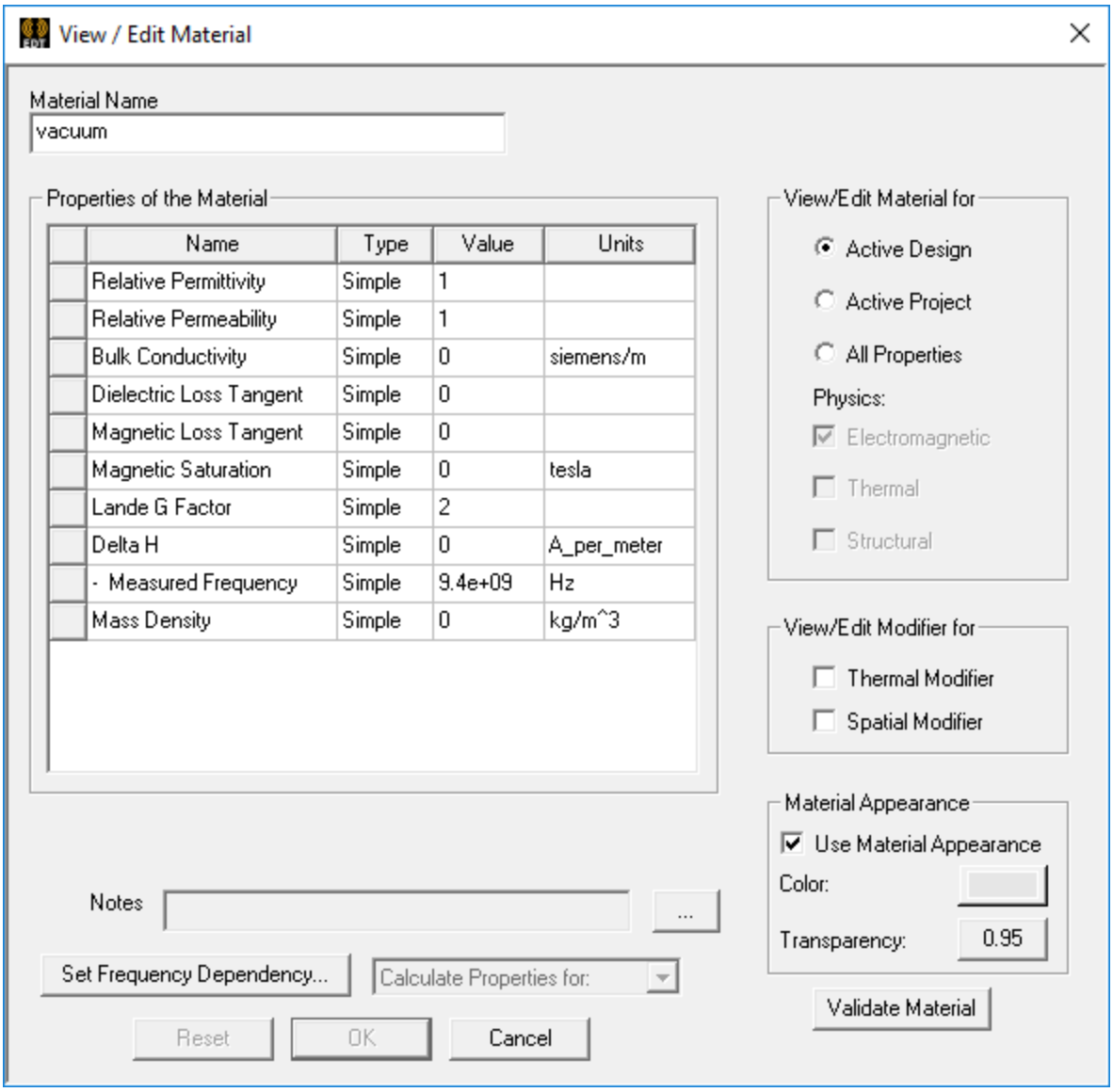


Viewing and Modifying Material Attributes

1. From the **Select Definition** window, select a material to view or modify, and click **View/Edit Materials**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **View/Edit Material** window appears. The material name and its property values are listed.



Note:

By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed.

To view the complete table of properties, see Filtering Materials.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Under **View/Edit Material for**, select:
   - **Active Design** to display properties used in the active design.
   - **Active Project** to display properties used in the active project.
   - **All Properties** to display all properties available. This enlarges the table of properties to show all properties possible. You can use the scroll bars or size the dialog to see all properties. When **All Properties** is selected, the following **Physics** classification options are enabled to show or hide properties based on simulation type:
     - **Electromagnetic**
     - **Thermal**
     - **Structural**

   > **Note:**
   >
   > If a material is edited in a design type for which the **Physics** type has not been set (e.g., an HFSS design but Electromagnetic physics type was not set), the **Physics** type will be automatically set in the material.

3. You can modify the material as follows:
   a. Provide a new name for the material in the **Material Name** text box.
   b. Under **Type**, specify whether a material property is **Simple** , **Anisotropic**, **Vector** and **Vector Mag**, or for Relative Permeability, **Non-linear**, as required for that property.

      For **Simple**, you provide a value or variable.

      For **Anisotropic**, you provide tensor values.

      For **Vector**, you provide a **Vector Mag**.

      For **Non-Linear**, you provide a Data Set.

   c. Provide new material property values in the **Value** boxes.
   d. Change the units for a material property.
   e. Enable the **Use Material Appearance** check box to specify a color and transparency.

   > **Note:**
   >
   > Materials stored in the global material library cannot be modified.

4. If you want to add descriptive notes for the new material, click the ellipsis button [...] to the right of the **Notes** field. This opens a dialog box in which you can enter text. Pressing

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enter or clicking **OK** saves the note. To enter multiple lines of notes, use **Ctrl+Enter** at the start of each new line.

5. Click **OK** on the **View/Edit Materials** dialog to add the new material to the material library.
6. Click **OK** to save the changes and return to the **Select Definition** window.

> **Warning:**
>
> If you modify a material that is assigned in the active project after generating a solution, the solution will be invalid.

Validating Materials

The Ansys Electronics Desktop can validate a material's property parameters for an Ansys EM software product. For example, the software will check if the range of values specified for each material property is reasonable.

If a material's property parameters are invalid, an error message will appear in the lower-right corner of the **View/Edit Material** window. If the parameters are valid, a green check mark will appear there.

To validate the material attributes listed in the **View/Edit Material** window:

- Select **Active Design**, **Active Project**, or **All Properties**, and then click **Validate Material**.

Copying Materials

1. In the **Select Definition** window, select the material you want to copy, and then click **Clone Material** or right-click the selected material and select **Clone** from the shortcut menu.
2. To modify the material's attributes, follow the directions for modifying materials.
3. Click **OK** to save the copy in the active project's material library.

Removing Materials

1. In the **Select Definition** window, select one or more materials you want to remove from the active project's material library.
2. Click the **Remove Material (s)** button or right-click the selected material and click **Remove** on the short-cut menu.

The material is deleted from the project material library.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The following materials cannot be deleted:

- Materials stored in Ansoft's global material library.
- Materials that have been assigned to objects in the active project.

In a project library, you may want to use the **Tools** > **Project Tools** > **Remove Unused Definitions** command to remove selected materials definitions that your project does not require.

Exporting Materials to a Library

1. In the **Select Definition** window, select the material you want to export.
2. Click the **Export Material to Library** button, or right-click the selected material and click **Export** from the short cut menu.

   The **Export to material library** file browser appears.
3. Click **PersonalLib** to export the material to a local project directory, accessible only to the user that created it.

   Click **UserLib** to export the material to a library that is shared by more than one user, usually in a central location.
4. Type the library's file name and then click **Save**.

Sorting Materials

You can change the order of the materials listed in the **Select Definition** window. You can sort the list of materials by name, library location, color, or material property value. To change the order of the listed materials:

- Click the column heading by which you want to order the materials.

If the arrow in the column heading points up, the material data will be listed in ascending order (1 to 9, A to Z) based on the values in the column you chose. If you want the material data to be listed in descending order (9 to 1, Z to A), click the column heading again. The arrow will point down.

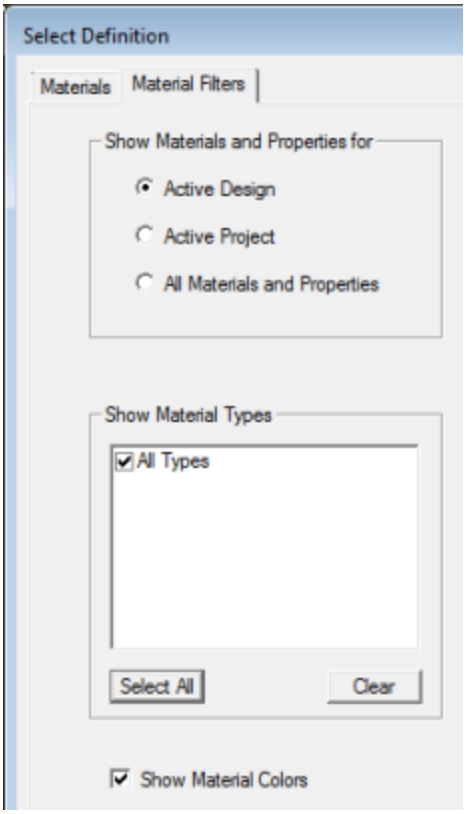
Note:

By default, not all of the available properties are displayed in the materials table. Only the properties commonly used by the product are displayed. To view the complete table of properties, see Filtering Materials.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Filtering Materials

If you want to remove certain materials or material properties from the list in the **Select Definition** window, use the filter options under the **Material Filters** tab. You can filter out materials based on the product or library with which they are associated. You can also filter out material properties and types of material properties. And you can remove the filtering in order to see all available material properties.



To filter materials or material properties listed in the **Select Definition** window, using the choices in the **Materials** tab:

1. The text field under Libraries lists the libraries for the project. Selecting the listed library highlights it and cause the table to display the materials in that library.
2. Above the Libraries area, you can check or uncheck boxes to show or hide Project Definitions and All Libraries.

- With both unchecked, nothing appears in the materials table. With both checked, the table shows all materials and highlights those used in the project.
- With only Project Definitions checked, the materials table shows only the materials used in the project.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- With All Libraries checked, the table displays all materials, but may not show all available properties.

To filter out or show additional materials/properties in the **Materials** tab:

1. Click the **Material Filters** tab.
2. Under **Show Materials and Properties for**, select one of the radio buttons:
- **Active Design** to display materials/properties used in the active design.
- **Active Project** to display materials/properties used by the active project.
- **All Materials and Properties** to display all materials and properties available. Selecting this enlarges the table of materials shown under the **Materials** tab to show all materials possible. You can use the scroll bars or size the dialog to see all materials.
3. Under **Show Material Types**, select materials types to display on the **Materials** tab. Click **Select All** to select all of the types listed. Click **Clear** to clear all selections.
4. If you check Show Material colors, the Materials tab of the **Select Definition** window includes a color for material colors. See Viewing and Modifying Material Attributes.
5. Click the **Materials** tab to save your selections.

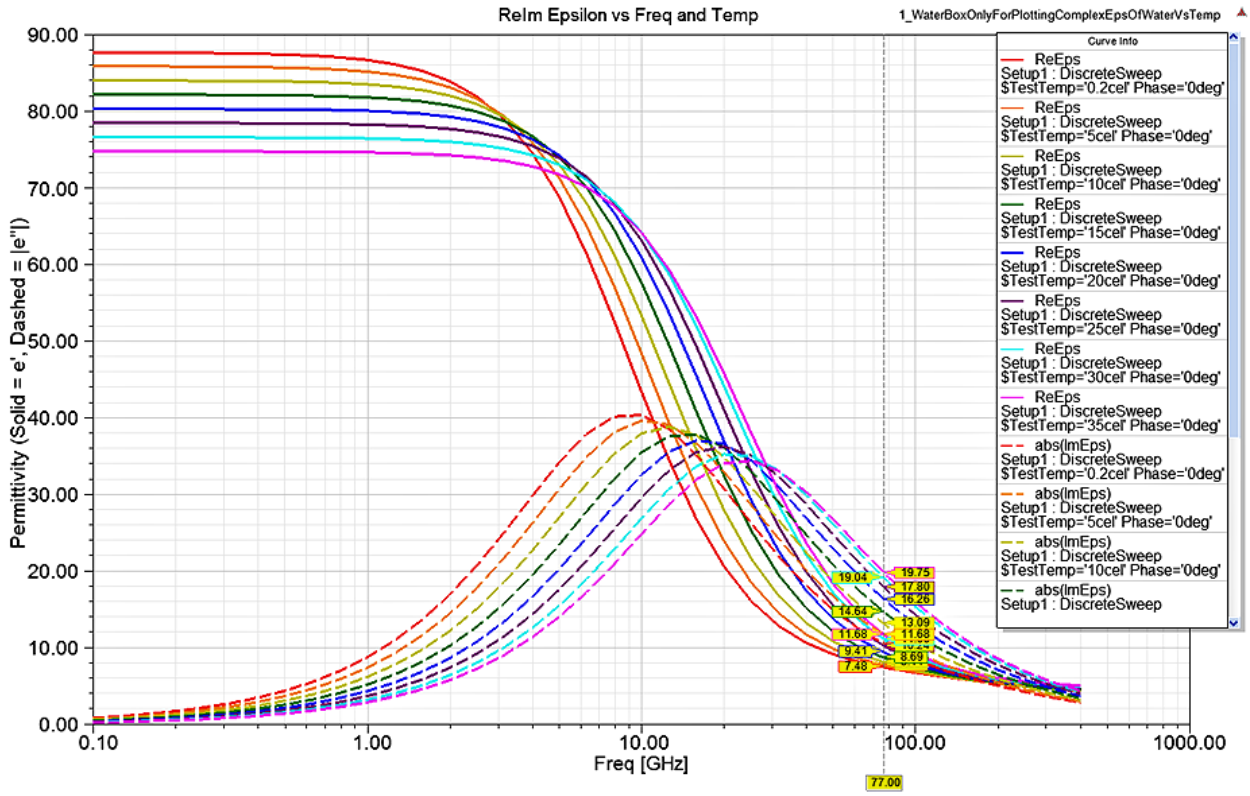
Click **Cancel** to revert back to the last saved selections.

Water (Temperature- and Frequency-Dependent Properties)

Applicability: The temperature-dependent and frequency-dependent water properties are applicable to HFSS, Q3D, and 2D Extractor designs.

The complex permittivity of water changes significantly in the 1 – 100 GHz range. Many HFSS applications involve frequencies within this range and also involve water. Appropriate water properties are needed for accurate electromagnetic modeling. The following image demonstrates the widely varying properties of water versus temperature and frequency:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



These varying properties of water are included in HFSS through the addition of the material, *Water 0.2-35C 0.1-400GHz*.l

Application Example: Companies test automotive radar at 24 GHz or 77 GHz with thin layers of water applied on the target fascia. A thin layer of water may increase both signal reflection and attenuation, significantly affecting the radiation pattern.

A relative permittivity (ϵ_r) of 81 is used as a typical value for water. If you were to assume this value of ϵ_r for the radar example just described, the simulation would produce inaccurate results. To solve this problem, a temperature-dependent and frequency-dependent material library, *Water_0p2-35C_0p1-400GHz*, has been included within the *syslib* folder of the Ansys Electronics Desktop application. Rather than containing multiple versions of water suitable for different temperature and frequency combinations, the complex permittivity is expressed as a function of temperature and frequency in a single material choice, *Water 0.2-35C 0.1-400GHz*.

Application Procedure

As with any material, you can choose the temperature-dependent and frequency-dependent water as the *Default material* or apply it to specific solid objects, as follows:

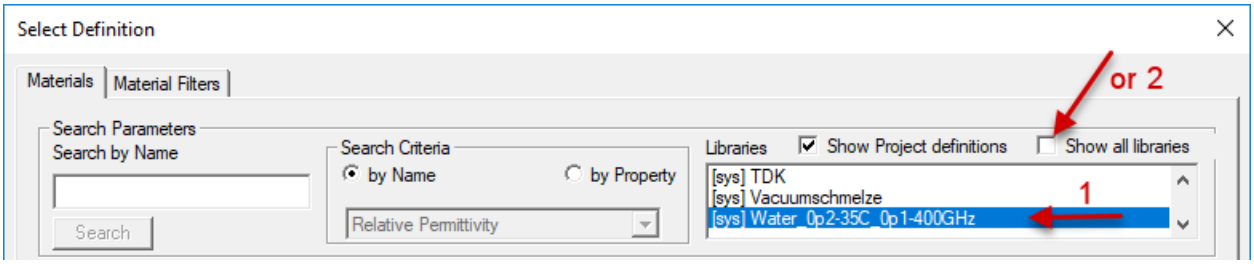
1. To apply the material for all subsequently created objects:
    - On the **Draw** ribbon tab, choose **Select** from the **Default material** drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Alternatively, to apply the material to one or more selected objects, use *one* of the following three methods:
   - From the **Attribute** tab of the *Properties* dialog box or docked *Properties* window, choose **Edit** from the **Material** drop-down menu.
   - Select one or more solid objects in the History Tree. Then, right-click a selected object and choose **Assign Material** from the shortcut menu.
   - Graphically, in *Select Objects* mode, select one or more objects in the Modeler window. Then, right-click in the window and choose **Assign Material**.

   Regardless of whether you use the step 1 method or one of the three methods under step 2, the same *Select Definition* dialog box appears.

3. In the **Libraries** section within the **Materials** tab, take *one* of the following three actions:
   - With **Show all libraries** *not* selected, click the library **[sys] Water_0p2-35C_0p1-400GHz** to select it (A single *SysLibrary* entry will appear in the material list along with any *Project* materials).
   - Select the **Show all libraries** option. (All *SysLibrary* and *Project* materials will be listed in the dialog box.



4. Select the material **Water 0.2-35C 0.1-400GHz**:

Name	Location	Origin	Relative Permittivity	Relat Permea
vacuum	Project	Materials	1	1
vacuum	SysLibrary	Materials	1	1
Water 0.2-35C 0.1-400GHz	SysLibrary	Water_0p2-35C_0p1-400GHz	1	1
water_distilled	Project	Materials	81	0.999991
water_distilled	SysLibrary	Materials	81	0.999991
water_fresh	SysLibrary	Materials	81	0.999991
water_sea	SysLibrary	Materials	81	0.999991
ZEONEX RS420 (tm)	SysLibrary	Materials	2.3	1

5. Click **OK**.

Include Temperature Dependence:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature dependency is an optional feature that is *not* enabled by default. For the subject water properties to work, you must select the *Include Temperature Dependence* option and define each water object's temperature. The additional procedure is as follows:

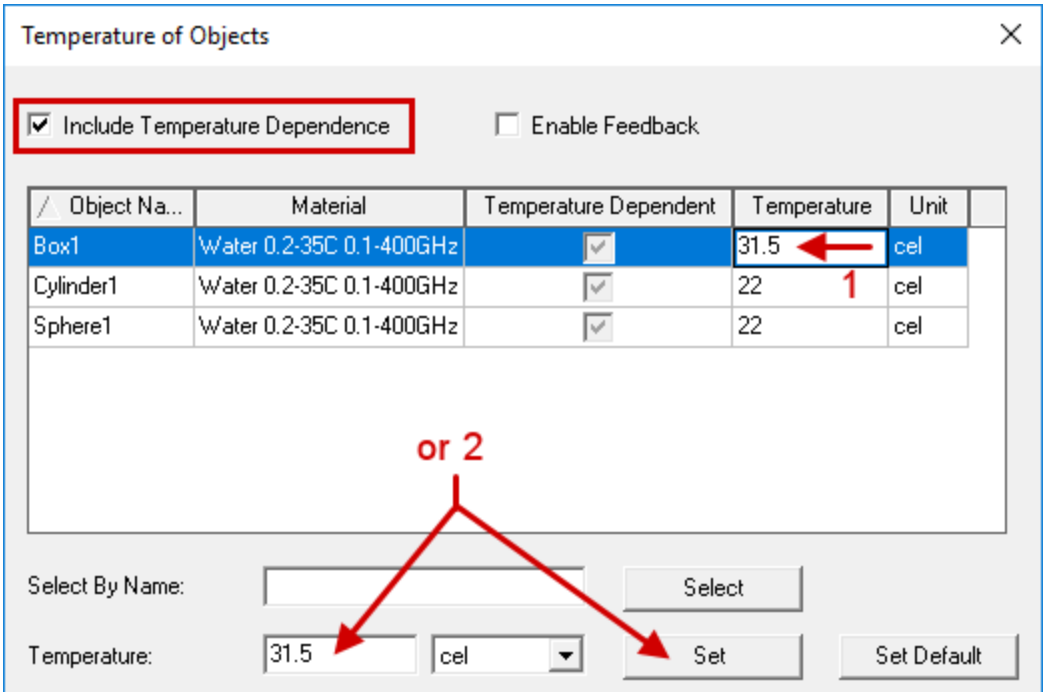
6. Using the menu bar, click one of the following commands, depending on the design type:
   - **HFSS** > **Set Object Temperature**
   - **Q3D Extractor** > **Set Object Temperature**
   - **2D Extractor** > **Set Object Temperature**

   Alternatively, right-click the top-level design entry in the Project Manager (*HFSSDesignx, Q3DDesignx,* or *2DExtractorDesignx*, by default), and choose **Set Object Temperature**.

   The **Temperature of Objects** dialog box appears.

7. Select the **Include Temperature Dependence** option.
8. In the **Temperature** column for all objects using the material *Water 0.2-35C 0.1-400GHz*, specify the object temperature in your preferred units.

   Alternatively, select one or more rows of the table, specify a value in the **Temperature** text box beneath the table, and click **Set**.



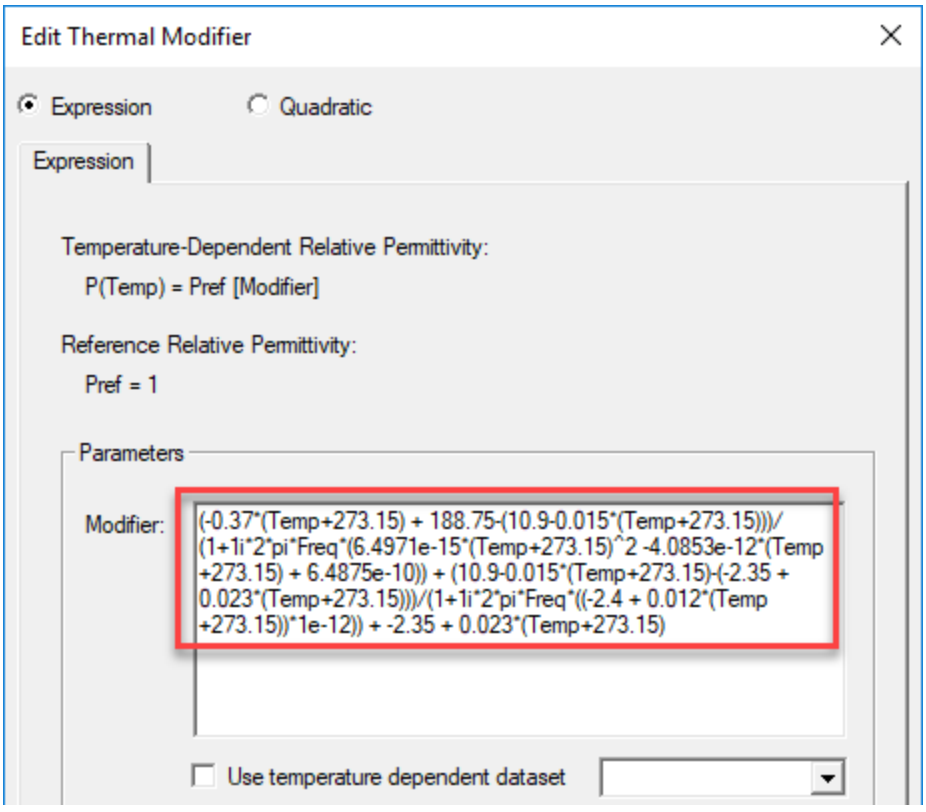
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The internal temperature unit in Ansys Electronics Desktop is Celsius. For any other temperature unit you might specify, the value of *Temp* is automatically converted to Celsius internally prior to the execution of the ϵ_r equation, which ensures that the equation is valid for all temperature units.

9. Click **OK**.

Relative Permittivity Calculation

A nominal value of 1 is specified for the *Relative Permittivity* of the water. However, this value is modified by the equation shown in the *Thermal Modifier* column of the *View / Edit Material* dialog box. While the equation is listed as a "thermal" modifier, it also includes frequency-dependent factors. To access this equation, select the material, as shown in step 4 above, and click **View / Edit Materials**. Then, click the *Thermal Modifier* equation and choose **Edit**:



This equation is comprised of several different factors, each defined by a specific part of the overall expression. The equations for each factor were determined using available empirical data and curve fitting techniques and were built upon previously published work (see the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference section near the bottom of this topic). Let's look at the Relative Permittivity (ϵ_r) equation and the individual parameters that comprise it.

Relative permittivity equation:

$\epsilon_r = [(\varepsilon - \varepsilon_2) / (1 + 1i * \omega * \tau_1^2) + (\varepsilon_2 - \varepsilon_\infty)] / (1 + 1i * \omega * \tau_2) + \varepsilon_\infty$

where ω is the frequency in radians/second (equal to 2π * Freq in Hz), and the other factors are as defined below.

Dielectric relaxation parameters of water:

- **Linear, temperature-dependent parameters:**

  $\varepsilon = -0.37 * T + 188.75$

  $\varepsilon_2 = 10.9 - 0.15 * T$

  $\varepsilon_\infty = -2.35 + 0.23 * T$

  $\tau_2 = (-2.4 + 0.12 * T) * 1E-12$

- **Quadratic, temperature-dependent parameter:**

  $\tau_1 = 6.4971E-15 * T^2 - 4.0853E-12 * T + 6.4875E-10$

For each of the preceding five equations, T is the absolute temperature in degrees Kelvin (°K). Since the internal calculation unit for temperature in Ansys Electronics Desktop is Celsius, 273.15° must be added to *Temp*, and this sum (Temp + 273.15) must be substituted for *T* in each equation. The resultant complete expression for the relative permittivity, formatted appropriately for the software, is as follows:

ϵ_r = (-0.37 * (Temp + 273.15) + 188.75 - (10.9 - 0.015 * (Temp + 273.15))) / (1 + 1i * 2 * pi * Freq * (6.4971E-15 * (Temp + 273.15)^2 - 4.0853E-12 * (Temp + 273.15) + 6.4875E-10)) + (10.9 - 0.015 * (Temp + 273.15) - (-2.35 + 0.023 * (Temp + 273.15))) / (1 + 1i * 2 * pi * Freq * ((-2.4 + 0.012 * (Temp + 273.15)) * 1e-12)) + -2.35 + 0.023 * (Temp + 273.15)

Reference:

4 June 1999, Chemical Physics Letters 306 (1999) 57-63

The dielectric relaxation of water between 0°C and 35°C
R. Buchner, J. Barthel, J. Stauber
Institut für Physikalische und Theoretische Chemie, Universität Regensburg D-93040 Regensburg, Germany

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Working with Material Libraries

There are three different materials libraries in HFSS: a *system library*, a *user library*, and a *personal library*.

The library files that ship with Electronics Desktop are stored under the **syslib** directory. These libraries are intended to be read-only and should not be modified. They are available for any material assignment in any project.

In addition to the system libraries, Electronics Desktop recognizes two user-configurable library structures, called the *User Library* and the *Personal Library*. These are used to add user (or company)-defined materials. Customarily, **userlib** is a network repository for proprietary or corporate definitions available to all seats in an enterprise, while **personalLib** contains project and design-specific libraries as needed by individual designs.

A root library directory is set up at installation. If none is specified, the default is the root Electronics Desktop directory.

Materials from all libraries are available for use in projects.

Editing Libraries

There are two different methods of editing libraries:

- Right-click **Materials** in the project window to display the **Edit Libraries** shortcut menu. Clicking displays the **Edit Libraries** window.

  Editing definitions from the project window does not modify the configured libraries for any particular design, since this is editing in general.

- Using **Tools** > **Edit Libraries** > **Materials** from the menu bar takes the current design into account and adds any new libraries to the configured list for the design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6 - Drawing a Model

After you insert a design into the current project, you can draw a model of the electromagnetic structure. The general strategy is to build the model as a collection of 3D objects. You can assign any single material to each 3D object.

You can create 3D objects by using the modeler's **Draw** commands or you can draw 1D and 2D objects, and then manipulate them to create 3D objects. Objects are drawn in the **3D Modeler** window. You can also import objects from other systems, use the 3D component libraries, the Antenna Design Toolkit, the SBR+ Parametric Antenna feature, as well as the Cable Modeling toolkits.

To open a new **3D Modeler** window, do one of the following:

- Insert a new design into the current project.
- Double-click an Ansys Electronics Desktop design that uses the 3D Modeler (HFSS, Q3D, Maxwell (2D and 3D), or Icepak) in the project tree.

If a **3D Modeler** window for an existing design is not open, do one of the following:

- Click **[solverName]** > **3D Model Editor**.
- Right-click the design name in the project tree, and then click **3D Model Editor** on the shortcut menu.

The model you draw is saved with the current project when you click **File** > **Save**.

> **Note:**
>
> If you access your machine via Remote Desktop, if Ansys Electronics Desktop is running and one or more modeler windows are open, those modeler windows automatically close. The message manager window displays a message indicating that Ansys Electronics Desktop closed the modeler windows.

When working with multiple projects, or when a project has multiple designs, you may have multiple **Modeler** windows available. To switch to the modeler window associated with a specific design:

1. In the Project Manager window, select the **Design** of interest.
2. Click **[solverName]** > **3D Model Editor** to focus the modeling window on the selected design.

If the menu command is unavailable, then the selected design is already in the modeler window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

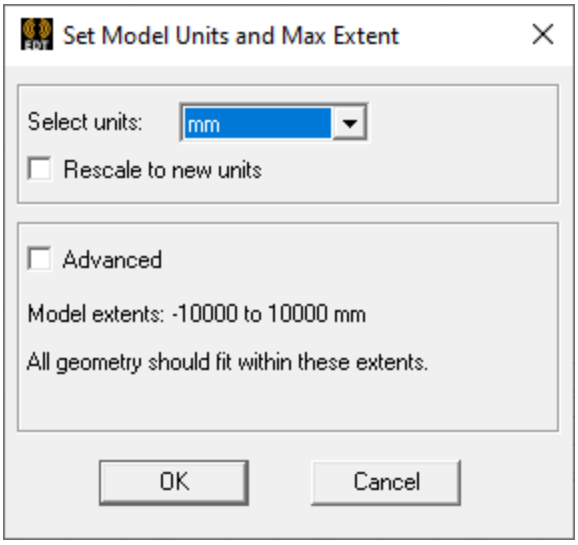
Setting Units of Measurement for the Model

You can specify the units of measurement for drawing geometric models. After the units of measurement have been specified, they are assigned to the objects in the **3D Modeler** window. You can then choose to display the model's dimensions in the new units, or rescale the model's dimensions to the new units.

To set the model's units of measurement:

1. Click **Modeler** > **Units**.

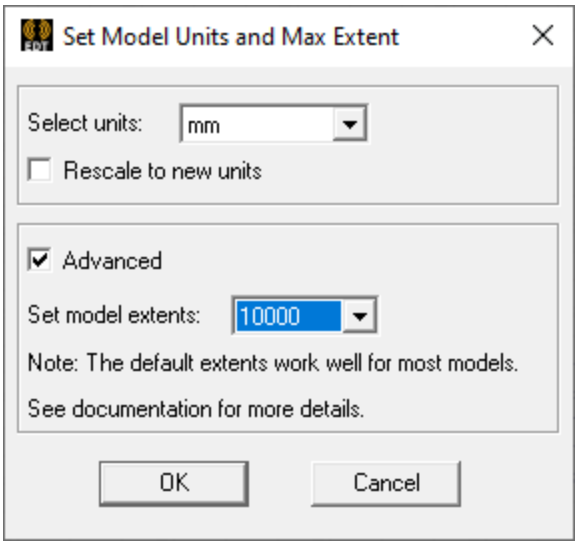
   The **Set Model Units** dialog box appears.



2. Select the new units for the model from the **Select units** drop-down menu.
3. Specify how the change in units affects the model:
   - Select the **Rescale to new units** option to rescale the dimensions to the new units. For example, selecting centimeters (cm) as the new unit of measurement results in a dimension of 10 millimeters (mm) becoming 10 cm.
   - Clear the **Rescale to new units** option (the default) to convert the dimensions to the new units without changing their scale. For example, selecting cm as the new unit of measurement results in a dimension of 10 mm becoming 1 cm.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. While most models should fit the default, you can check Advanced to enable Max model extent (E) to specify model extents in the 3D modeler.



.

Parasolid kernel has a strict size limit of +/- 500 units. As many AEDT models go beyond this limit in the desired units, we have implemented internal scaling to allow creation of models with a larger extent. With our default model extent of 10,000, all geometry will be internally scaled down by a factor of 100 to fit within parasolid size limit. We recommend using the default extent of 10,000 as much as possible. Modeling operations will output an error if model goes beyond 50,000 units.

A second extent option of 100 is available in the advanced options.

If model extent is set to 100, we do not scale geometry internally. Modeling operations will output an error if model goes beyond +/- 500 units. Extent of 100 should be only used in rare situations where it is determined that scaling is causing a failure, and there is no workaround.

5. Click **OK** to apply the new units to the model.

Drawing Objects

You can draw one-, two-, or three-dimensional objects using the **Draw** commands. You can alter objects individually or together to create the geometry of your structure. In the **Tools** > **Modeler Options**, **Drawing** tab, you can set a default to either draw objects directly with the mouse or by invoking a **Properties** dialog in which you can enter the values for the object dimensions. The **Dialog** mode drawing feature works with the equation based curve, equation based surface, and all two and three dimensional objects. You can toggle to **Point** mode via the **F3** function key and to **Dialog** mode via the **F4** function key. When you use the **Dialog** mode for drawing objects the Edit property of new primitives setting is ignored.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

One-dimensional (1D) objects in the modeler include straight line, arc line, center-point arc, and spline segments, or a combination of these - called polylines. One-dimensional objects are open objects; their boundaries do not enclose a region, unless you connect their endpoints. They have length, but no surface or volume. Generally they are used as temporary objects from which to create 2D objects.

Two-dimensional (2D) objects in the modeler include objects such as equation based surfaces, rectangles, ellipses, circles, and regular polygons. Two-dimensional objects are closed sheet objects; their boundaries enclose a region. You can create 2D sheet objects by covering the enclosed region. In many applications (FSS, antennas) it is essential to calculate net power flow through a surface.

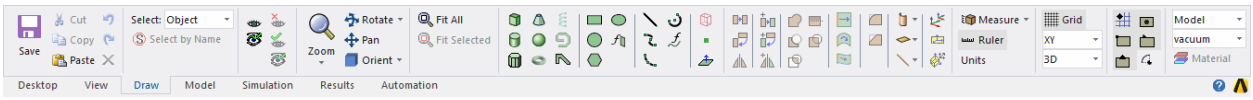
You can also edit the properties of a polyline from the history tree to assign it a Cross Section property as line or rectangular. If you then assign it either a height or a width, the polyline becomes a sheet object.

By default, the history tree organizes sheet objects according to their boundary assignments. To change this, select the **Sheets** icon, and right-click to display the **Group Sheets by Assignment** check box. Within the calculator sheet objects are listed under surface.

Three-dimensional (3D) objects in the modeler include objects such as boxes, cylinders, regular polyhedra, cones, spheres, torii, and helices. These objects have boundaries that enclose a region with volume.

You can create 3D objects by manipulating 2D objects along a plane or by using the appropriate **Draw** commands. You can also edit the properties of a polyline from the history tree to assign it a Cross Section property as circle rectangular. If you then assign it an appropriate diameter or both height or a width, the polyline becomes a 3D object.

You can access the **Draw** commands via icons on the **Draw** tab ribbon:



By default, the history tree organizes 3D objects by material. To change this, select the **Objects** icon, and right-click to display the shortcut menu to see **History Tree Layout commands**. You can also use Group Commands for Modeler Objects.

While you draw objects you can also:

- Select Movement Mode as 3D, In Plane, Out of Plane, Along X, Y or Z axis.
- Select Grid Plane as XY, YZ, or XZ.
- Set the Drawing Plane
- Select the Coordinate System and choose Cartesian, Cylindrical or Spherical coordinates.
- Change Snap Settings

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Set Reference Point for the movement mode
- Adjust the View

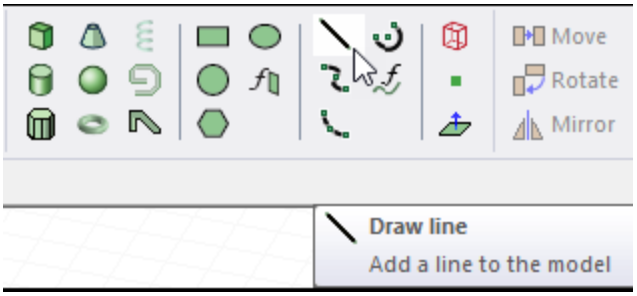
After you draw an object in the **3D Modeler** window, you can modify the object's properties, such as its position, dimensions, or color, in the **Properties** dialog box. Most model object properties can be assigned as Design variables when can then be manipulated during the solve to test their effect on the solution. For non-model objects, you can use Post Processing variables (default and Design variables,.

> **Note:**
>
> If you access your machine via Remote Desktop, if Ansys Electronics Desktop is running and one or more modeler windows are open, those modeler windows automatically close. The message manager window displays a message indicating that Ansys Electronics Desktop closed the modeler windows.

Drawing a Line Segment

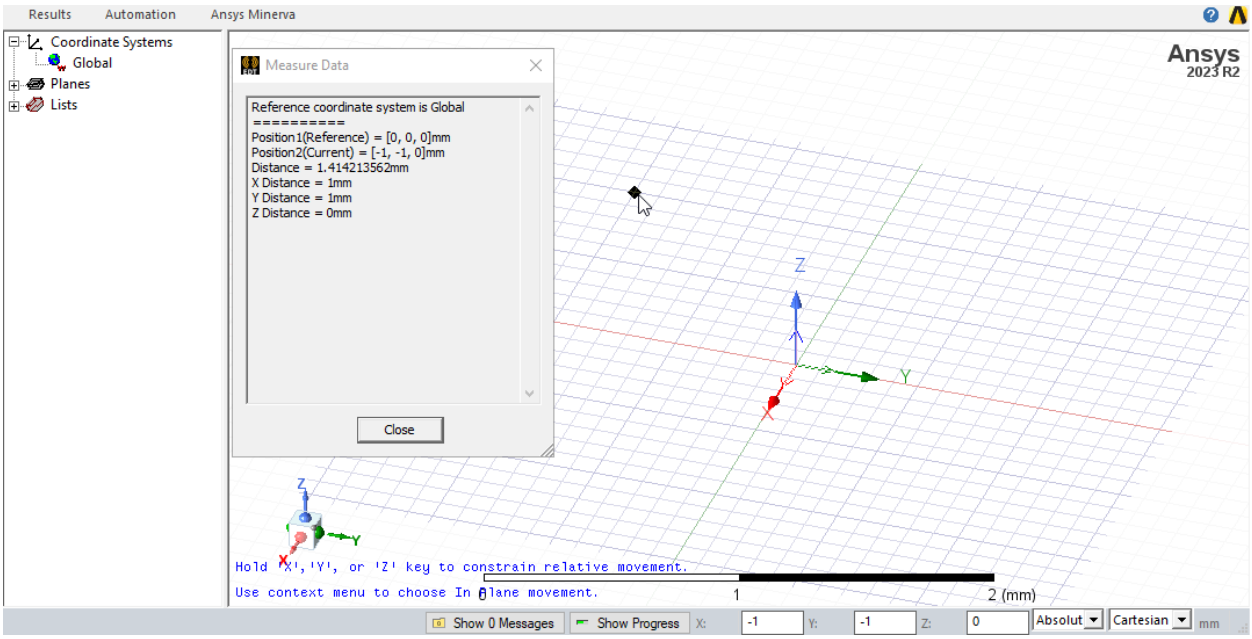
To create an object with one or more straight line segments, use the **Draw** > **Line** command.

1. From the menu bar, click **Draw**> **Line** or, on the **Draw** ribbon tab, click the **Draw line** icon:



2. Select the first point of the line in one of the following ways:
   - Click the point in the Modeler window, checking the Measure Data dialogue for the precise coordinates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

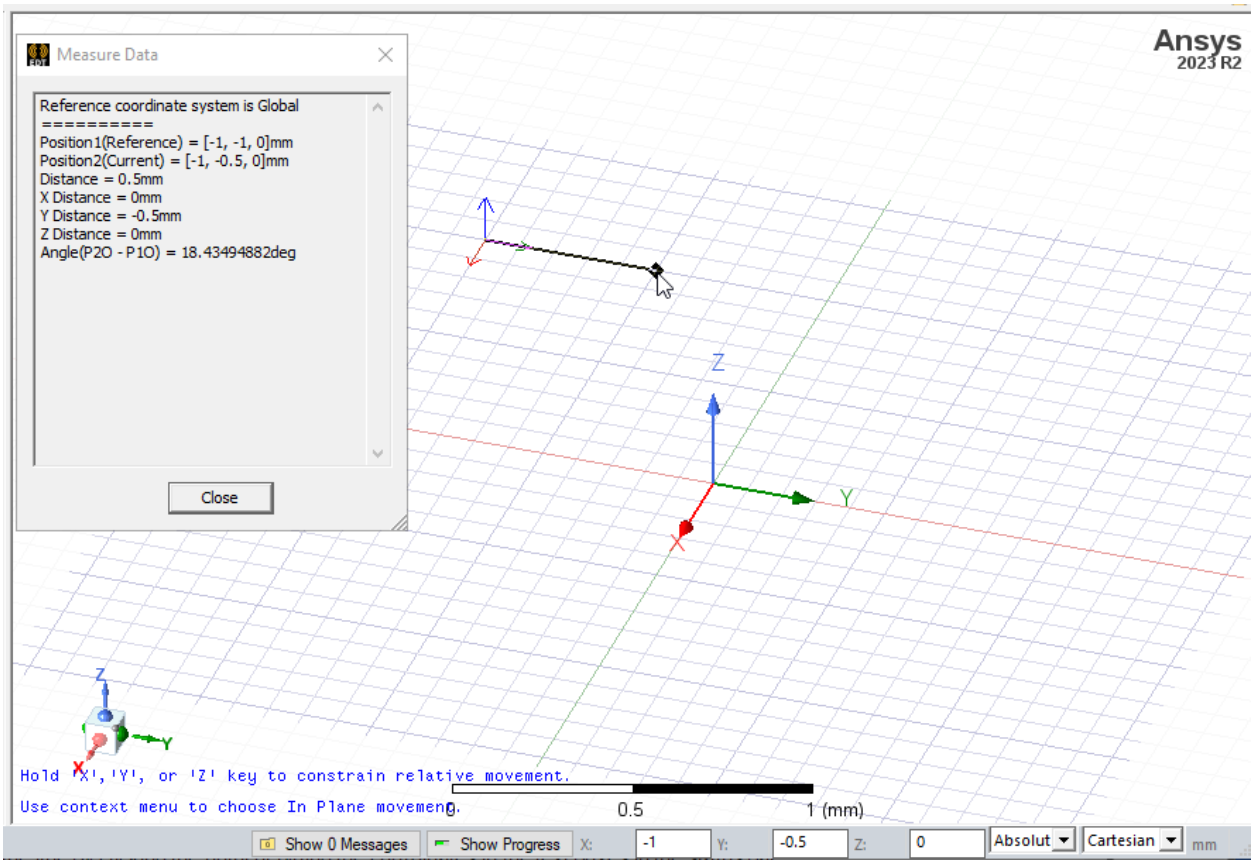


- You can accept the first point or change it by editing its coordinates in the X, Y, and Z boxes in the status bar. Note that the Status bar includes selections for Absolute or Relative Coordinates, as well as for Cartesian, Cylindrical and Spherical Coordinates.

To delete the last point that was entered, click **Undo Previous Segment** on the shortcut menu. After using the undo feature, you can also use **Redo Previous Segment** on the shortcut menu.

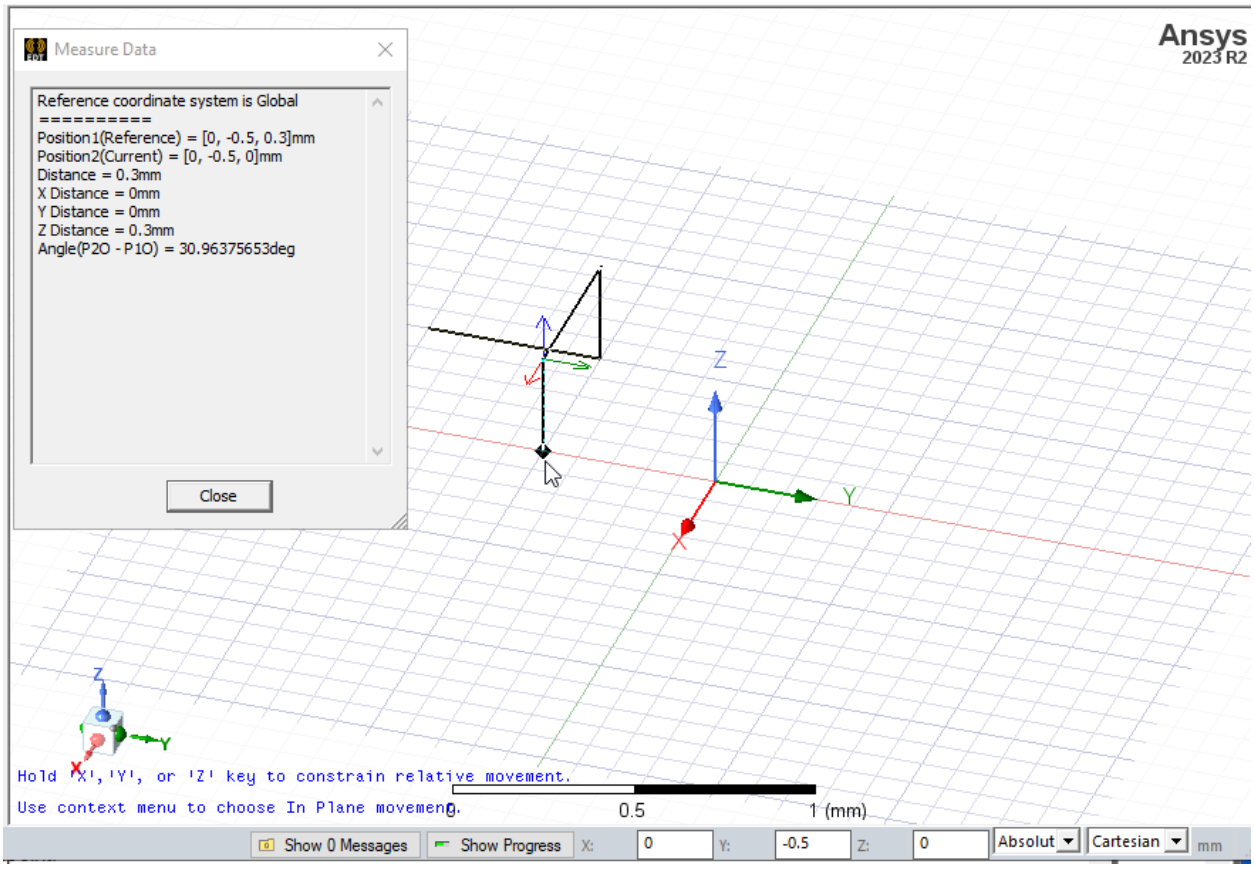
3. Select the next point of the line by clicking the point or typing the coordinates in the text boxes in the status bar.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The endpoint serves as the start point for a subsequent line segment. You can specify the plane through which the line passes by selecting the X, Y, or Z keys to constrain movement. For example, the following figure shows a line with five points that includes X, Y, and Z movements.

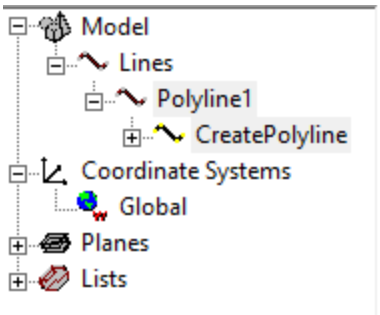
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To delete all points and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

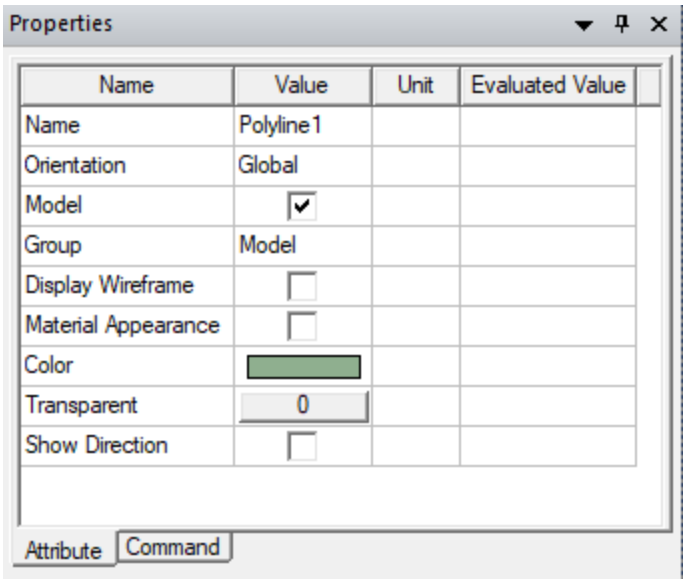
4. Complete the line in one of the following ways:
   - Double-click the endpoint.
   - Click **Done** on the context (right-click) menu.
   - Press **Enter**.

   The new line appears selected in the Modeler window and shown in the History tree.

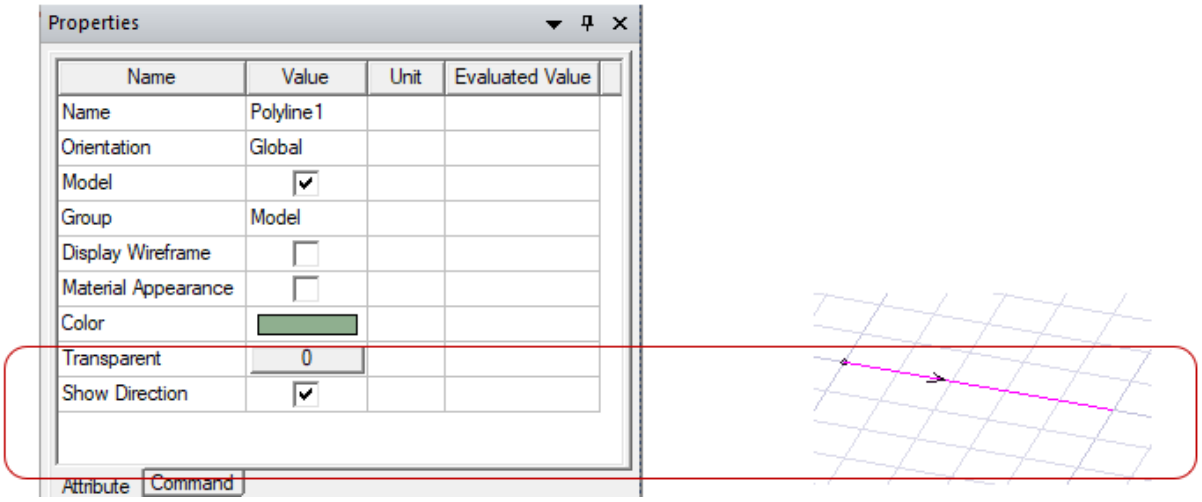


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, in which you can modify the object's attributes by editing the object's properties. You can also do this through the docked Properties window.

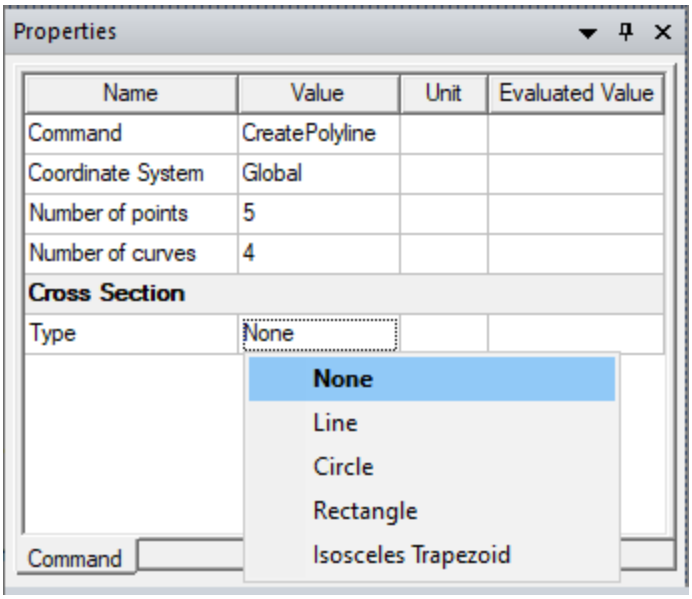


Those listed under the Command tab describe the commands used to create the object. These commands also appear in the History Tree. The properties listed as line attributes include Name, Orientation, whether a Model object, whether to Display Wireframe, Color, Transparency, and whether to Show Direction as arrows. The Show Direction property is most helpful to unambiguously show the line start orientation when plotting fields along a line.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. The Command tab **Properties** includes the Coordinate System that applies to the line, the number of points and curves, as well as the ability to assign a specific Cross Section to the line.



6. If you created the line with the Modeler option for editing properties of new primitives, click **OK** to close the **Properties** dialog box.

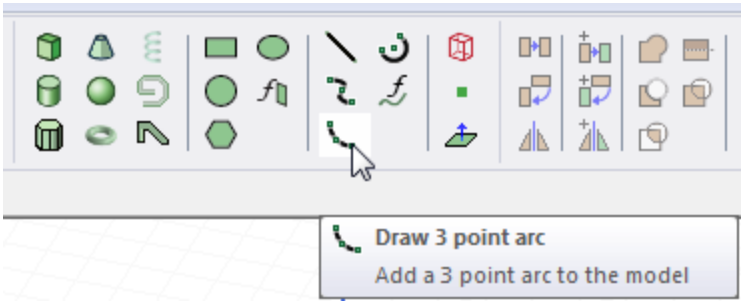
> **Note:** While drawing a polyline, you can switch between straight line, arc line, or spline segments using the **Set Edge Type** commands on the shortcut menu.

Drawing a Three-Point Arc Line

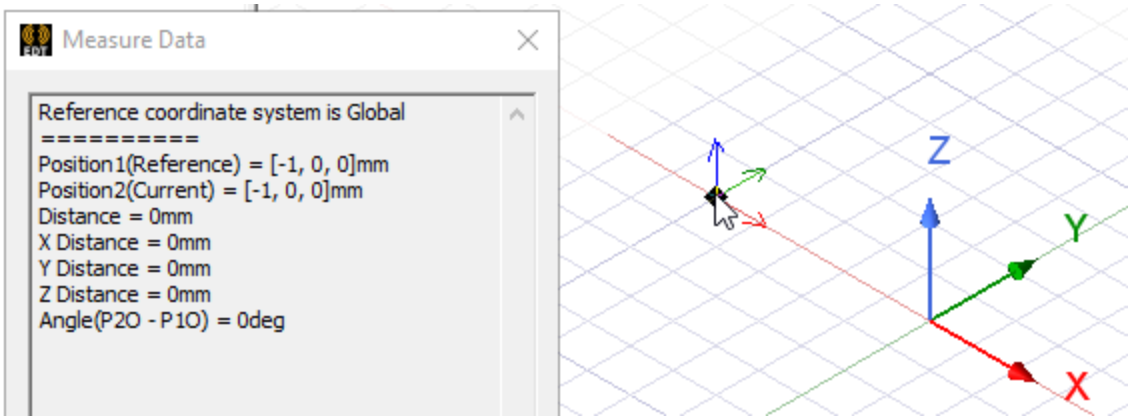
In the modeler, a three-point arc line segment is an arced line defined by three points on its curve. Use the **Draw** > **Arc** > **3 Point** command to create a polyline object with one or more arc line segments. Before you draw a three-point line arc, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Arc**> **3 Point** or, on the **Draw** ribbon tab, click the **Draw 3 point arc** icon:

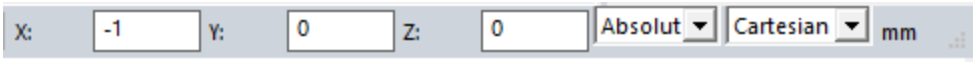
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



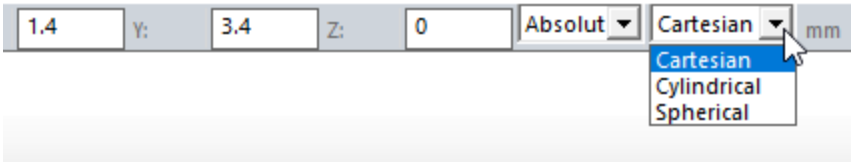
2. Select the start point of the arc in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the **X, Y,** and **Z** text boxes.



     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the line in Cartesian, Cylindrical, or Spherical coordinates.



3. Select the midpoint of the arc by clicking the point or typing the coordinates in the text boxes in the status bar.

   To delete the last point that was entered, click **Undo Previous Segment** on the shortcut menu. After using the undo feature, you can also use **Redo Previous Segment** on the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To delete all points and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

4. Select the endpoint of the arc by clicking the point or typing the coordinates in the text boxes in the status bar.

   The endpoint serves as the start point for a subsequent arc line segment.

5. If the endpoint is the last point of the polyline object, double-click the point to complete the polyline or click **Done** on the shortcut menu.

   If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, in which you can modify the object's attributes by editing the Properties. Those listed under the Command tab describe the commands used to create the object. These commands also appear in the History Tree. The Properties listed as line attributes include Name, Orientation, whether a Model object, whether to Display Wireframe, Color, Transparency, and whether to Show Direction as arrows. The Show Direction property is most helpful to unambiguously show the line start orientation when plotting fields along a line.

6. Click **OK**.

   Based on the three points you specified, the modeler calculates the center point and radius of the arc and draws an arced line through the three points.

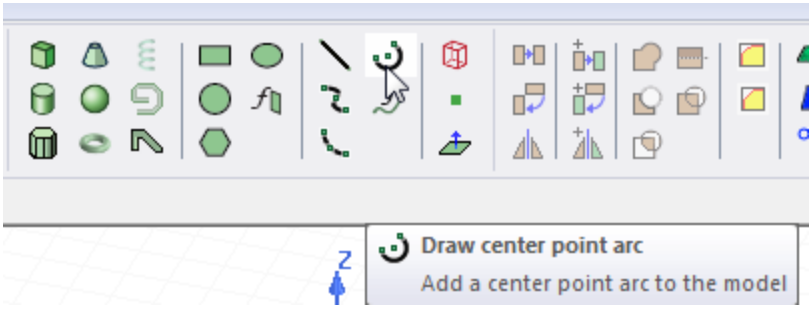
> **Note:**
>
> While drawing a polyline, you can switch between arc line, straight line, or spline segments using the **Set Edge Type** commands on the shortcut menu.

Drawing a Center-Point Arc Line

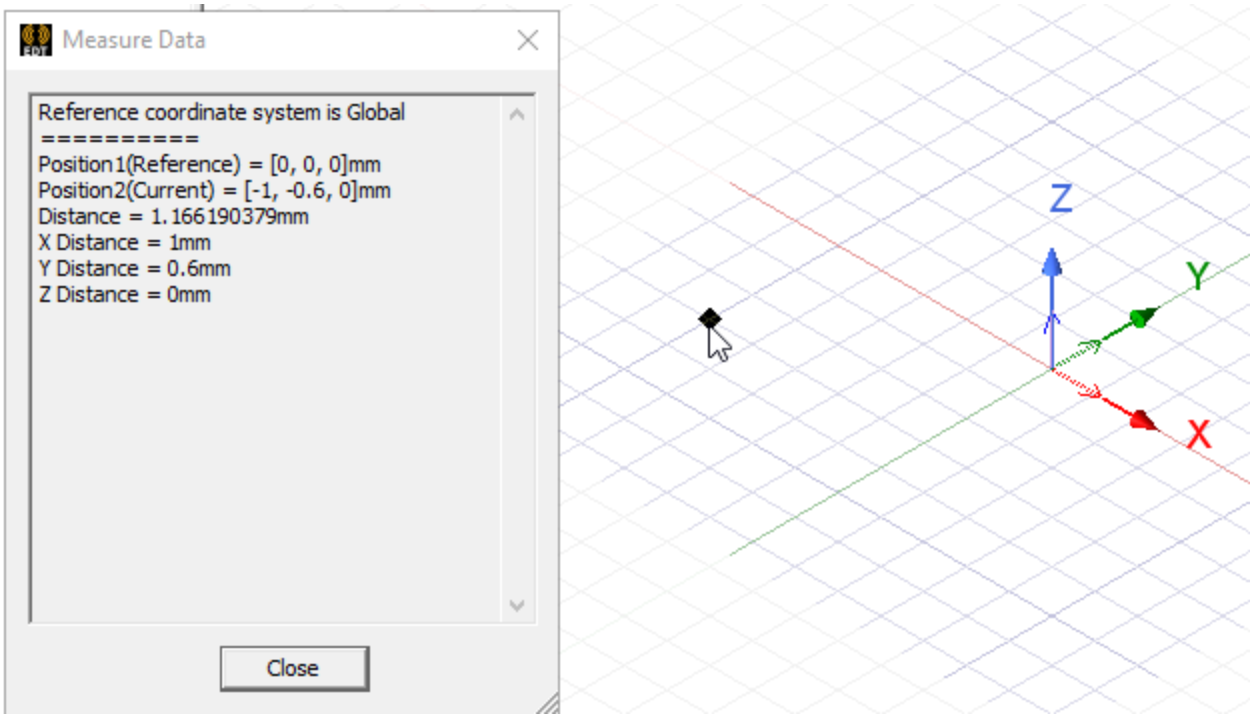
In the modeler, a center-point arc line segment is an arced line defined by a center point, start point and angle. Use the **Draw** > **Arc** > **Center Point** command to create a polyline object with one or more center-point arc line segments. Before you draw a center-pint arc line, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Arc**> **Center Point** or, on the **Draw** ribbon tab, click the **Draw center point arc** icon:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



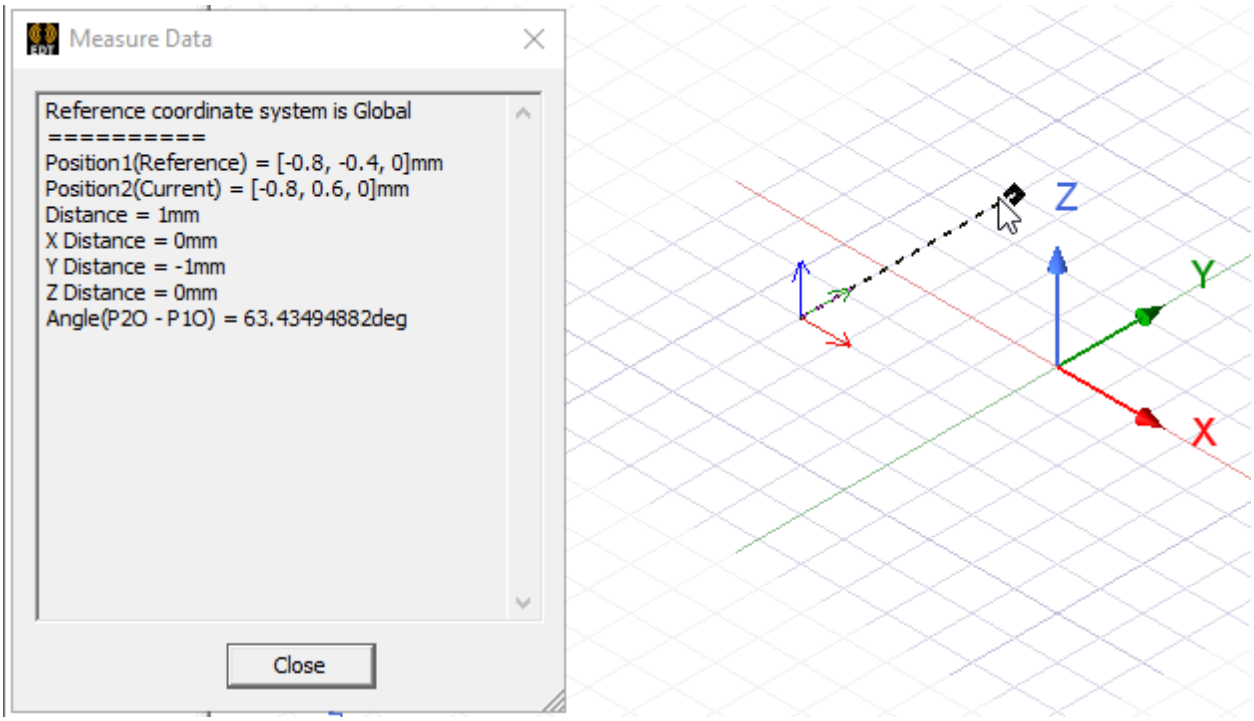
2. Select the center point of the arc in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the text boxes in the status bar.

3. Select the start point, or radius, of the arc by clicking the point or typing the coordinates in the text boxes in the status bar.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

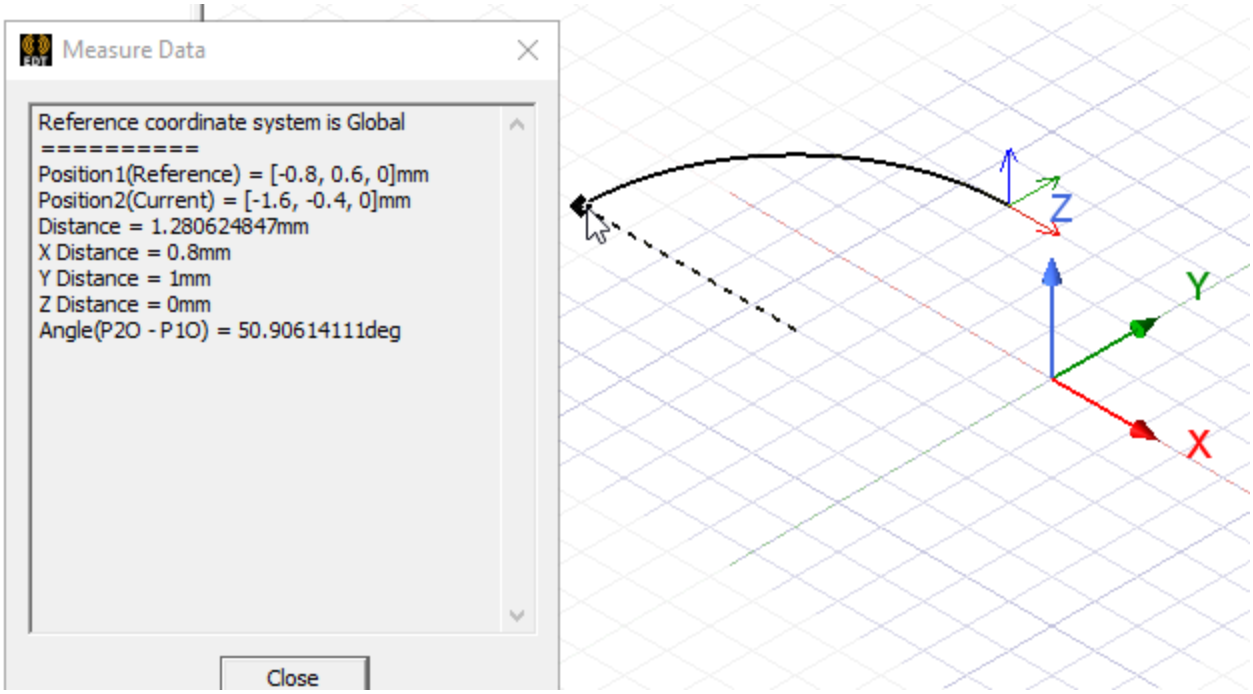


To delete the last point that was entered, click **Undo Previous Segment** on the shortcut menu. After using the undo feature, you can also use **Redo Previous Segment** on the shortcut menu.

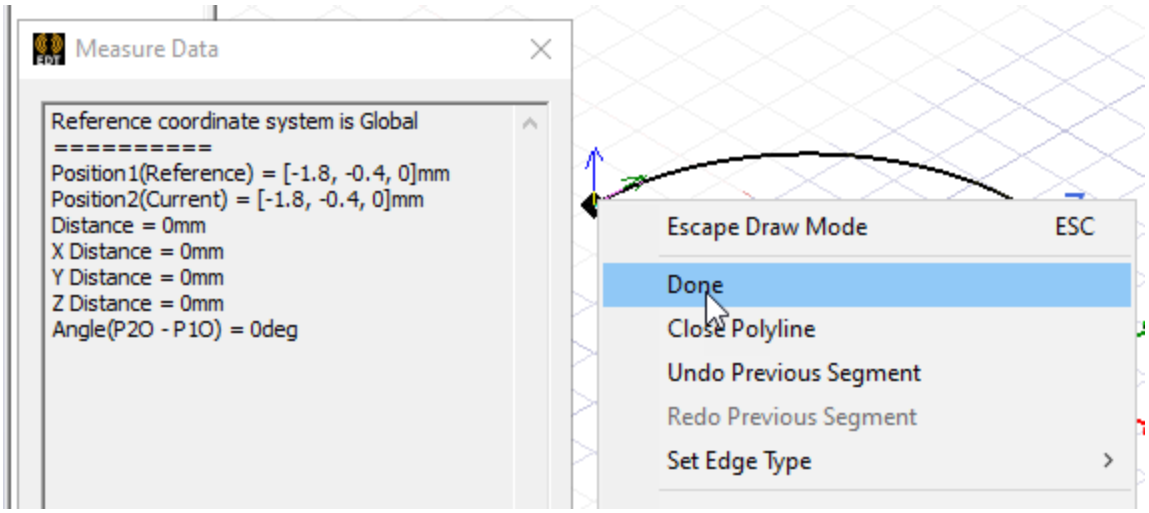
To delete all points and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Sweep the angle, or endpoint, of the arc by clicking the point or typing the coordinates in the text boxes in the status bar.



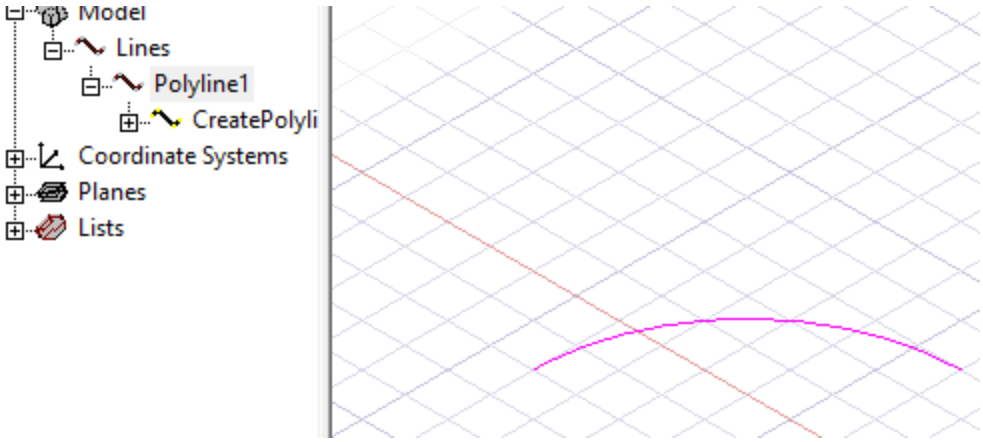
5. If the endpoint is the last point of the polyline object, double-click the point to complete the polyline or click **Done** on the shortcut menu.



If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's attributes. The new polyline

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

appears in the modeler window and in the History tree.



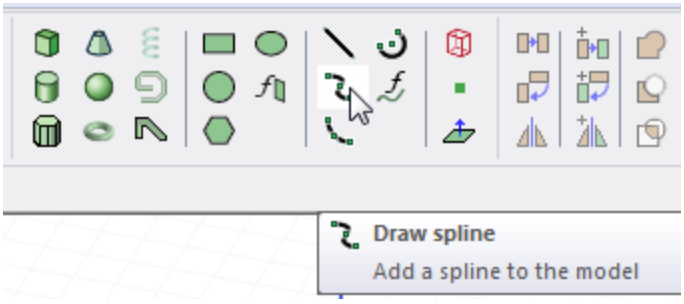
6. Click **OK**.

> **Note:**
>
> While drawing a polyline, you can switch between arc line, straight line, or spline segments using the **Set Edge Type** commands on the shortcut menu.

Drawing a Spline

A spline is a curved line defined by three points. The modeler uses a natural spline type: a piece wise cubic spline with an end condition that has a derivative of zero. Use the **Draw** > **Spline** command to create a polyline object with one or more spline segments. Before you draw a spline, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

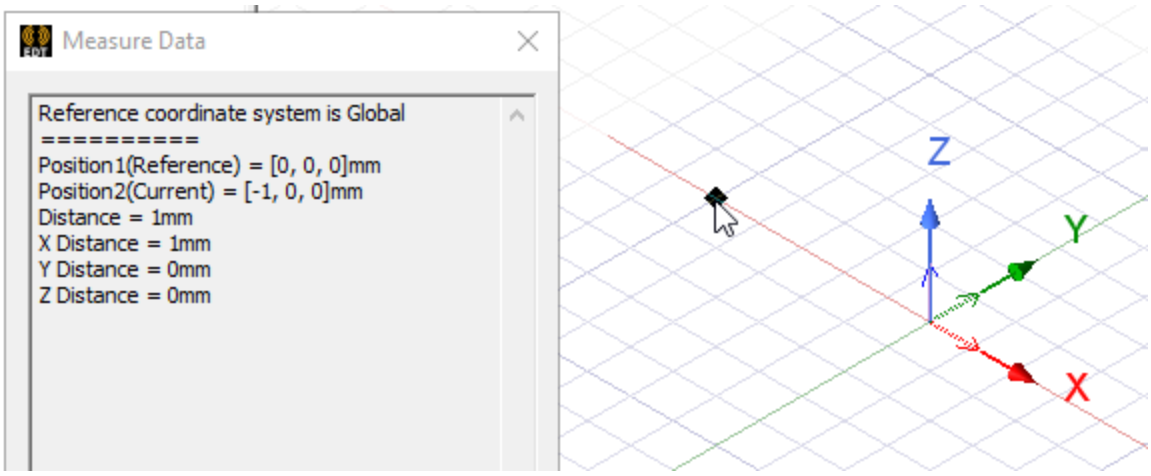
1. From the menu bar, click **Draw**> **Spline** or, on the **Draw** ribbon tab, click the **Draw spline** icon:



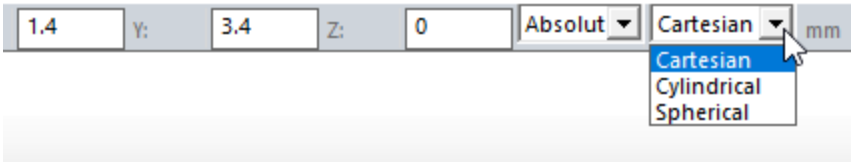
2. Select the spline's start point in one of the following ways:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the point.



- Type the point's coordinates in the text boxes in the Status bar, and then press **Enter**.
  The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the spline in Cartesian, Cylindrical, or Spherical coordinates.

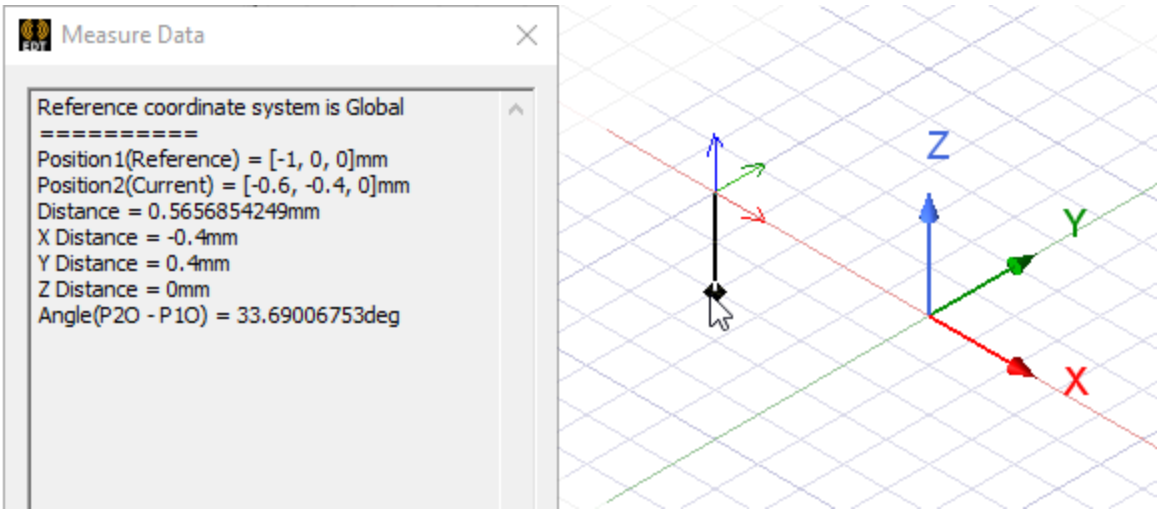


To delete the last point entered, click **Undo Previous Segment** on the shortcut menu. After using the undo feature, you can also use **Redo Previous Segment** on the shortcut menu.

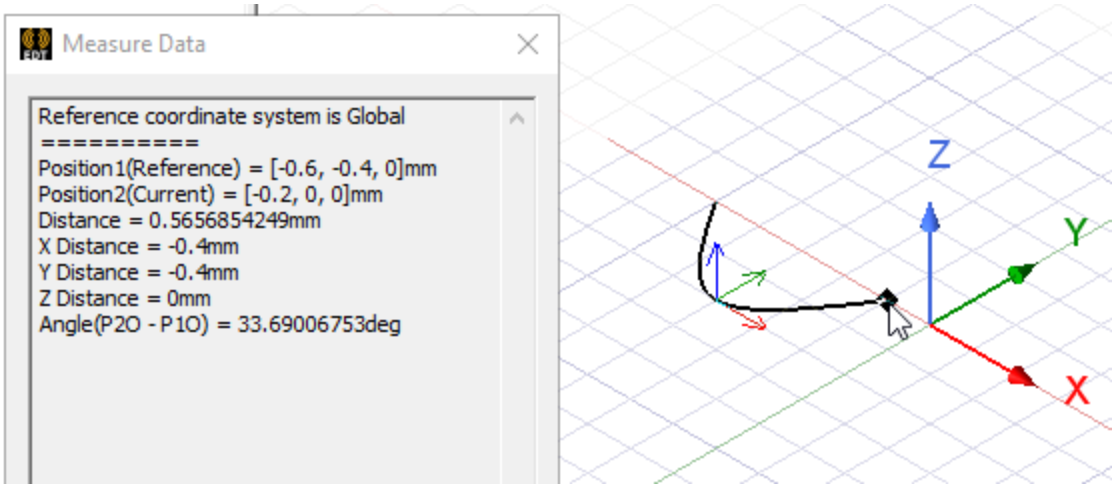
To delete all selected points and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the midpoint of the spline by clicking the point or typing the coordinates in the text boxes in the status bar.



4. Select the endpoint of the spline by clicking the point or typing the coordinates in the text boxes in the status bar.

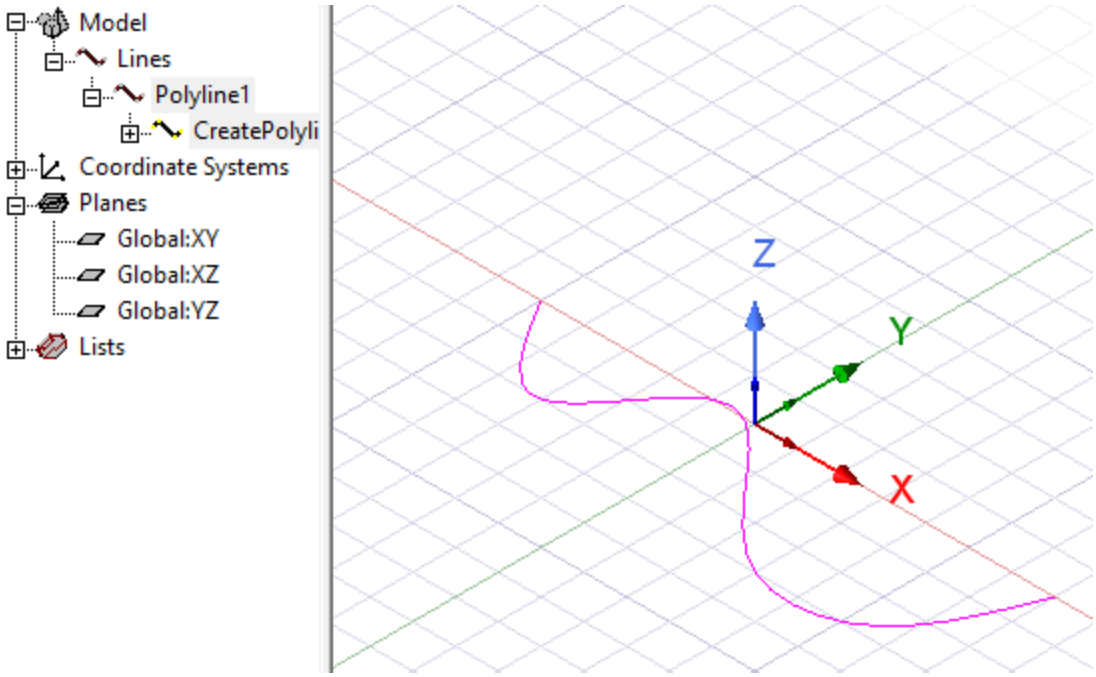


The endpoint serves as the start point for a subsequent spline segment. You can complete the spline at this point, or add additional segments.

5. Complete the spline in one of the following ways:
   - Double-click the endpoint.
   - Click **Done** on the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Press **Enter**.
  The spline appears in the Modeler window and the History tree.



If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, in which you can modify the object's attributes by editing the Properties. Those listed under the Command tab describe the commands used to create the object. These commands also appear in the History Tree. The Properties listed as line attributes include Name, Orientation, whether a Model object, whether to Display Wireframe, Color, Transparency, and whether to **Show Direction** as arrows. The **Show Direction** property is most helpful to unambiguously show the line start orientation when plotting fields along a line.

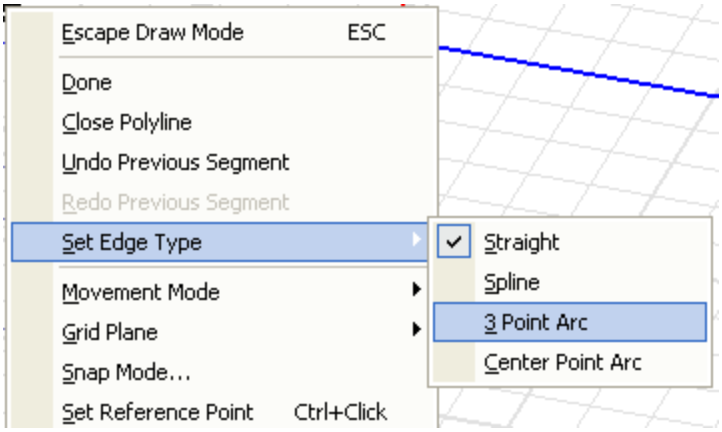
6. If you have edited Properties, click **OK**.

> **Note:**
>
> While drawing a polyline, you can switch between spline, straight line, or arc line segments using the **Set Edge Type** commands on the shortcut menu.

Drawing a Polyline

A polyline is a single object that includes any combination of straight line, arc line, or spline segments. The endpoint of one segment is the start point for the next segment. Use the shortcut menu's **Set Edge Type** commands to switch between straight line, arc line, or spline segments while drawing a polyline.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

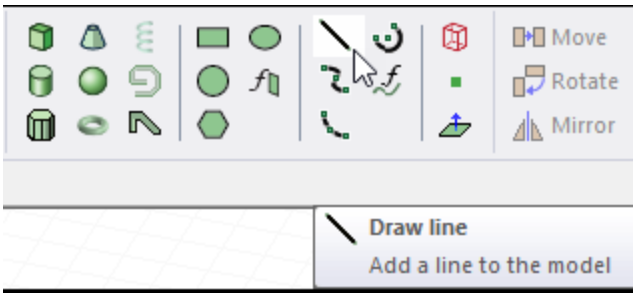


Before drawing a polyline, determine how closed polylines will be handled in the 3D Modeler options, as follows:

Using the menu bar, click **Tools** > **Options** > **General Options**. In the *Options* dialog box that appears, choose **3D Modeler** > **Operation** from the tree on the left side. Then, in the **Polyline Creation** section of the operation options, select or clear the **Automatically cover closed polylines** check box.

If checked, closed polylines become sheet objects and are listed under *Sheets* in the History Tree. If unchecked, closed polylines are listed under *Lines* in the History Tree.

1. From the menu bar, click **Draw**> **Line** or, on the **Draw** ribbon tab, click the **Draw line** icon:



2. Right-click in the **3D Modeler** window to access the shortcut menu, and then point to **Set Edge Type**.
3. Click **Straight, Spline, 3 Point Arc,** or **Center Point Arc**, depending on which type of polyline segment you want to draw.
4. Depending on your selection in the previous step, do one of the following:
   - If you clicked **Straight**, follow the procedure for drawing a straight line.
   - If you clicked **Spline**, follow the procedure for drawing a spline.
   - If you clicked **3 Point Arc**, follow the procedure for drawing a three-point arc line.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you clicked **Center Point Arc**, follow the procedure for drawing a center-point arc line.

5. Repeat steps 2 and 3 for each segment of the polyline object. The endpoint of the previous segment serves as the start point for the next segment.

   The shortcut menu lets you do the following for each segment:

   Undo Previous Segment or **Redo Previous Segment**.

6. Complete the polyline in one of the following ways:
   - Double-click the endpoint of the final segment.
   - Click **Done** on the shortcut menu.

   > **Note:**
   >
   > To connect the polyline's start and endpoints, click **Close Polyline** on the shortcut menu.

   If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, in which you can modify the object's attributes by editing the Properties. Those listed under the Command tab describe the commands used to create the object. These commands also appear in the History Tree. The Properties listed as line attributes include Name, Orientation, whether a Model object, whether to Display Wireframe, Color, Transparency, and whether to Show Direction as arrows. The Show Direction property is most helpful to unambiguously show the line start orientation plotting fields along a line. Notice that by going to the History Tree and selecting Create Polyline for that object, you can assign a cross section and dimensions to a polyline.

7. Click **OK**.

   If you select a polyline in the History Tree, you can use the Measure mode to see the total length.

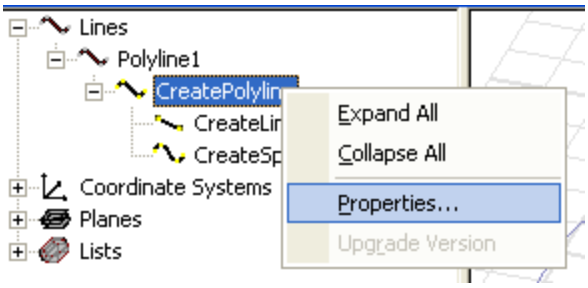
Assigning a Cross Section and Dimensions to a Polyline

By viewing the properties of a polyline from the History Tree, you can assign either a Line, Circle, Rectangle or Isosceles Trapezoid cross section to a polyline. This assignment enables editable dimension properties of width for a line, diameter for a circle, and height and width for a rectangle or trapezoid. To assign a cross section to a polyline:

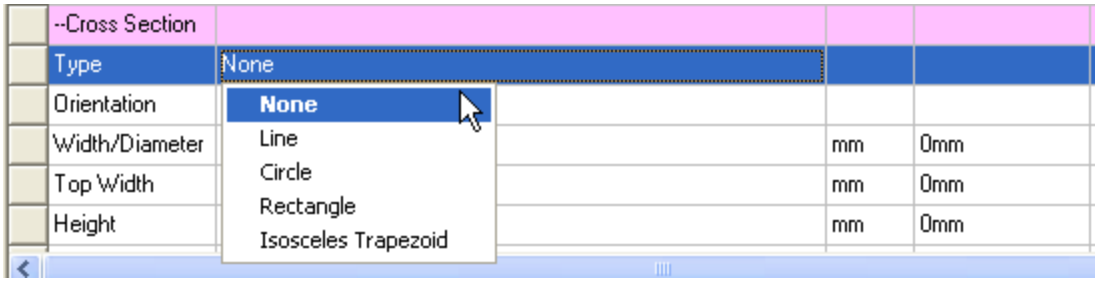
1. In the History Tree of the Modeler window, select the **CreatePolyline** entry for the polyline that you want to give a cross section.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This action displays the polyline properties in the docked **Properties** window (if you have it displayed). Alternatively, right-click the **CreatePolyline** entry and choose **Properties...** from the shortcut menu to display the **Properties** dialog box.



2. In the **Cross Section** section of the properties, in the second column of the **Type** row, click **None** to display the cross section choices.



3. Choose a **Line, Circle, Rectangle,** or **Isosceles Trapezoid** cross section.
   - Selecting **Line** causes the Cross Section area of the polyline properties to display editable fields for **Orientation** and **Width**.
   - Selecting **Circle** causes the Cross Section area of the polyline properties to display an editable field for **Diameter**.
   - Selecting **Rectangle** causes the Cross Section area of the polyline properties to display editable fields for **Orientation**, **Width**, and **Height**.
   - Selecting **Isosceles Trapezoid** causes the Cross Section area of the polyline properties to display editable fields for **Orientation**, **Width/Diameter**, **Top Width**, and **Height**.
4. If you select **Line**, **Rectangle,** or **Isosceles Trapezoid** you can edit the **Orientation** as **Auto, X, Y,** or **Z**. This setting specifies the direction in which the **Height** dimension extends.
5. Specify the dimensions and select the units for the Cross section.

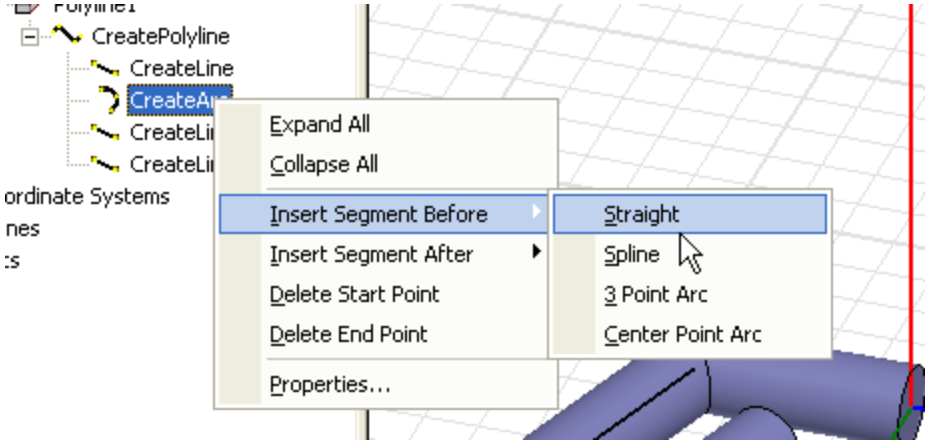
   Type a value in the dimension field(s) and select units from the drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The dimensions must be reasonable relative to the specified shape and orientation of the polyline. If the polyline cannot be extended into current Orientation for the given dimension(s), you will receive a warning. If you receive a warning, check the Orientation, dimension and units.

When the modeler can extend the dimensions legally, it displays the modified object, and lists it in the History Tree as either a Sheet object (Line or one dimensional Rectangle cross section) or as a Solid object (Circle or two dimensional Rectangle cross section).

6. You can modify the new polyline either by editing the properties, or by using the History Tree to select one of the segments that make up the polyline and right-clicking to display the shortcut menu showing commands to **Insert**, **Delete**, or display the segment **Properties**.



A segment that you select in the History Tree is indicated in the Modeler window by black highlighting. If you insert a new segment, it adopts the dimensions you specified for the polyline object.

Inserting Line Segments

You can insert line segments of various kinds for existing line, arc, or spline objects.

1. Select a line segment in the History Tree (not the modeler window). The individual segments entries are **CreateLine**, **CreateArc**, **CreateAngularArc**, or **CreateSpline**.

   This action highlights the object and enables the **Insert Line Segment** subcommands in the **Draw** menu.

2. Using the menu bar, go to the **Draw** > **Line Segment** submenu and then choose either the **Insert Segment Before** or **Insert Segment After** submenu.

   Alternatively, right-click a polyline segment in the History Tree and, from the shortcut menu, choose either the **Insert Segment Before** or **Insert Segment After** submenu

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Use this submenu to specify the kind of segment to add. The inserted segment can be: **Straight, Spline. 3 Point Arc,** or **Center Point Arc**.
4. Depending on your selection in the previous step, do one of the following:
   - If you clicked **Straight**, follow the procedure for drawing a straight line.
   - If you clicked **3 Point Arc**, follow the procedure for drawing a three-point arc line.
   - If you clicked **Center Point Arc**, follow the procedure for drawing a center-point arc line.
   - If you clicked **Spline**, follow the procedure for drawing a spline.

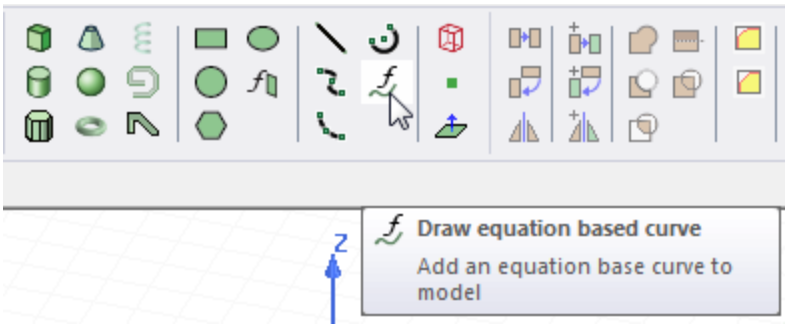
   > **Note:**
   >
   > When inserting straight segments or arcs, the insertion operation will terminate as soon as you complete the definition of a single new segment. For splines, you can insert a spline with any number of defining points, and the insertion operation terminates as soon as you terminate the spline.

5. Repeat steps 1 through 4 for each segment of the polyline before or after which you want to insert a new segment. The endpoint of the previous segment serves as the start point for the next segment.

Drawing an Equation-Based Curve

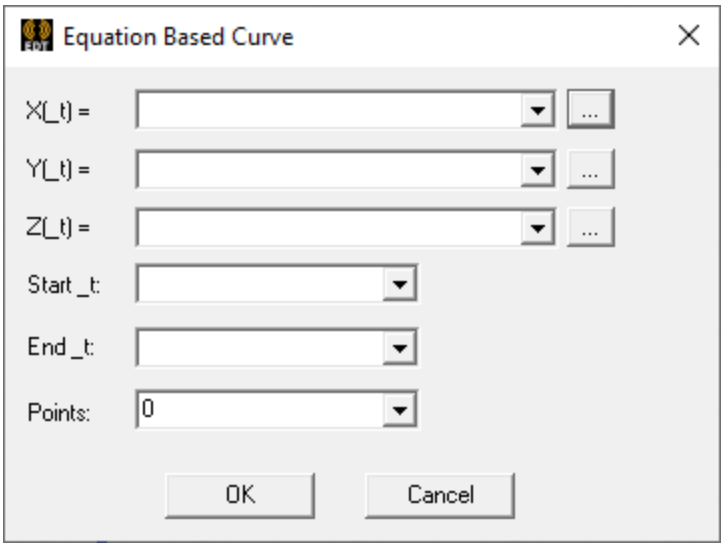
Any line that can be described by an equation in three dimensions can be drawn. Before you draw am equation-based curve, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Equation Based Curve** or, on the **Draw** ribbon tab, click the **Draw equation based curve** icon:



   The *Equation Based Curve* dialog box opens:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

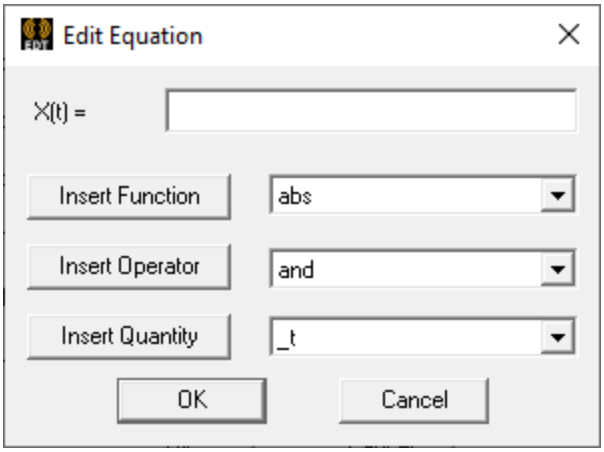


2. Type equations for **X(_t)**, **Y(_t)**, and **Z(_t)**

   You can also define an equation by doing the following:

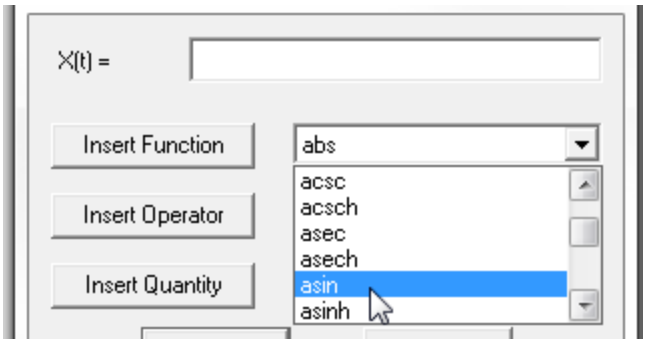
   a. Click an ellipsis button **[...]**.

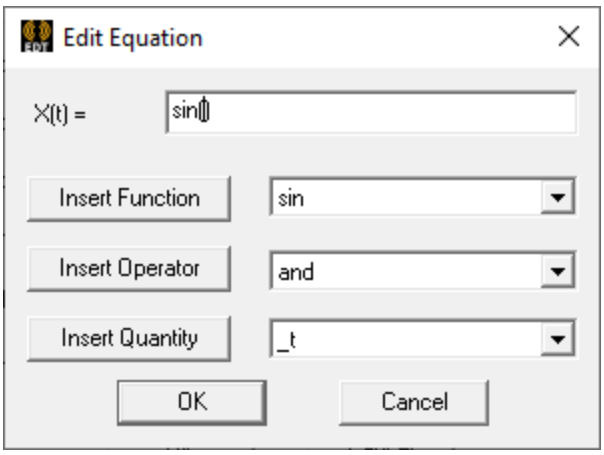
      The **Edit Equation** dialog box appears:



   b. Do one or more of the following to define the equation:
      - Type a numerical value, variable, or expression directly in the text box.
      - Build an equation for X(_t), Y(_t), or Z(_t) by specifying appropriate function, operator, and quantity to insert into the expression from the three drop-down menus:

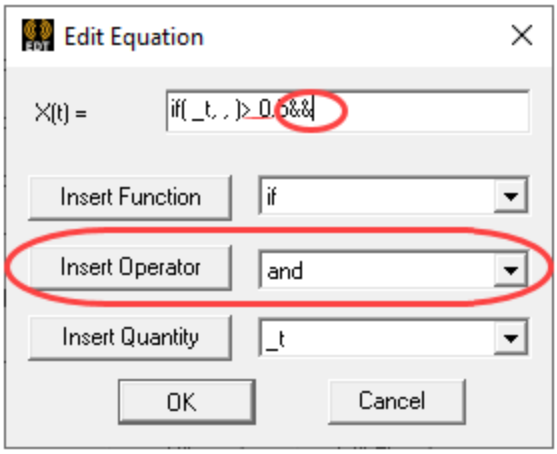
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

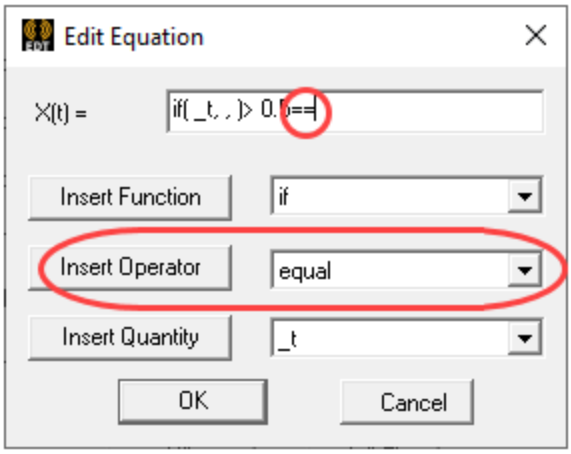




Click the corresponding **Insert Function, Insert Operator,** or **Insert Quantity** buttons to insert the selections into the equation text box at the current cursor location. The if(, ,) function has been added under **Insert Function**.

- The **Insert Operator** button inserts corresponding operator:





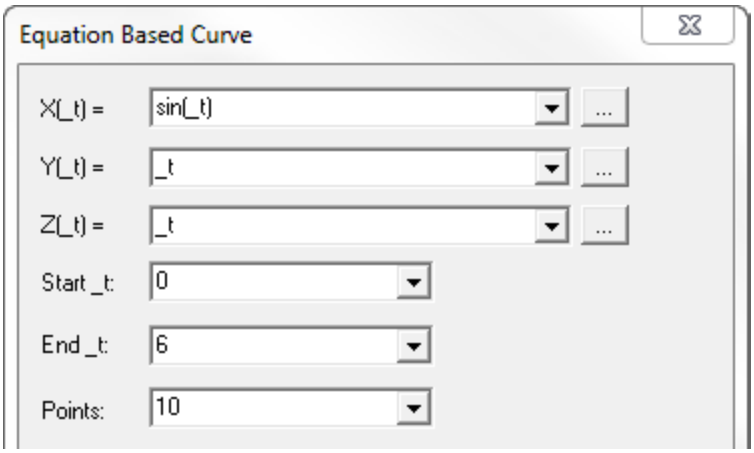
- Use a combination of the preceding two methods to build complex equations.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. When you are finished defining the equation, click **OK** to close the **Edit Equation** dialog box and return to the **Equation Based Curve** dialog box.

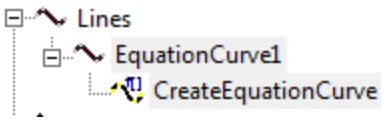
> **Note:**
>
> - The equations are taken to be meters. Use of units in equations is not properly supported.
> - While parsing expressions, equation based curves convert each variable separately to model units and assume that the resulting expression is in model units.
> - Equation based curves depend on the variable value library to correctly evaluate the units of expression.
> - Functions that do not support intrinsic (_t for curve, _u and _v for surface) will give an error if not supported.

3. Select a start value from the **Start_t** drop-down menu.
4. Select an end value from the **End_t** drop-down menu.
5. Type in or select the number of points in the curve from the **Points** drop-down menu.



6. Click **OK** on dialog to close it and create the curve.

   The curve appears in the modeler window, and the History tree shows the curve object and the command.



If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, in which you can modify the object's attributes by editing the Properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Properties listed as line attributes include Name, Orientation, whether a Model object, whether to Display Wireframe, Color, Transparency, and whether to Show Direction as arrows. The Show Direction property is most helpful to unambiguously show the line start orientation when plotting fields along a line.

Functions, Operators and Quantities for the Edit Equation Dialog Box

This table lists the functions, operators and Quantities for the Edit Equation dialog box.

Item	Definition
Functions	
abs	Absolute value of the simulation quantity which results in a number that is always positive.
acos	Arc cosine (the inverse function of a cosine).
acosh	Inverse hyperbolic arc cosine.
acot	Inverse cotangent
acoth	Inverse hyperbolic cotangent
acsc	Inverse cosecant
acsch	Inverse hyperbolic cosecant
asec	Inverse secant
asech	Inverse hyperbolic secant
asin	Arc sine (the inverse function of sine)
asinh	Inverse hyperbolic sine
atan	Arc tangent (the inverse function of a tan)
atanh	Inverse hyperbolic tan
cos	Cosine
cosh	Hyperbolic cosine
cot	Cotangent
coth	Hyperbolic cotangent
csc	Cosecant
csch	Hyperbolic cosecant
ln	Natural logarithm.
log	Natural logarithm (same as ln).
sin	Sine.
sinh	Hyperbolic sine.
sqrt	Square root of the selected simulation quantity.
tan	Tangent.

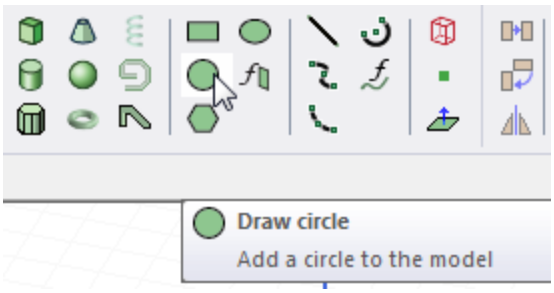
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item	Definition
tanh	Hyperbolic tangent.
Operators	
and	
cross	
division	
dot	
equal	
exp	
exponent	
greater_than	
greater_than_ or_equal	
if	if(, ,)
not	
not_equal	
or	
Quantities	
_t	For equation based lines
_u,_v	For equation based surfaces.
PI	

Drawing a Circle

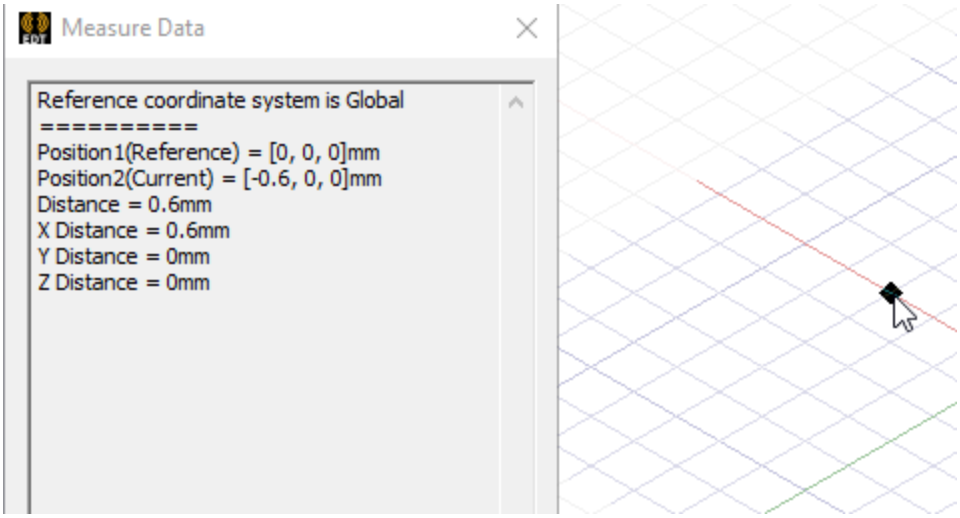
Draw a circle by selecting a center point and a radius. Circles are drawn as true surfaces in the modeler. Before you draw a circle, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Circle** or on the **Draw** ribbon tab, click the **Draw circle** icon:



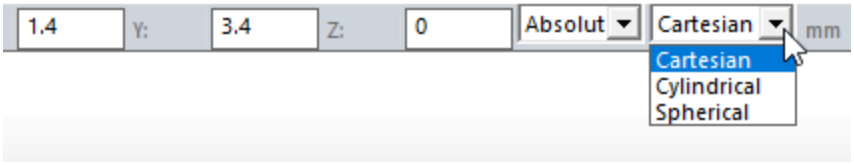
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the center point of the circle in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the text boxes in the status bar.

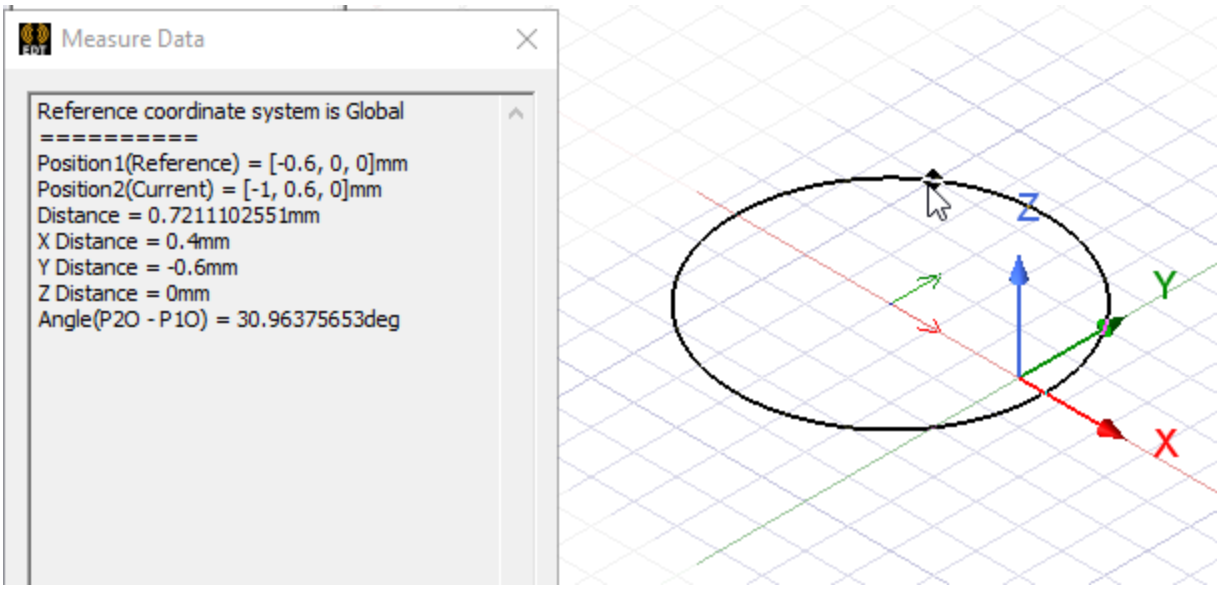
     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the circle in Cartesian, Cylindrical, or Spherical coordinates.



3. Specify the radius by selecting a point on the circle's circumference in one of the following ways:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

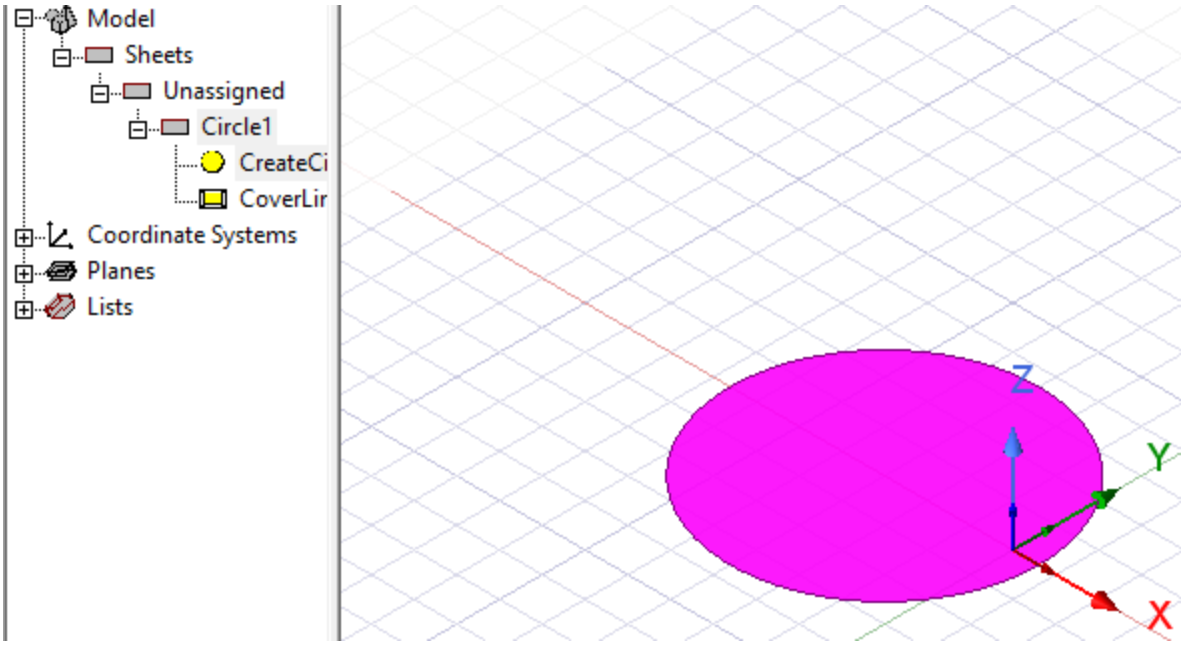
- Click the point.



- Type the coordinates of the point relative to the center point in the **dX**, **dY**, and **dZ** boxes, where **d** is the distance from the previously selected point.

4. Double-click or right-click and select Done from the shortcut menu. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. If the **Automatically cover closed polyline** option is selected in the **Modeler Options** window, the circle will be covered, resulting in a 2D sheet object. Otherwise it will be a closed 1D polyline object. The new circle appears in the Modeler window and History tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands see:

Surface Approximation and related sections:

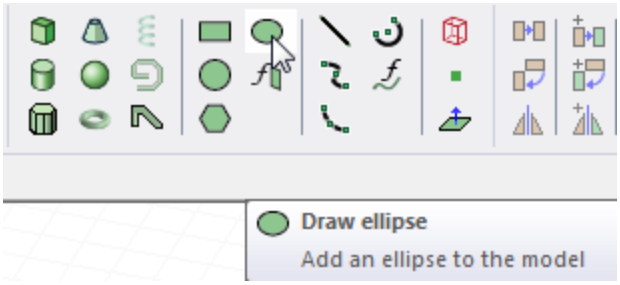
Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing an Ellipse

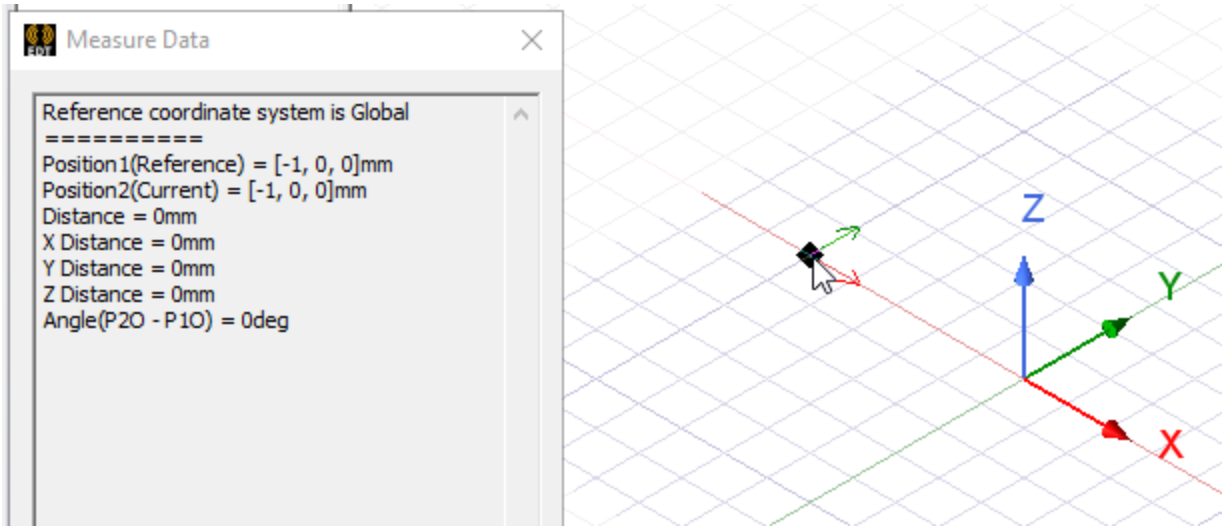
Draw an ellipse by specifying a center point, base radius, and secondary radius. Before you draw an ellipse, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw>** **Ellipse** or, on the **Draw** ribbon tab, click the **Draw ellipse** icon:

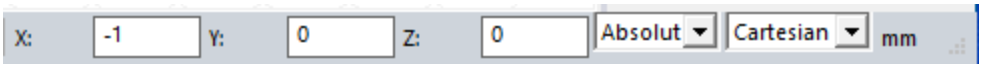
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



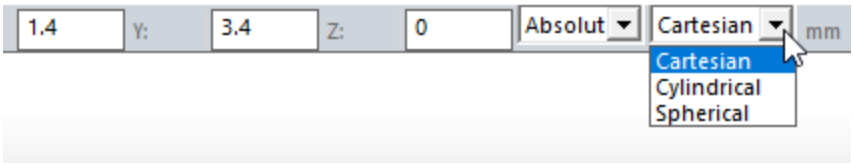
2. Select the center point of the ellipse in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the text boxes in the status bar.



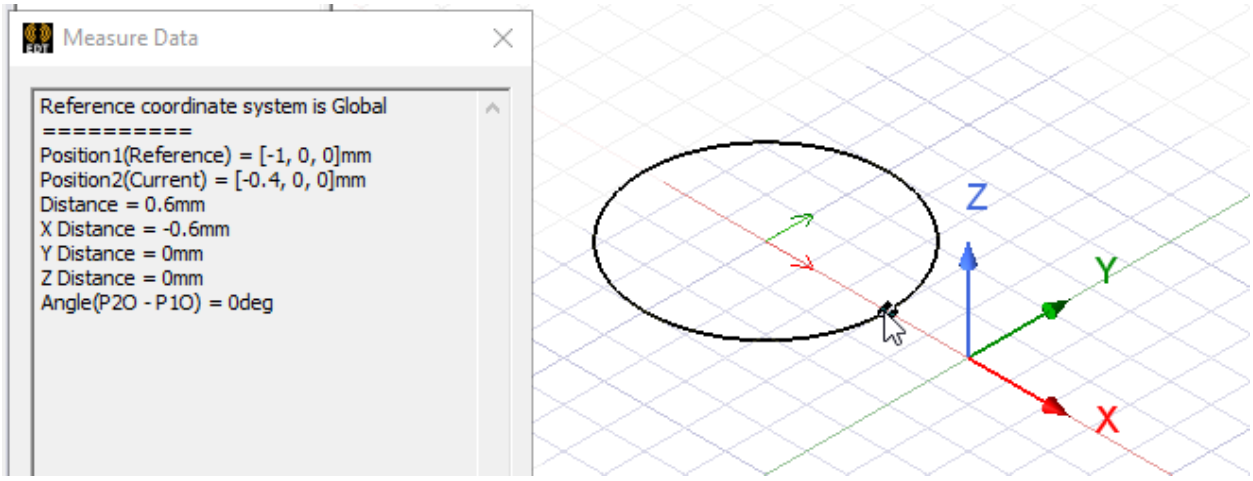
     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the ellipse in Cartesian, Cylindrical, or Spherical coordinates.



3. Specify the base radius of the ellipse. If the current drawing plane is xy, then x is the base radius direction. If the drawing plane is yz, then y is the base radius direction. If the drawing plane is xz, then z is the base radius direction. Select the point in one of the following ways:

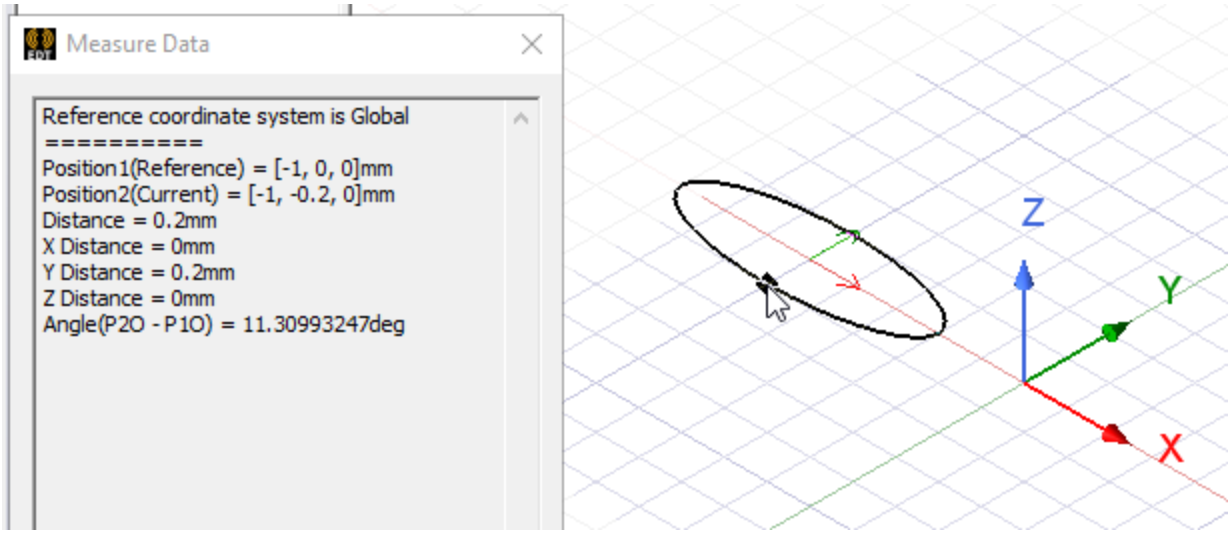
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the point. Ansys Electronics Desktop constrains mouse movement to the base radius direction.



- Type the coordinates of a point relative to the center point in the **dX, dY,** or **dZ** box, where **d** is the distance from the previously selected point.

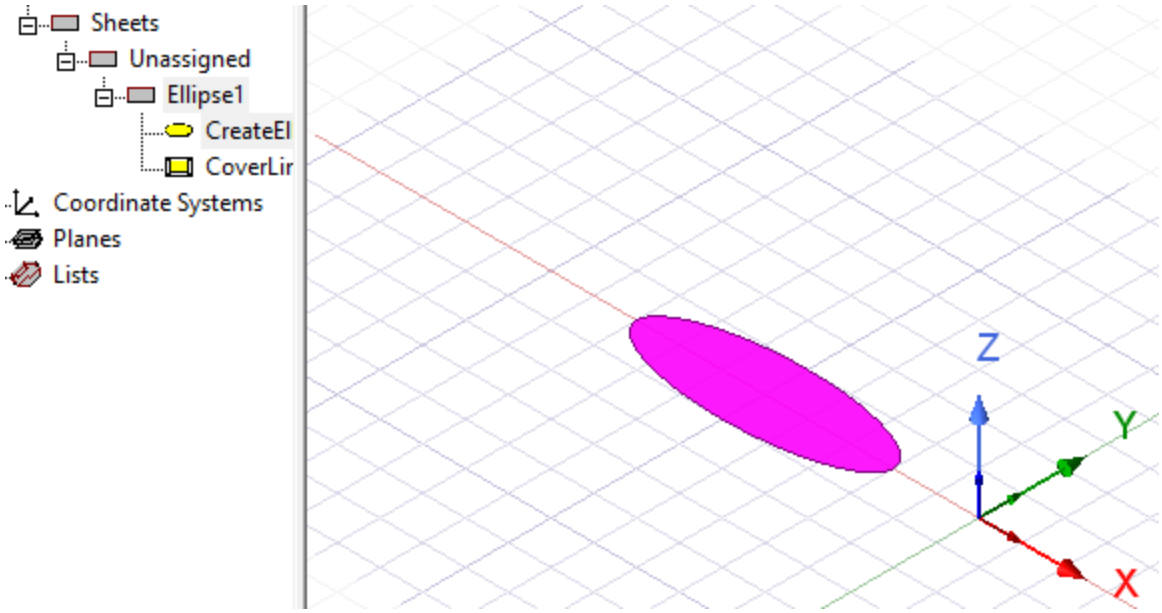
4. Specify the secondary radius of the ellipse. Select the point in one of the following ways:
   - Click the point. Ansys Electronics Desktop constrains mouse movement to a point on the plane orthogonal to the base radius direction.



   - Type the coordinates of a point relative to the center point in the **dXm dY,** or **dZ** box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Double click, or right click and select **Done** from the short cut menu. The ellipse appears in the Modeler window and in the History Tree.



6. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. The **Ratio** value represents the aspect ratio of the secondary radius to the base radius.

   If the **Automatically cover closed polyline** option is selected in the **Modeler Options** window, the ellipse will be covered, resulting in a 2D sheet object. Otherwise it will be a closed 1D polyline object.

If the base radius is larger than the secondary radius, the ellipse's longer axis will lie along the default base radius direction. If the secondary radius is larger than the base radius, the ellipse's longer axis will lie perpendicular to the default base radius direction. To create an ellipse with an arbitrary orientation, rotate or move the ellipse after drawing it.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands see:

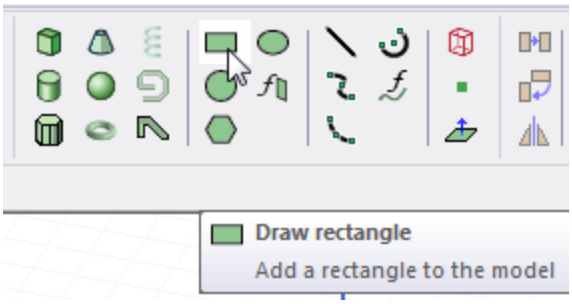
Surface Approximation and related sections:

Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing a Rectangle

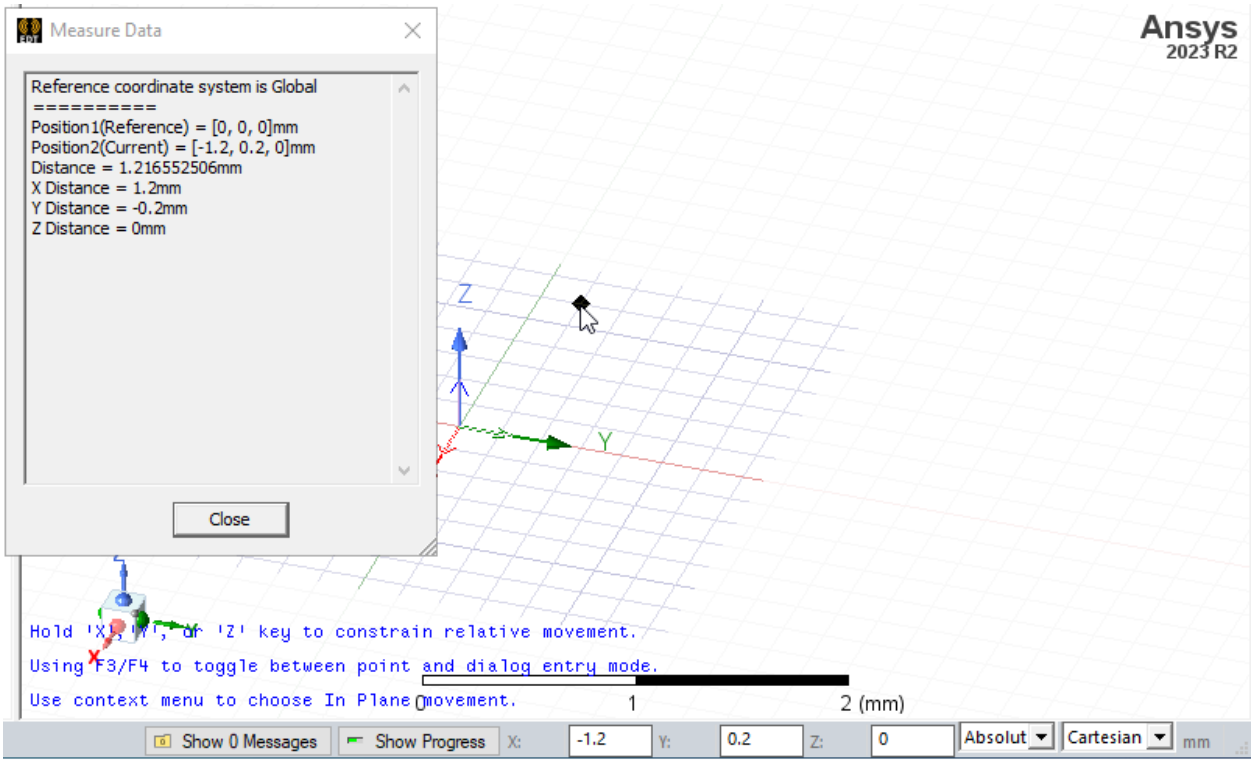
Draw a rectangle (or square) by selecting two diagonally opposite corners. Before you draw a rectangle, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Rectangle** or, on the **Draw** ribbon tab, click the **Draw rectangle** icon:

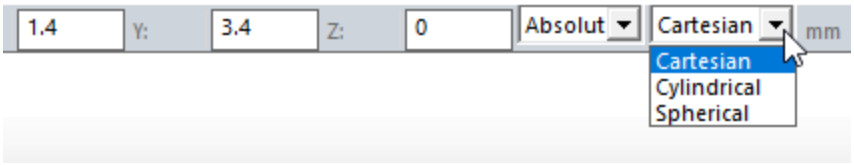


2. Select the first diagonal corner in one of the following ways:
   - Click the point. The Measure Data dialog shows the coordinates and other information.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



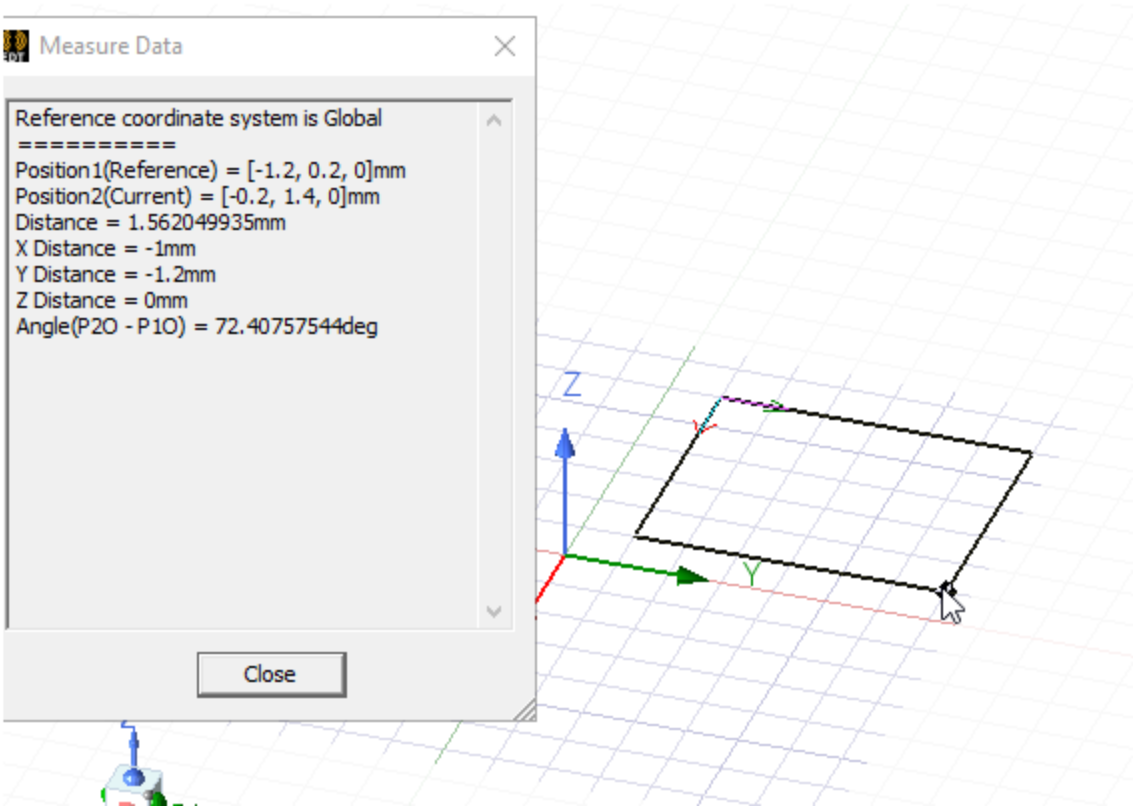
- Type the point's coordinates in the text boxes in the status bar. The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the cylinder in Cartesian, Cylindrical, or Spherical coordinates.



To delete the selected point and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the second corner of the rectangle in one of the following ways:
   - Click the point.

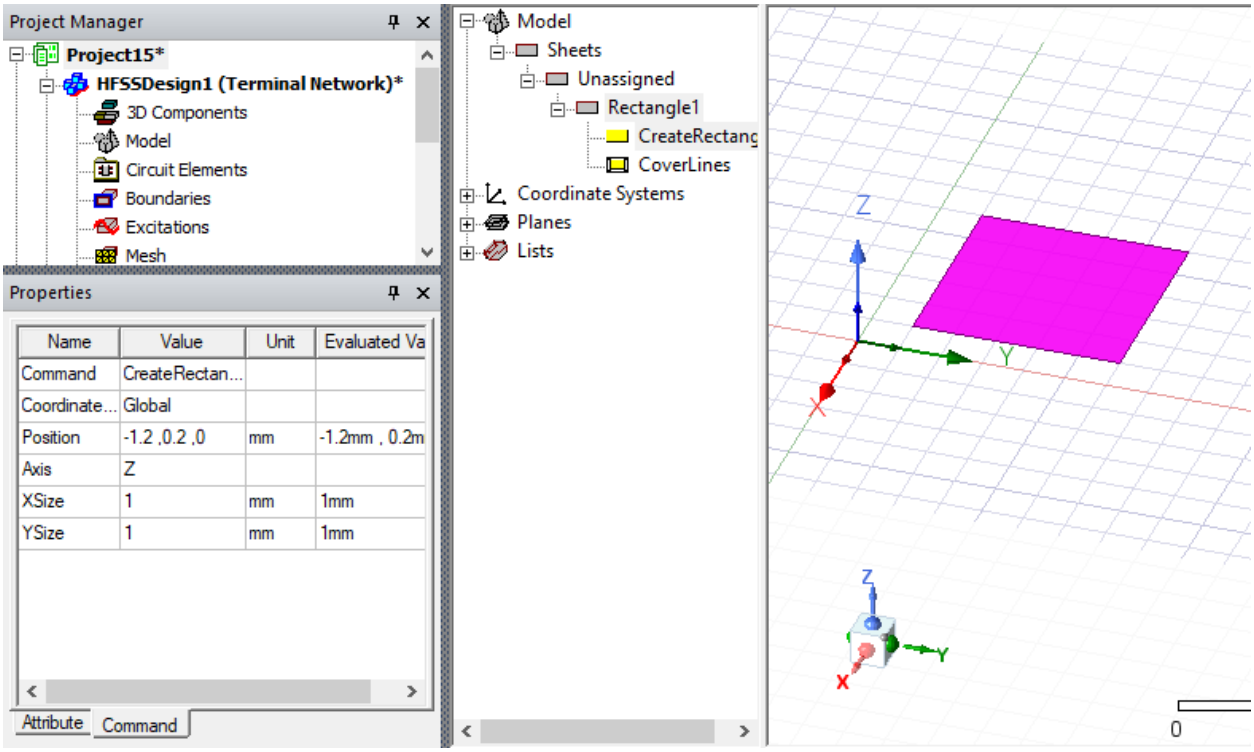


   - Type the coordinates of the point relative to the first diagonal corner in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

   If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. You can also use the docked Properties window.

4. Click **OK**. If the **Automatically cover closed polyline** option is selected in the **Modeler Options** window, the rectangle will be covered, resulting in a 2D sheet object, as shown in the History tree in the figure below. Otherwise it will be a closed 1D polyline object.

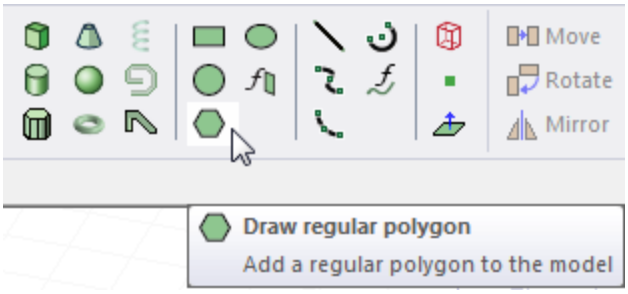
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Drawing a Regular Polygon

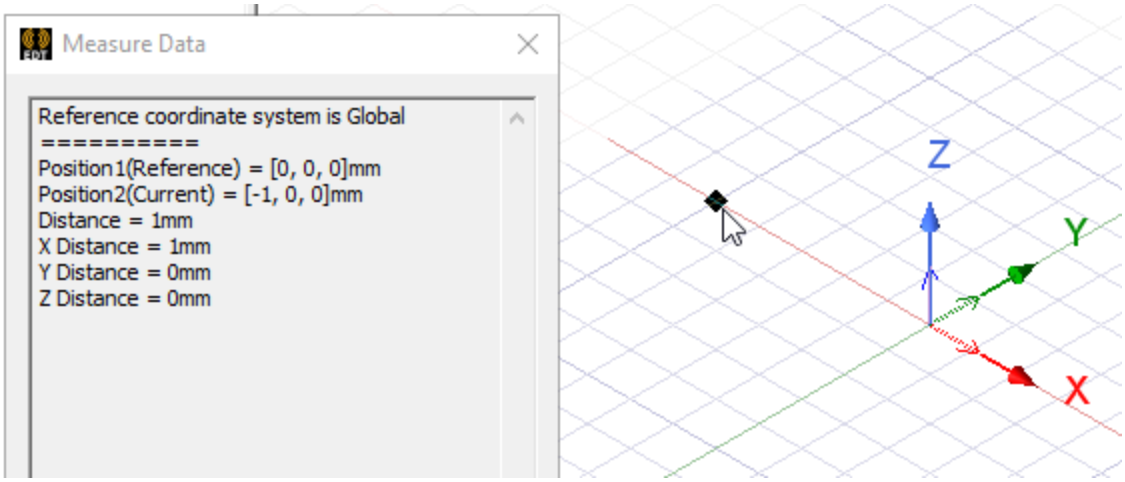
A regular polygon is a 2D object with three or more equal sides. Regular polygons are useful for drawing faceted 2D objects. Before you draw a regular polygon, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. Click **Draw>** **Regular Polygon** or, on the **Draw** ribbon tab, click the **Draw regular polygon** icon:



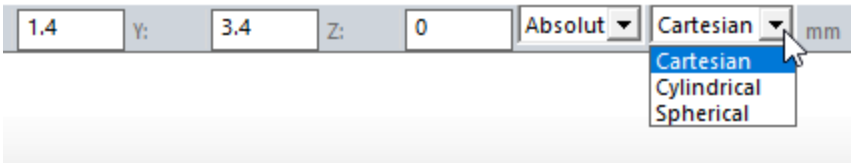
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the center point of the polygon in one of the following ways:
   - Click the point.



   - Type the point's coordinates in the text boxes in the status bar.

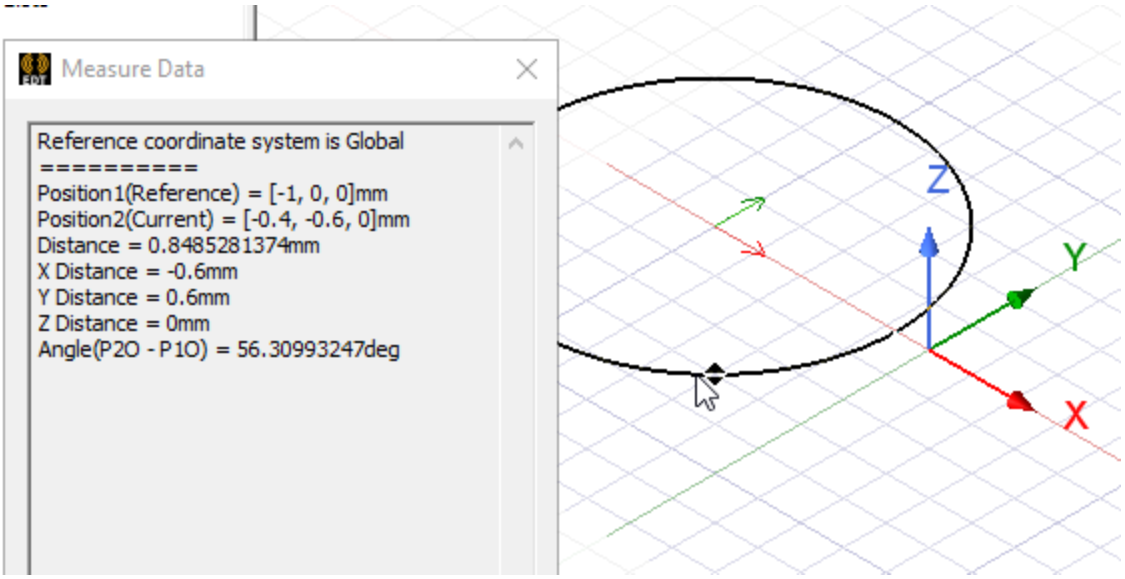
     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the regular polygon in Cartesian, Cylindrical, or Spherical coordinates.



3. Specify the polygon's radius, the distance from the center point to one of the polygon's vertices, in one of the following ways:

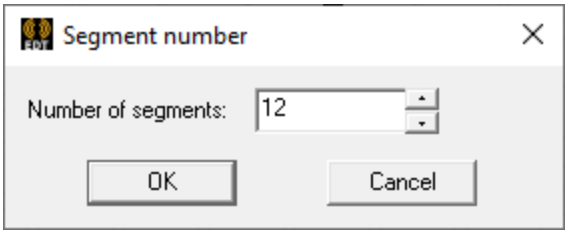
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the point.



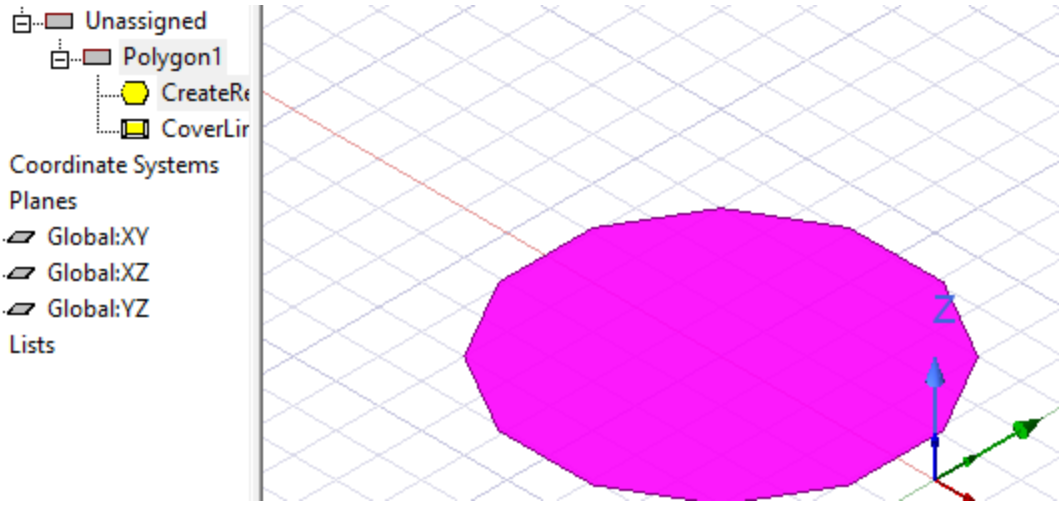
- Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

4. In the **Segment number** dialog box, enter the **Number of segments** in the polygon, and then click **OK**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click OK. The polygon appears in the Modeler window and in the History tree.



5. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. If you have edited the Properties dialog, click **OK**.

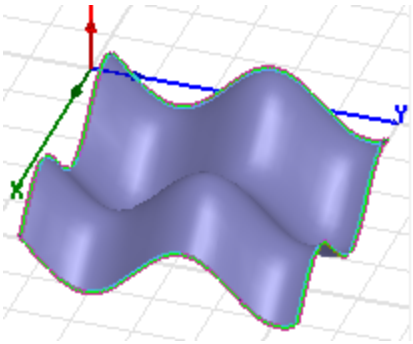
> **Note:**
>
> The radius is measured from the center point to a corner of the polygon, or the intersection of two edges. It is *not* measured from the center point to the midpoint of an edge.

If the **Automatically cover closed polyline** option is selected in the **Modeler Options** window, the polygon will be covered, resulting in a 2D sheet object. Otherwise it will be a closed 1D polyline object.

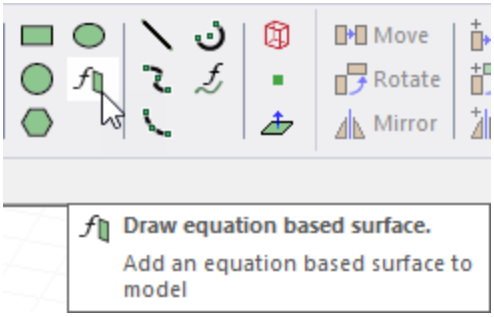
Drawing an Equation-Based Surface

Any surface that can be described by an equation in three dimensions can be drawn. Before you draw an equation-based surface, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

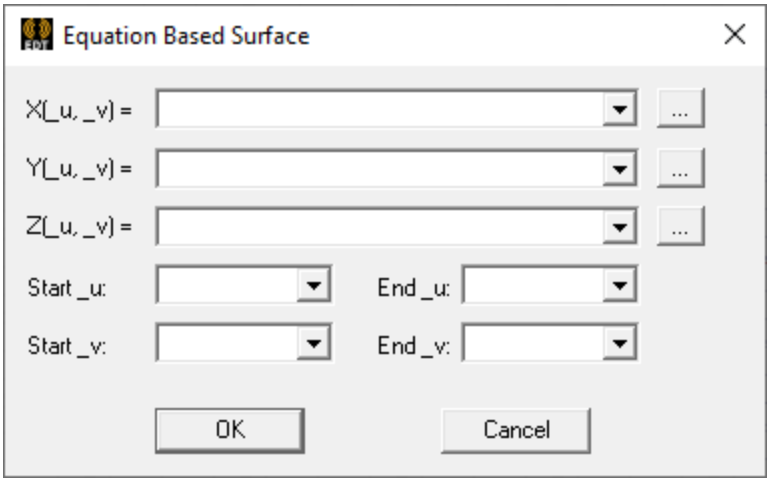
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



1. From the menu bar, click **Draw**> **Equation Based Surface** or, on the **Draw** ribbon tab, click the **Draw equation based surface** icon:



The **Equation Based Surface** dialog box opens.



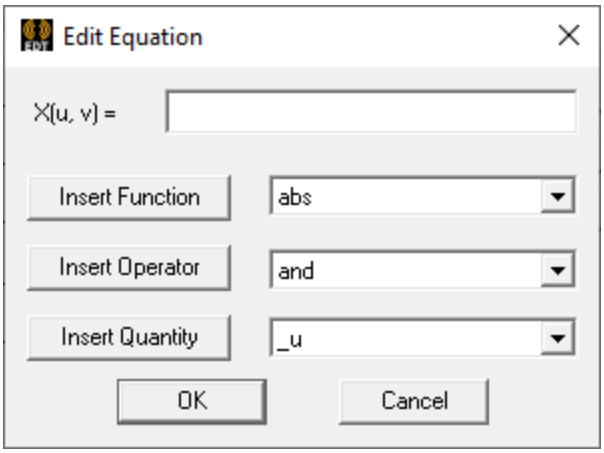
2. Type equations for **X(_u, _v)**, **Y(_u, _v)**, and **Z(_u, _v)**.

   You can also define an equation by doing the following:

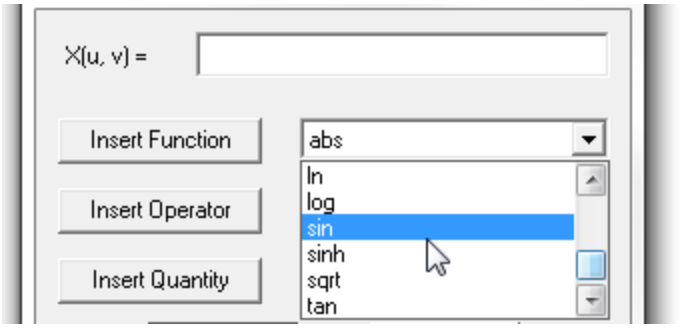
   a. Click the ellipsis button (**...**).

      The **Edit Equation** dialog box appears:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



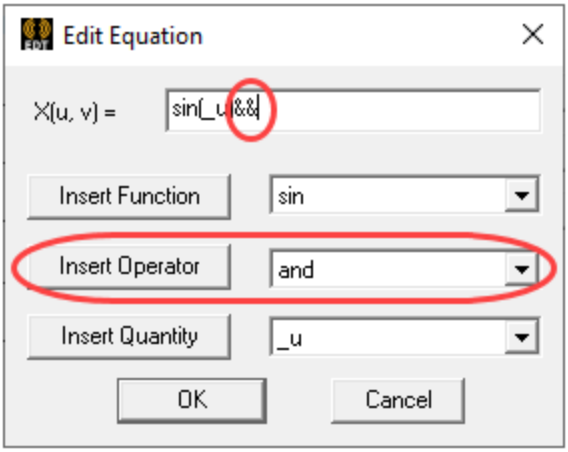
b. Do one or more of the following to define the equation:
   - Type a numerical value, variable, or expression directly in the text box
   - Build an equation for X(u, v), Y(u, v), or Z(u, v), by selecting the function, operator, and quantity to insert into the equation from the three drop-down menus:

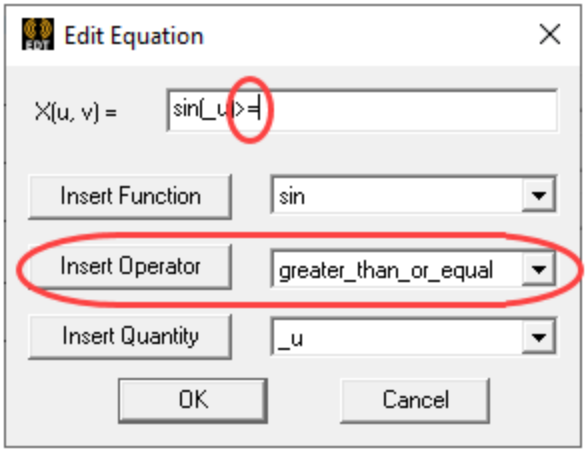


     Click the corresponding **Insert Function, Insert Operator,** or **Insert Quantity** buttons to insert the selections into the equation text box at the current cursor location. The if(, ,) function has been added under **Insert Function**.

   - The **Insert Operator** button inserts corresponding operator:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



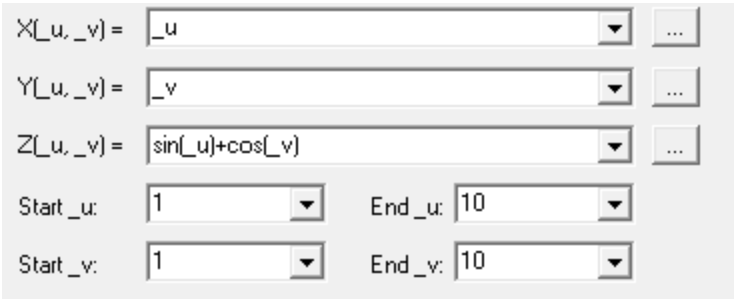


- You can combine the preceding two methods to build complex equations.

c. When you are finished defining the equation, click **OK** to close the **Edit Equation** dialog box and return to the **Equation Based Surface** dialog box.
   - The equations are taken to be meters. Use of units in equations is not properly supported..
   - While parsing expressions, equation based curves convert each variable separately to model units and assume that the resulting expression is in model units.
   - Equation based curves depend on the variable value library to correctly evaluate the units of expression.
   - Functions that do not support intrinsic (_t for curve, _u and _v for surface) will give an error if not supported.

3. Specify start values for the **Start_u** and **Start_v** fields.

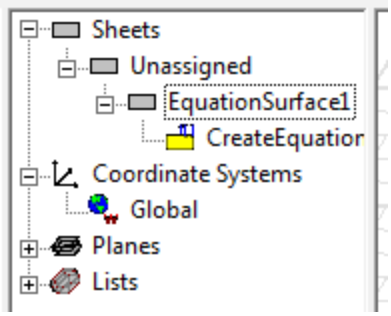
   Select end values for **End_u** and **End_v** fields.



4. Click **OK** on the **Properties** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The surface is drawn in the Modeler window. The History tree contains the Equation Surface object and the command properties.

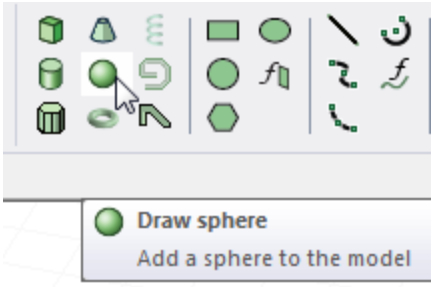


If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's attributes.

Drawing a Sphere

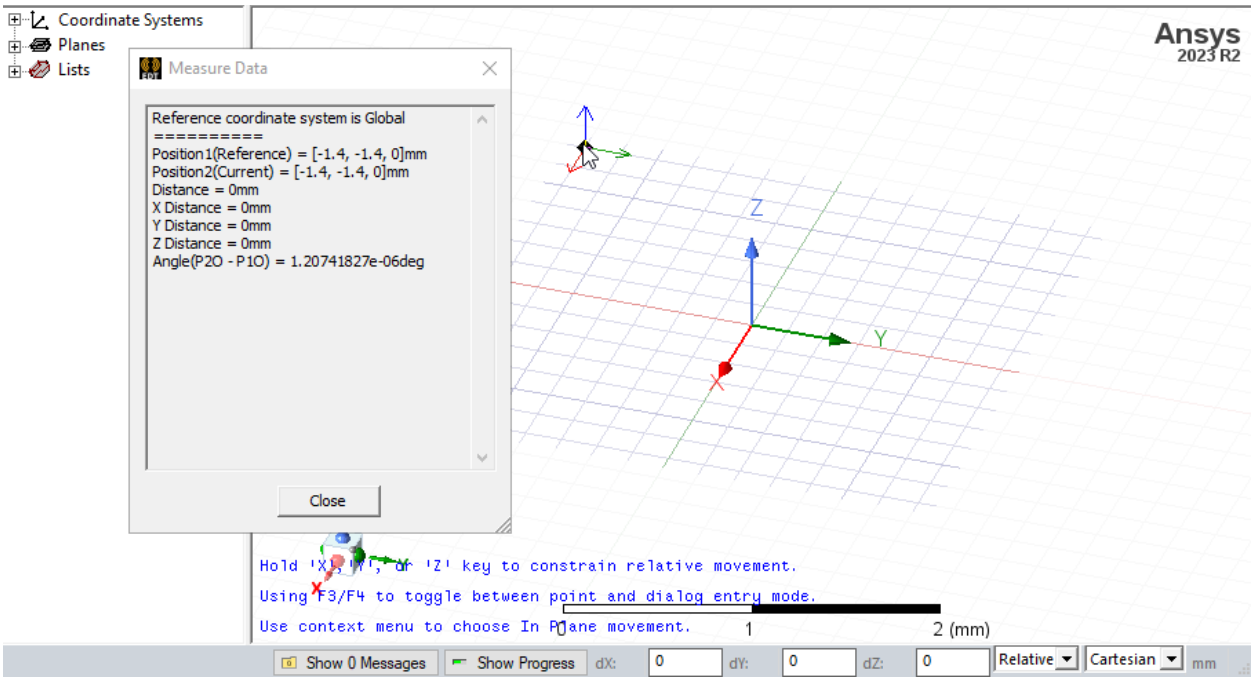
Draw a sphere, a 3D circle, by selecting a center point and a radius. Spheres are drawn as true surfaces in the modeler. Before you draw a sphere, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Sphere** or, on the **Draw** ribbon tab, click the **Draw sphere** icon:



2. Select the center point of the sphere in one of the following ways:
   - Click the point. The Measure Data dialog shows the coordinates and other information.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

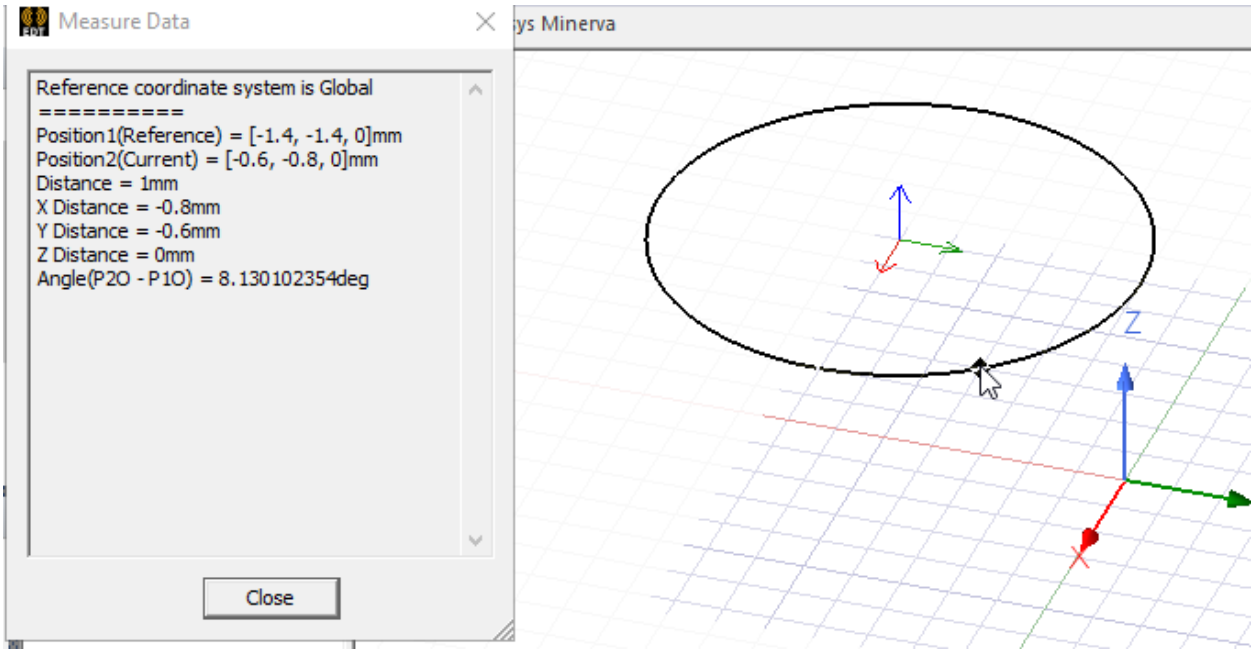


- Type the point's coordinates in the text boxes in the status bar. The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the sphere in Cartesian, Cylindrical, or Spherical coordinates.

3. Specify the radius by selecting a point on the sphere's circumference in one of the following ways:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

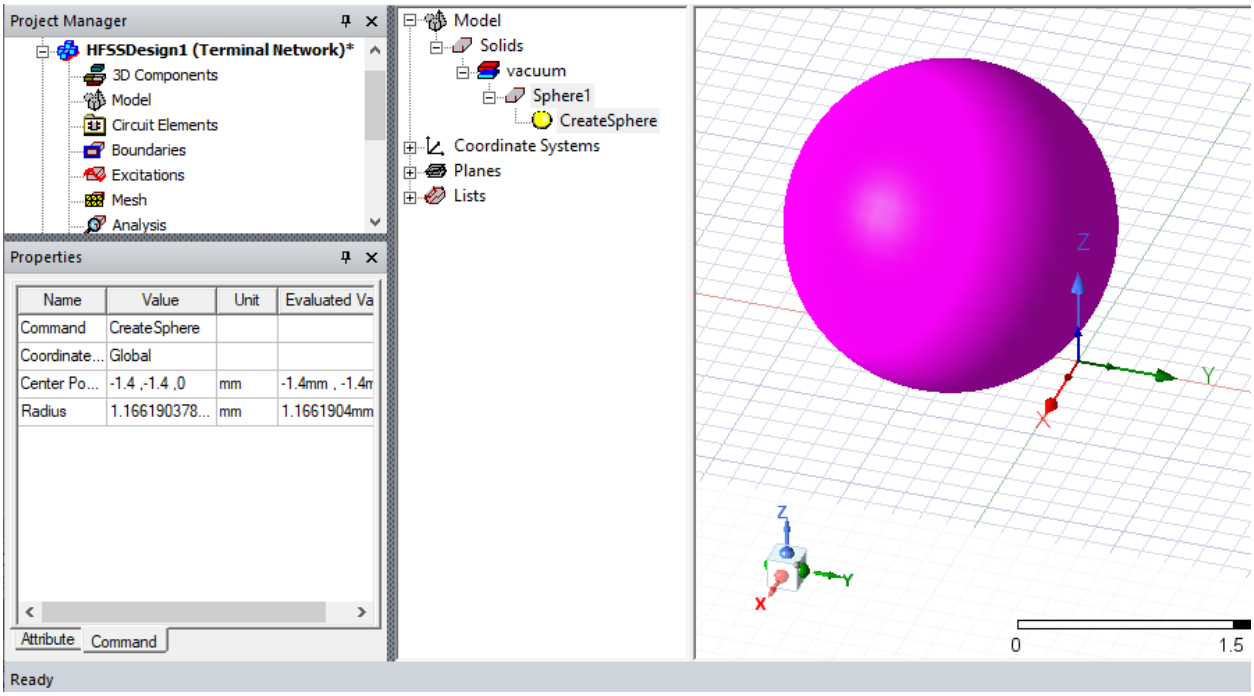
- Click the point



- Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. You can also use the docked Properties window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click **OK**.

> **Note:**
>
> The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands see:
>
> Surface Approximation and related sections:
>
> Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Related Topics

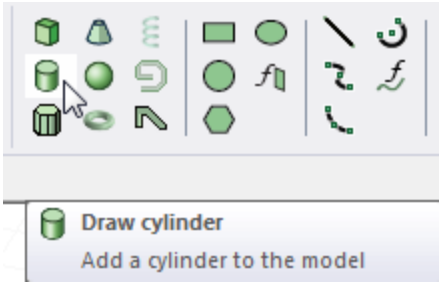
Modifying Objects

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

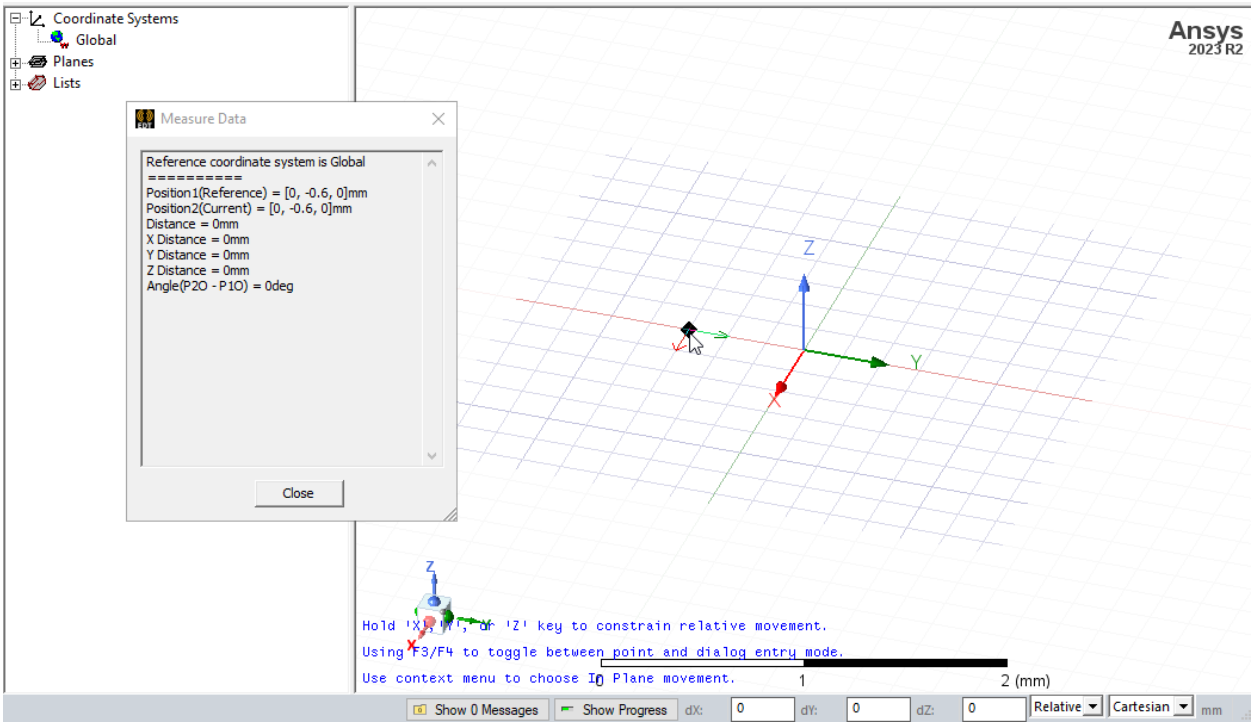
Drawing a Cylinder

Draw a cylinder by selecting a center point, radius, and height. Cylinders are drawn as true surfaces in the modeler. Before you draw a cylinder, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Cylinder** or, on the **Draw** ribbon tab, click the **Draw cylinder** icon:



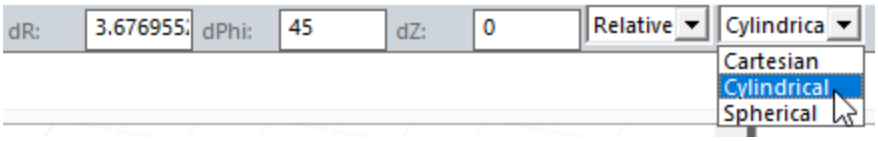
2. Select the center point of the cylinder's base circle in one of the following ways:
   - Click the point. The Measure Data dialog shows the coordinates and other information.



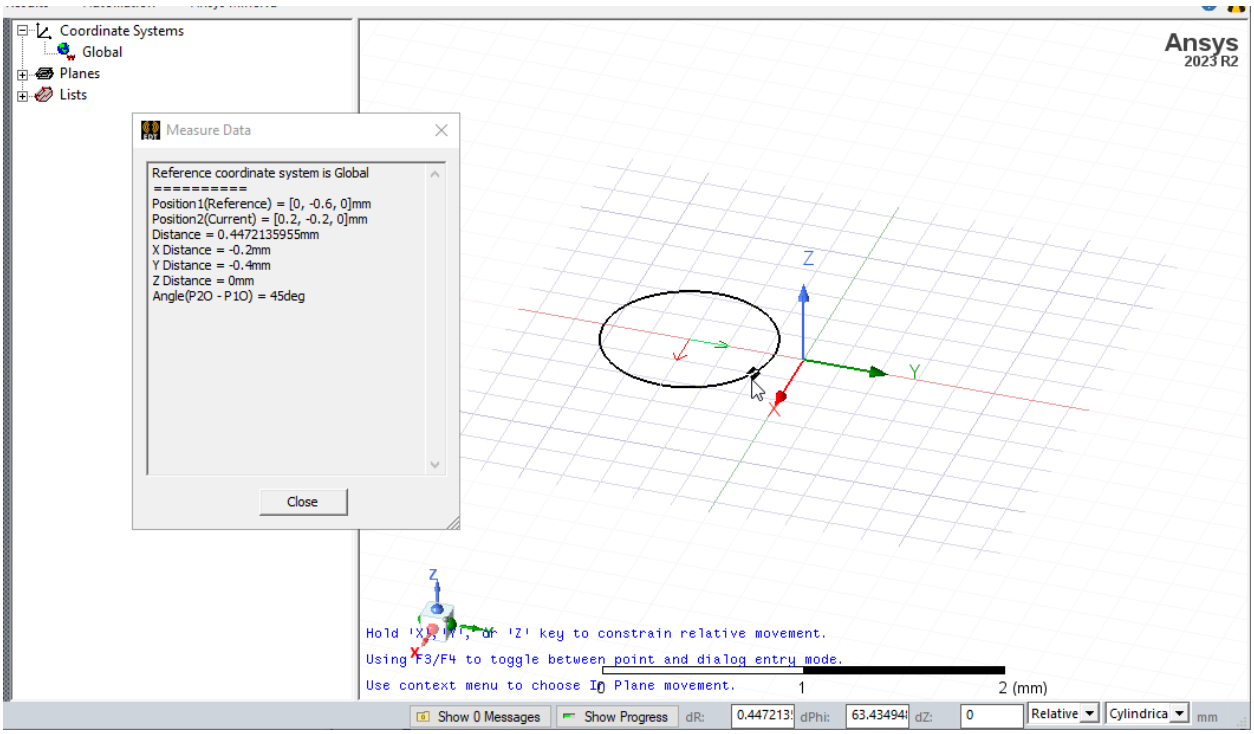
   - Type the point's coordinates in the text boxes in the status bar. The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the cylinder in Cartesian, Cylindrical, or

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Spherical coordinates.

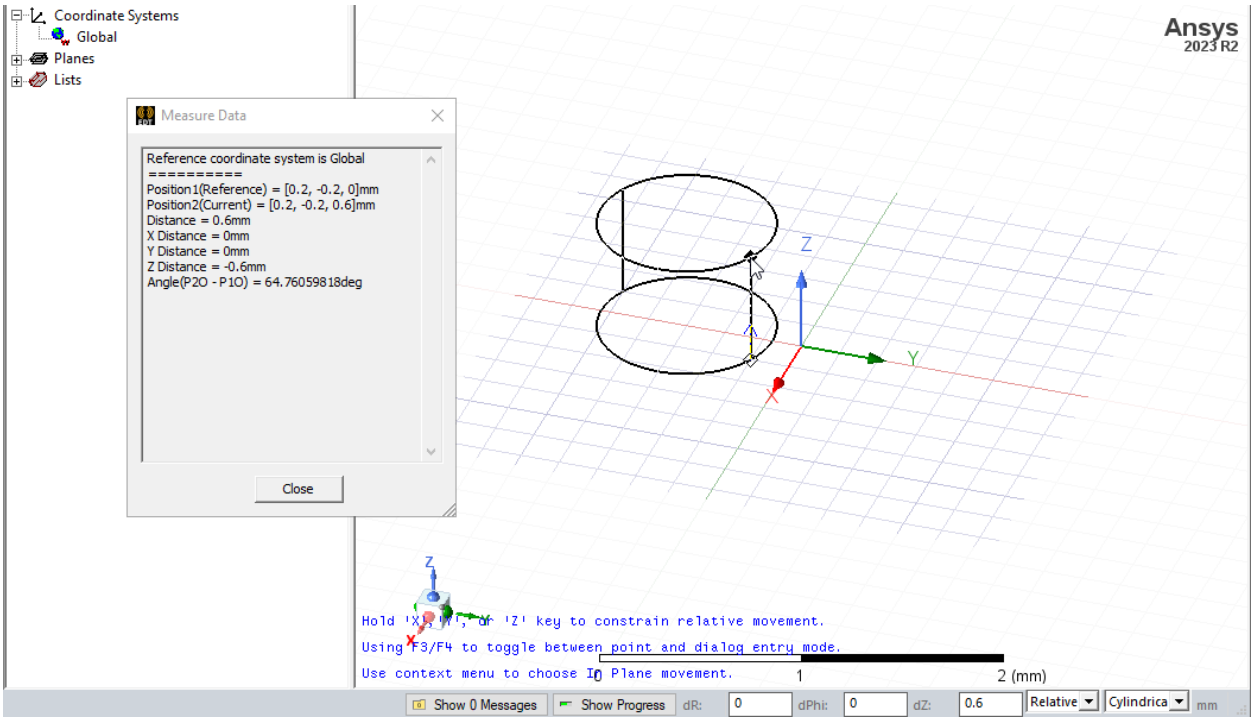


3. Specify the radius by selecting a point on the base circle's circumference in one of the following ways:
   - Click the point.



   - Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.
4. Specify the cylinder's height by selecting a point on the axis perpendicular to the base circle's plane. Select the point by clicking the point or typing the coordinates in the text box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

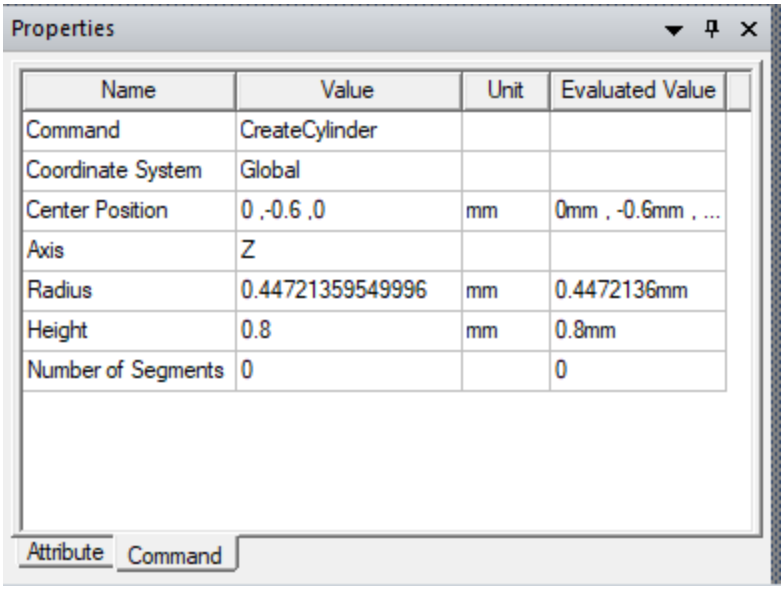


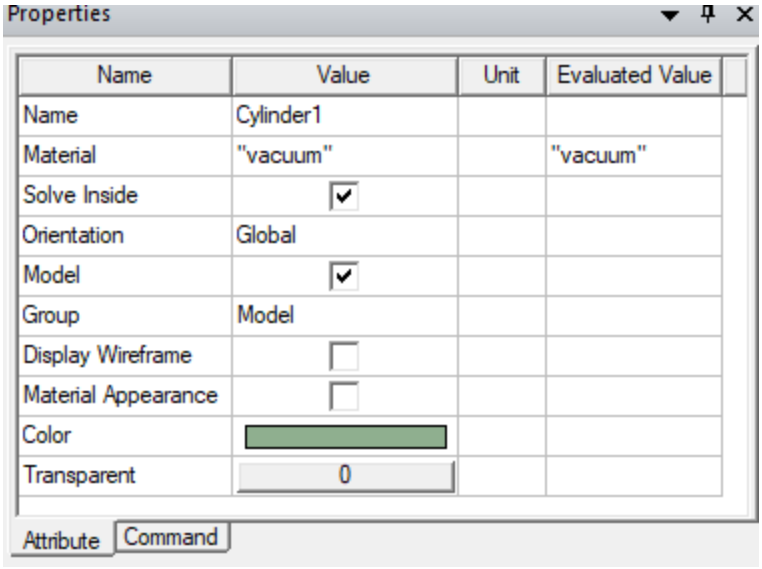
> **Note:**
>
> If you create a cylinder with a height of zero, Ansys Electronics Desktop draws a circular sheet object.

Double click, or right click and select **Done** from the short cut menu. If the Modeler option for editing properties of new primitives is checked, the *Properties* dialog box appears, enabling you to modify the object's Command and Attribute properties. You can also use

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

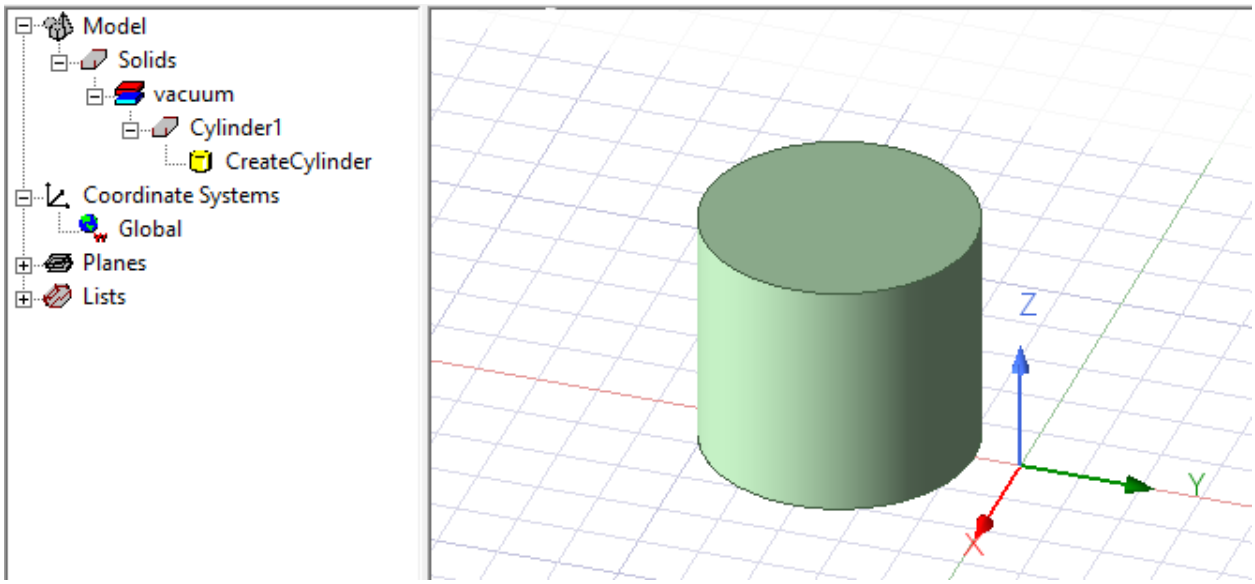
the docked Properties window.





5. If you used the Properties window, click **OK**. The cylinder appears in the Modeler window and is listed in the History tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands see:

Surface Approximation and related sections:

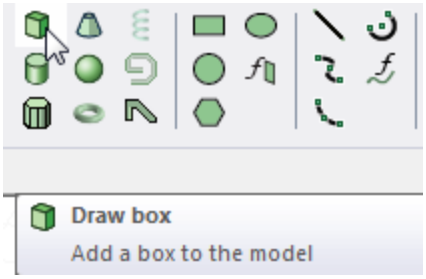
Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing a Box

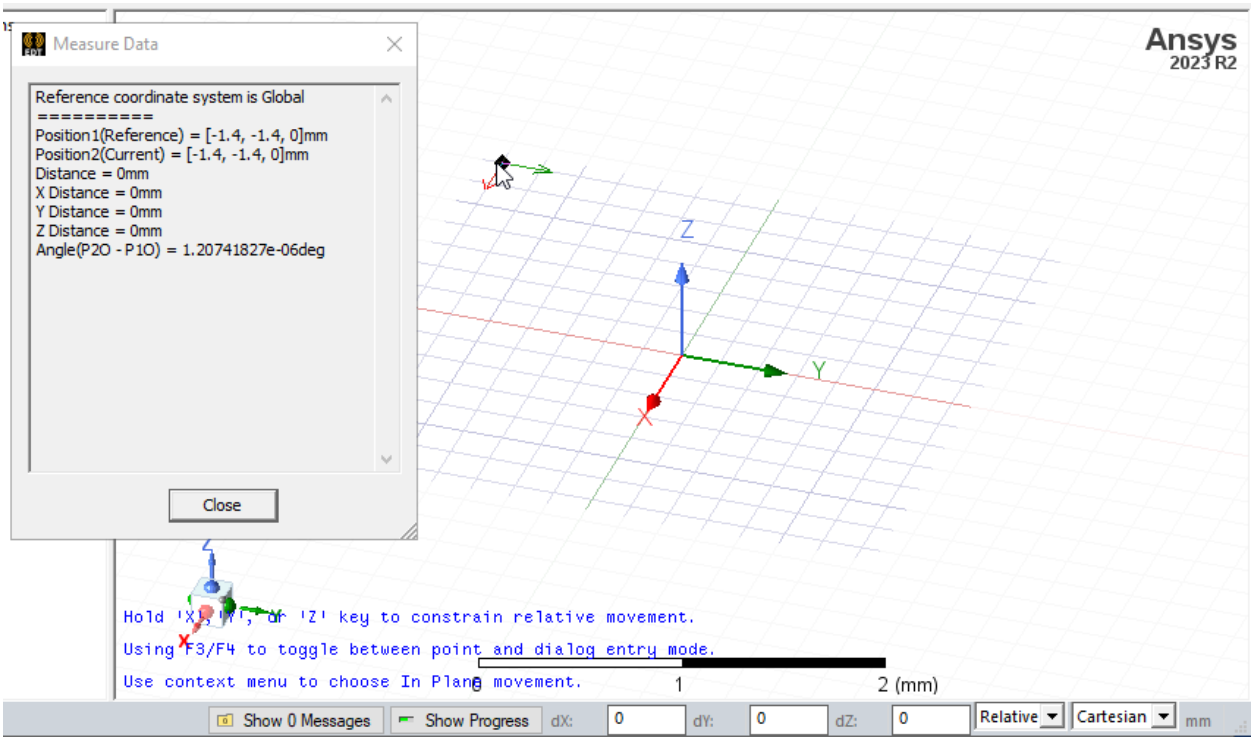
Draw a box by selecting two diagonally opposite corners of the base rectangle, then specifying the height. Before you draw a box, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw>** **Box** or, on the **Draw** tab of the ribbon, click the **Draw box** icon:

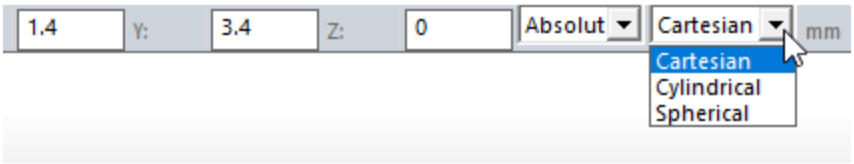
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select the first diagonal corner of the base rectangle in one of the following ways:
   - Click the point. The Measure Data dialog shows the coordinates and other information.



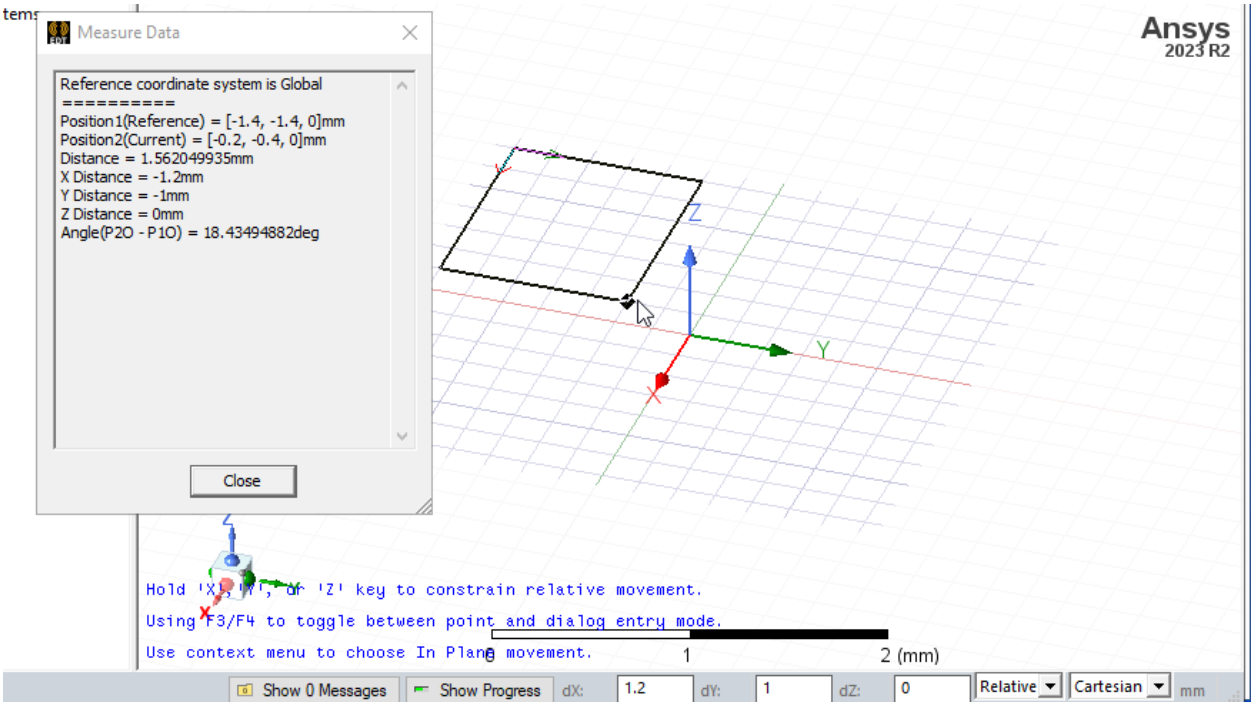
   - Type the point's coordinates in the text boxes in the status bar. The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the box in Cartesian, Cylindrical, or Spherical coordinates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

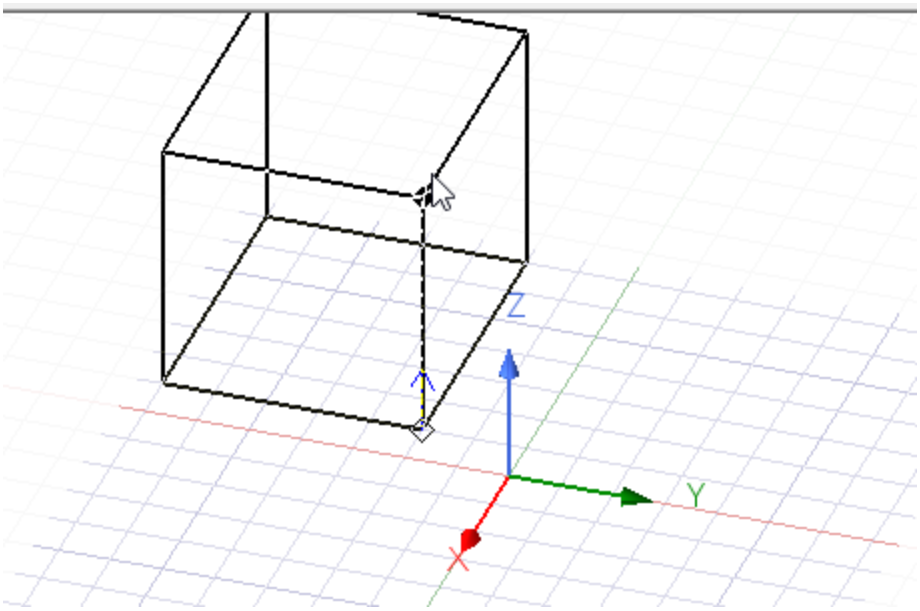
To delete the selected point and start over, press **Esc** or click **Escape Draw Mode** on the shortcut menu.

3. Select the second corner of the base rectangle in one of the following ways:
   - Click the point.



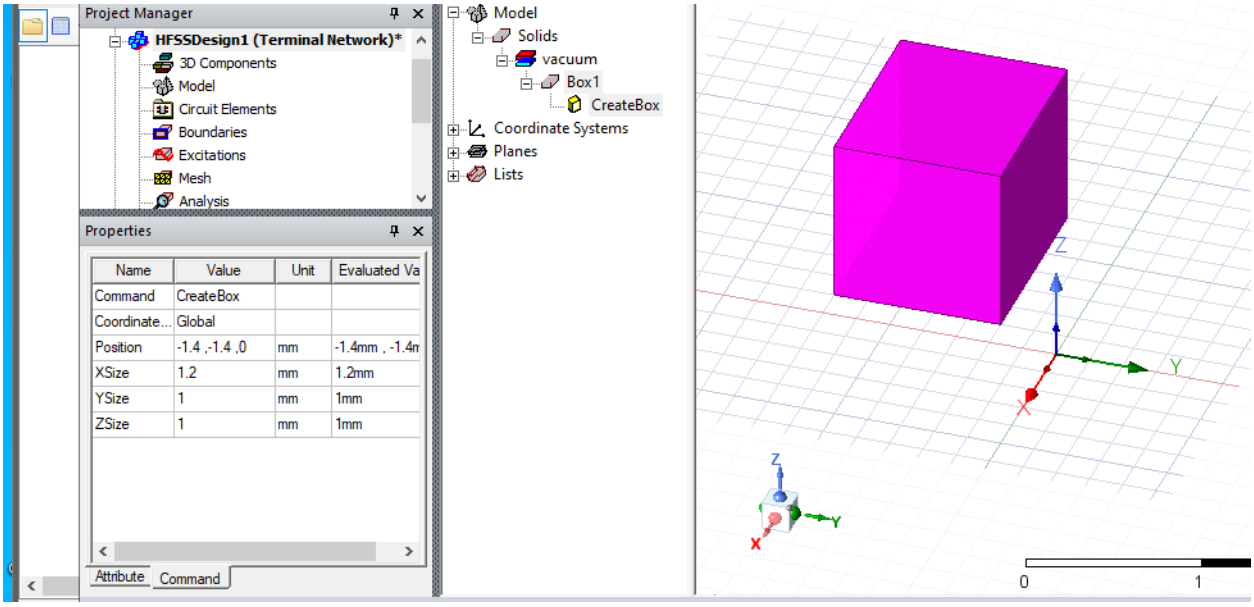
   - Type the coordinates of the point relative to the first diagonal corner in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.
4. Specify the height of the box by selecting a point on the axis perpendicular to the base rectangle. Select the point by clicking the point or typing the coordinates in the **dX, dY,** and **dZ** boxes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Double -click or right-click and select **Done** from the shortcut menu.

   If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. You can also use the docked Properties window.

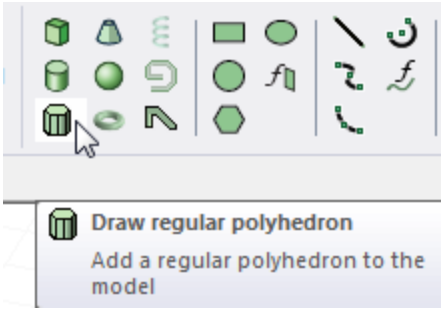


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

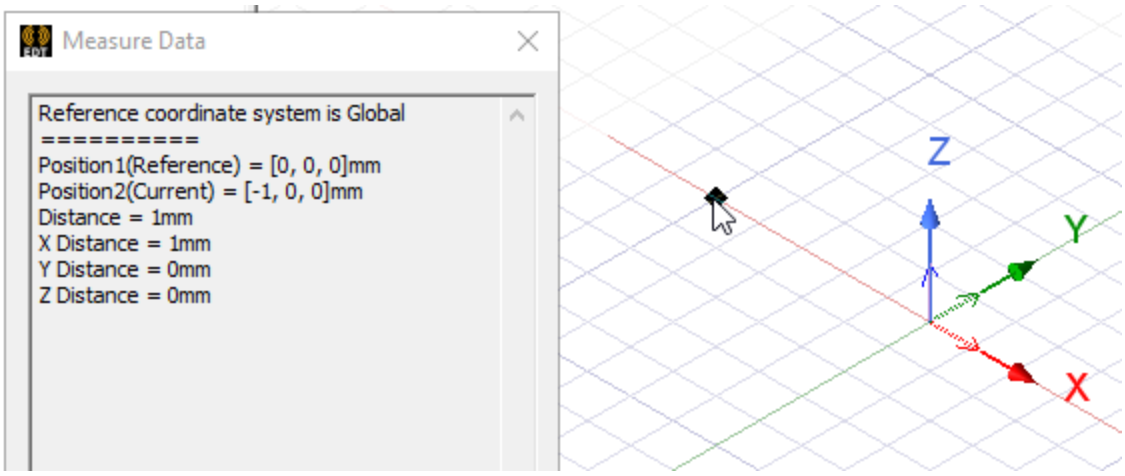
Drawing a Regular Polyhedron

In the modeler, regular polyhedrons are 3D objects with regular polygon faces; each face has three or more equal sides. Regular polyhedrons are useful for drawing faceted 3D objects. Before you draw a regular polyhedron, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

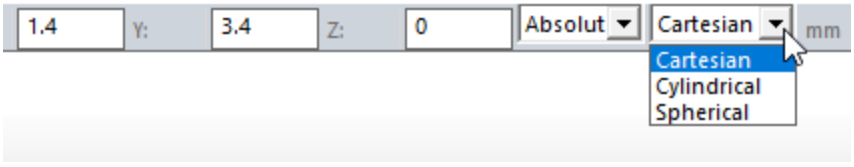
1. From the menu bar, click **Draw**> **Regular Polyhedron** or, on the **Draw** ribbon tab, click the **Draw regular polyhedron** icon:



2. Select the center point of the polyhedron in one of the following ways:
   - Click the point.

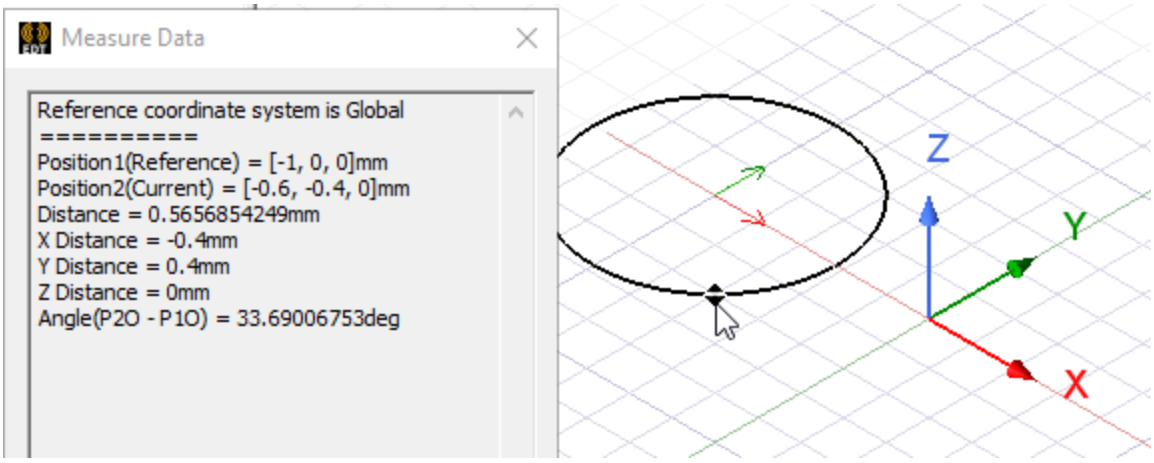


   - Type the point's coordinates in the text boxes in the status bar.
     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the regular polygon in Cartesian, Cylindrical, or Spherical coordinates.



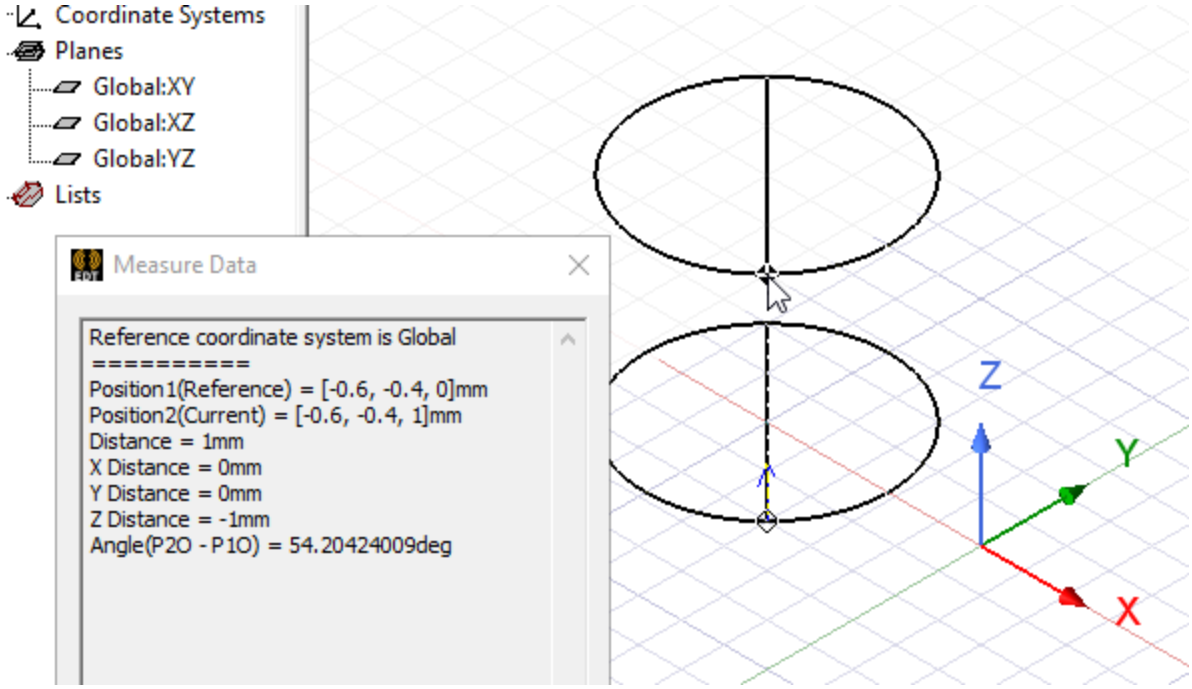
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the radius of the polyhedron, the distance from the center point to one of the polyhedron's vertices, in one of the following ways:
   - Click the point.



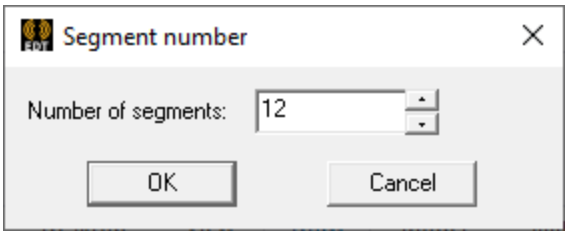
   - Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

4. Select the height of the polyhedron, the distance from the origin plane along the normal from the radius point.

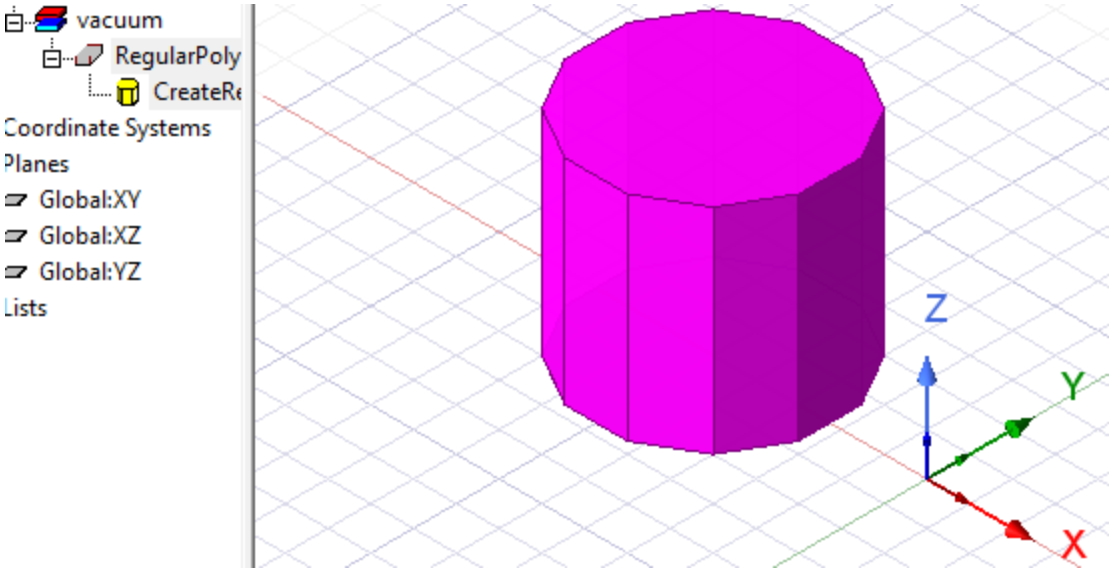


5. In the **Segment number** dialog box, enter the **Number of segments** in the polyhedron, and then click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Click **OK**. The **Segment number** dialog closes and the regular polyhedron appears in the Modeler window and in the History Tree.



6. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. Click **OK**.

> **Note:**
>
> The radius is measured from the center point to a corner of the polygon, or the intersection of two edges. It is *not* measured from the center point to the midpoint of an edge.

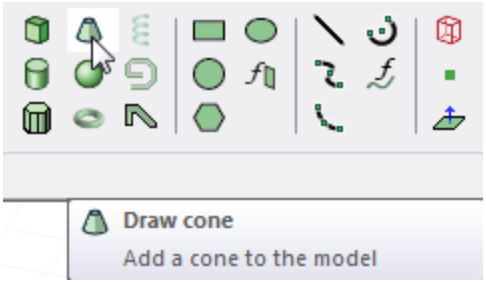
If the **Automatically cover closed polyline** option is selected in the **Modeler Options** dialog box, the polygon will be covered, resulting in a 2D sheet object. Otherwise it will be a closed 1D polyline object.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

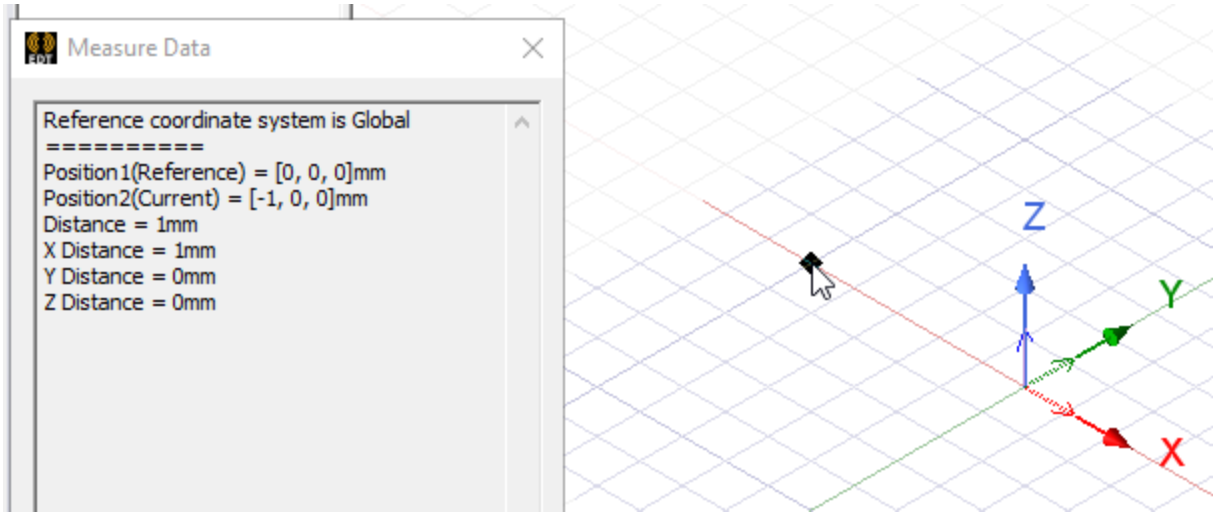
Drawing a Cone

Draw a cone by selecting the center point and radius of the cone's base circle, then specifying the radius of the cone's top circle and the cone's height. Cones are drawn as true surfaces in the modeler. Before you draw a cone, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

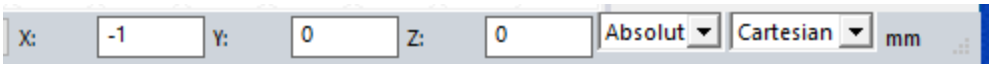
1. From the menu bar, click **Draw**> **Cone** or, on the **Draw** ribbon tab, click the **Draw cone** icon:



2. Select the center point of the cone's base circle in one of the following ways:
   - Click the point.



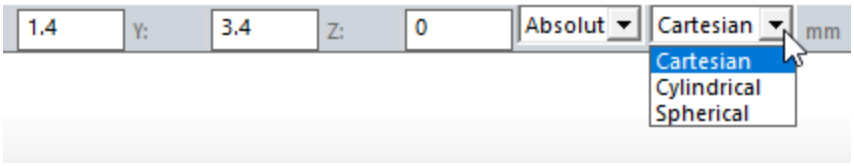
   - Type the point's coordinates in the text boxes in the status bar.



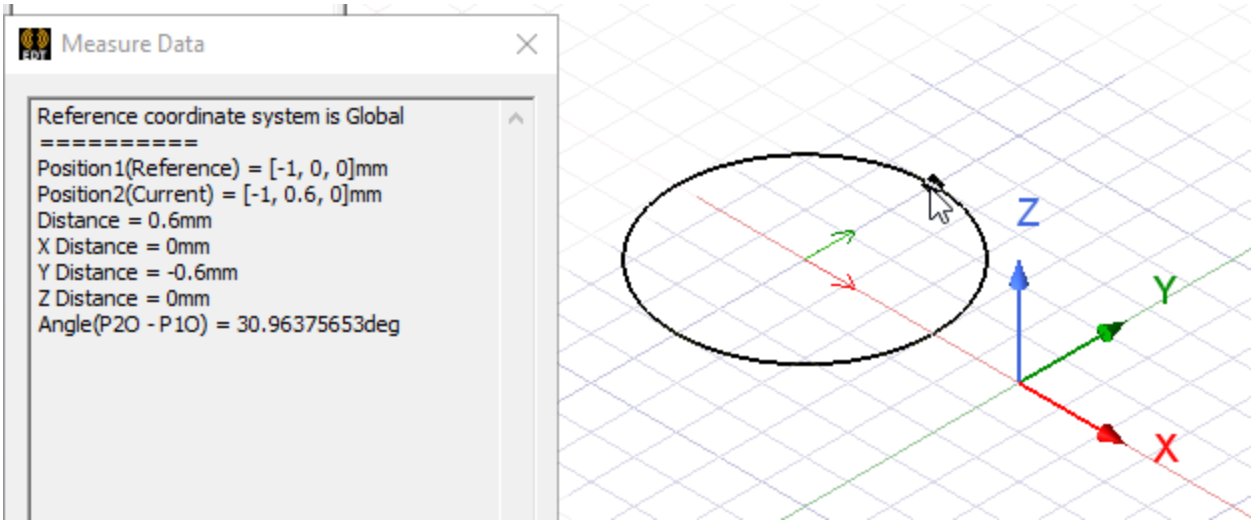
   The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the cone in Cartesian,

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

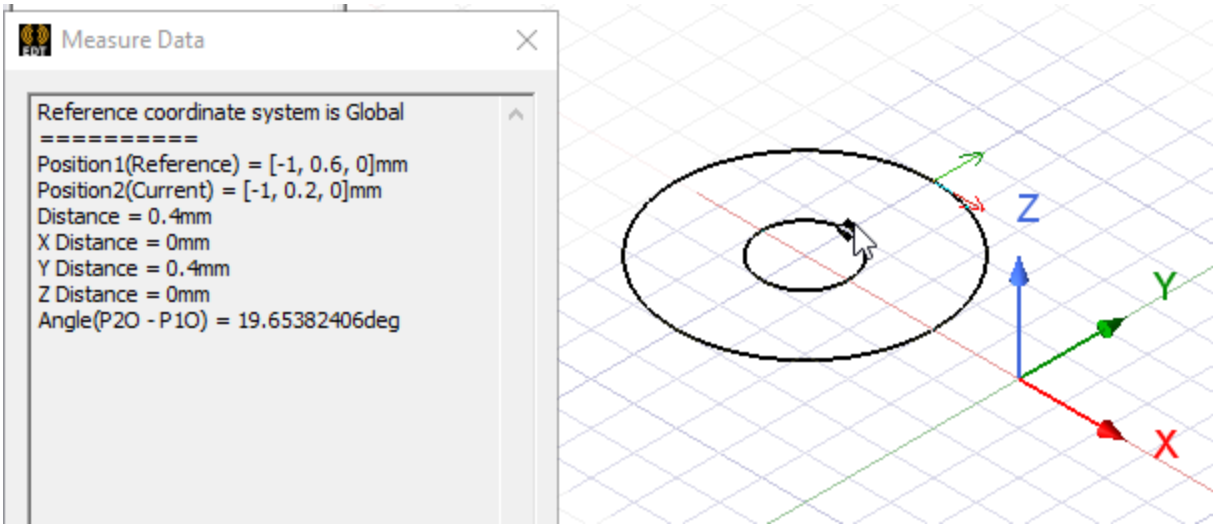
Cylindrical, or Spherical coordinates.



3. Specify the radius of the cone's base circle by selecting a point on the base circle's circumference. Select the point in one of the following ways:
   - Click the point.

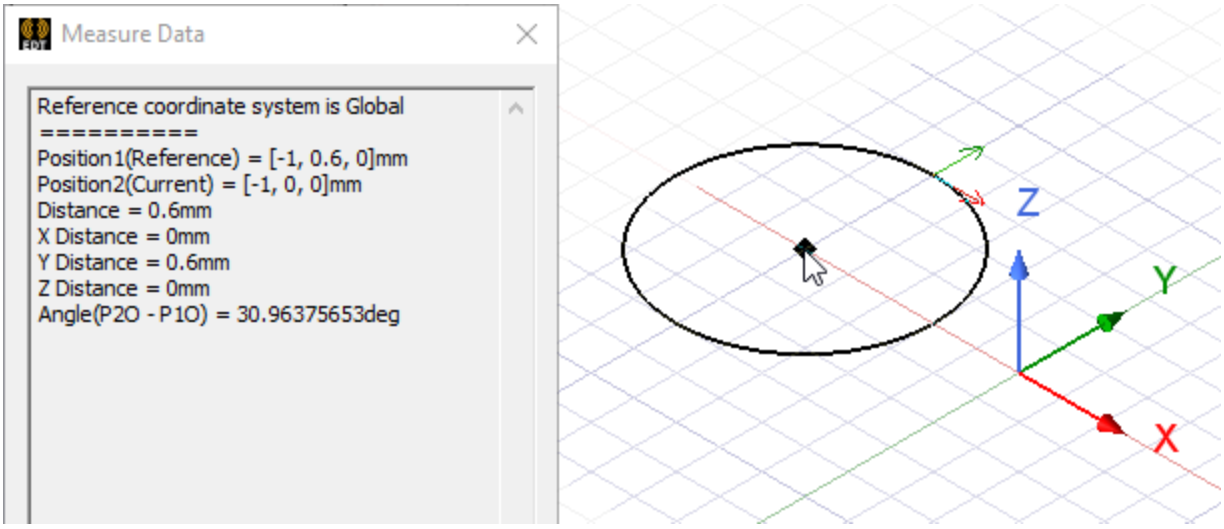


   - Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.
4. Specify the radius of the cone's top circle by selecting a point on its circumference. Select the point by clicking it or typing its coordinates in the **dX, dY,** and **dZ** boxes.

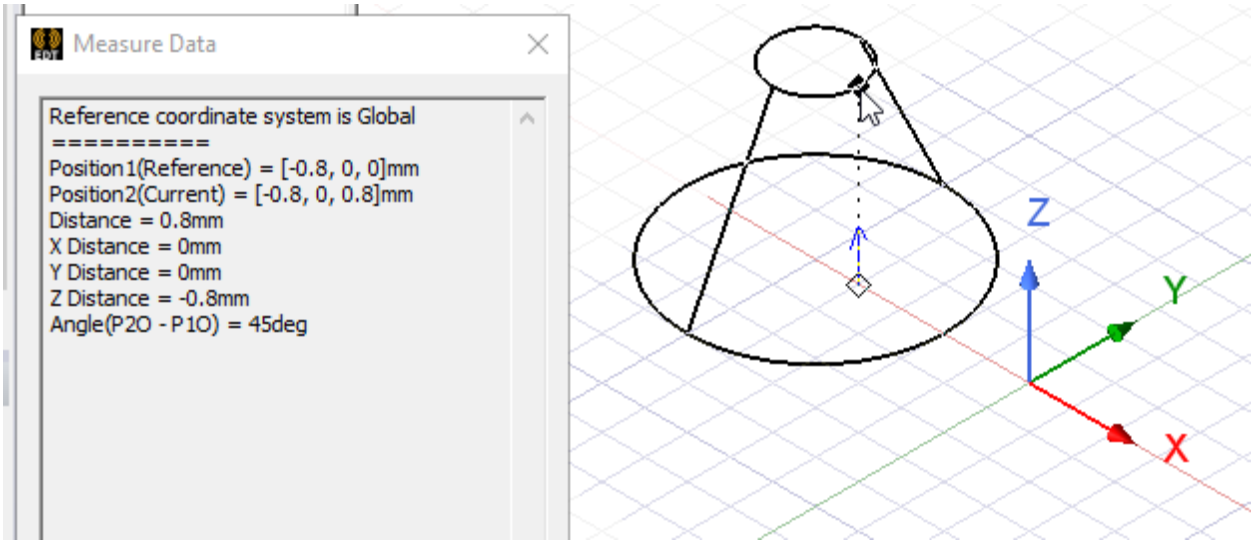


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To create an apex, select the same center point as the cone's base circle.



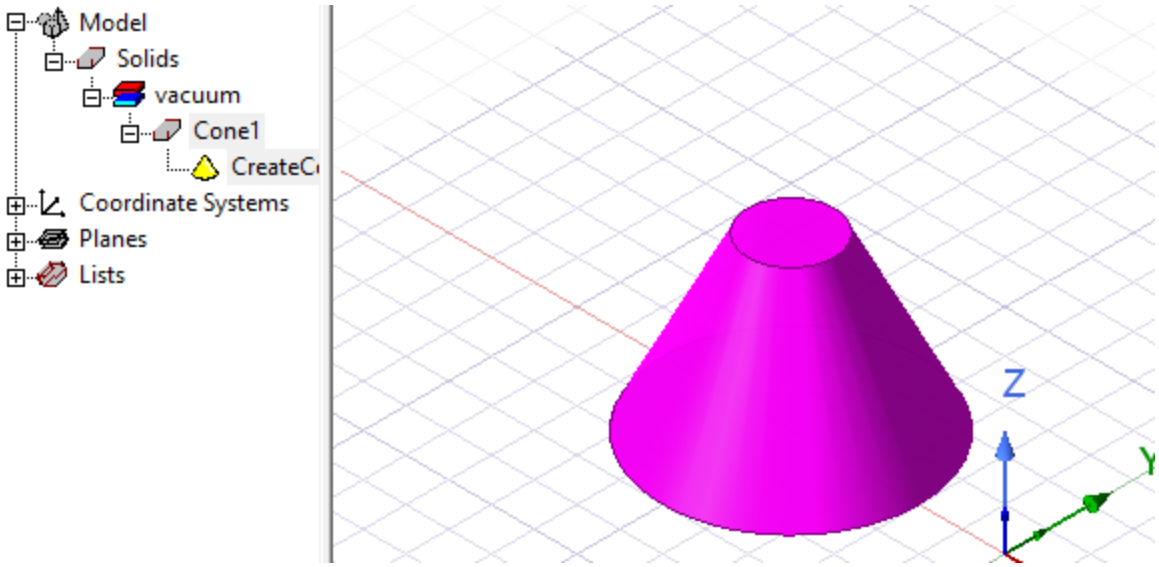
5. Specify the height of the cone by selecting a point on the axis perpendicular to the base circle's plane. Select the point by clicking the point or typing the coordinates in the **dX, dY,** and **dZ** boxes.



6. Double click, or right click and select **Done** from the short cut menu. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. The cone appears in the Modeler window

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and in the History Tree.



Note:

The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands see:

Surface Approximation and related sections:

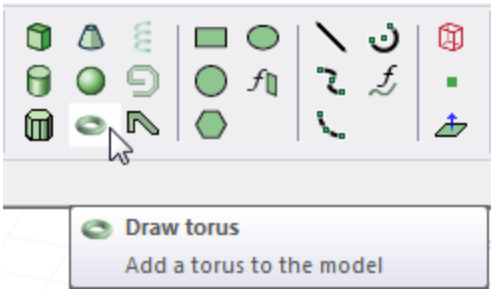
Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing a Torus

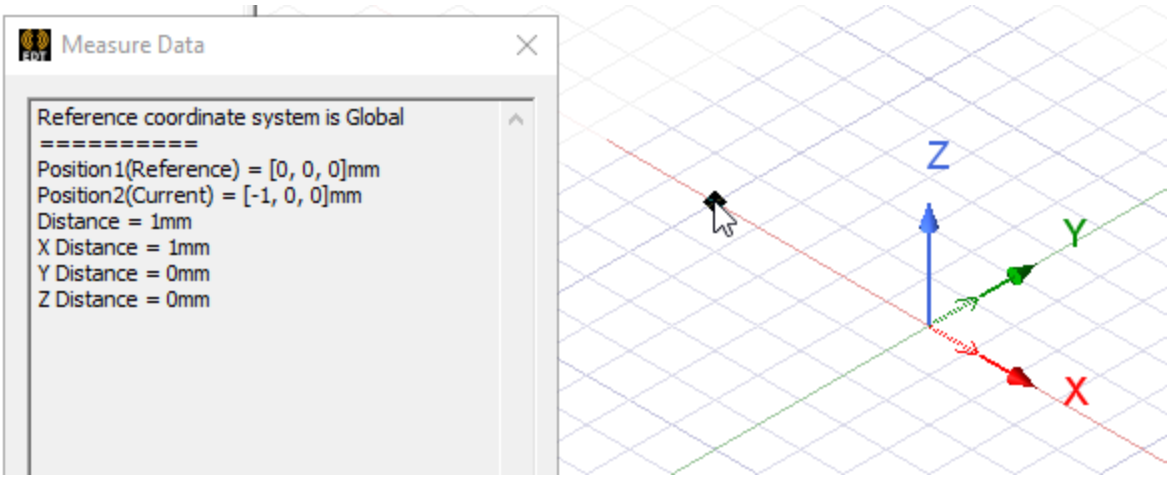
Draw a torus by selecting its center point, major radius, and minor radius. The modeler then sweeps a circle around a circular path. A torus is drawn as true surfaces in the modeler. Before you draw a torus, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Torus** or, on the **Draw** ribbon tab, click the **Draw torus** icon:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

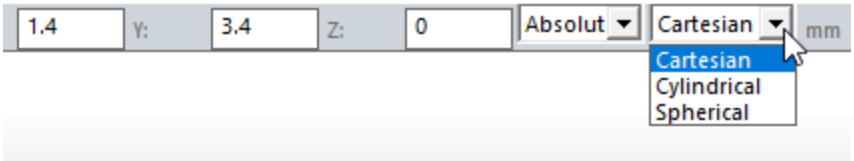


2. Select the center point of the torus in one of the following ways:
   - Click the point.



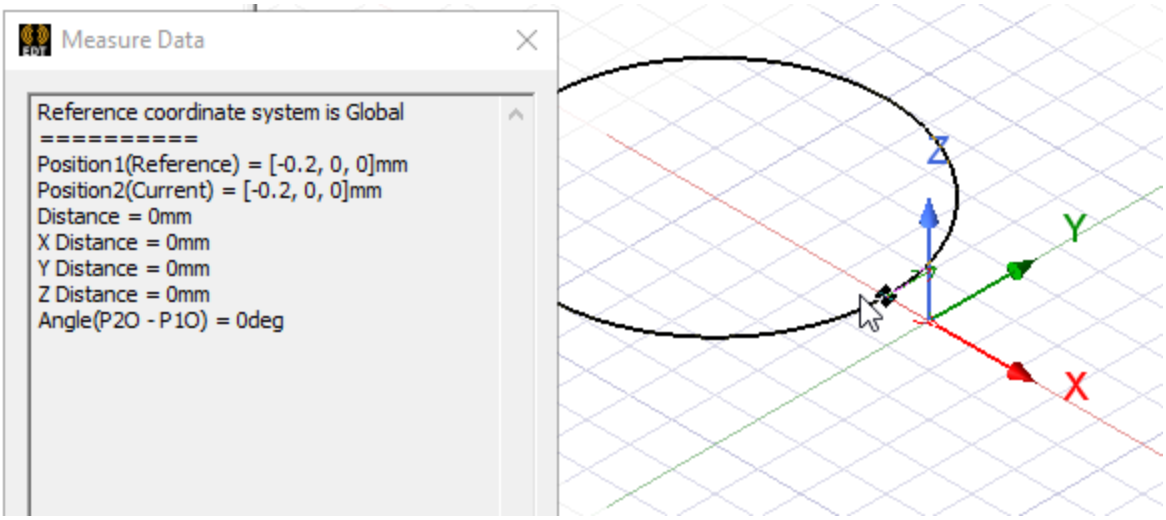
   - Type the point's coordinates in the text boxes in the status bar.

     The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the torus in Cartesian, Cylindrical, or Spherical coordinates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Specify the major radius by selecting a point in one of the following ways:
   - Click the point.

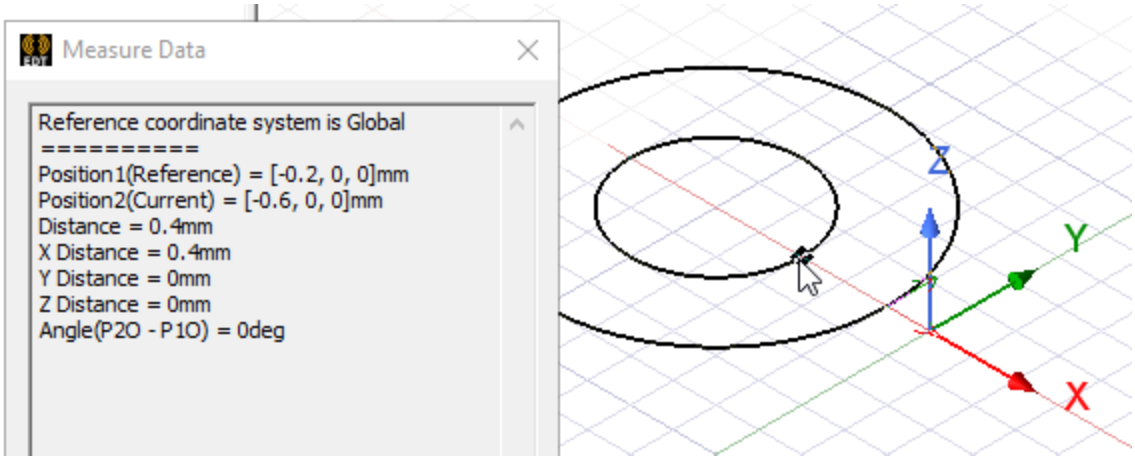


   - Type the coordinates of the point relative to the center point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

   The major radius determines the diameter of the torus.

4. Specify the minor radius by selecting a point relative to the major radius point.

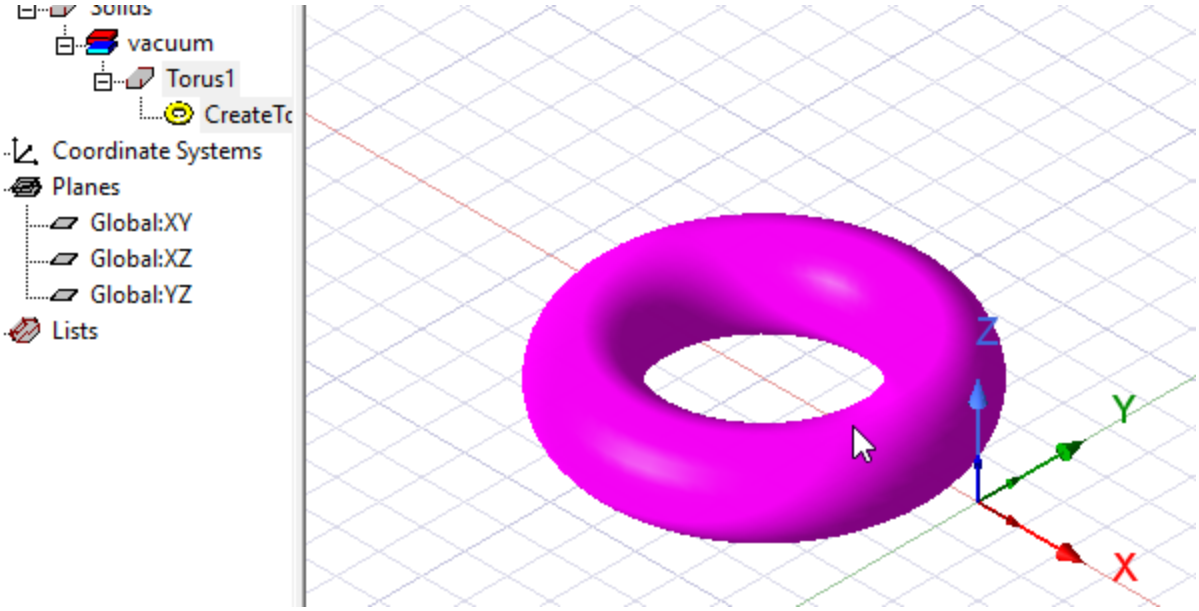
   The minor radius determines the diameter of the "donut hole".



   If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Double-click or right-click to open the shortcut menu and click **Done**. The torus appears in the Modeler window and in the History Tree.



6. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. Click **OK**.

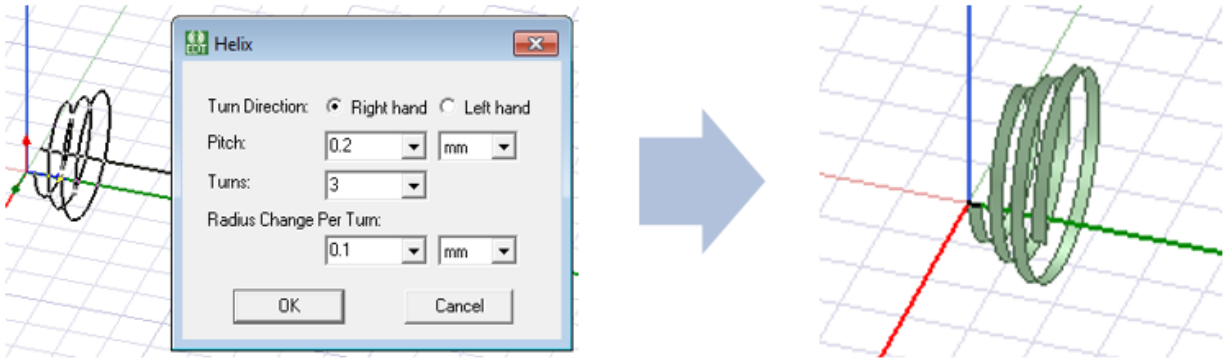
> **Note:**
>
> The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands see:
>
> Surface Approximation and related sections:
>
> Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing a Helix

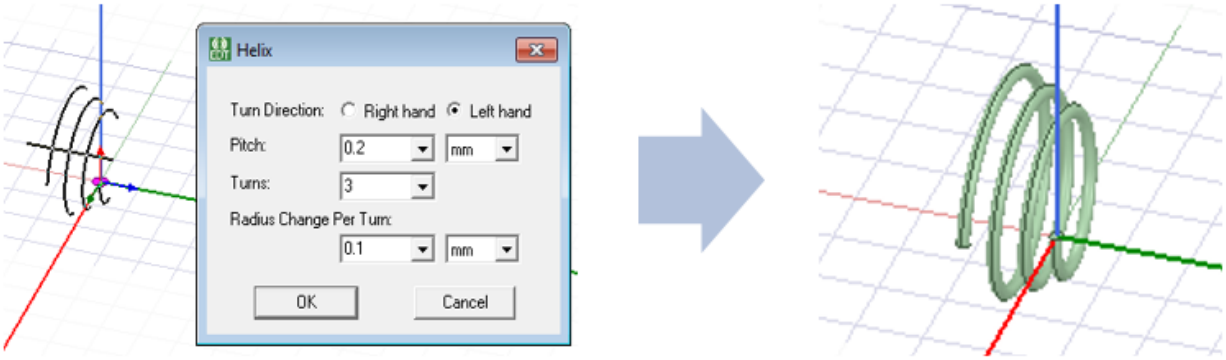
A helix is a 3D spiral object created by sweeping a line or 2D sheet object that you create along a vector that you define. The center of the helix is defined by the placement of the vector relative to the swept object. You must define the vector so that the swept object will not self-intersect, the rule being, do not draw the vector so that it centers on or overlaps a selected 2D object. After you

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

create the vector, the **Helix** dialog opens, where you then specify the turn direction, pitch, number of turns, and radius of the turn to define the helix.

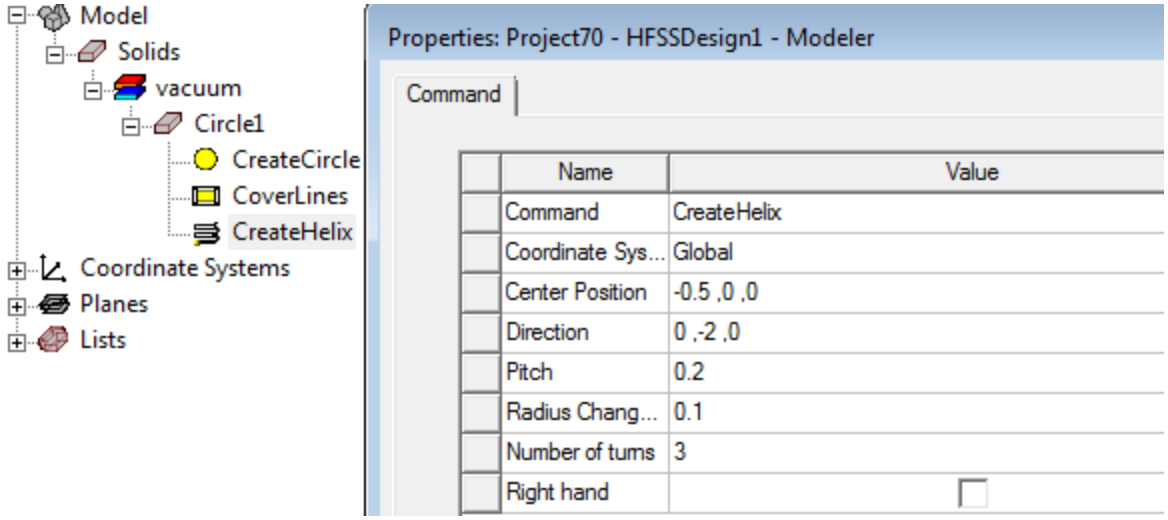


Sweeping a 2D object results in a hollow 3D object. Sweeping a 2D sheet object results in a 3D solid object.



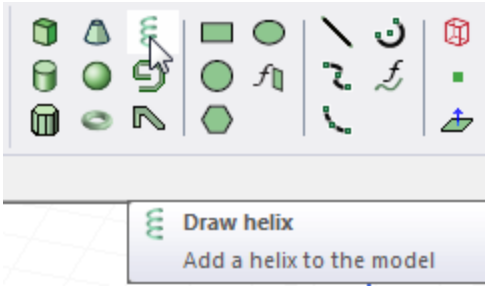
Once you have created a helix, you can changing it by edit the properties of the original object, and by editing the helix command properties for coordinate system, center position (which corresponds to the vector placement), direction, pitch, turns, radius change, and direction.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To create a helix:

1. Create and select the line or 2D object you want to sweep to form a helix. Selecting a valid object enables the **Helix** icon on the **Draw** tab of the ribbon.
2. From the menu bar, click **Draw**> **Helix** or, on the **Draw** ribbon tab, click the **Draw helix** icon:

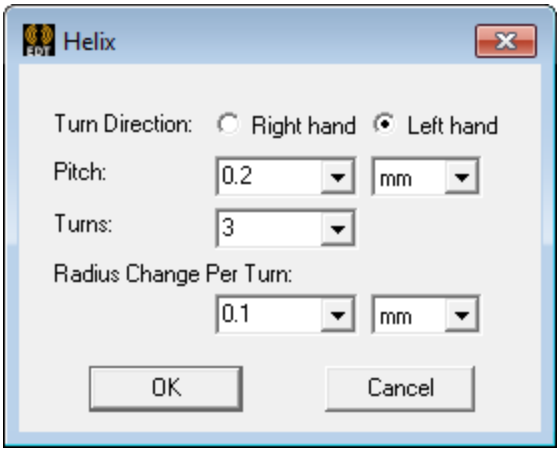


3. Draw the vector you want to sweep the object around. The two points that describe the vector affect axis location and direction only and not the helix length. (The vector definition corresponds to the **Center Position** and **Direction** properties of the **CreateHelix** command in the History tree.) The helix length is determined when you enter the pitch and number of turns in the **Pitch** and **Turns** text boxes. The initial radius of the helix is determined by the axis position relative to the object being swept. It is important that the swept object cannot intersect itself. For a 2D object, you should ensure that the vector you draw does not intersect the interior. If it does, the object cannot be drawn due to self-intersection and you receive a message that "Body could not be created because of invalid parameters."
    a. Select the start point by clicking the point or typing its coordinates in the **X, Y,** and **Z** text boxes. (Remember that you can also edit the parameters of a completed helix).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Select the endpoint by clicking the point or typing its coordinates relative to the start point in the **dX, dY,** and **dZ** boxes.

The **Helix** dialog box appears:



4. For **Turn Direction**, select **Right hand** if the turn direction is clockwise and **Left hand** if the turn direction is counter-clockwise.
5. In the **Pitch** text box, type the distance between each turn in the helix, and click a unit in the drop-down menu.
6. In the **Turns** text box, type the number of complete revolutions the object will make along the vector.
7. In the **Radius Change per Turn** text box, type a number for the increase in the radius and select the units from the drop-down menu.
8. After you set these values, the selected object is swept along the vector to form a helix. The original object you swept is deleted.
9. Click **OK** to create the Helix.

   If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties. You can also select the commands for the object and the helix command in the History Tree to access and edit their properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands see:

Surface Approximation and related sections:

Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing a Segmented Helix with Polygon Cross-Section Using a User Defined Primitive

Ansys provides you with a DLL to define the parameters of a segmented helix with a polygon cross-section. Before you draw a helix, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. Click **Draw** > **User Defined Primitive** > **SegmentedHelix** > **PolygonHelix**.

   The **Create User Defined Part** dialog box appears. The **Parameters** tab permits you to edit the parameters. An **Info** tab contains information about the user defined primitive, its purpose, the company/author who created it, the date created and the version number.

2. Specify the values for the following parameters:

PolygonSegments	Number of segments in the polygon cross-section. Enter zero (0) for true circle
PolygonRadius	Radius of the polygon cross-section.
StartHelixRadius	The radius of a segmented helix is defined from the helix center of rotation to the center of the helix cross-section at segment transitions. The first and last segments of the helix are half segments. See this figure.
RadiusChange	The radius change per turn of the helix.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pitch	Distance between helix turns.
Turns	The number of turns in the helix.
SegmentsPerTurn	The number of segments constructing each turn. Enter zero (0) for true curve.
RightHanded	Helix winding direction. Enter non-zero value for right-handed helix.

3. Click **OK**.

Drawing a Segmented Helix with Rectangular Cross-Section Using a User Defined Primitive

Ansys provides you with a DLL to define the parameters of a segmented helix with a rectangular cross-section. Before you draw a segmented helix, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. Click **Draw** > **User Defined Primitive** > **SegmentedHelix** > **RectHelix**.

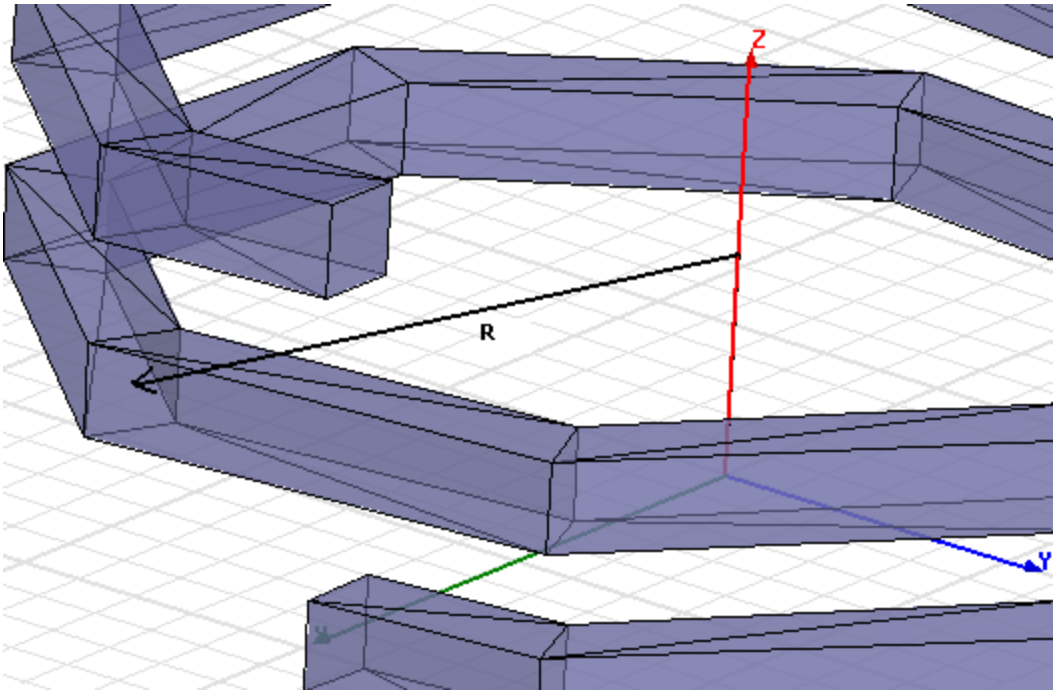
   The **Create User Defined Part** dialog box appears. The **Parameters** tab permits you to edit the parameters. An **Info** tab contains information about the user defined primitive, its purpose, the company/author who created it, the date created and the version number.

2. Specify the values for the following parameters:

RectHeight	Height of rectangular cross-section.
RectWidth	Width of rectangular cross-section.
StartHelixRadius	The radius of a segmented helix is defined from the helix center of rotation to the center of the helix cross-section at segment transitions. The first and last segments of the helix are half segments. See this figure.
RadiusChange	The radius change per turn of the helix.
Pitch	Distance between helix turns.
Turns	The number of turns in the helix.
SegmentsPerTurn	The number of segments constructing each turn. Enter zero (0) for true curve.
RightHanded	Helix winding direction. Enter non-zero value for right-handed helix.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

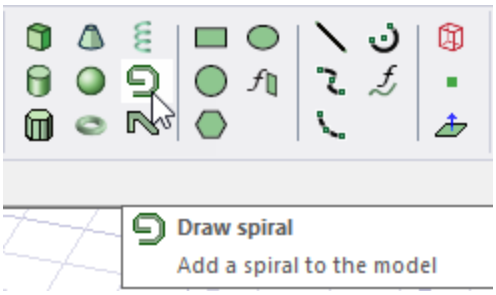
3. Click **OK**.



Drawing a Spiral

A spiral is a 2D or 3D spiral object created by sweeping an object around a vector. Sweeping a 1D object results in a 2D sheet object. Sweeping a 2D sheet object results in a 3D solid object. Before you draw a spiral, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. Select the 1D or 2D object you want to sweep to form a spiral.
2. From the menu bar, click **Draw**> **Spiral** or, on the **Draw** ribbon tab, click the **Draw spiral** icon:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Draw the vector you want to sweep the object around:
   a. Select the start point by clicking the point or typing its coordinates in the **X, Y,** and **Z** text boxes.
   The Status bar also includes options to specify the Coordinate System as Absolute or Relative, and drop down menu options to specify the sphere in Cartesian, Cylindrical, or Spherical coordinates.

   b. Select the endpoint by clicking the point or typing its coordinates relative to the start point in the **dX, dY,** and **dZ** boxes.

   The **Spiral** dialog box appears.

4. Select **Right hand** if the turn direction is clockwise and **Left hand** if the turn direction is counter-clockwise.
5. In the **Radius Change** text box, type the difference in radius between each turn of the spiral.

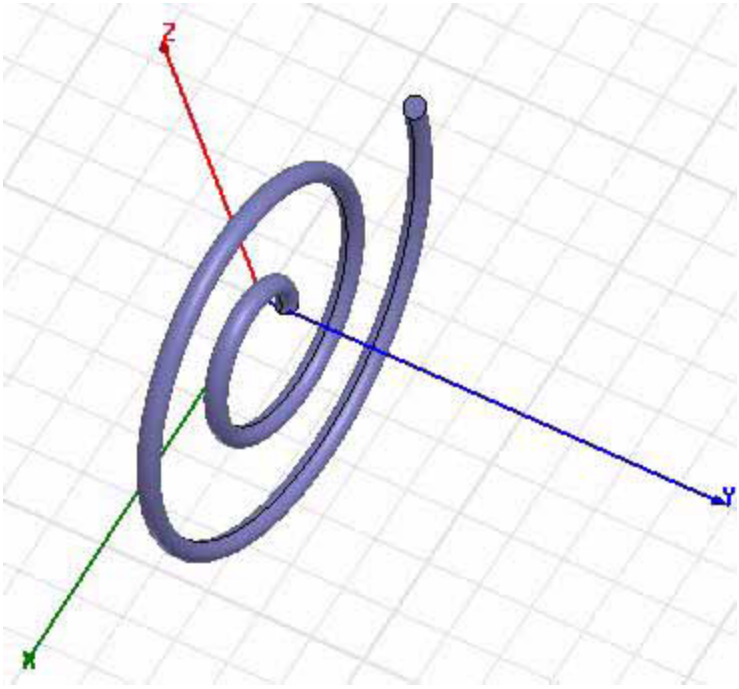
   The radius of the first turn is measured from the center point of the 1D or 2D object you are sweeping to the vector you drew.

6. Click a unit for the radius in the pull-down list.
7. In the **Turns** text box, type the number of complete revolutions the object will make around the vector.

   The selected object is swept around the vector to form a spiral. The original object you swept is deleted. If the Modeler option for editing properties of new primitives is checked, the **Properties** dialog box appears, enabling you to modify the object's properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Click **OK**.



This 3D spiral was created from a 2D circle drawn at z = 0. The turn direction was right hand, the radius change was set at 2, and the number of turns was set at 2.

> **Note:**
>
> The 3D Modeler permits drawing true curved objects. However, the solution is obtained using a tetrahedral mesh, which conforms to the true surface only within the limits identified by certain mesh settings. The modeler has default settings for this conformance, which is a reasonable trade-off between solution speed and solution quality for most objects but may not be ideal for all such objects. High-aspect ratio curves structures (such as helices with narrow and curved cross-sections) may benefit from user control of the faceting values. For details about these commands see:
>
> Surface Approximation and related sections:
>
> Rectilinear Elements and Curvilinear Elements and Guidelines for Modifying Surface Approximations.

Drawing a Spiral Using User Defined Primitives

Ansys provides you with a DLL and a Python script to define the parameters of a rectangular spiral. Before you draw a spiral, you can specify the coordinate system, and you can set the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw** > **User Defined Primitive** > **Examples** > **RectangularSpiral (DLL)** or **Rectangular Spiral (Python)**.

   The **User Defined Primitive Operation** dialog box appears. The **Parameters** tab permits you to see and edit the parameters. An **Info** tab contains information about the user defined primitive, its purpose, the company/author who created it, the date created and the version number.

2. Specify the values for the following parameters:

Xpos	X location of the starting point.
Ypos	Y location of the starting point.
Dist	The separation distance between turns.
Turns	The number of complete revolutions the object will make around the vector
Width	The width of the spiral.
Thickness	The thickness/height of the spiral. If you specify the thickness as zero, the modeler draws a sheetobject.

3. Click **OK**.

   This creates the primitive and displays the **Properties** dialog box for the new object.

Tip:

To see newly created DLLs, click **Draw** > **User Defined Primitive** > **Update Menu**.

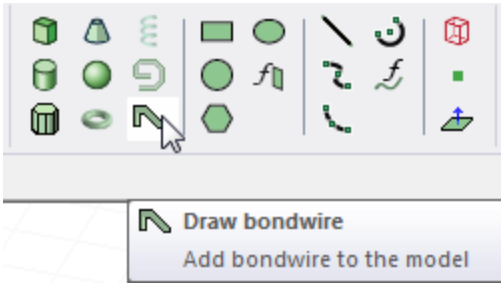
To see the primitives that you have created, click **Draw** > **User Defined Primitive** > **UserLib.**

Drawing a Bondwire

A bondwire is a thin metal wire that connects a metal signal trace with a chip. Please see the topic Bondwires in the *HFSS: Technical Notes* before drawing a bondwire.

1. From the menu bar, click **Draw**> **Bondwire** or, on the **Draw** ribbon tab, click the **Draw bondwire** icon:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select the bond pad point in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the text boxes in the status bar.
3. Select the lead point by clicking the point or typing the coordinates in the text boxes in the status bar.

   The **Bondwires** dialog box appears.
4. In the **Type** list, click the JEDEC modeling standard shape you want the bondwire to have: **JEDEC 4-point**, **JEDEC 5-point, or Low**.

   The **Type** selection changes the dialog bondwire graphic, and shows options for that type.
5. Enter the number of facets in the bondwire in the **No. of Facets** text box.

   The minimum value is 3. The value describes the number of faces that make up the circumference of the bondwire.
6. In the diameter field, specify a diameter value and select the units from the drop-down menu.
7. Enter the height between the bond pad and the top of the loop in the **h1** text box. Include the height's unit of length.
8. The value in the **h2** text box is the height between the bond pad and the lead point. It was calculated by Ansys Electronics Desktop based on the lead point you selected. If you modify the value of **h2**, the lead point will be modified.

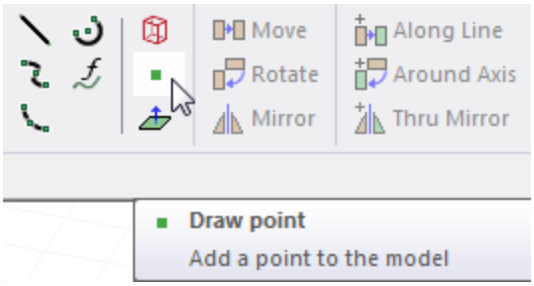
   Optionally, type a new value in the **h2** text box. Include the height's unit of length.
9. If you selected **JEDEC 5-point** or **Low**, do the following:
   a. Type the angle between the horizontal plane and the wire at the bond pad point in the **alpha** text box.
   b. Type the angle between the horizontal plane and the wire at the lead point in the **beta** text box.
10. Click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Drawing a Point

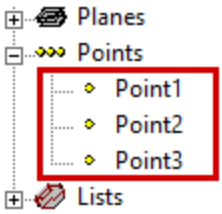
Drawing a point object within the problem region enables you to plot fields or perform field computations at that point. Points are always considered non-model objects by the modeler. Before you draw a point, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

1. From the menu bar, click **Draw**> **Point** or, on the **Draw** ribbon tab, click the **Draw point** icon:



2. Select the point in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the text boxes in the status bar.

   The point is listed under **Points** in the History Tree:



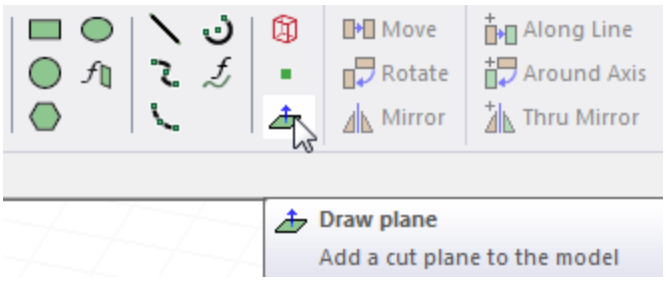
> **Note:**
>
> There is no visible indication of the point in the drawing area unless it is selected in the History Tree. Also, the point cannot be selected graphically once added to the model. It must be selected from the History Tree or from the **Geometry** window within the **Fields Calculator**.

Drawing a Plane

A plane object is a cut plane through the problem region. You can use plot fields to perform field computations on its surface. Planes are always considered non-model objects by the modeler. Before you draw a plane, you can specify the coordinate system, and you can set the drawing plane as Z, Y, or Z, or you can edit the plane in the properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. From the menu bar, click **Draw**> **Plane** or, on the **Draw** ribbon tab, click the **Draw plane** icon:

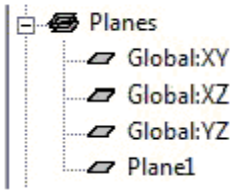


2. Select the origin in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the text boxes in the status bar.

   To delete the selected point and start over, press **Esc**.

3. Select a normal point in one of the following ways:
   - Click the point.
   - Type the coordinates of the point relative to the origin in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

   The plane is created. Its center point is located at the origin you specified and oriented perpendicular to the normal point you specified. The new plane is listed under **Planes** in the History Tree.

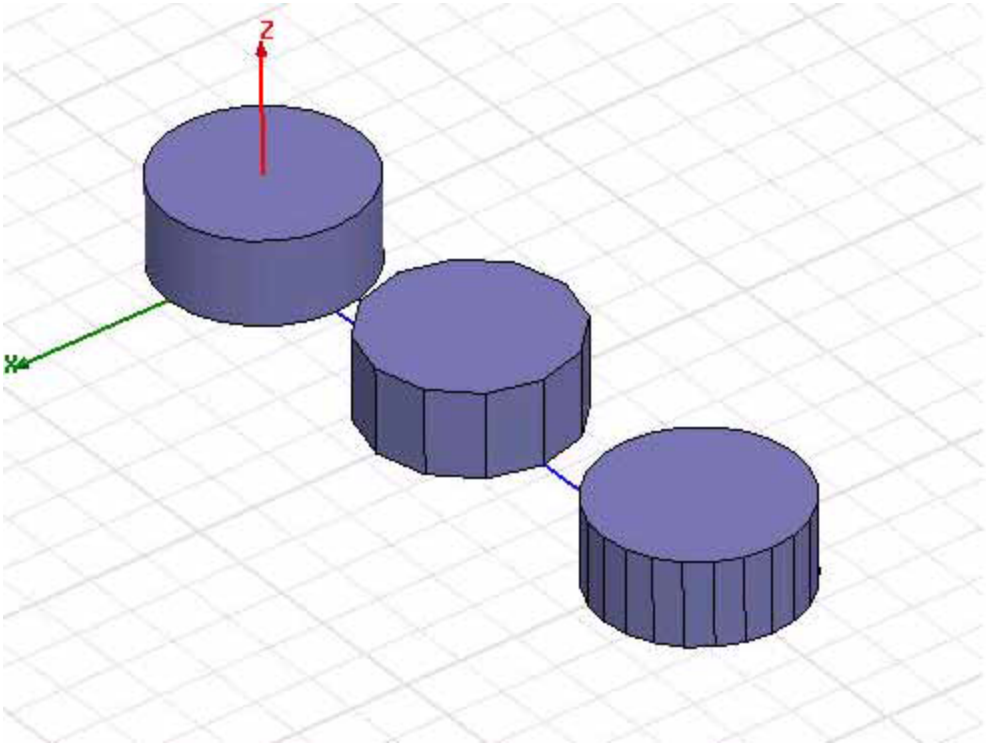


> **Note:**
>
> You only need to draw a plane that does not lie on a predefined xy, yz, and xz plane. Default planes are created on the xy, yz, and xz planes of the global coordinate system as well as any new coordinate system you create.

Creating Segmented Geometry

For some structures, you may want to create segmented as opposed to smooth (or True) surfaces. The following figure shows a comparison of a cylinder created with true surfaces and with segmented surfaces.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following model objects can be created as segmented structures:

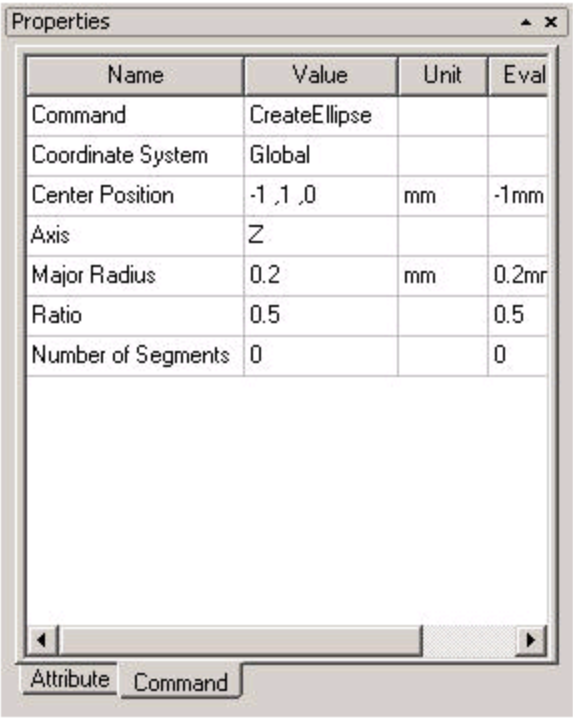
Circle, Ellipse, Cylinder	See: Segmented Objects
Polyline, Arc, Line Segment	See: Converting Polyline Segments

Segmented Objects

To create segmented circles, ellipses, and cylinders use the **Number of Segments** parameter on the Command Tab of the **Properties** dialog as shown below. To convert an object from true surface to segmented, do the following:

1. Select the **circle**, **ellipse**, or **cylinder** in the modeler window or in the History Tree.
2. In the command tab of the properties window (shown docked below), change the **Number of Segments** to an integer value of three or greater and press **Enter**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Values of 1 and 2 are not valid values for the circle, ellipse, or cylinder command and will cause an error.

Drawing Non-Model Objects

If you want to create an object that does not affect the geometric model, define the object as *non-model*. This ensures that the object is used for analysis only; it will not affect the solution process. After drawing the non-model object, assuming it lies in the problem region, you can use it in the reporter as a place on which to plot field quantities. For example, draw a non-model line across the design, then (in the reporter) plot Mag_E on every point along that line.

You can assign output variables (default) and design variable as property values for non-model objects.

Following are examples of using non-model objects to analyze a solution:

- Draw a polyline along which to plot fields or perform field computations. Note that when you create a value versus distance plot, by default, the line will be divided into 100 equally spaced points. You can modify the number of points into which the line is divided in the **Edit Sweeps** dialog box. For more information, see: Specifying Variable Values for a Sweep Definition.
- Draw a rectangle upon which to plot fields in the reporter or perform field computations.
- Draw a volume box to analyze fields in areas of the problem region that are not occupied by an object or that consist of parts of several objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Draw a plane, which is always a non-model object.
- Draw a point, which is always a non-model object, in order to plot fields in the Reporter or perform field computations at that point.

What do you want to do?

Switch to non-model drawing mode. Objects you draw in non-model mode will not be included in the solution process.

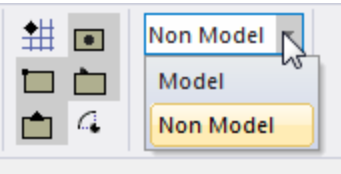
Modify an existing model object to be a non-model object.

Selecting Non-Model Drawing Mode

To switch to non-model drawing mode:

1. Click **Modeler**> **New Object Type**> **Non Model**.

   Alternatively, on the **Draw** ribbon tab, choose **Non Model** from the **Set to model non model type** drop-down menu:

2. Draw the object.

Changing an Object to Non Model

To modify an existing object to be a non-model object:

1. Select the object you want to modify.
2. In the docked **Properties** window, clear the **Model** option.

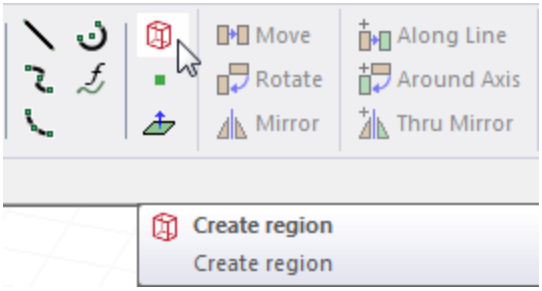
   The object will not be included in the solution process. If the object lies in the problem region, you can plot solution quantities on it.

Drawing a Region

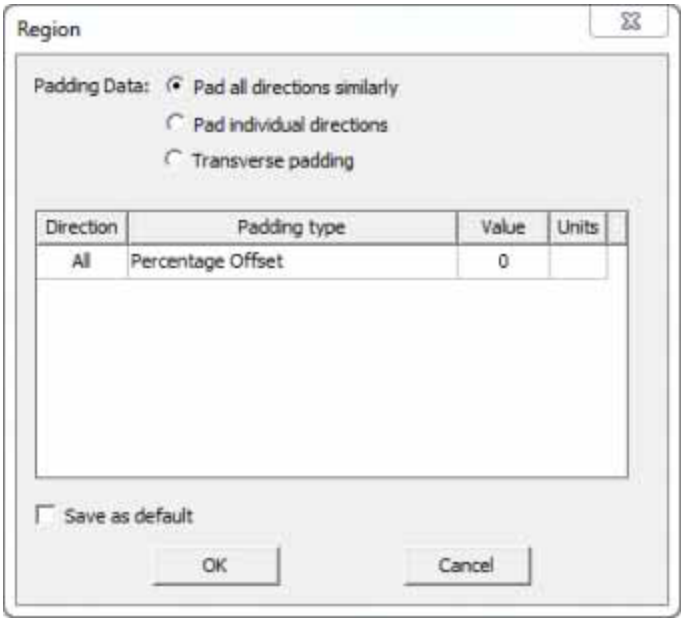
To draw a region encompassing the objects in the current project:

1. From the menu bar, click **Draw** > **Region** or, on the **Draw** ribbon tab, click the **Create region** icon:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

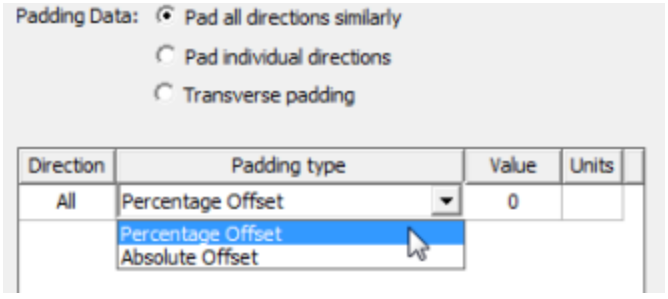


This displays the **Region** dialog box.



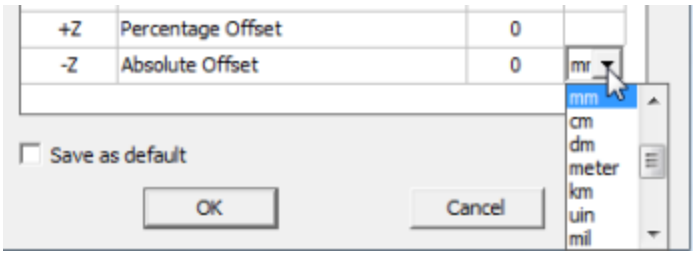
2. For the Padding data, click the Padding Data radio button as **Pad all directions similarly, Pad individual directions**, or **Transverse padding**.

   Selecting **Pad all directions similarly** leaves the Padding type field as requiring a single value that affects all directions. In this case, you can specify the Padding type by selecting **Percentage Offset** or **Absolute Offset** from the drop-down menu.

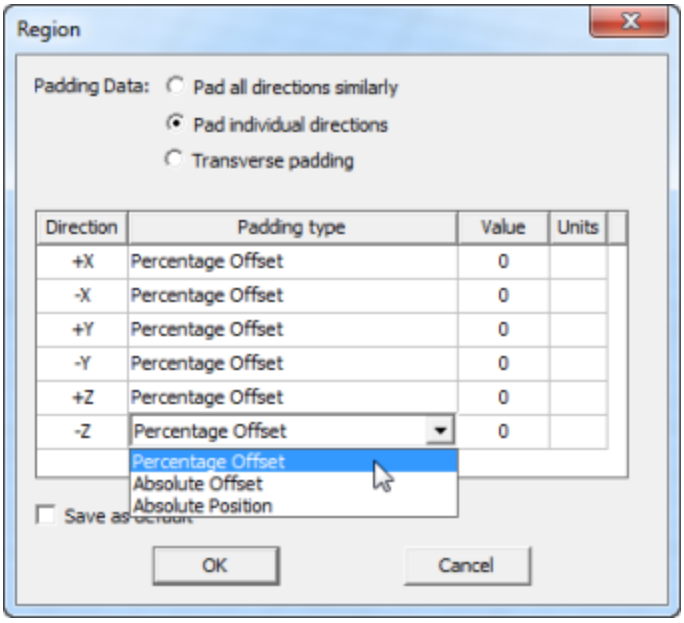


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you select **Absolute Offset**, you also specify the units by selecting from a drop-down menu.



Selecting **Pad individual directions** displays the Padding Percentage as a table of Positive and Negative X,Y, and Z coordinates, permitting you to specify padding for each direction. In this case, you can specify the Padding type by selecting **Percentage Offset** or **Absolute Offset** or **Absolute Position** from the drop-down menu.

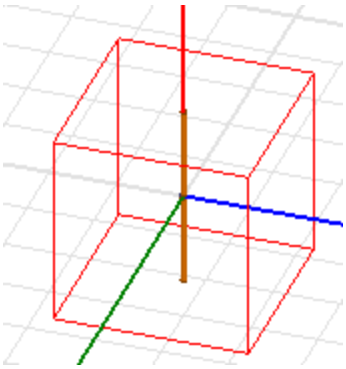


Selecting **Transverse padding** means that padding in one direction is controlled by dimensions in other two directions. Each direction (for example, X) is padded with user-specified percentage of diagonal length of the other two directions (Y and Z).

For a single antenna element, something that is radiating does not radiate with reasonable efficiency until it is lambda/2 in length, L, and it radiates in the direction transverse to the dimension L. So we get the dimension L from the region command and pad it 50% which would then correspond to the lambda/4 of the frequency at which the device is lambda/2 but only in the direction in which energy radiates, the transverse direction.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A basic example is a thin resonate dipole oriented in z. There is zero radiation in the z-direction so no need to pad the abc in that direction since no energy goes in that direction. But in X and Y transverse to the length L we do need to pad lambda/4 (the 50% setting) because it is that direction that the device radiates.



3. Specify the Padding values in the fields and select the units from the drop-down menu.
4. If desired, select the check box to save the values as Default.
5. Click **OK** to close the dialog and create the region.

The region is drawn, selected, and displayed in the History tree. It is created using the current coordinate system. The Properties dialog for the Create Region icon in the History tree shows the coordinate system and Padding values. The Properties dialog for the selected model lists attributes tab including Name, Material (Default, vacuum), Solve inside, Orientation, Model, Color, Display Wireframe, and Transparency. You can edit all of these values.

If you try to create a region that does not contain all of the objects in your model, the modeler automatically expands the region to cover all objects. The region also updates automatically as your geometry changes.

Only one region can be created for a single project using the **Draw** > **Region** command. If you try to create a second region, the **Properties** window appears for the existing region, allowing you to change operation parameters and attributes.

Related Topics

Drawing a Sub-Region

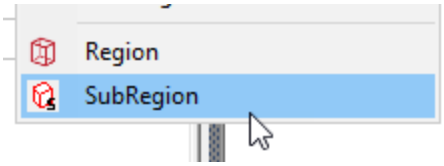
Drawing a SubRegion

The Modeler allows you to define a subregion that encompasses a set of objects inside previously defined region. Like regions, subregions automatically resize if the objects contained in these subregions change. You can create nested subregions, and reassign the content of a subregion.

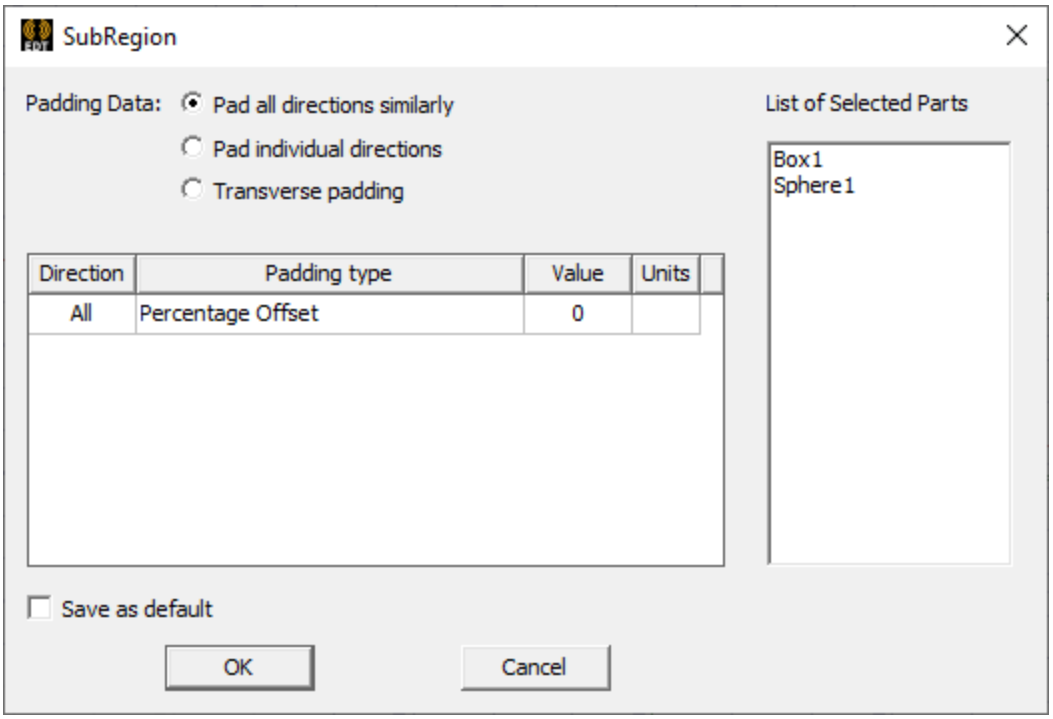
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To draw a sub-region in the current project:

1. Select the parts of components that you want to include in a subregion.
2. From the menu bar, click **Draw** > **SubRegion**:



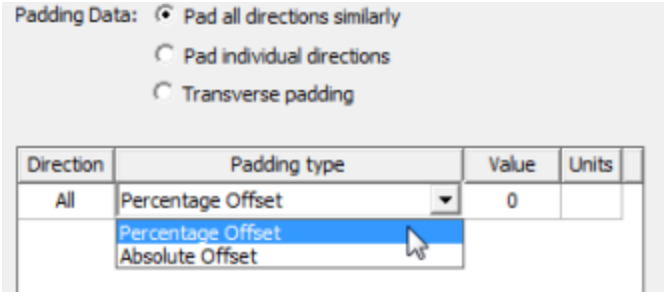
This displays the **SubRegion** dialog box.



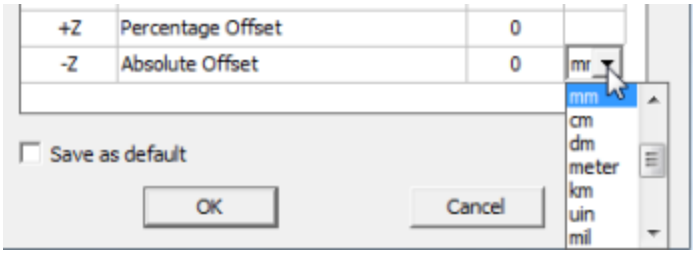
3. For the Padding data, click the Padding Data radio button as **Pad all directions similarly, Pad individual directions**, or **Transverse padding**.

   Selecting **Pad all directions similarly** leaves the Padding type field as requiring a single value that affects all directions. In this case, you can specify the Padding type by selecting **Percentage Offset** or **Absolute Offset** from the drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

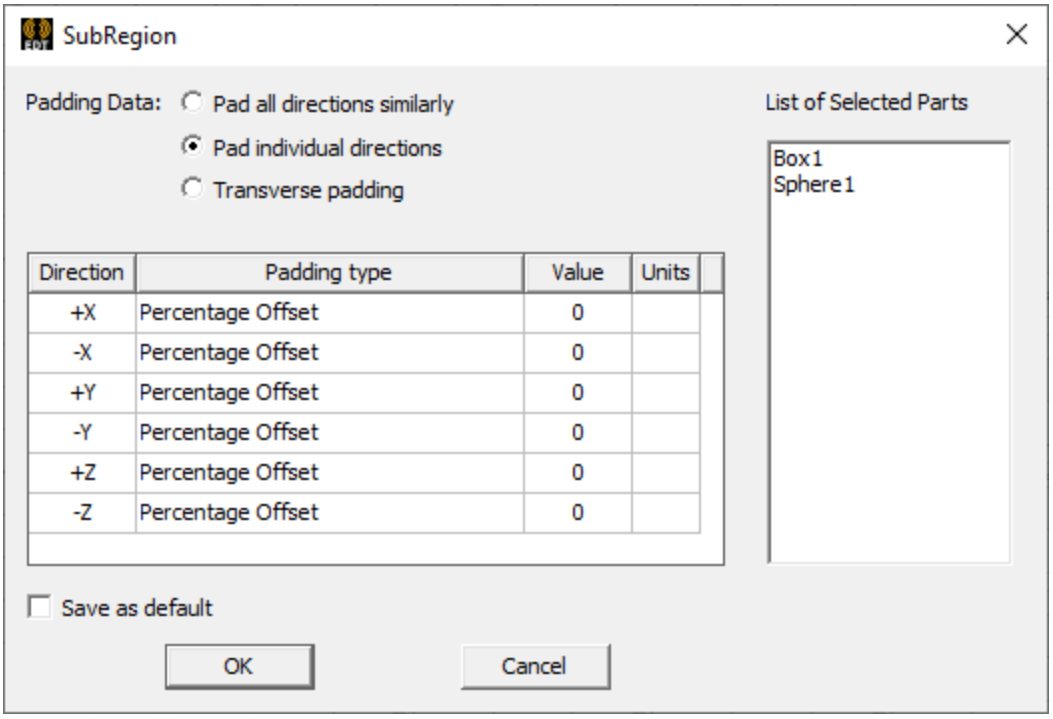


If you select **Absolute Offset**, you also specify the units by selecting from a drop-down menu.



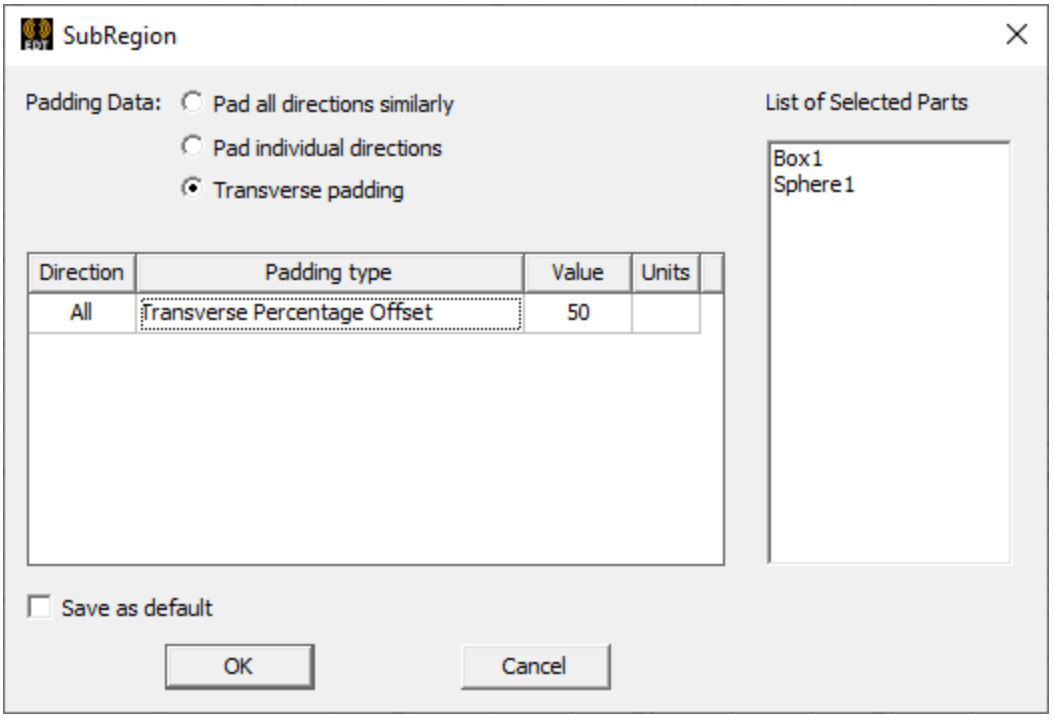
Selecting **Pad individual directions** displays the Padding Percentage as a table of Positive and Negative X,Y, and Z coordinates, permitting you to specify padding for each direction. In this case, you can specify the Padding type by selecting **Percentage Offset** or **Absolute Offset** or **Absolute Position** from the drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



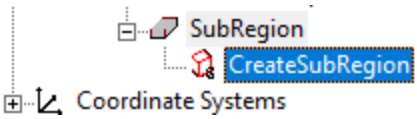
Selecting **Transverse padding** means that padding in one direction is controlled by dimensions in other two directions. Each direction (for example, X) is padded with user-specified percentage of diagonal length of the other two directions (Y and Z).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Specify the Padding values in the fields and select the units from the drop-down menu.
5. If desired, select the check box to save the values as Default.
6. Click **OK** to close the dialog and create the sub-region.

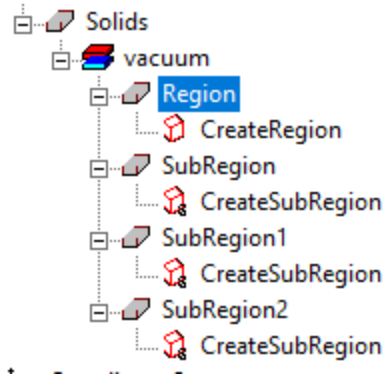
The sub-region is drawn, selected, and displayed in the History tree. It is created using the current coordinate system.



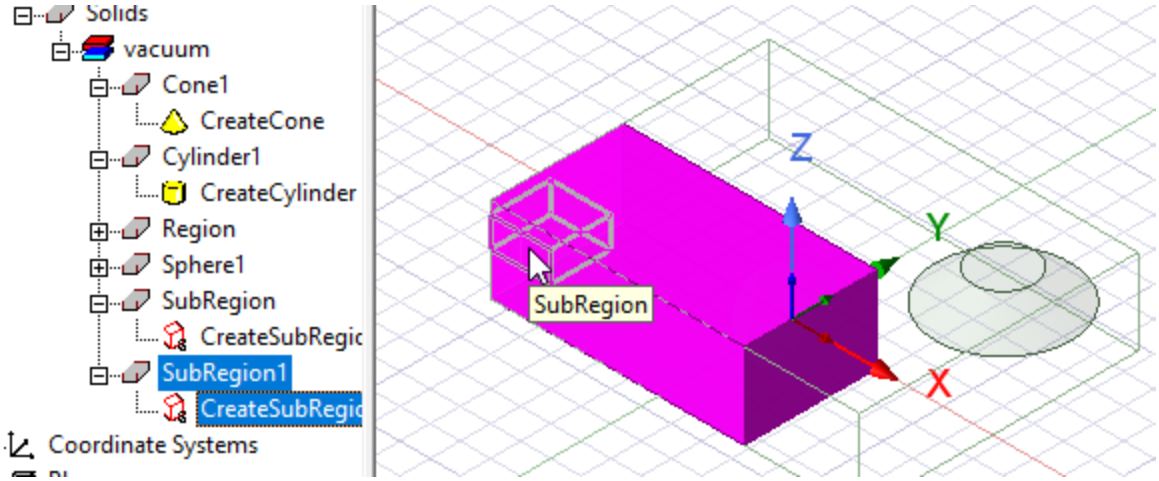
Additional SubRegions and Nested SubRegions.

If you create additional sub-regions, these appear in the History Tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



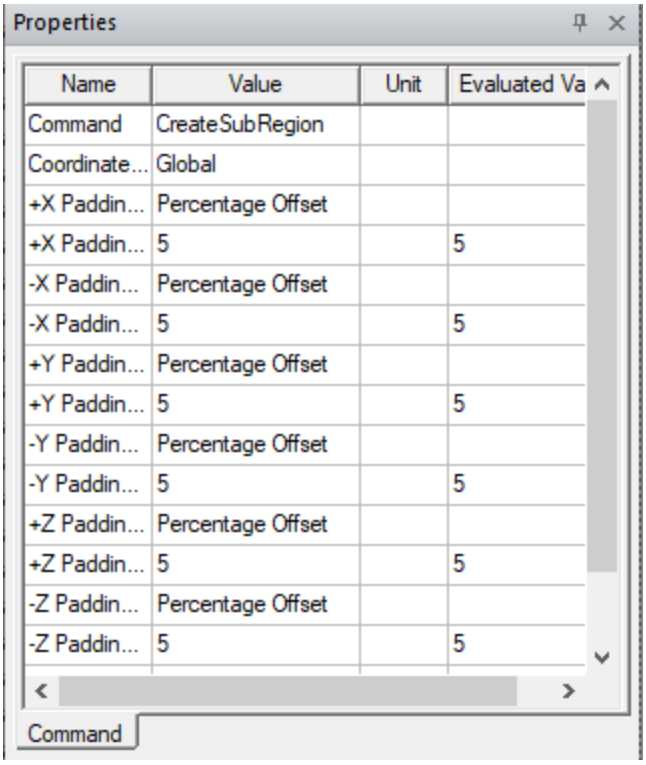
SubRegions can contain other SubRegions, that is, SubRegions can be nested.



Properties for SubRegions

The Properties dialog for the Create SubRegion icon in the History tree shows the coordinate system and Padding values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Properties dialog for the selected model lists attributes tab including Name, Material (Default, vacuum), Solve inside, Orientation, Model, Color, Display Wireframe, and Transparency. You can edit all of these values.

You can use modeling operations on these sub region parts with following rules

- Geometry of non-subregion parts should not depend on sub-region parts. That is, sub-region parts should not be allowed to be used as a tool, not even after cloning them
- Simple local operations like face move, fillet, and arrange operations should be allowed on sub-region parts.

Global region should take the sub-regions into account in its computation.

Sub-Regions can be included in components.

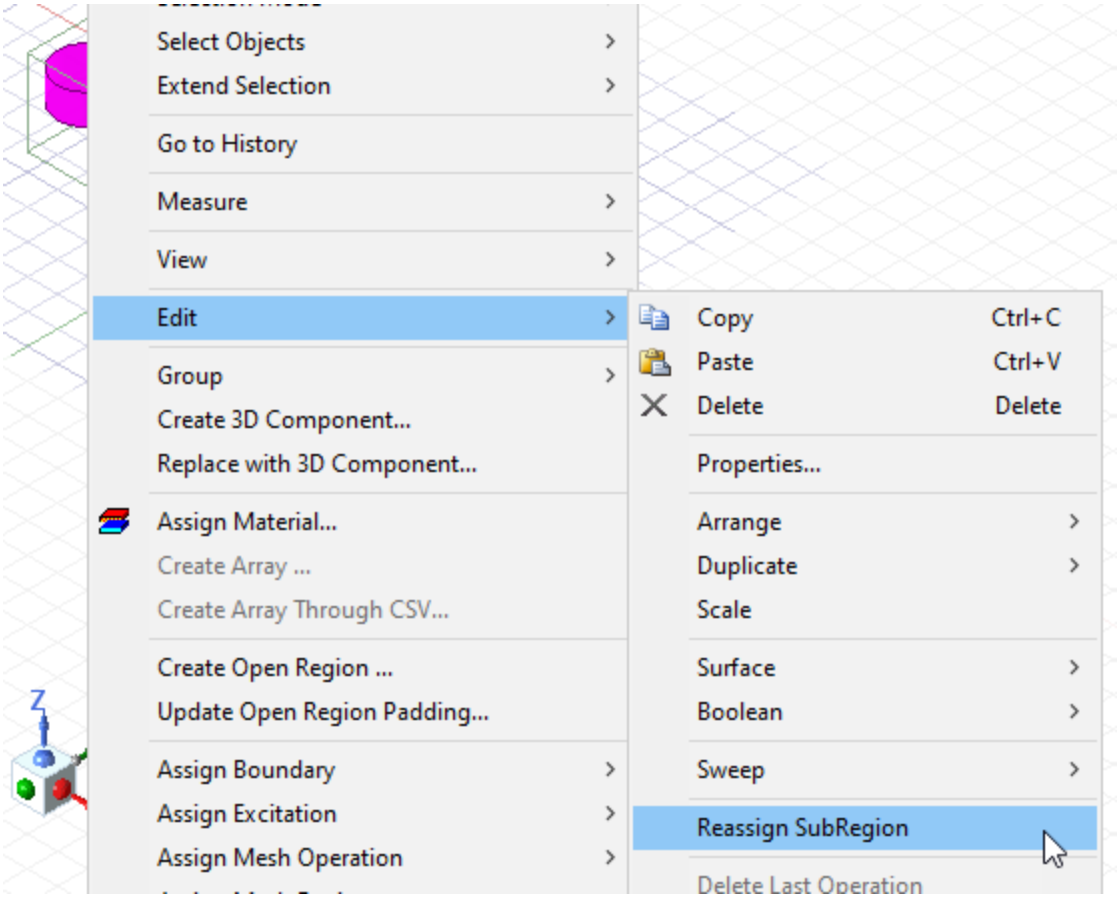
Reassigning a SubRegion

The Modeler allows you to reassign a subregion that encompasses a set of objects inside previously defined region. Like regions, subregions automatically resize if the objects contained in these subregions change.

To reassign a sub-region in the current project:

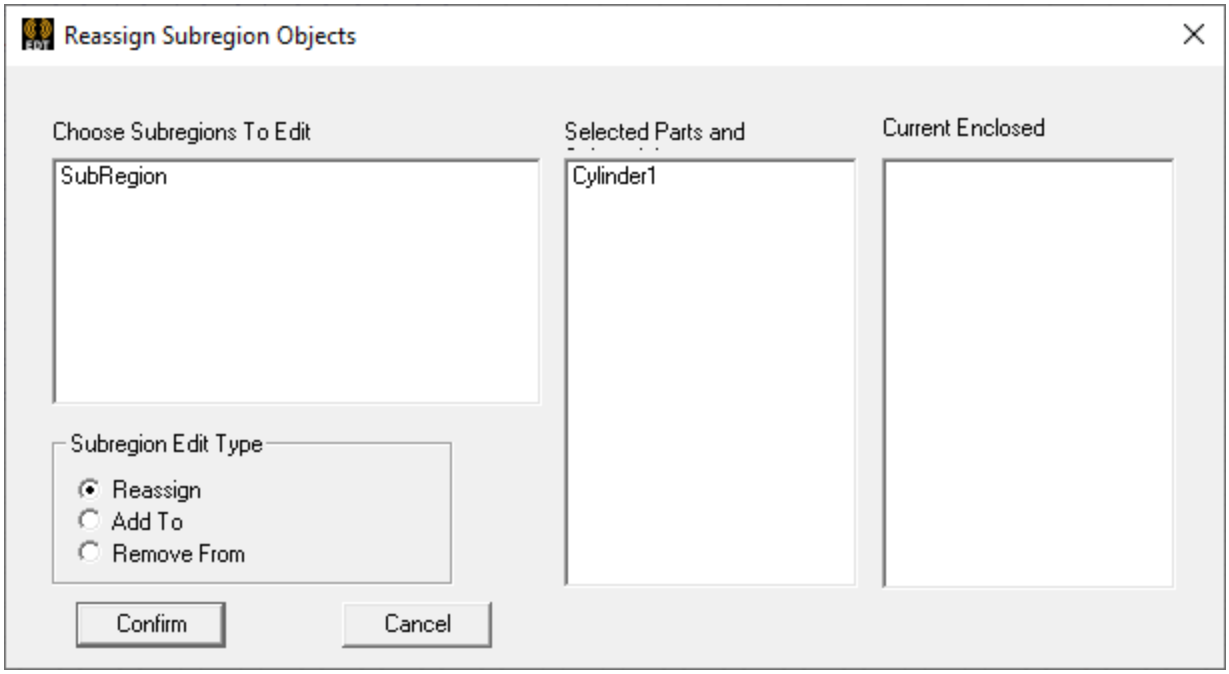
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the components or parts that you want to assign to a subregion.
2. Right-click in the Modeler window to invoke the shortcut menu and click **Edit** > **Reassign SubRegion**:

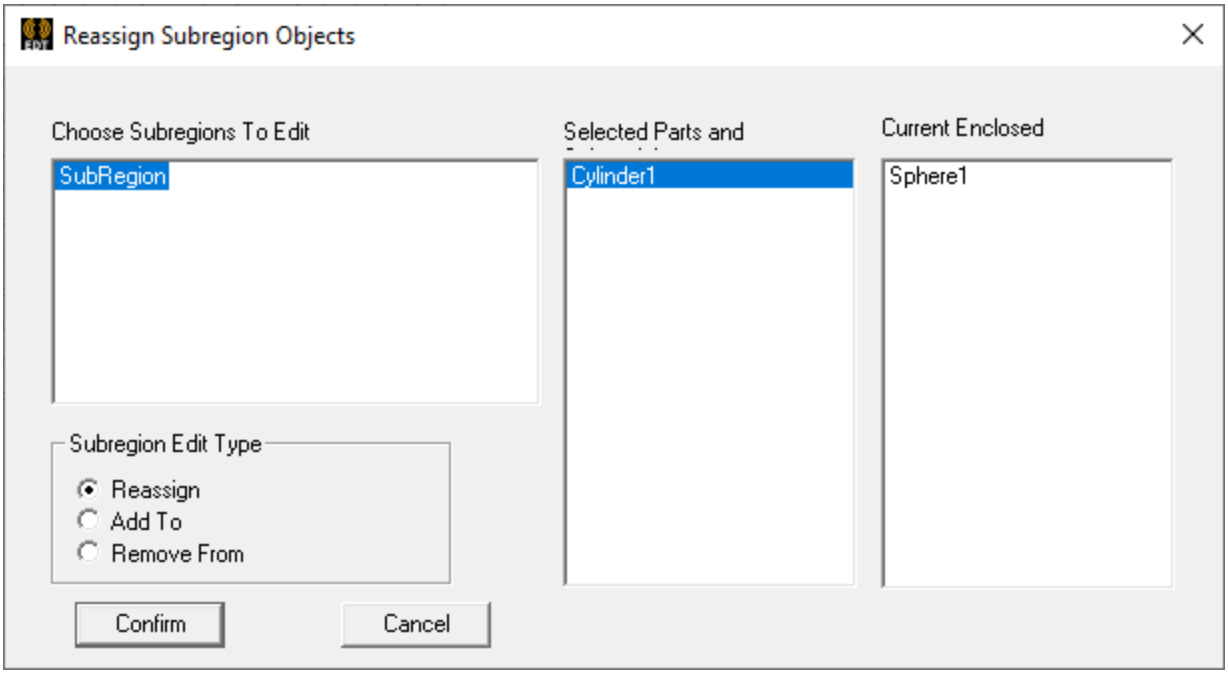


This displays the **SubRegion** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



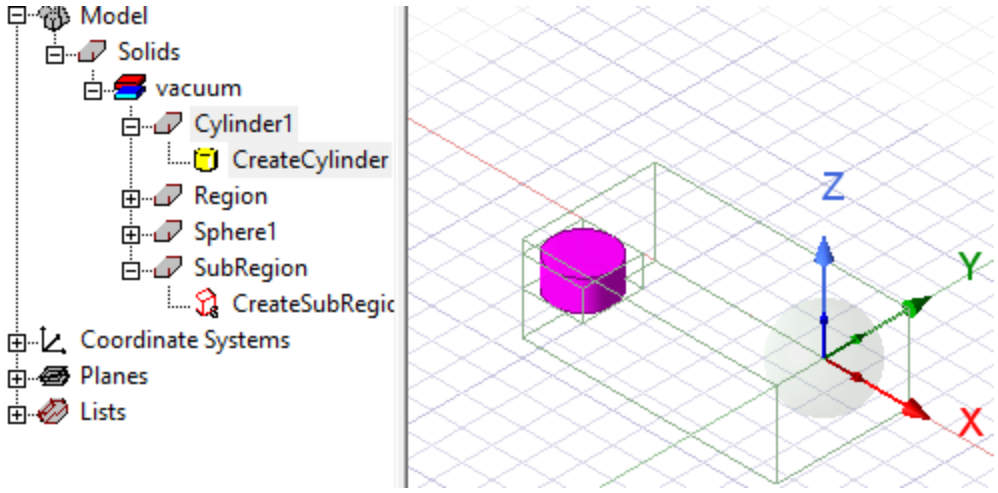
3. Select the SubRegion to Edit and the Parts to Reassign. If there are multiple SubRegions, these will be listed. When you have selected a SubRegion, the Currently Enclosed components are listed.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

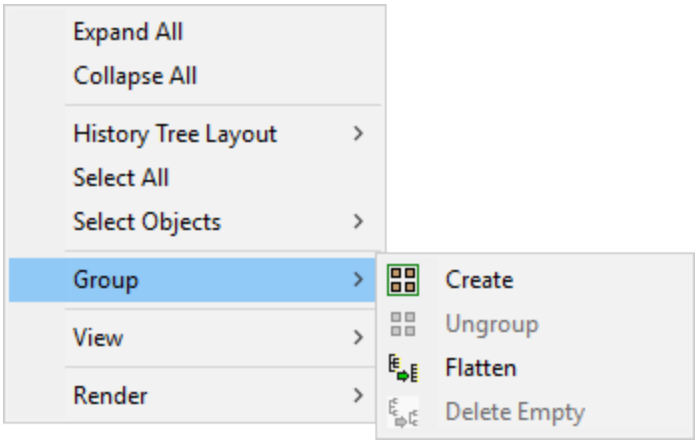
4. Specify whether to Reassign the selected parts to the SubRegion, or Add To the SubRegion, or Remove from the SubRegion.
5. Click **Confirm** to close the dialog complete the reassignments.

The sub-region is redrawn, selected, and displayed in the History tree. It is created using the current coordinate system.



Group Commands for Modeler Objects

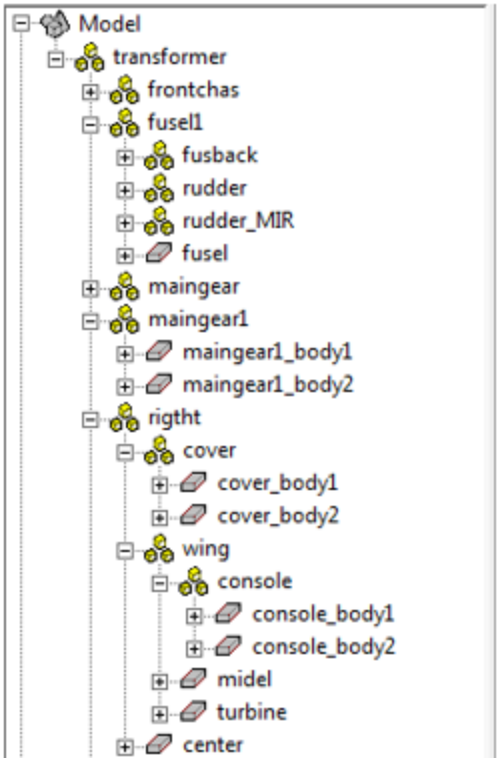
The 3D Modeler allows you to group objects in the History tree. Besides predefined folders under **History Tree Layout** for solids, sheets, and material assignments, a set of **Group** commands allows you to create groups, ungroup objects, flatten a selected group's hierarchy, and delete empty groups. Groups are not compatible with 3D components, although, you can organize 3D components by definition in the History tree.



The **Group** commands support the following features:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Group objects in the history tree. When objects are grouped, they show up under a sub-item in the history tree.
- Groups can contain sub-groups and sub-group containing further sub-groups.
- Groups permit moving objects from one group to any other group at any time.
- Groups are purely for organization of history tree. They do not affect solution in any way.
- Groups permit you to bring in MCAD assemblies and sub-assemblies as groups. See: Importing 3D Model Files.



- Groups can contain objects, submodels (for example, UDM, 3D Component, CAD links) and groups. Coordinate systems, planes etc will not have any parent group.

Object groups have following important distinctions from 3D components:

- Groups do not encapsulate history of objects. They do not encapsulate parameters used by those objects.
- Groups do not have history tree operations. For example, Arrange operations apply to all objects of the group rather than the group.
- Groups are not independent. Delete of an object in one group could cause object in another group to be also deleted.

Accessing Group Commands

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can access Group commands in several ways: via **Modeler > Group**, via the right-click shortcut menu in the History tree, via the right-click shortcut menu in the Modeler window, and via the Group icons on the Model tab.

Group Command Descriptions

Group > **Create** works with or without a selection. If the History Tree Layout>Show Groups command is disabled, a dialog box reports that groups are currently hidden, and that if you OK to continue, groups will be shown. If there is no selection of objects for new group, an empty group is created under Model. If there are selections, a new group is created under same group as all the selections, provided all selections are under one group. Once a group is created, all selections move under the new group. If selections are under different groups, then a new group is created under Model. Selections could be objects, groups, 3D Components, or User Defined Model. The **Group** > **Create** command tries to find a more suitable group name where possible, based on names of all selections. If not, default group names are Group1, Group2, and so on. You can edit group names later.

Group > **Ungroup** is enabled only when you select one or more groups. Upon ungroup, all the contents of the selected group move under group's parent and the selected group is deleted. Note that **Group** > **Ungroup** is not recursive; it only affects the selected group is and all its children groups remain intact. Ungroup is also different from **Delete** (under **Edit** > **Delete**), which deletes a selected group and everything under it.

Group > **Flatten** is enabled when you select a group. You can select multiple groups at different levels. All of the selected group's contents (objects, sub-groups, and so forth) come directly under selected group. In a sense this is a recursive ungroup operation as all the children and grandchildren groups of selected group are ungrouped. If the History Tree Layout > Show Groups command is disabled, a dialog box reports that groups are currently hidden, and that if you OK to continue, groups will be shown.

Group > **Delete Empty** deletes all empty groups under the selected group. If a selected group is empty it is deleted. This command is helpful to clean up empty groups after you have moved objects from one group to another. If the History Tree Layout > Show Groups command is disabled, a dialog box reports that groups are currently hidden, and that if you click **OK** to continue, groups will be shown.

Operations on Groups

Edit/Delete

Delete operations delete group and all the children, including sub-groups, under the group.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copy and Paste of Objects in Groups

When you copy and paste objects with groups to another design, group information is carried over, that is, new group in target design is created. If a group with the exact name exists, then the objects are moved under that group and no new group is created. When pasted in the same design, the pasted object moves under same parent group as original object.

Copy and Paste of Groups

When you copy and paste one or more groups in same design or to another design, a new group is created with unique name derived from original group name. Everything under that group, including any sub-groups, is copy/pasted as well.

Copy and Paste of Groups and Objects

If you multi-select a few objects and groups, all of them are pasted correctly. If you select a group and few objects under that group, copy and paste includes the complete group structure. This means that selection of objects under the selected group is ignored. If you want to only copy and paste few objects from group, you should only select those objects and NOT select the group.

Arrange Operations on Groups

You can select group to enable arrange operations like move, rotate and mirror. Note that arrange operations works directly on objects under selected group. So after arrange operation is done, the History tree shows the arrange operation under every object of that group.

Duplicate Operations on Groups

You can select groups to enable duplicate operations. Afterward, the History tree shows a duplicate operation under every object of that group. Newly created objects appear under a new group. There will be new group for every instance of duplicate.

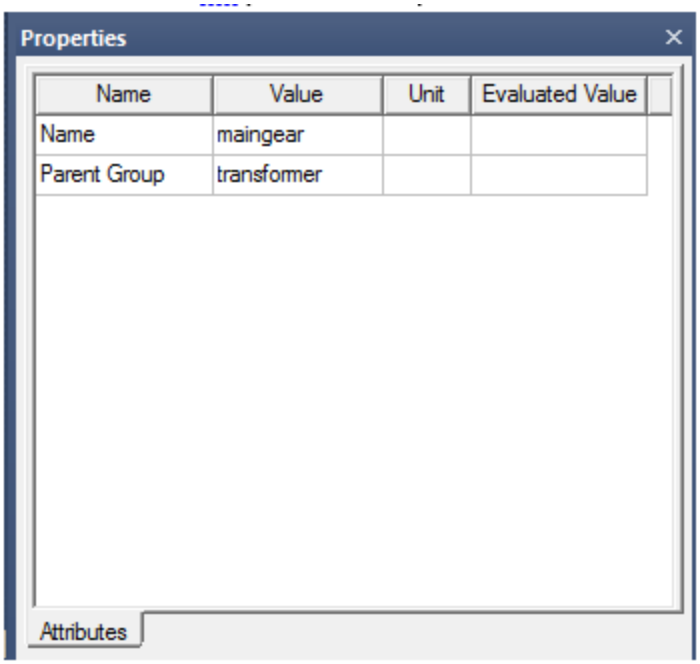
Other Operations on Groups and Objects in Groups

Modeling operations that create new objects, such as Create from Face or Edge, as well as Simplify and Separate. The newly created objects appear under a new group created under the original object's group, based on the tool option settings. See: 3D Modeler Options: Group Options.

Group Properties

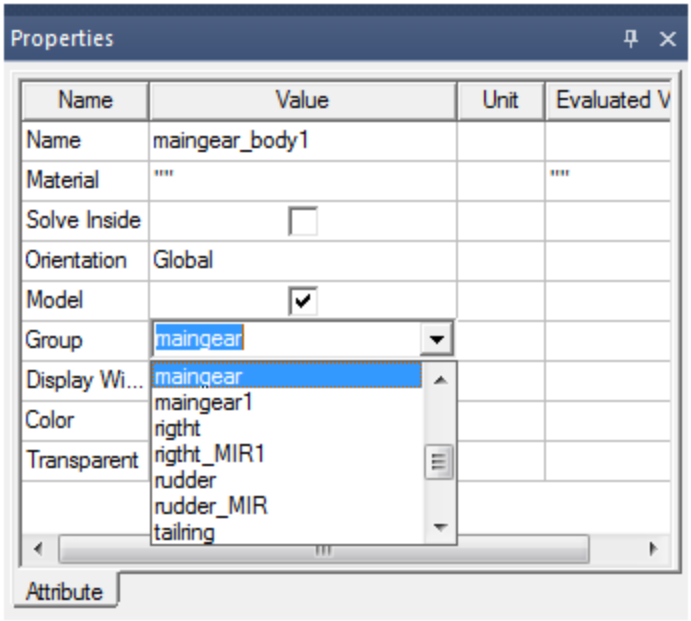
A Group’s Property tab is shown when you select a group in the history tree. The Group’s properties include Name and parent Group.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Group Property Window

Object, group and submodel properties have a Group property. You can edit the Group Property and this provides another way of setting an object's (or group's) group. You can select any of the existing groups listed or create a new group.



Group assignment could be changed in Object's property window

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Model Analysis

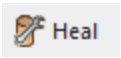
For some models it may be beneficial to remove unnecessary small entities and to fix object misalignments to avoid potential mesh issues. The Modeler window includes Model Analysis functions to help you evaluate models you have imported or created. Select **Modeler** > **Model Analysis** to see the menu options. Depending on the design and the current selection, some features may not be enabled. The menu includes the following commands.

- **Analyze Objects**
- **Analyze InterObject Misalignment**
- **Analyze Surface Mesh**
- **Show Analysis dialog**

> **Note:**
>
> Before running model analysis, you must remove all command history for the selected object by using the Purge History command. If you need to save the object history, save a separate copy.

After import, you typically perform validation check. This lets you focus on objects and object pairs that have errors and or warnings. The objects that fail should be analyzed by using the **Modeler** > **Model Analysis** > **Analyze Objects** menu item.

1. Select the objects and invoke **Modeler** > **Model Analysis** > **Analyze Objects**; or, on the **Model** ribbon, click **Heal**.

   This displays the **Analysis Options** dialog box, which allows you to specify settings for entity check level and small feature detection.

   When you **OK** this dialog box, the initial analysis executes and the **Model Analysis** dialog box is displayed.

2. Choose the objects that have "Invalid Entities Found" and **Perform** > **Heal Objects**.

   In most cases, the objects will be healed and the errors fixed.

3. If errors still persist, choose the edges and faces and click on **Delete**.

   This will replace the selected face/edge object by a tolerant edge/vertex respectively. In some cases the replacement of the face/edge by tolerant edge/vertex will fail.

When models pass the initial validity checks, mesh generation could still fail. The following errors can be present in models: (See Error Detection.)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Non-manifold topology. These are non-manifold edges and vertices that are present in the model.
- Object pair intersection. This detects whether pairs of objects intersect.
- Small feature detection - small edge length, small face area and sliver face detection.
- Misaligned entities detection - detects pairs of faces from objects that can be aligned to remove object intersections. This improves the probability of mesh success.
- Mesh failure error display. This is available for single object, object pairs and last simulation run (all objects in a model). Errors reported by the meshing module are reported to the user.

Small feature errors and misaligned entities should be resolved before you invoke the meshing for the model.

By default, the **Heal** command is automatically applied to imported objects.

Analysis Options Dialog Box

To perform analysis on an object according to specified features and tolerance values:

1. Select the object you want to analyze and click **Modeler** > **Model Analysis** > **Analyze Objects**.

   This displays the **Analysis Options** dialog box, with the **Analysis Options** tab selected. Selecting **Modeler** > **Model Analysis** > **Heal** also displays this dialog box. If, during **Modeler** > **Import...** you select **Heal Imported Objects** and **Manual**, you also see this dialog box.

2. If desired, check the **Perform Entity Check Errors** check box.

   This enables the **Check Level** menu. The setting can be **Basic**, **Strict**, or **Comprehensive**. See: Modeler Validation Settings.

3. If desired, click the check boxes to enable and set the **Detect Feature** settings:
   - **Detect Holes**, and specify the Maximum Radius.
   - **Detect Chamfers**, and specify the Maximum Width.
   - **Detect Blends**, and specify the Maximum Radius.
4. If desired set the **Detect Small Entities** features and tolerance values.
   - **Small Edges**, length less than
   - **Small Faces**, area less than
   - **Sliver Faces**, which enables:
     - Object Bounding Box Scale Factor
     - Sliver Edge Width
5. Click the **Properties** tab to see a listing of the geometric properties of the selected object.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Clicking **OK** on this dialog displays the **Model Analysis** dialog box, which contains the results of the analysis.

Analyzing the Surface Mesh

To set the options to analyze the surface mesh:

1. Select an object of interest.

   This enables the Analyze Surface Mesh command in the menu.

2. Click **Modeler** > **Model Analysis** > **Analyze Surface Mesh**.
   The **Surface Mesh Analysis Options** dialog box appears. This dialog box allows you to set parameters to remove.

   You can also open the **Surface Mesh Analysis Options** dialog box from the **Model Analysis** dialog box via the **Perform** drop-down menu on the **Objects** tab.

3. Select or clear the **Perform Object Pairs Analysis** check box. Selecting this option evaluates the mesh for all combinations of the selected objects.
4. Select or clear the **Ignore Objects Separated by greater than** check box, and enter a value in the text box. Selecting this option means that object pairs are disregarded from analysis if their separation is greater than the specified value.
5. Click **OK** to perform the analysis with the selected options.
   The **Model Analysis** dialog box appears, displaying the results of the analysis.

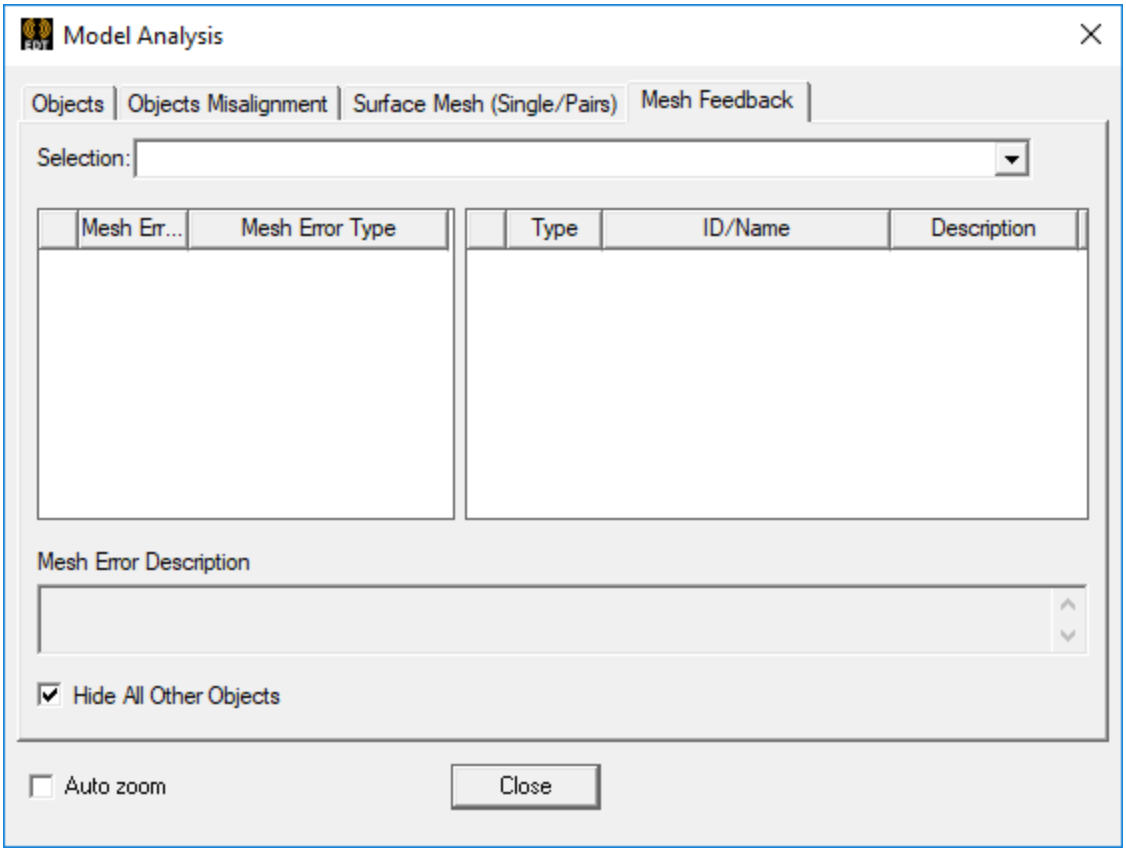
Model Analysis Dialog Box

This dialog box contains results for all model analysis, including diagnostic information relating to mesh issues. To view the analysis options:

1. Click **Modeler** > **Model Analysis** > **Show Analysis Dialog**.
   A submenu appears.
2. Select one of the following from the submenu:
   - **Objects**
   - **Objects Misalignment**
   - **Surface Mesh**
   - **View Mesh Feedback**

   The **Model Analysis** dialog box appears (This dialog box also appears automatically after clicking **OK** in the **Analysis Options** dialog box).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. You can select **Hide All Other Objects** to suppress display of objects, faces, or surfaces in the Modeler window not selected in the **Mesh Feedback** panel
4. Select the **Auto zoom to selection** check box to automatically zoom to the item selected on the **Objects** tab.
5. Make the desired changes on each tab in the **Model Analysis** dialog box.
    - **Objects** tab
    - **Objects Misalignment** tab
    - **Surface Mesh (Single/Pairs)** tab
    - **Mesh Feedback** tab
6. Click **Close** to close the **Model Analysis** dialog box.

Objects Tab

All results relating to model analysis of specific objects are presented under the **Objects** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. The results table contains the following information.
    - **Object** – column listing the objects in the current design.
    - **Last Analysis status** – column giving the analysis status of the listed objects. Objects can have the following status:
        - **Good** – the object contains no invalid geometry entities given the tolerance values specified in the **Analysis Options** dialog box.
        - **Null Body** – the object is non-existent.
        - **Analysis not performed** – the object was not selected for analysis.
        - **Invalid entity errors** – these are api_check_entity() errors and non-manifold errors which Ansys EM recommends that you fix before meshing.
        - **Small entity errors** – small faces, sliver faces and small edges that are optionally detected based on the tolerance limits specified in the Analysis Options dialog box.
2. Select any object name in the table which contains errors to display a table with a list of Error types, corresponding Entity List IDS for associated faces, edges and vertices, and error descriptions. Selecting the row for each error causes the Modeler window to highlight the associated entity.

    > **Note:**
    >
    > **Auto Zoom to Selection** – if this option is checked, the modeler automatically zooms to the item selected in the **Model Analysis** dialog box.

3. Select the face, edge or vertex entity from the list to view the error description in the **Description** field.
4. Select the **Delete** button if you want to remove a selected face or edge entity.
5. Select the **Perform** button to list the commands that you can execute on the selected objects in the Results table.
    - **Heal Objects** – repairs invalid geometry entities for the selected objects within the specified tolerance settings. The **Healing Analysis** dialog box will appear.
    - **Analyze Objects** – evaluates the object status. Selecting this displays the Analysis Options dialog.
    - **Analyze Surface Mesh** – invokes a mesh for each selected object and reports analysis results under the **Surface Mesh (Single/Pairs)** tab. Selecting this option displays a dialog with radio buttons to select.
        - **Perform Object Pairs Analysis** – evaluates mesh for all combinations of the selected objects.
        - **Ignore objects separated by greater than a specified value** – object pairs are disregarded from analysis if their separation is greater than the specified

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

value.

- Click **OK** to perform the analysis with the selected options.

- **Analyze Interobject Misalignment** – determines any misalignments between two selected objects in the results table. The results are reported under the **Objects Misalignment** tab.
- **Display Healing Log** – checking this causes the **Model Analysis** dialog box to display a healing log which includes information about operations performed on an object during the healing process.

Object Misalignment Tab

The table in this panel displays results of an **Interobject Misalignment** analysis. It contains a list of Alignable Faces, described in a list of Object Sets, and corresponding Misaligned Faces.

All misaligned face pairs corresponding to the analyzed objects are listed in the table.

- **Align Faces** – select a face pair in the table and click **Align Faces** to align selected faces.
- **Clear All Analysis Data** – this button removes all information from the tables.
- **Display Log for Object Set** – checking this causes the **Model Analysis** dialog box to display a healing log which includes information about operations performed on an object set during the healing process.
- **Auto Zoom to Selection** – if this option is checked, the modeler automatically zooms to the item selected in the table.

After validation check is performed, the pairs of objects that intersect are chosen for analysis. Use the analysis results to find whether objects have faces that can be aligned.

Choose all the bodies that intersect with another body.

1. From the **Model Analysis** dialog box choose perform/Analyze Interobject misalignment. Or you can run **Modeler** > **Model Analysis** > **Analyze Interobject Misalignment**.

   If the analysis finds object pairs that can be aligned, they will be displayed in the **Objects Misalignment** tab.

2. You can select individual or multiple rows and perform Align Faces. In some cases, face alignment will fail if the topology of the body changes by a large factor after alignment.
3. Identify individual bodies and body pairs that fail to mesh.
4. Perform Mesh analysis on individual objects and object pairs.
5. Review the reports and fix the errors.

Surface Mesh (Single/Pairs) Tab

The panel displays the results of a surface mesh analysis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. You can display results for:
   - Individual Objects
   - Object Pairs

   > **Note:**
   >
   > **Auto Zoom to Selection** – if this option is checked, the modeler automatically zooms to the object or object pair selected.

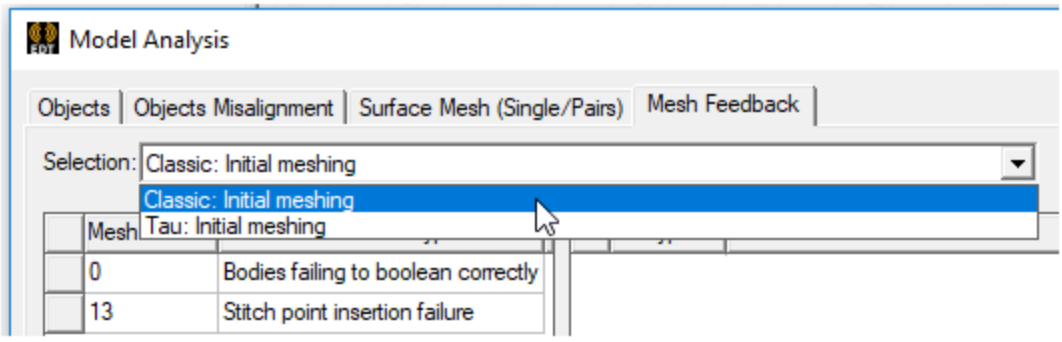
2. The results table contains the following information:
   - **Object** – column listing object name or a pair of object names.
   - **Last Analysis Status** – column stating the meshing status of the object or object pair.
     - Mesh Success
     - Mesh Failure
   - **Error Type** – this column gives the category of error that caused the mesh failure.
   - **Error Detail** – provide specific geometry information regarding mesh error location.

   Display options include:

   - **Display Mesh Analysis** – checking this displays further details concerning each error to be listed.
   - **Auto Zoom to Selection** – checking this causes the modeler to automatically zoom to objects or faces corresponding to the error.

Mesh Feedback Tab

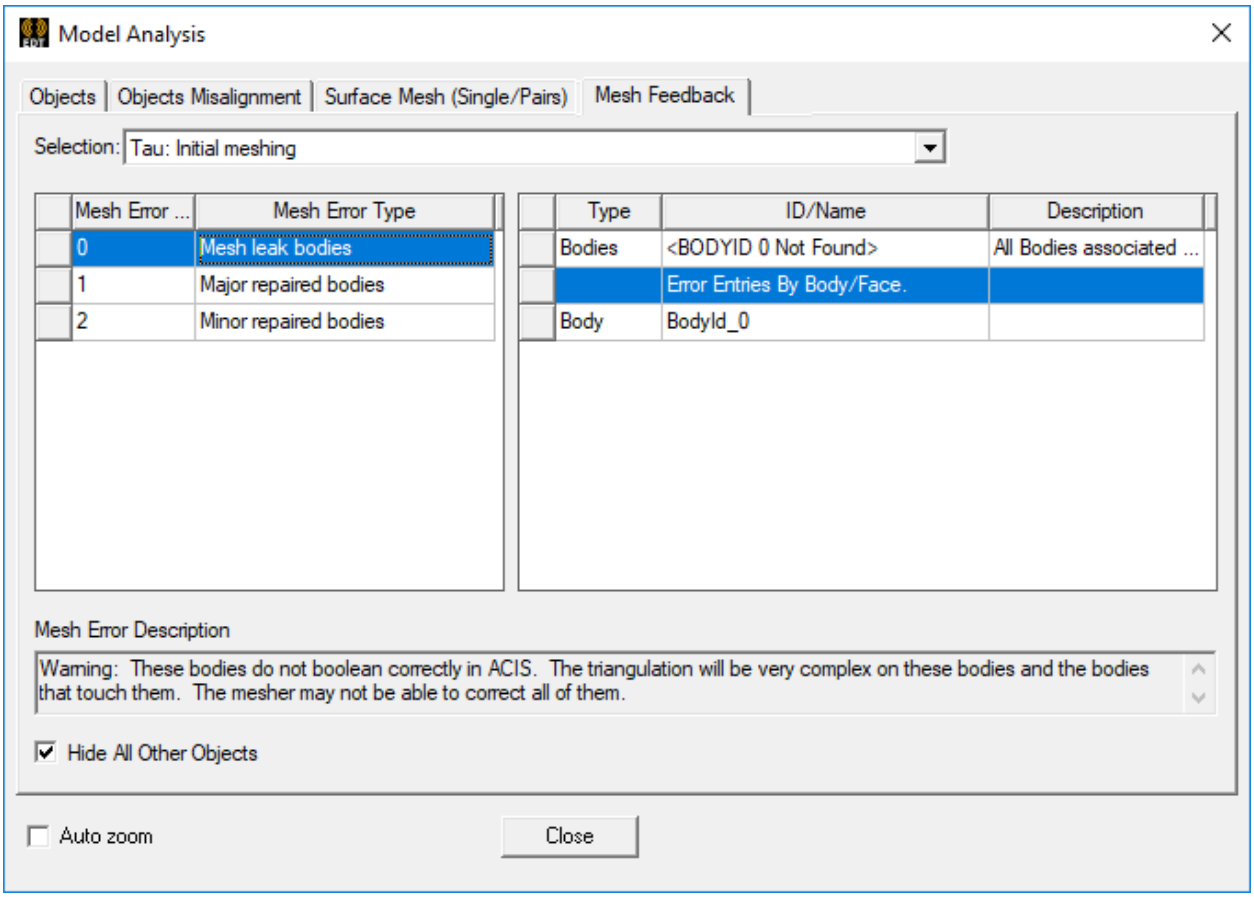
The tables in this panel list model errors as viewed by the mesher. If the project has solutions by different solvers, the **Selection** drop down lets you select which to view.



The left side table has columns for Mesh Error Serial ID and Error Type.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mesh Error Serial ID** – the 0 item is the body with which the subsequently listed entries are associated.
- **Mesh Error Type** – this column gives the category of error that caused the mesh failure, for example, Non-Manifold Edge, or Point Insertion Failure.



The display in the table on the right side of the **Model Analysis** window depends on your selections from the left hand table. When you select a row in errors panel (the left panel as shown in above figure), all the faces are selected (second row in right panel) and if you enable **Auto Zoom**, the view fits to the errors:

- **Type** – whether the error applies to Bodies, Faces, or Triangles. If you select a particular error type, the columns for Type, ID/Name and Description are filled in.
- **ID/Name** – the object name or object ID for the error type.
- **Description** – the first three rows are for Bodies, Faces, and mesh Triangles. Subsequent rows, are of Error Entities By Body.Face.

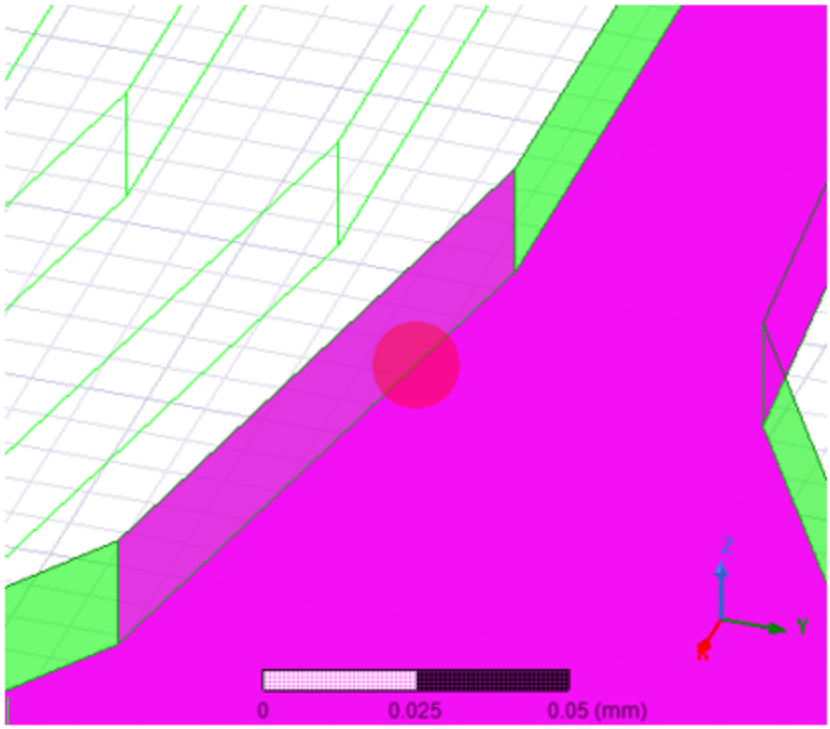
The **Mesh Error Description** field describes error message describing the nature of the selected Mesh Error Serial ID, the implications, and provides a recommended response.

Display options include:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Hide All Other Objects** – hides objects in the Modeler window not associated with the selected Bodies, Faces, or Triangles.
- **Auto Zoom to Selection** – checking this causes the modeler to automatically zoom to objects or faces corresponding to the selected Mesh Error Serial ID (on the left side table) or the ID/Name (on the right side table).

Use of these selections let you more easily view and respond to the errors. If you enable **Auto Zoom**, any selection whether it is body(ies), face(s) or error triangle(s) or error segment(s) will fit to view. If all the faces are also selected (second row in right panel), then errors and selected entities will fit to view.



The error shown in the figure above is a contact error between the faces. Focusing only on the errors when an error is selected in right panel, improves visualization of these meshing errors and provides you with precise information in order to take corrective action quickly.

Mesh Error Markers

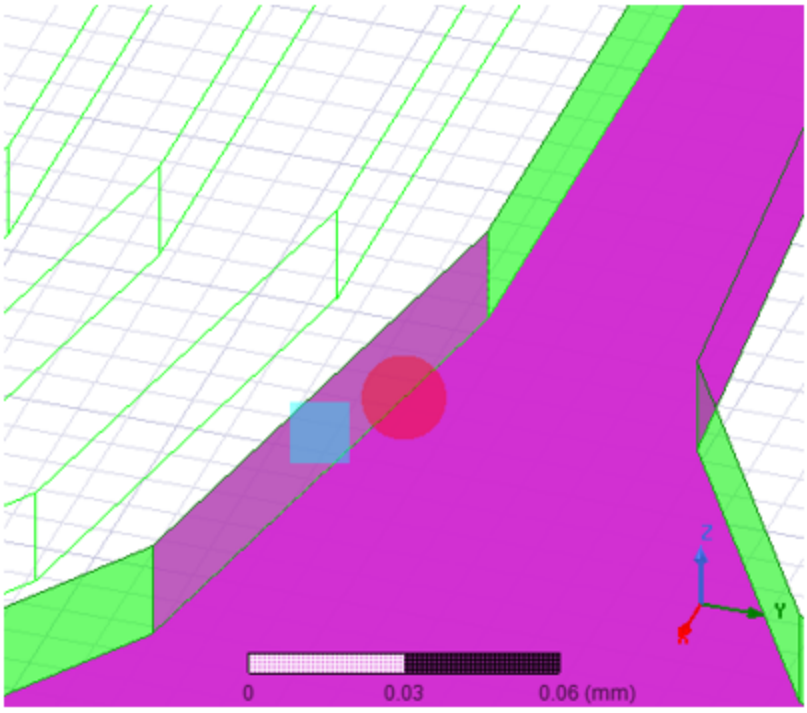
As shown above the mesh errors are marked by ‘Red Circles’. The following table describes all such markers:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Mesh errors
	Mesh warnings
	Mesher information
	Selected Faces

These markers are located at the center of errors or center of faces and their size correspond to the maximum side size of the entity, such as error triangles, error segments or faces. The size of these markers is clamped to a maximum and a minimum size. Additionally they have some transparency. The size and transparency of these markers cannot be changed by the user.

If you select an individual face in the right panel, then the face is marked using a blue square as shown below:



If many individual faces are selected in the right panel, each face is marked by a blue square. In order to find which blue square corresponds to which face in the right panel in the dialog box,

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

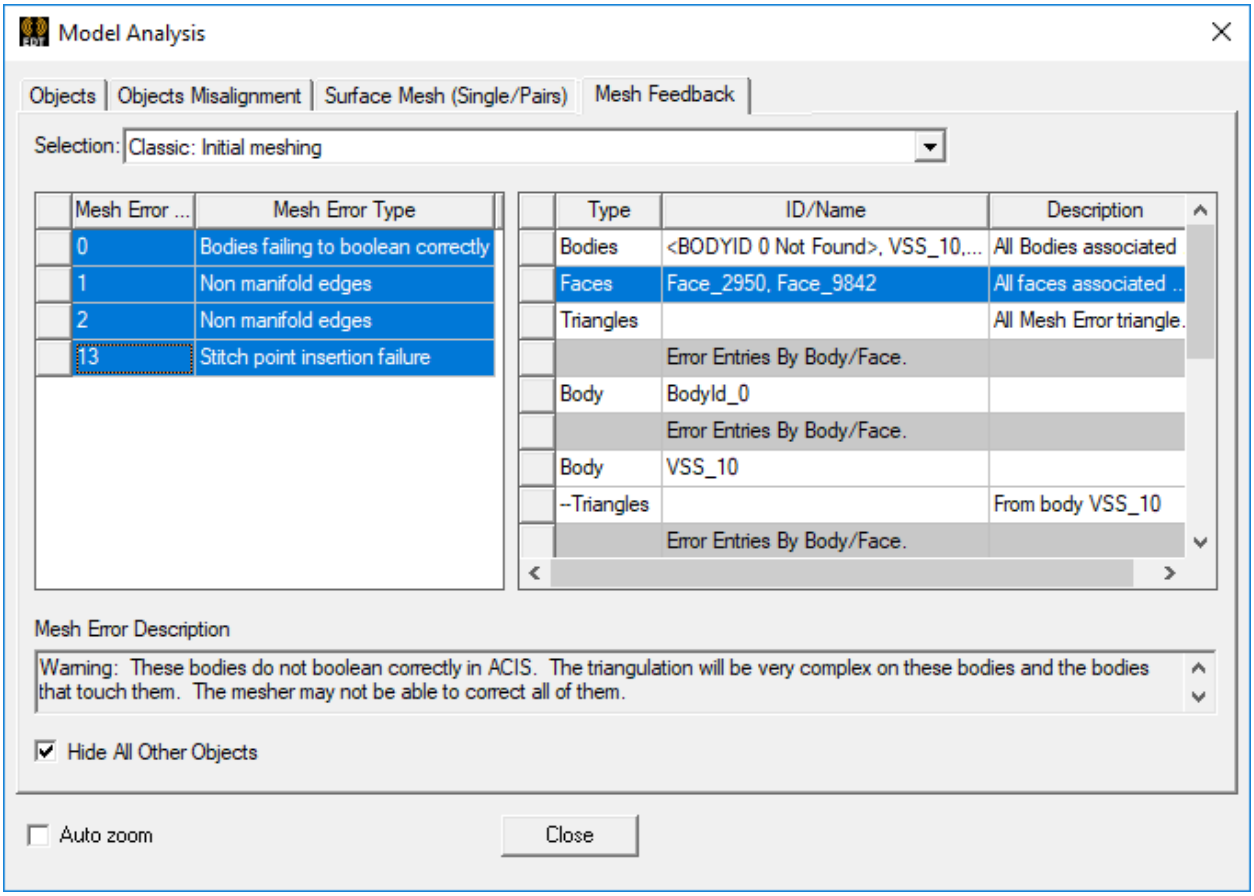
click on the blue square. This action selects the corresponding face row in the right panel of the dialog.

Face Selection

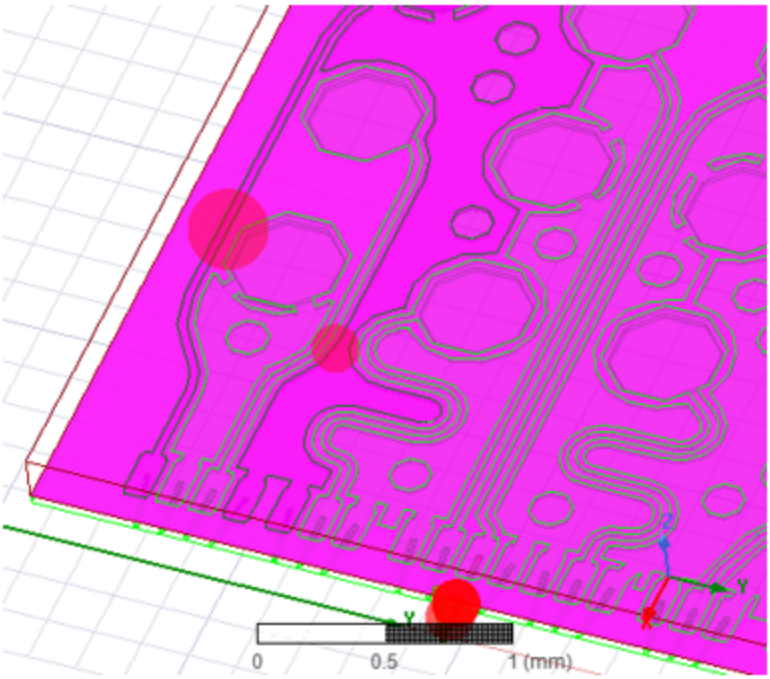
When the all faces row or individual face rows are selected, the Modeler selects the face(s). This enables the face related commands in the Modeler window. Also when you select faces this way, it makes the corresponding bodies of the selected faces visible, even when 'Hide all other objects' is enabled. This provides some contextual information about the face location with respect to its body.

Multiple Selection of Mesh Errors

You can select multiple rows in the right pane of the **Mesh Feedback** tab. To view all the meshing errors, select all the rows and enable **Auto Zoom**, as shown in figures below:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With **Auto Zoom** enabled, the view fits to all the errors, providing an overview of all the mesh errors, as shown in above figure.

Also, when multiple errors are selected, right panel shows the merged information about all bodies, faces, error triangles etc. And Mesh Error Description text box shows description of all the errors.

Design Settings

The tabs and settings within the *Design Settings* dialog box are dependent on the design type, and there is a product-specific topic for each applicable design type. Please click the appropriate link below for the product of interest:

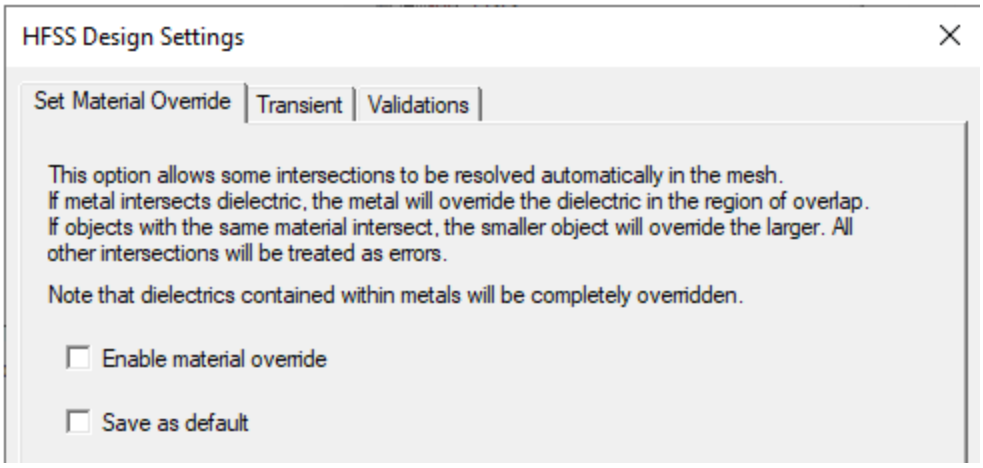
- HFSS
- HFSS Transient
- HFSS 3D Layout
- Icepak
- Mechanical
- Q3D
- 2D Extractor

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Design Settings for HFSS

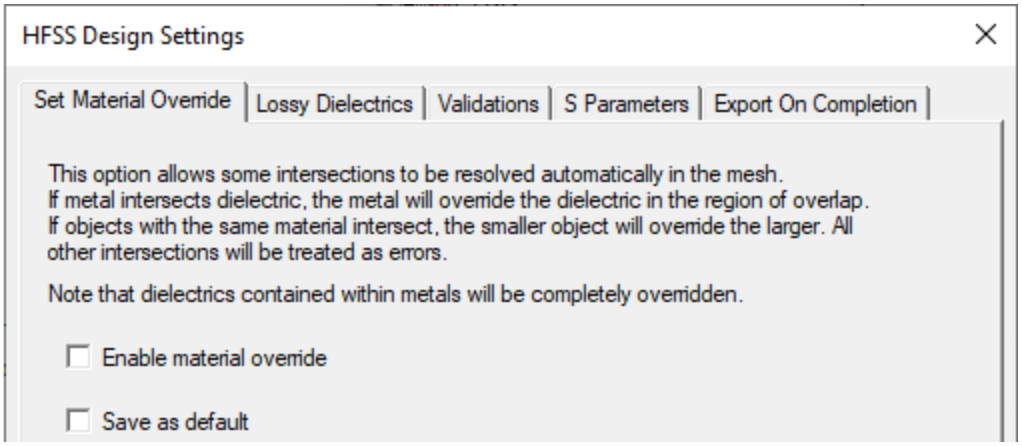
The **HFSS>Design Settings** command displays a dialog with tabs appropriate to the HFSS Solution settings. For HFSS driven solutions, using Modal or Terminal options, the tabs include **Set Material Override,** for automatic **Lossy Dielectrics**, **DC Extrapolation**, **Validations**, for **S-Parameter** definition, for **Export On Completion**, and for **Adaptive Mesh**.

For Transient designs, the dialog contains Set Materials Override, Validations, and a Transient tab.



For SBR+ Designs,

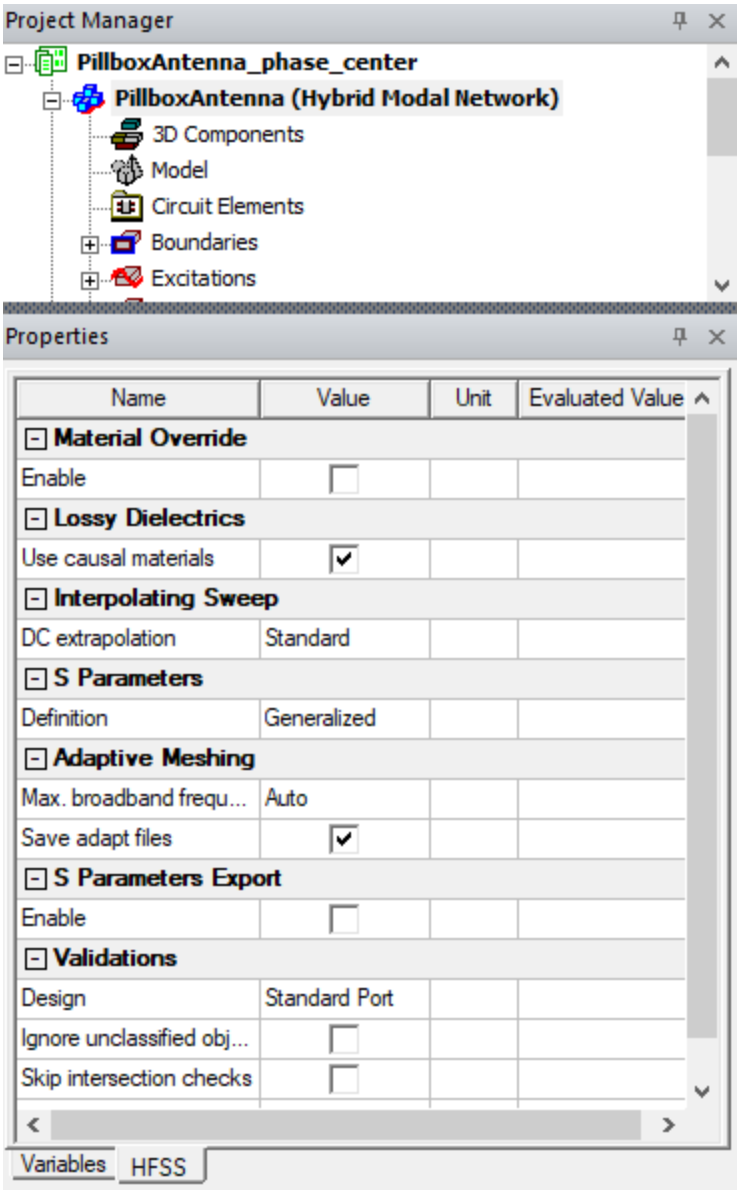
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For Eigenmode or Characteristic Mode, only the Set Materials Override and Validations tabs are needed.

Selecting a current design in the **Project** tree also displays the Design Settings in **Properties** window under the solver tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Set Material Override Tab

The **Set Material Override** tab includes text note and a check box to **Allow metals to override dielectrics**. The purpose of this feature is to allow you to avoid doing explicit subtraction in the modeler. One example application is a via that passes through many dielectric layers--with the option turned on, the via does not have to be subtracted from the layers.

The **Set Material Override** option allows some intersections to be resolved automatically in the mesh. If metal intersects dielectric, the metal overrides the dielectric in the overlap region. (That is, the metal object is subtracted from the dielectric.) If objects with the same material overlap,

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the small object overrides the larger. (That is, the small object is subtracted from the larger.) All other intersections are treated as errors. Normally, the modeler considers any intersection between 3D objects to be an error.

To use this feature, check **Enable material override**.

In the meshing process, the dielectrics are locally overwritten by the metals in the intersecting region. That is, the part of the dielectric that is inside the metal is removed, and if the dielectric is completely inside, the whole object disappears.

Lossy Dielectrics Tab

This option applies frequency dependent lossy materials for the solver and post processor. The materials are not modified in the design. Instead, the Djordjevic-Sarkar model is applied before the material is passed to the solver or used for post processing.

- Automatically use causal materials. (Default, unchecked).

If the assigned material is already frequency dependent, automatic creation of frequency dependent lossy materials is ignored.

This feature addresses cases where you only have simple constant material properties available, but want to automatically apply a general-purpose frequency dependence to ensure causal solutions when solving frequency sweeps. Automatic causal material calculations are not performed under the following circumstances:

- If the solution type is eigenmode
- If the material permittivity or loss tangent is anisotropic
- If the material permittivity or loss tangent is spatially dependent
- If the material permittivity or loss tangent is frequency dependent
- If the material itself is not a lossy dielectric

Otherwise, when enabled the Djordjevic-Sarkar model is applied to all constant lossy dielectrics. These are defined as having a constant permittivity that is greater than one and a constant loss tangent that is greater than zero. The inputs to the Djordjevic-Sarkar model are the material's constant permittivity and loss tangent, plus the standard default values of measurement frequency (1 GHz), DC conductivity (1e-12 S/m), and DC permittivity (none). The outputs from the Djordjevic-Sarkar model are the expressions for permittivity and conductivity. These expressions, plus zero loss tangent, are used in place of the material's constant properties.

When reading legacy designs (HFSS 12 and earlier), this feature is unchecked.

DC Extrapolation Tab

The **DC Extrapolation** tab option affects Interpolation Sweep setup. If you select Standard DC Extrapolation, the software computes values automatically, and the tab for DC Extrapolation

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

does not appear on the Interpolating Sweep setup.

For Advanced DC Extrapolation, the Interpolating Sweep setup includes the DC Extrapolation tab, so you can set a Minimum solved frequency.

The **Design Settings** dialog also contains check boxes to **Save As Default**.

> **Note:**
>
> Users must be careful: this setting changes the "ground rules" of the modeler, and may have unexpected results.

Validations Tab

The **Validation** tab offers choices for Model validation and HFSS validations to control the extent of validations performed, and therefore the time involved.

Model Validation choices are whether to:

- Ignore unclassified objects
- Skip intersection checks

You also control the Entity check level as Strict, Basic, Warning Only, or None.

The HFSS choices are whether to:

- Perform full validations with standard port validations. Standard validation does not check for internal or floating ports.
- Perform full validations with extended port validations. The Extended validation checks to see if a wave port has been applied to an internal face, or if a wave or lumped port does not have solve inside geometry on either side.
- Perform minimal validations. This choice disables port validation options (neither is performed), and skips boundary overlap validation.

The default is to Perform Full Validations. However you can set your own default choices by using the Save as Default check box.

S Parameters Tab

The **S Parameters** tab offers choices for S parameter definition for post processing as Generalized or Power. Power S parameters are calculated for the reporter, S parameter export, and the Matrix display. Changing the setting between Generalized or Power updates reports and exported data, but not the solve status.

Generalized S parameters are based on the relationship between scattered and incident fields; hence, it is possible to have S parameters greater than one if there is a strong imaginary component of the reference impedance. While correct, some circuit designers may find this

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

counter-intuitive and prefer instead to use Power S parameters. Power S parameters are based on the transfer and reflection of power, which must be less than unity for any passive device.

Note: Setting Power S using UpdateRegistry requires a workaround.

Adaptive Mesh Tab

Maximum Number of Frequencies for Broadband Adapt: Normally the decision for how many frequencies to use for broadband adaptive meshing is set automatically. Because the decision is not straightforward, this option is for advanced users to manually set the maximum number of frequencies to solve during broadband adaptive meshing. This feature works for both auto and advanced simulation setups.

Save Adaptive Mesh Control Files: Allows you to disable saving adaptive mesh control files to save disk space. However, if unchecked, reverting the mesh to a previous pass requires adapting from the initial mesh. Changing the setting reverts to the initial mesh.

Export On Completion

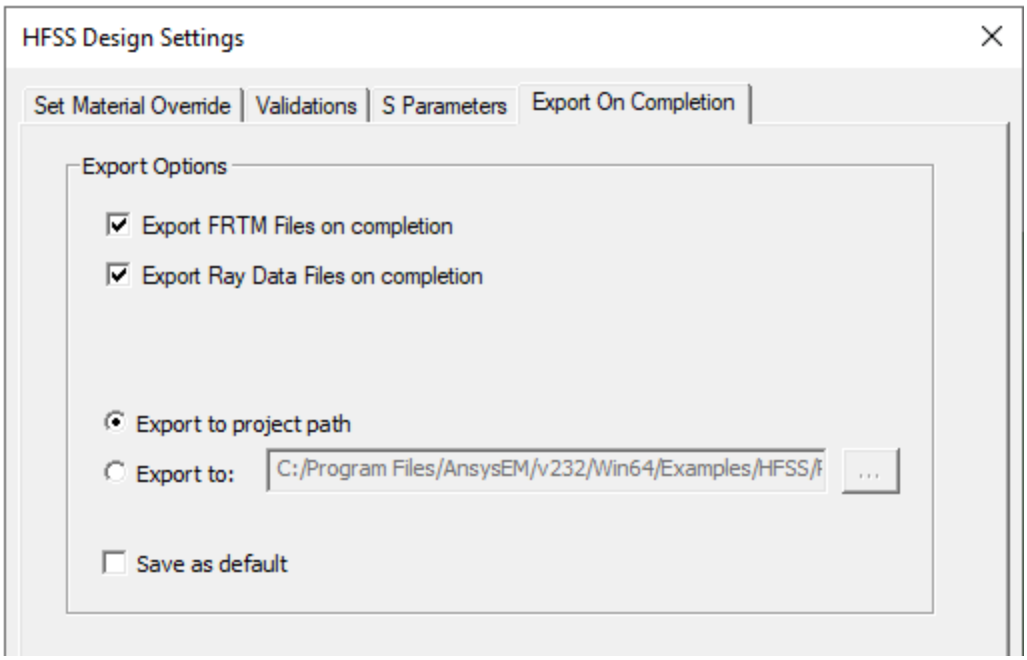
This applies to HFSS 3D Driven Modal/Terminal and HFSS 3D Layout and, in a simpler way, to SBR+ or Hybrid solution type designs.

The SBR+ solver, when examining range-doppler problems, produces its raw data in a proprietary file format with the extension ".frtm". The **Export On Completion** setting lets you specify that output, and enabled, lets you either export to the project path or a specified file location.

In addition, the SBR+ ray data export feature is available for Hybrid SBR+ and SBR+ solution-type, for any frequency domain definition (single point, discrete sweep, interpolating sweep, FFL, etc.), source and/or radiation domain configuration, in range-Doppler simulations, and for VRT plots, in a proprietary file format with the extension ".hdm". The ray export feature also supports distributed simulations both in Hybrid and SBR+ solution-type. The solution distribution differs between Hybrid and SBR+ solution-type. In Hybrid, SBR+ distributes jobs according to frequency domain subsets of the original domain, but all distributed jobs shoot the same number of rays. In SBR+ solution-type, all distributed jobs share the same complete frequency domain, but each job only performs a subset of the ray tracing algorithm.

The simulation Ray Export settings appear under the **Export On Completion** tab in the HFSS Design Settings.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



By default, the SBR+ files appear in a sub-folder along side the aedtresults folder, and is named .aedtexport. Within that folder, exported files are segregated by design name, setup name, parametric variation, and date/time of solve.

It is your responsibility to ensure that there is enough available space on disk to perform ray export both for solver and VRT. The ray export algorithm performs very limited disk-space availability checks.

If you name a specific folder, all exports (named distinctly) are placed in that folder.

A frequency group represents a subset of frequencies chosen from the frequency domain of the simulation. In Hybrid SBR+, each group simulates up to *N* frequencies, with *N* the number of tasks allowed for each distributed simulation. In SBR+ solution-type, all frequencies are simulated at once (e.g., all files are under Group_0), but each node only processes a subset of the total number of rays. There is currently no way of knowing what frequencies belong to what group other than partially parsing one of the files in the export directory.

Ray Export files are exported either under a default <project_name>.aedtexports folder, or in a folder of the users choice. Exports are grouped by export date and time, and then by design, solution/variation type, and lastly by group number. A group represents a subset of frequencies chosen from the frequency domain of the simulation. In Hybrid SBR+, each group simulates up to N frequencies, with N the number of tasks allowed for each distributed simulation. In SBR+ solution-type, all frequencies are simulated at once (e.g., all files are under Group_0), but each node only processes a subset of the total number of rays. There's currently no way of knowing

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

what frequencies belong to what group other than partially parsing one of the files in the export directory.

Ray export file names follow a specific format:

`(sbr|cw)export_src<`*`srcName`*`>_rank<`*`rankNumber`*`>_b<`*`batchNumber`*`>.hdm`

The export type (`sbr|cw`) indicates if the .hdm contains SBR/UTD or CW ray exports. `srcName` follows the following convention:

FEBI or NF/FF links: _<*excitation*>_<*port/mode*>

parametric antennas or FFD/NFD files: _<*excitation*>

bistatic plane-wave: integer index <*N*> starting at 0, incidence angle

monostatic plane-wave: 0

The other two file-name groups are simply numerical identifiers to disambiguate ray exports.

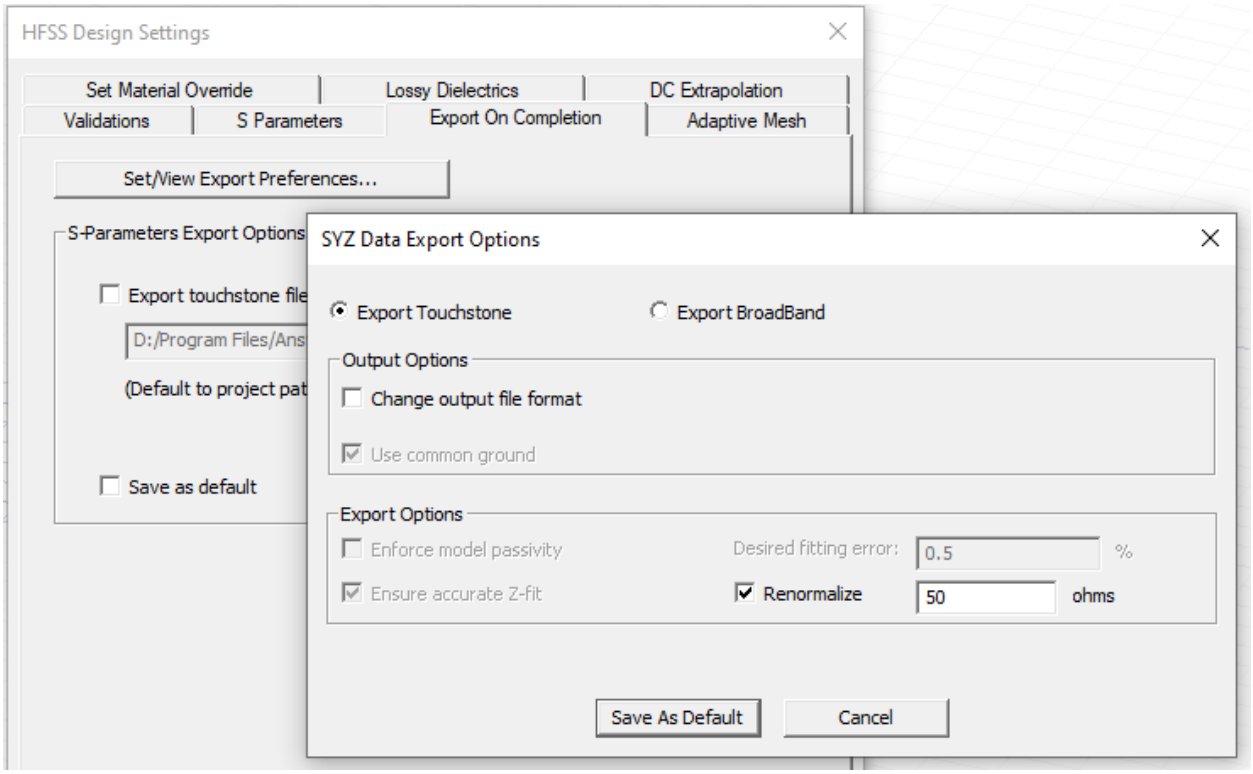
The .hdm format begins with an ASCII header that completely describes the file format and content in both a human-readable way and a machine-readable way. Parsers for the hdm format are available in the pyAEDT package. A binary section follows the ASCII header. Please refer to the available parsers and to a hdm header for more information.

For **HFSS driven solutions**, or HFSS 3D Layout, after a sweep (discrete, interpolating, or fast) simulation is successfully completed, you can choose to automatically save a Touchstone file for the sweep's S-parameter data into a folder. By default, this folder is named after the project. The exported directories/files are not managed by Ansys Electronics Desktop. That is, you can rename the project, design, setup, etc. and previously exported directories/files will NOT be renamed. For Optimetrics simulation, export is per solved variation. Touchstone files are exported only when the simulation completes without error

The **Export S Parameters** tab lets you **Set/View Export Preferences** and specify whether to Export Touchstone file after completing frequency sweep. Checking the export feature enables you choose to override the default project path. You can save your preferences as the default. Preference is per design type and per user. This means that the same user can have one set of preferences for 3D layout design and another set for HFSS 3D design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clicking the **Set/View Export** Preferences button opens the **SYZ Data Export Options** dialog.



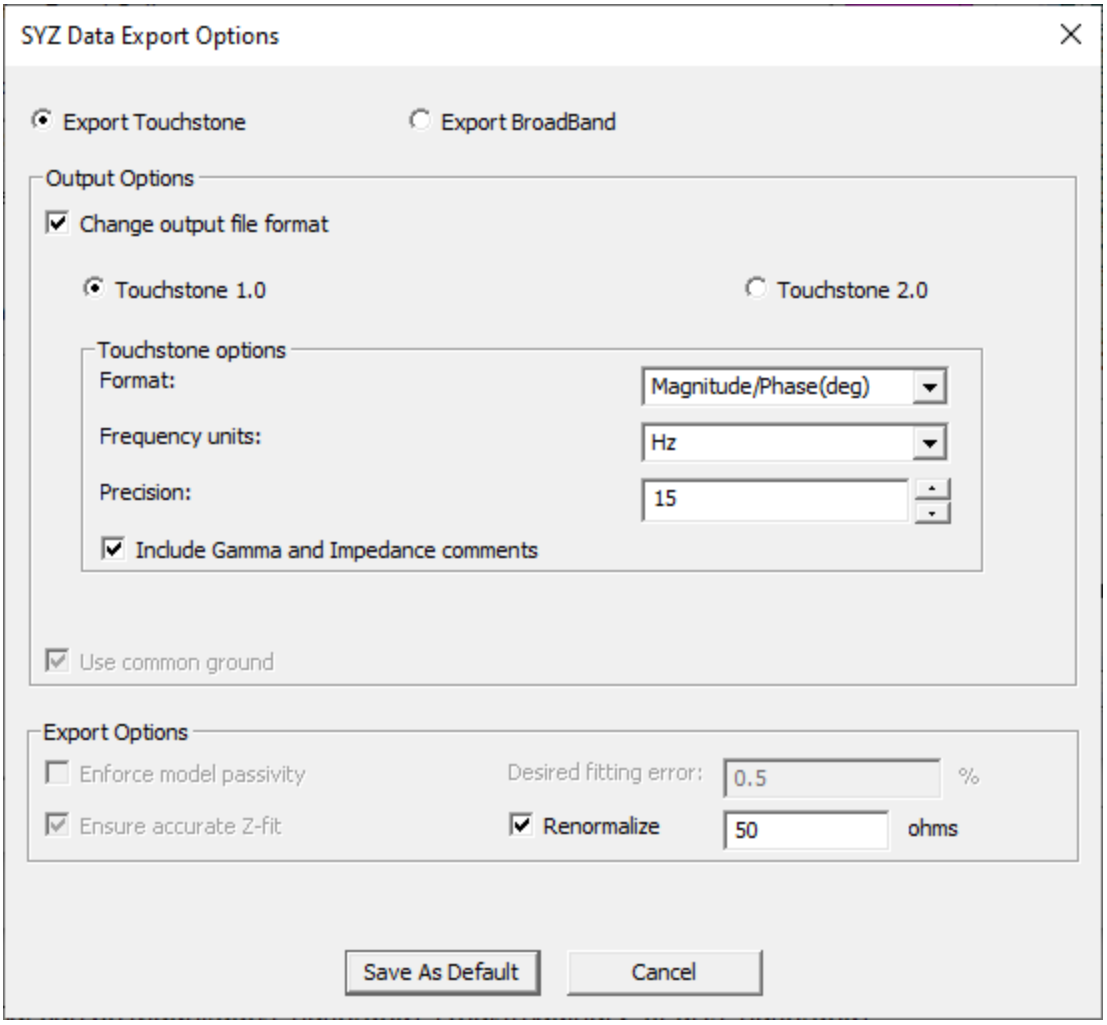
The **SYZ Data Export Options** dialog contains a range of choices for Output format, including advanced options.

Macromodel Output Options.

Checking Change output file format displays additional choices that lets you select Touchstone 1.0 or Touchstone 2.0, as well as additional Touchstone options. Unchecking conceals the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

output file format choices.



- Format can be Magnitude/Phase(deg), Real/Imaginary, or dB/Phase(deg)
- Frequency units can be Hz, KHz, MHz, or GHz.
- Precision can be to the number of digits you specify.
- Include Gamma and Impedance comments. Unchecking allows you to suppress writing Gamma and Impedance.

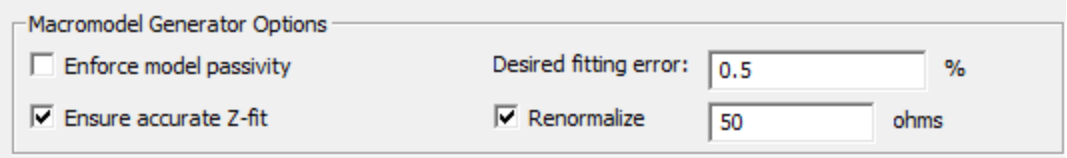
You can also specify whether to Use common ground.

Macromodel Generator Options include:

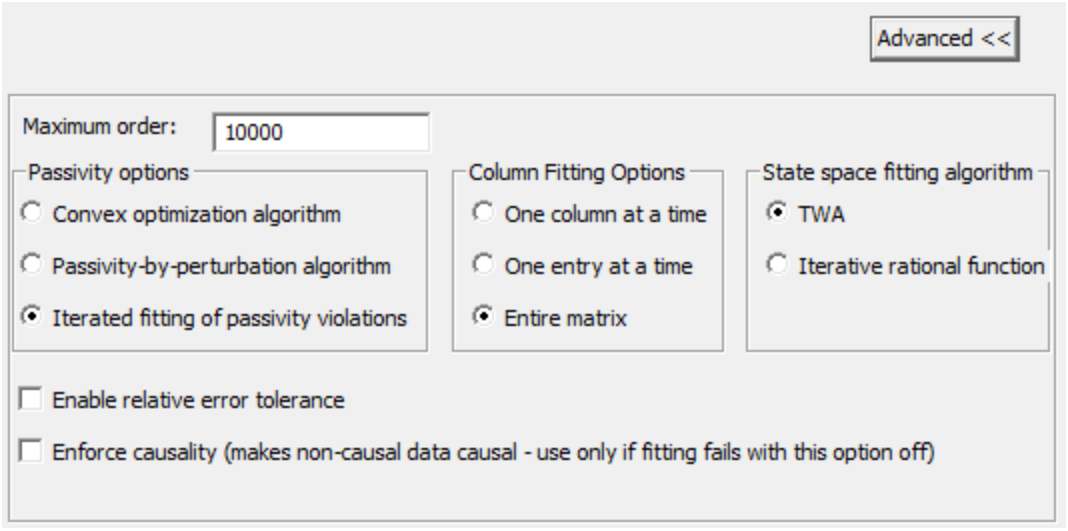
- Enforce model passivity
- Desired fitting error percent
- Ensure accurate Z-fit

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Renormalize to a specified value



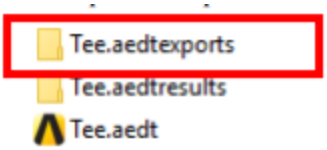
Clicking the **Advanced** option button displays or hides the Advanced options.



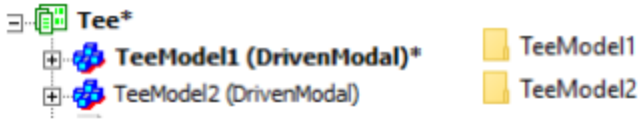
Choosing **Save As Default** saves the selected options as your user preferences for automatic Touchstone export and for subsequent launch of the **SYZ Data Export options**.

Automatic Touchstone File Exports

The top level export folder is named after the project file.

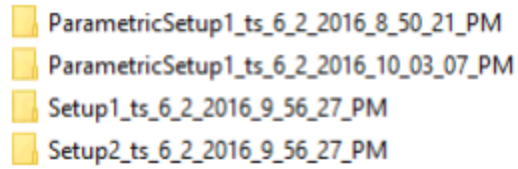


Within the top level export folder, a separate folder is created for each design that has exported touchstone files.

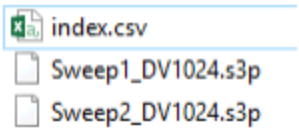


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

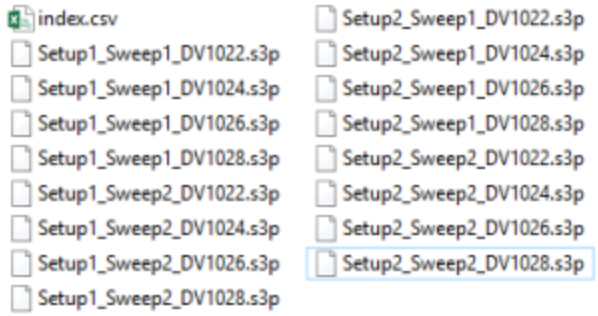
In each design export folder, exported files are further organized based on the setup that is being simulated. For non-Optimetrics simulation, the sub-folder is named after the solve setup and followed by a “time-stamp” string2. For Optimetrics simulation, the sub-folder is named after the Optimetrics setup and followed by a “time-stamp”. string.



Touchstone files are exported into these sub-folders and an index file is created per sub-folder to help users track the exported data. For non-Optimetrics export, files are named as <Sweep>_ DV<unique ID>.



For Optimetrics export, files are named as <Setup>_<Sweep>_DV<unique ID>.



The index.csv can be opened in Excel. Variation information is listed in row/column format for readability.

A	B
Var_ID	offset
1024	0.2in
1026	0.3in
1022	0.1in
1028	0.4in

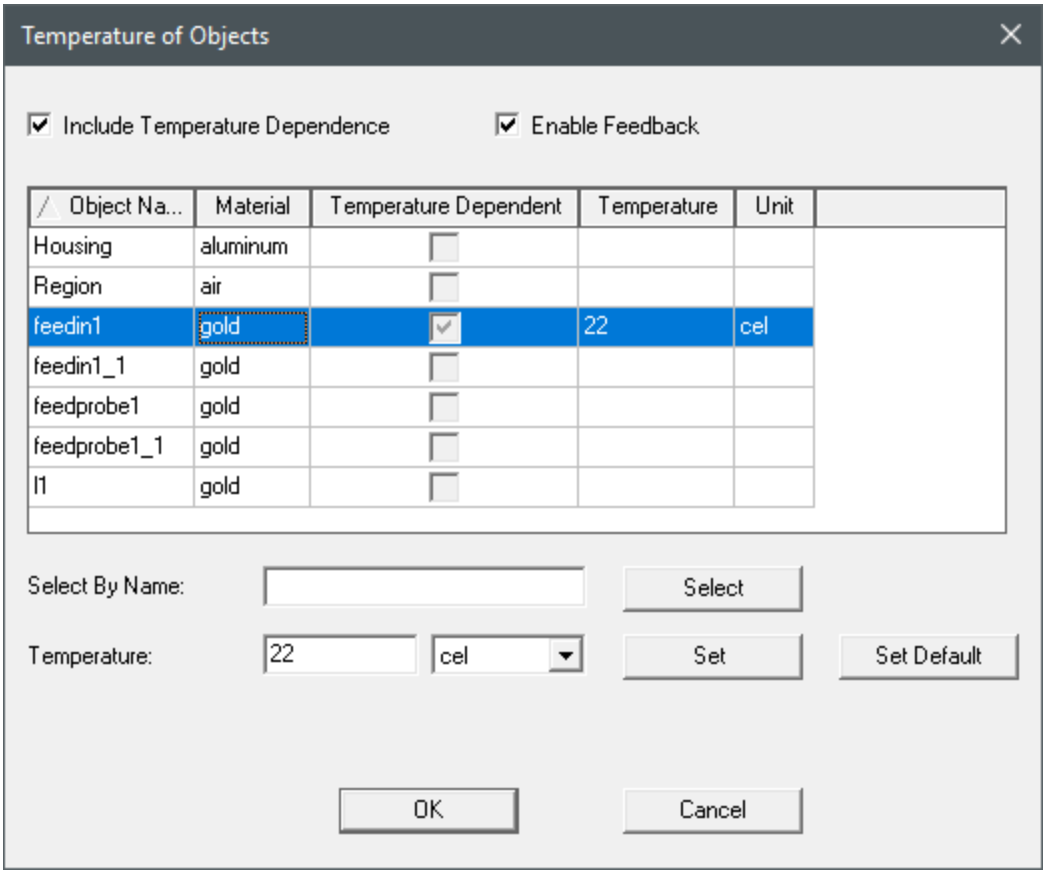
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Temperature of Objects

To set the temperature of objects and to enable feedback for use in two-way coupling:

1. Use the **HFSS** > **Set Object Temperature** command to display the **Temperature of Objects** dialog box. You can also right-click the design in the project tree and select from the shortcut menu.

   This window includes a table of the objects in the design. In order, the table's columns list: object name, material, a check box to show whether each object has temperature-dependent features, Temperature (if applicable), and unit. To enable temperature dependence, you must edit the material properties and include a thermal modifier. See: Specifying Thermal Modifiers.



2. To enable editing, select **Include Temperature Dependence**.

   This makes the table objects selectable. The headers for the **Object Name** column and the **Material** column include sort direction arrows. You can invert the sort direction in each column by clicking the header. If the list is longer than the display, a scroll bar will appear.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

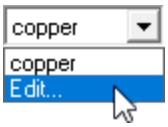
You can also use the **Select by name** field to find objects by name. Enter the name of the object you want and click **Select**.

Selected objects are highlighted, and you can make multiple selections.

3. To set the temperature for the selected object(s), type a value or an existing variable name in the text field.
4. Select the units from the drop-down menu.
5. Click **Set** to apply the value to the selected objects, or click **Set Default** to make the specified values the default.

   If you click **Set Default**, the row(s) for the selected object(s) display the temperature value and units.

6. To edit the material for an object, click the object's material to display a drop-down menu.



   Click **Edit...** to display the Materials dialog box.

7. To enable coupling with thermal designs, select **Enable Feedback**.
   - You can perform two-way coupling between an HFSS, Q3D, or Maxwell design and an Icepak design within the Electronics Desktop. In an Icepak design, you can specify the number of coupling iterations to be run. EM Loss data calculated by the source HFSS, Q3D, or Maxwell design is consumed by Icepak when calculating thermal data in the current coupling iteration. The thermal data is then consumed by the source design in the next coupling iteration's calculation.
   - Alternatively, you can perform thermal static and transient analysis based on a high frequency solution by coupling with other Ansys thermal systems in Ansys Workbench. The solver can then use per mesh element temperatures for the object's material properties. If **Enable Feedback** is enabled but feedback is not available, the temperature specified in the dialog is used for the temperature dependent properties in the design.

   Note:

   When **Enable Feedback** is selected, the right-click menu for **Analysis** and the analysis setup includes the **Revert to Initial Temperature** command. Selecting this command invalidates the current solution.

8. To close the dialog box and accept changes, click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Model Preparation Commands

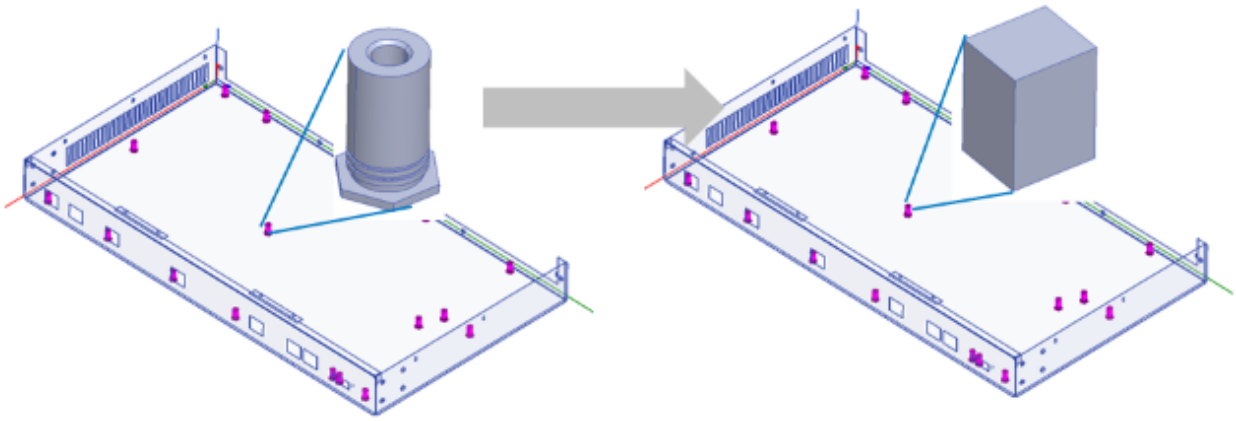
Use the **Modeler** > **Model Preparation** commands to prepare a model.

These commands include:

- Simplify
- Heal
- Stitch Sheets
- Align Faces
- Remove Faces
- Remove Edges
- Perform Explicit Subtractions

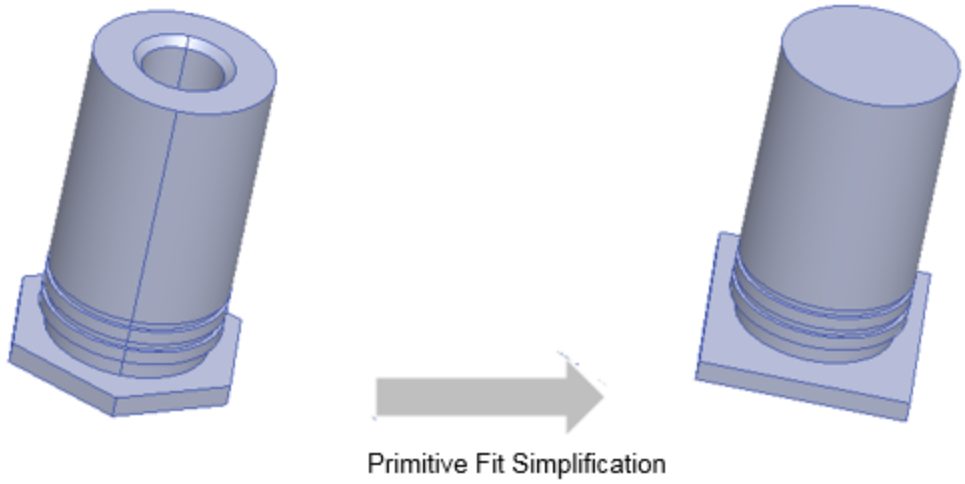
Simplify Command

Use the **Modeler** > **Model Preparation** > **Simplify** command to converts a complex MCAD object into simpler primitives which are easy to mesh and solve. The operation can be applied on any selected object and not just imported objects. You can specify the type of simplification as Bounding Box, Primitive Fit, or Polygon Fit.



Bounding Box Simplification

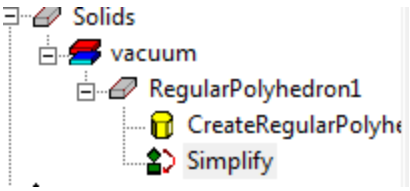
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Primitive Fit Simplification

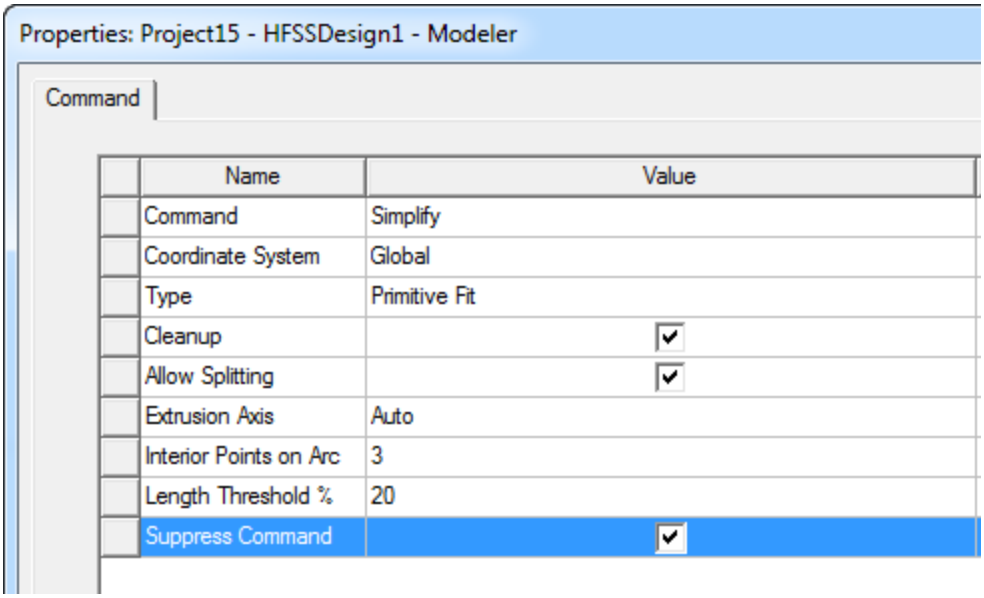
Polygon Fit Simplification

The History tree shows the **Simplify** command as being applied to the object.



Any parameters you specify in the dialog you can also edit in the Properties dialog for the History tree **Simplify** command to get a different simplification. The properties display in the docked properties window and include only the parameters that are used by your currently selected **Simplify** command type. None of the parameters of simplify operation accept variables.

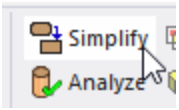
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



1. Select an object.

   This enables the Simplify command on the **Modeler** > **Model Preparation** submenu.

2. Click **Modeler** > **Model Preparation** > **Simplify** or on the **Model** ribbon, select **Simplify**.



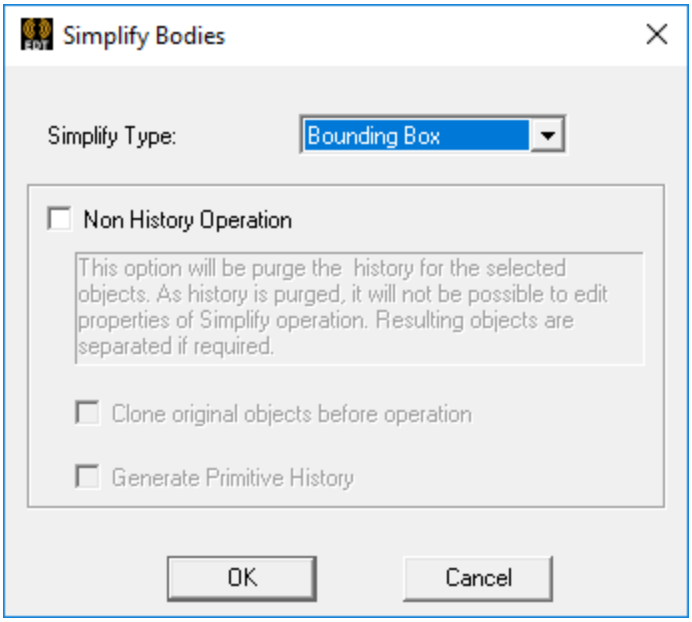
This opens the **Simplify Bodies** dialog box. A drop-down menu lets you select the degree of simplification, from Bounding Box, Primitive Fit, or Polygon Fit.

Bounding Box Simplify Type

If you select Bounding Box as the Simplify Type, there are no additional parameters. The object is replaced by its exact bounding box as computed in operation's coordinate system. You can change the operation CS to get a bounding box in an appropriate orientation. For the **Non**

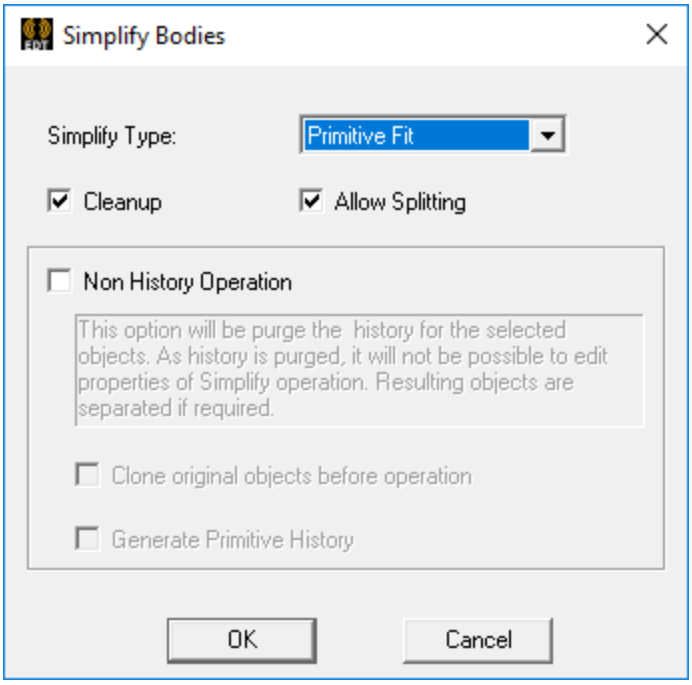
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

History Operation options, see below.



Primitive Fit Simplification

For Primitive Fit simplification the object is replaced by a set of primitives shapes like Prism, Cylinder, Cone and so forth. This simplification type typically produces the simplest geometry with highest number of primitives. Primitive fit has the following options: Cleanup and Allow Splitting, whether to Separate Bodies and Purge History, and if so, whether to clone original objects before the operation.



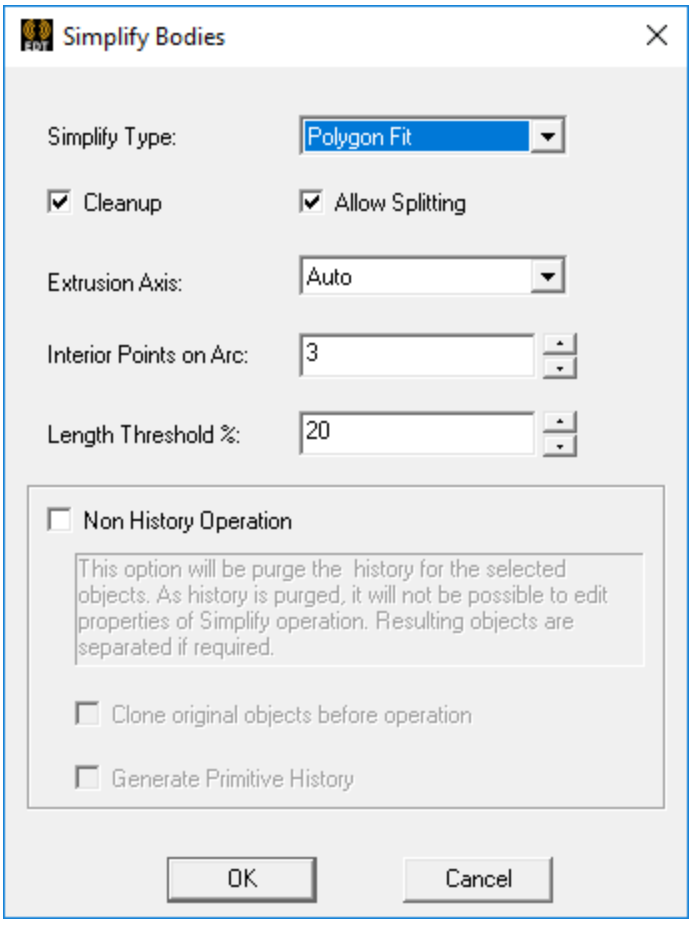
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For Primitive Fit, the parameters are:

- **Cleanup** - This option allows you to clean the model before performing simplification. Cleanup include removing small features like rounds, fillets, chamfers in a solid body. It is recommended to set this option to get more simplified result.
- **Allow Splitting** - This option controls if object should be split during simplification. If this option is selected, complex object will first split into multiple pieces and each piece will be further simplified. Setting this option will result in an object which will match more closely with original object. It is recommended to set this option.
- For the **Non History Operation** options, see below.

Polygon Fit Simplification

For Polygon Fit simplification, the object is replaced by set of polygon swept along normal and other primitives like box and cylinder. This simplification type typically produces geometry closest to original object with highest number of primitives. If you select Polygon Fit as the Simplify Type, the parameters are Clean Up, Allow Splitting, Extrusion Axis, Interior Points on Arc, and Length Threshold. For the **Non History Operation** options, see below.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

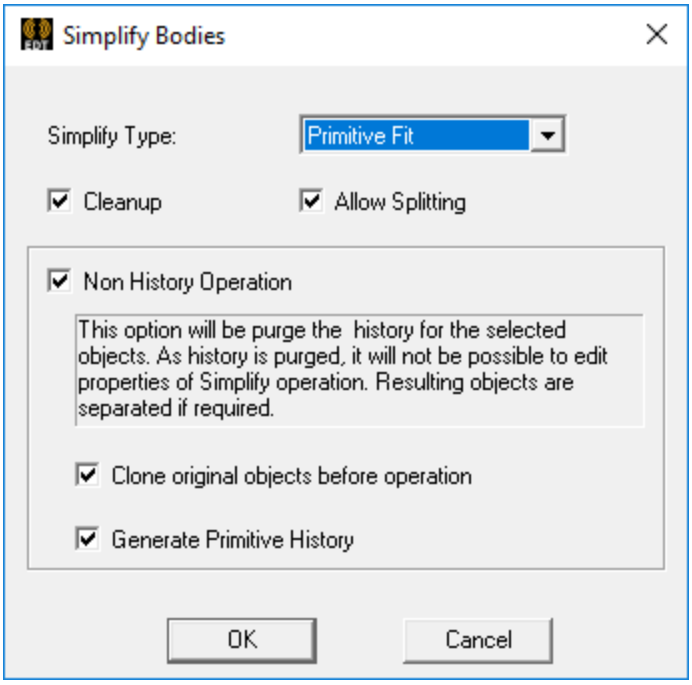
For Polygon Fit, you specify parameters for

- **Cleanup** - This option allows you to clean the model before performing simplification. Cleanup include removing small features like rounds, fillets, chamfers in a solid body. It is recommended to set this option to get more simplified result.
- **Allow Splitting** - This option controls if object should be split during during simplification. If this option is selected, complex object will first split into multiple pieces and each piece will be further simplified. Setting this option will result in an object which will match more closely with original object. It is recommended to set this option.
- **Extrusion Axis**, whether Auto selected, or X, Y, or Z. - This option specified a normal plane in which polygon profile is looked for. The default is Auto which allows an algorithm to determine best possible plane for polygon profile. You can specify a axis to help the algorithm, particularly when there is a draft or chamfer in the extrusion direction. The polygon profile is then swept in the extrusion direction.
- **Interior Points on Arc** - This specifies number of interior points created when a curve on the polygon profile is represented by line segments. The range is 1 to 100 with default being 3. Total number of points used to represent a curve is 5 when number of interior points is 3. This number specifies the maximum number of interior points to add. The number of points added could be less if the Length Threshold is already met.
- **Length Threshold %** - This specifies the length of edge as percentage of maximum length of an edge in the profile. It is specified as percentage of maximum edge length. If length of edge is more than the specified threshold, more interior points are added until the number of “Interior Points on Arc” is met.
- For the **Non History Operation** options, see below.

Non History Operation Options

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Non History Operation options are available for all Simplify Types.



With this option selected, the input body is simplified into multiple primitives (or polygon shapes) and a new part is created for each simplified shape. This option also purges the history of original part and it is not possible Edit Properties of the Simplify operation. You can choose to clone original object before simplification to retain history of original part. You can also choose to generate Primitive History.

Simplify result with separate bodies option

Use of Coordinate System

By default, the current working CS is used as the Simplify operation CS, but you can also change operation CS through the **Simplify** command property window or Property dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heal

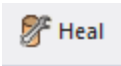
The **Heal** command provides a way to correct geometric violations and to remove specific kinds of small features. When models are imported, two types of errors can occur - geometry errors and topology errors. Geometry errors are errors in definition of the underlying geometry while topology errors are errors in how the underlying components like faces, edges and vertices are connected. Ansys Electromagnetics recommends that these be fixed before you invoke mesh generation.

Imported objects which have only one operation on the history tree, can be healed. (Use the Purge History command to remove unwanted history operations before using **Heal**.)

> **Note:**
>
> If you need to save the object history, save a separate copy for that purpose before you heal the object.

Healing can be invoked in different ways.

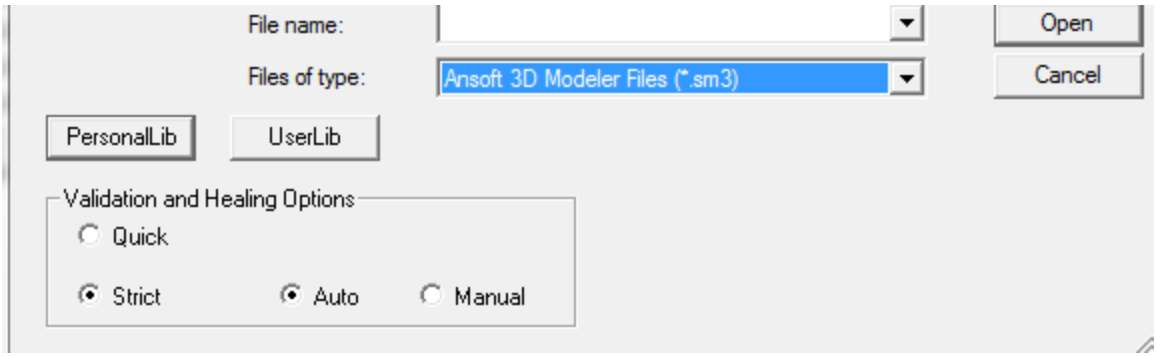
- The menu command **Modeler** > **Model Preparation** > **Heal** command, or the **Model** ribbon **Heal** icon applies to a selected object.

- Some formats permit healing during **Modeler** > **Import**. These are:
  - 3D Modeler file (*.sm3),
  - SAT file (*.sat),
  - STEP file (*.step,*. stp),
  - IGES file (*.iges, *.igs),
  - CreoParametric files (*.prt, *.asm),
  - CATIA (*.model, *.CATpart), and
  - Parasolid file (*.x_t, and *.x_b).

  Selecting these formats enables a check box at the bottom of this window, **Validation and Healing Options**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



See this table for details. The Quick option allows you to switch off healing to speed up the import process.

- The **Model Analysis** dialog box that appears after running **Modeler** > **Model Analysis** > **Analyze Objects**, or **Modeler** > **Model Analysis** > **Show Analysis** dialog includes a **Perform** action menu with **Heal Objects** as a selection.

Any of these approaches leads to the same heal process.

Basic Steps in the Heal Process

There are several steps that are performed on selected objects.

1. Entity check, according to the **Analysis Options** settings.
2. Basic healing. This is done for all selected objects. Basic healing consists of fixing surface normals in the object and updating the orientation (to avoid having an object with negative volume).
3. Advanced healing. This is auto-heal. This is invoked on objects that require healing, that is, bodies that have errors, including have non-manifold errors.
4. Feature Removal. If you choose in the **Healing Options** to remove small holes, chamfers, blends, small edges, small faces and/or sliver faces, the actions are performed on all selected objects. There is no guarantee that small feature removal will be successful. (Also see Specifying the Model Resolution for defeaturing through the Auto Simplify and Model Resolution settings there.)

The above actions are performed on the selected objects. If you choose objects for healing which have not been analyzed, analysis is performed to determine its state (that is, whether it has invalid entities, small entities, and so forth). Invalid objects have all the above steps performed. Advanced healing is not performed on objects that do not require it.

While working on analyzing complex bodies, it is sometimes useful to examine faces, edges and vertices. In particular it is useful to find the connected faces for a face or edge or vertex, connected edges for a face/edge/vertex and connected vertices for a face/edge/vertex. The additional selection modes are available under **Edit** > **Selection Mode**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Validation and Healing Options for Import File

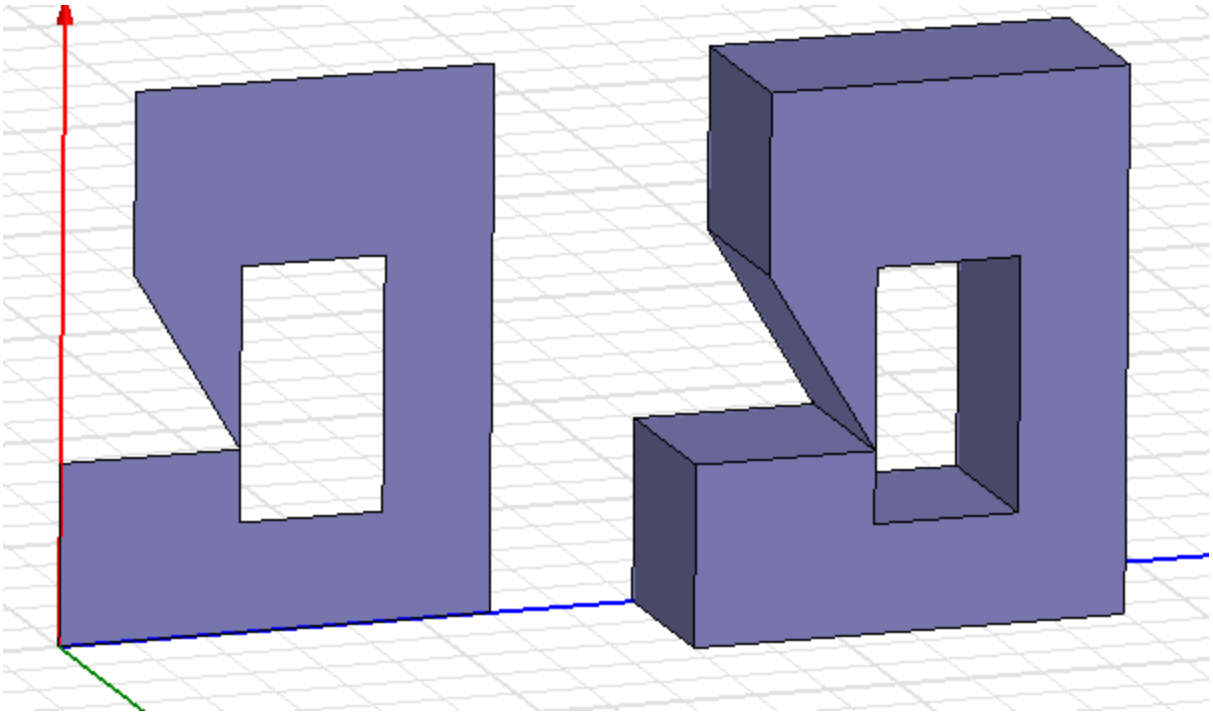
The **Import File** dialog box contains shows different Validation and Healing options for the seven formats listed here. The Quick option allows you to import these formats without healing.

Format	Options Available			
3D Modeler file (*.sm3),	Quick	Strict		
SAT file (*.sat),	Quick	Strict	Auto or manual	
STEP file (*.step,*.stp),	Quick	Strict	Auto or Manual	Stitch Tolerance
IGES file (*.iges, *.igs)	Quick	Strict	Auto or Manual	Stitch Tolerance
Creo Parametric files (*.prt, *.asm)	Quick	Strict	Auto or Manual	Import Free Surfaces. This imports such surfaces as well as parts.
CATIA (*.model, *.CATpart)	Quick	Strict	Auto or Manual	Stitch Tolerance
Parasolid file (*.x_t, and *.x_b)	Quick	Strict	Auto or Manual	

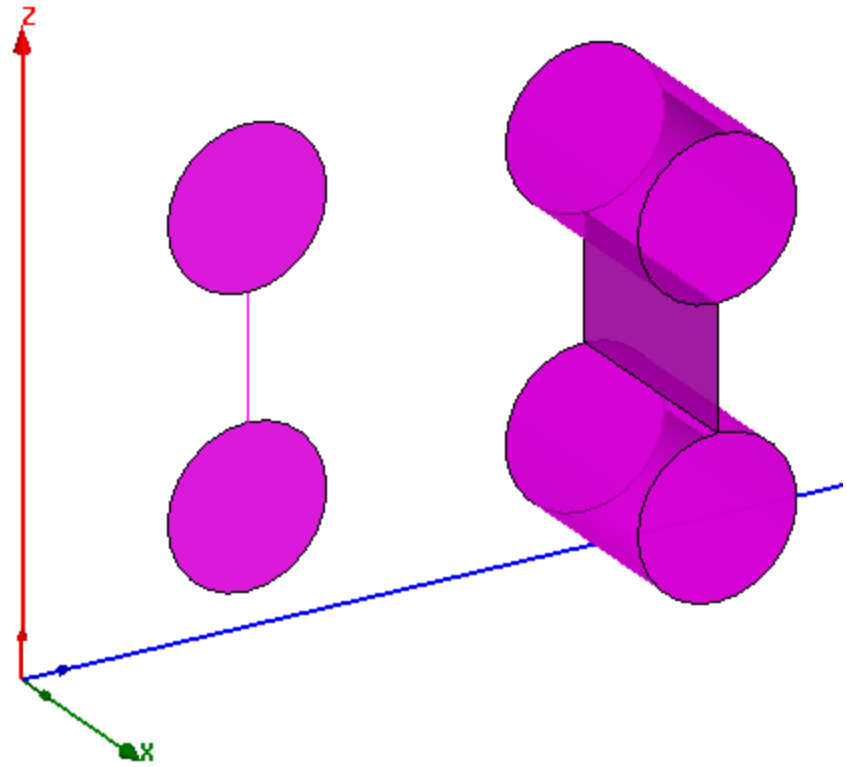
Healing Non-Manifold Objects

Non-manifold objects, in simple terms, are non-physical objects or objects that cannot be manufactured. For example, objects that intersect themselves (like the symbol for infinity in 2D) are clearly non-manifold. In addition objects that touch themselves may be non-manifold such as when a 2D object touches itself at a vertex, or a 3D object touches itself at a point or edge. These cases are shown in the following figure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Another type of non-manifold object has mixed dimensionality. For example, a pair of 2D objects connected by a 1D line segment, or a pair of 3D objects connected by a 2D sheet object. These cases are illustrated below.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The criteria for manufacturability is a simple manifestation of a complex mathematical concept that must be adhered to in the solid modeling system. When creating geometry, either directly, or through boolean operations, you should always consider whether or not the resulting operation will result in an object that could not be manufactured. If this is the case, then the object will cause an error in the modeler or in the meshing system.

To heal non-manifold objects:

1. Identify an edge that is non-manifold.
2. Select the connected faces.

   You can use the face selection toolbar icons.
3. Create a face coordinate system on the planar face.
4. Create a small box to cover the non-manifold edge.
5. Either do a union or a subtraction to remove the faces that contain the non-manifold edge.

   The non-manifold edge is now removed. You may also remove or add a small portion of the model.
6. Repeat for all non-manifold edges.

Setting the Healing Options

The **Healing Options** let you control how healing proceeds with respect to a variety of features and issues.

1. Click **Modeler** > **Model Healing** > **Heal** to open the **Healing Options** dialog box. You can also open the **Healing Options** dialog box from the **Model Analysis** dialog via the **Objects** tab drop down menu.

   The **Healing Options** dialog contains three tabs:

   - **Healing Options**
   - **Feature Removal Options**
   - **Properties**, which lists the geometric properties of the currently selected object.
2. Select the **Healing Options** tab to specify the following:
   - **Heal Type** as: **Auto Heal** (default), **Manual Heal**, or **No Heal**. Selecting **Manual Heal** enables the **Manual Heal Options**:
     - **Perform Tolerant Stitching** check box: This enables a field for the **Stitch Tolerance** value, and a check box to **Stop After First Error**.
     - **Perform Geometry Simplification**: This enables fields for **Simplification Tolerance** and **Maximum Generated Radius** values. You can also select radio buttons to **Simplify Curves**, **Surfaces**, or **Both**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Tighten Gaps** settings: A check box to select **Perform Tighten Gaps**. A field to specify **Tighten Gaps Within** a given value in mm.

3. Select the **Feature Removal Options** tab to specify the following:

   Feature Removal Options:

   - **Remove Holes** check box and **Maximum Radius** value.
   - **Remove Chamfers** check box and **Maximum Width** value.
   - **Remove Blends** check box and **Maximum Radius** value.

   Remove Small Entity Options:

   - **Small Edges, Length Less Than**, less than a specified value.
   - **Small Faces Area e Less Than**, less than a specified area.
   - **Sliver Face Width Less Than**, less than either:
   - **Object Bounding box Scale Factor**, less than a specified scale factor
   - **Sliver Edge Width**, less than a specified value.

   > **Note:** Sliver faces have a maximum distance among the long edges that is smaller than the specified tolerance and have at least one short edge and at most three long edges. A short edge has a length less than the specified tolerance. A long edge has a length greater than the specified tolerance. You can give the tolerance as a absolute value or a factor of the bounding box containing the face.

   Control Object Properties Change options:

   - **Allowable Change in Surface Area** check box, and percent value.
   - **Allowable Change in Volume** check box, and percent value.

4. Select the **Properties** tab to view the geometric properties of the currently selected object.

5. Click **OK** to apply the specified Healing options and to open the Analysis dialog box.

Stitch Sheets

Use the **Modeler** > **Model Preparation** > **Stitch Sheets** command to stitch selected sheets.

1. Select two or more sheet objects.

   This enables the **Stitch Sheets** command on the **Modeler** > **Model Preparation** submenu.

2. Click **Modeler** > **Model Preparation** > **Stitch Sheets**; or, on the **Model** ribbon, click the **Stitch Sheets** icon.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

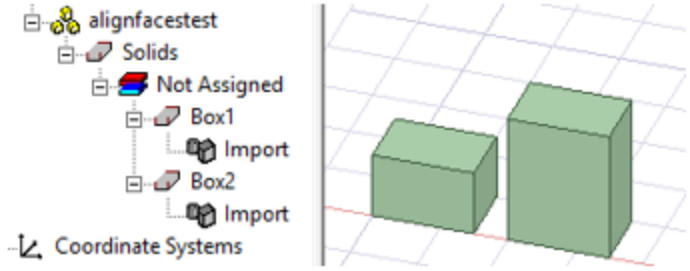
This displays a **Stitch** dialog box with a **Maximum Stitch Tolerance** field. The default value (auto) comes from the **Healing** dialog box's **Options** tab with Manual Healing selected. You may edit the value in the **Stitch** dialog box or in the **Healing Options**.

3. Click **OK**.

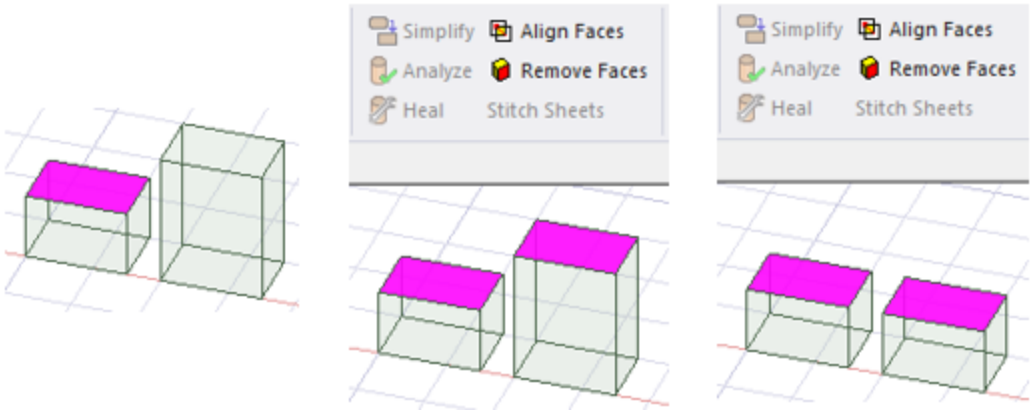
   This closes the dialog box and attempts to perform stitching on the selected sheets. If the sheets are separated beyond the stitch tolerance, stitching is not performed and a warning is issued.

Align Selected Faces

Use the **Modeler** > **Model Preparation** > **Align Faces** command to align the adjacent selected faces of imported objects that have only one operation in their History Tree. You can use **Undo/Redo** on this command.

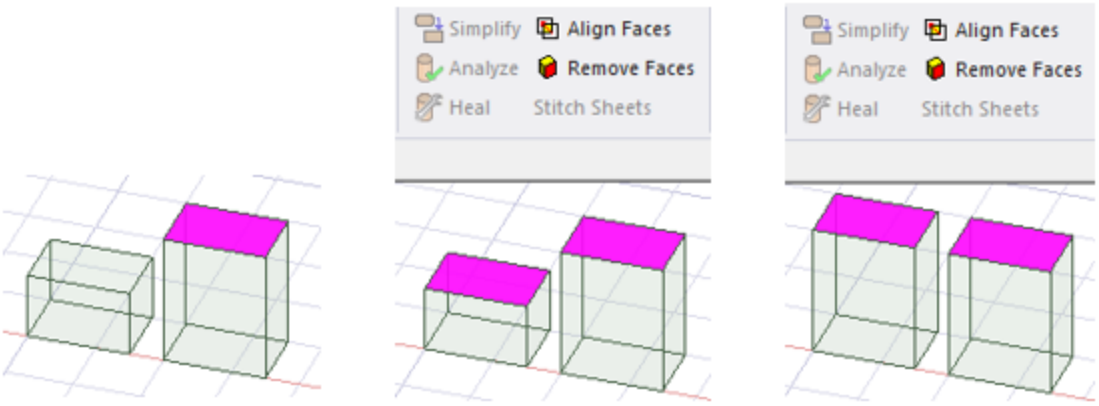


The **Model** ribbon also includes an **Align Faces** icon, which is enabled when you make appropriate face selection. The first selected face defines the plane of alignment for subsequent selection, that is, tool faces to blank faces.

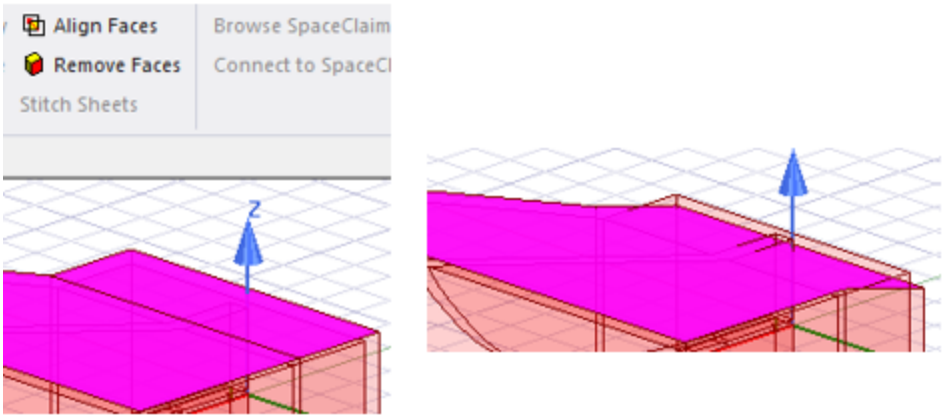


Selecting faces in a different order changes the alignment.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

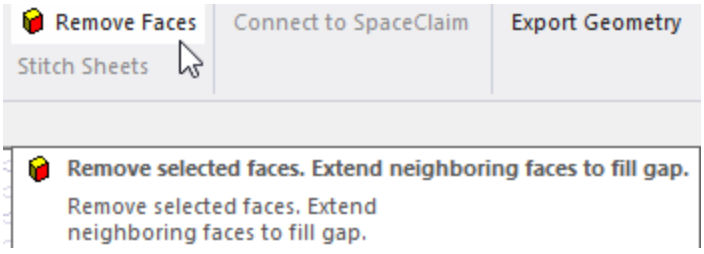


The selected faces can touch.



Remove Selected Faces

Use the **Modeler** > **Model Preparation** > **Remove Faces** command to remove the selected faces, extending the remaining faces to fill gaps. You can also use the **Model** ribbon **Remove Faces** icon when you have made an appropriate face selection.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you find object-pair intersections that healing does not fix, or that can be fixed (by alignment), you can correct the problem by one of the following methods.

1. Use the **Remove Faces** command (**Modeler** > **Model Preparation** > **Remove Faces**) or by performing Boolean subtract.
2. If overlap between objects is too large to be fixed by healing or by face alignment. Boolean intersect shows the common portion between the bodies. In this case, use a subtract operation to remove overlaps.

Remove Selected Edges

Use **Modeler** > **Model Preparation** > **Remove Edges** to remove selected edges. You can also use the toolbar icon when you have made an appropriate edge selection.

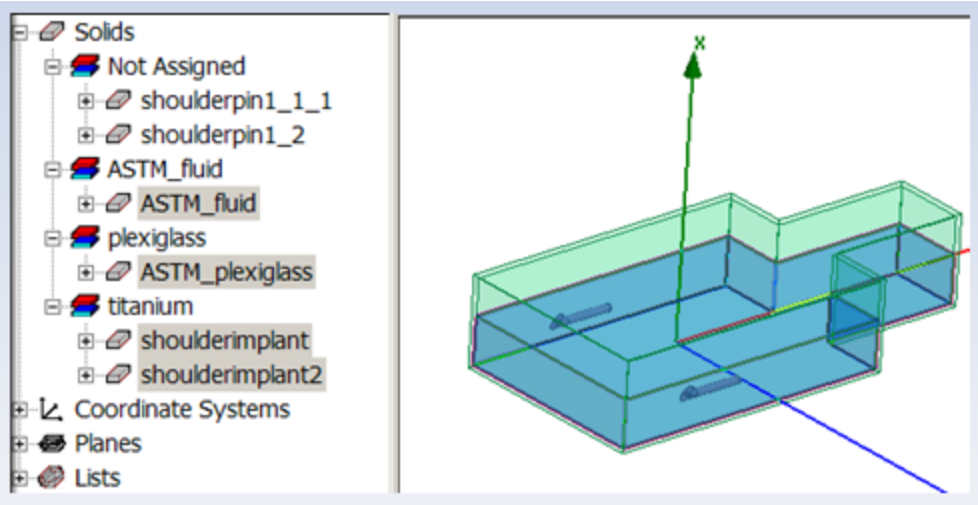
Perform Explicit Subtractions

Use **Modeler** > **Model Preparation** > **Perform Explicit Subtractions** to subtract selected History Tree objects that are fully contained by other parts.

1. Select two or more history objects.

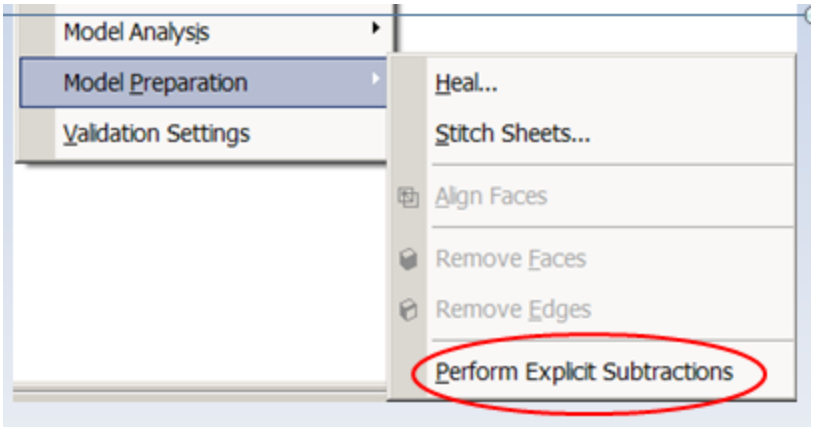
   This enables the **Modeler** > **Model Preparation** > **Perform Explicit Subtractions** submenu.

   For example, consider the following model.

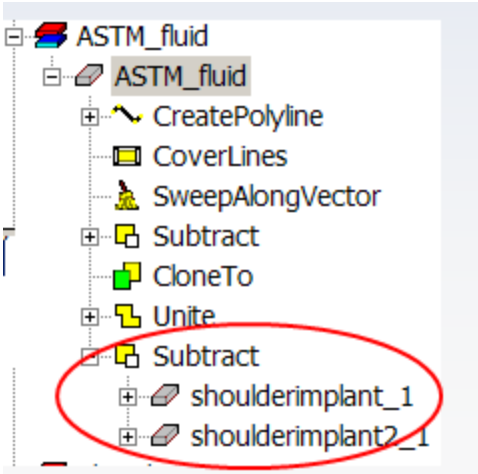


2. **Perform Explicit Subtraction** can be performed.

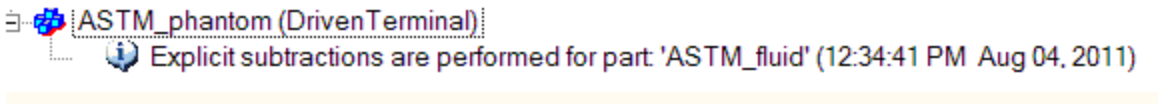
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The results appear in the History tree as shown:



The Message window reports this successful action.



If no parts are fully contained by other parts, a message explains that no explicit subtraction is needed.

Creating a User-Defined Primitive

The modeler allows you to generate user-defined primitives, primitives customized to suit any application. User-defined primitives are accessed using DLLs or IronPython scripts that you

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

build and compile. When user-defined primitives exist in your UserLib or PersonalLib directory (given the paths specified in **Tools** > **General Options**, Project Options tab), they appear in the menu bar under **Draw** > **User Defined Primitives**. Newly created UDPs will appear after a restart or after executing the command **Draw** > **User Defined Primitive** > **Update Menu.**

For Python-based primitives, see the **HFSS Scripting Guide**.

The modeler includes example C++ source and header files that can be used to generate DLLs. The files are located in the **UserDefinedPrimitives/Examples** subdirectory under the directory.

As an example, create the primitive **myUDP.dll** using Microsoft Visual C++ Developer Studio:

1. Create a directory to store all of the workspace information, call it UDPDir.
2. Use the sample workspace **RectangularSpiral.dsw** as a template:
    a. Copy **RectangularSpiral.dsw** and **RectangularSpiral.dsp** from the **UserDefinedPrimitives/Examples** directory to this new directory.
    b. Make sure the new files have write permissions.
    c. Rename the files to **myUDP.dsw** and **myUDP.dsp** respectively.
    d. Open the **.dsw** and **.dsp** files in a text editor, and replace every occurrence of **RectangularSpiral** with **myDLL**.
    e. Save **myUDP.dsp** and **myUDP.dsw**.
3. In the **UDPDir** directory, create a **Headers** subdirectory.
4. Copy the **UserDefinedPrimitiveStructures.h** and **UserDefinedPrimitiveDLLInclude.h** files from the **UserDefinedPrimitives/Headers** directory.

> **Note:**
>
> The header files include information on the methods that are available for use in your source code. They must be included when you compile the DLL.

5. In the **UDPDir** directory, create a **Source** subdirectory.
6. Use the sample source file **RectangularSpiral.cpp** as a template:
    a. Copy **RectangularSpiral.cpp** from the **UserDefinedPrimitives/Examples** directory to this new directory.
    b. Make sure the new file has write permission.
    c. Rename the file to **myUDP.cpp**.

    The resulting directory structure will resemble the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
UDPDir/
        myUDP.dsw
        myUDP.dsp
        Headers/
          UserDefinedPrimitiveDLLInclude.h
          UserDefinedPrimitiveStructures.h
        Sources/
          myUDP.cpp
```

7. Open **myUDP.dsw** using Microsoft Visual C++ Developer Studio, and edit the source code to create your desired primitive. You may also add additional headers and source files as appropriate.

   The UDP dll contains a data structure called UDPPrimitiveTypeInfo. This contains information about the udp, its purpose, company/author who created it, date created and the version number. When you select a primitive from your library, you see the **Create Primitive** dialog with a **Parameters** tab for setting the parameters, and an **Info** tab with the information from this data structure.

8. Build **myUDP.dll** using the **Win32 Release** configuration.
9. Copy the resulting file **myUDP.dll** to the **[*ProductInstallationPath*]/userlib/UserDefinedPrimitives** directory or the **[*PersonalLibPath*]personallib/UserDefinedPrimitives** directory.
10. To view your primitives, click **Draw** > **User Defined Primitive** > **Update Menu** and then click **Draw** > **User Defined Primitive** > **[*PrimitiveToDraw*]**.

> **Note:**
>
> On Linux, you may use the same example directory structure, source, and header files to build and compile a shared library using C++. The resulting shared library will have a **.so** extension and needs to be placed in the same */userlib/UserDefinedPrimitives* directory.
>
> As with the Windows DLL, the compiled library will work only on the operating system on which it was built.

User Customization through User Defined Primitives (UDPs)

User Defined Primitives (UDPs) allow users to add customized geometric modeling commands to the Ansys Electronics Desktop. UDPs are C++ compiled or Python libraries that can be added to the desktop interface and shared between users with common modeling needs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

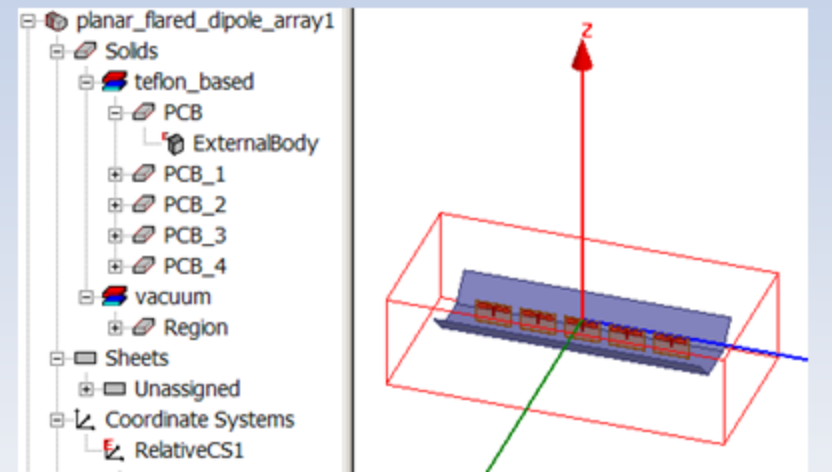
To create a UDP, see Creating a User Defined PrimitiveCreating a User Defined Primitive for requirements and the procedure for building a proper DLL or Python library.

In order to share UDPs between users, an existing DLL or Python library may be copied into the **UserLib** > **User Defined Primitives** subdirectory which can be given the paths specified in **Tools** > **General Options** under *General* > *Directories*. Placing an appropriately constructed DLL or Python library in this subdirectory and executing **Draw** > **User Defined Primitives** > **Update Menu** adds a new menu item in the **Draw** > **User Defined Primitives** menu to allow access to the UDP.

User Defined Model (UDM)

A User Defined Model (UDM) is collection of externally defined parts imported into an Ansys EM 3D Modeler or created using a C or Python script.

- UDM includes part attributes (like name, color etc) and material assignment
- UDM can also have external coordinate systems and corresponding planes
- UDM parts can be parameterized and manipulated in Ansys EM modeler just like any other part



UDM can either represent Static geometry models or

- Dynamic links to models of external geometry editors
- Used for supporting CAD integration in WorkBench

UDM uses same plugin technology as User Defined Part (UDP)

See the following sections:

UDM compared to User Defined Primitives

Insert UDM Command on Draw Menu

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDM Properties

Library of Models for CAD Integration

Ansys EM to Ansys Geometry Transfer

Material Assignment Transfer

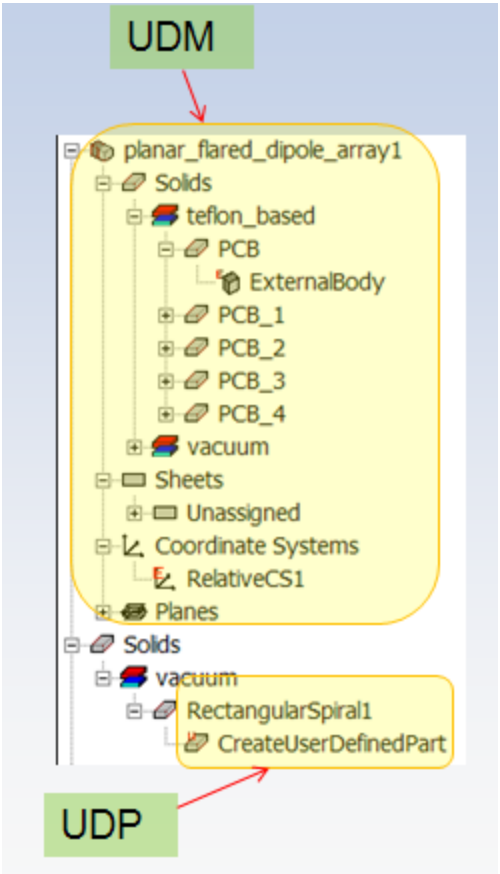
Geometry Transfer through Ansys DesignModeler (DM)

CAD Integration Functionality

See the **HFSS Scripting Guide** for information about Creating User Defined Primitives and User Defined Models in Python Scripts.

UDM Compared to User Defined Primitives

User Defined Models (UDM) resemble User Defined Primitives (UDP):



- Ansys EM products can be extended by users through new UDMs
- UDM plugins are discovered by searching standard directory paths

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Plugins for static UDM can build model using 'callback interfaces' (like create-box, create-cylinder, subtract etc) similar to UDP.
- UDMs run inside Ansys EM applications
- UDMs provide geometry, topology, persistence and parameters

  In contrast to UDP:

- UDM provides multiple Parts/CS/etc.

  UDP provides primitive operation only for a single Part

- UDM provides part attributes and material assignment

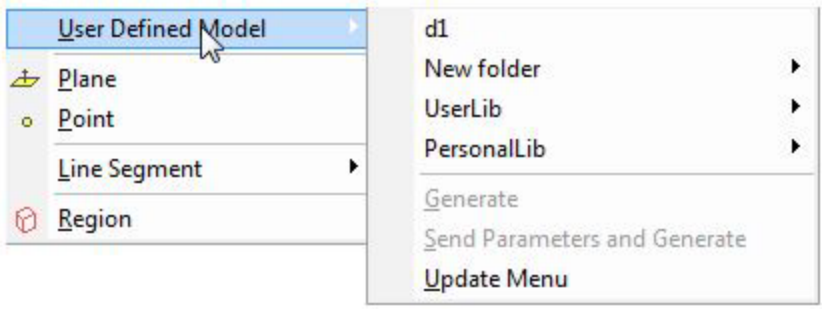
  UDP does not define part attributes or material

  UDM Properties have four tabs - Definitions, Parameters, Options and Info

- Definition tab has:
  - UDM name
  - Coordinate system used to position UDM
  - May have external reference to file
- Info tab has:
  - UDM dll or .py name, dll or .py location, version etc
- Option tab:
  - may have options if any

Insert UDM Command on Draw Menu

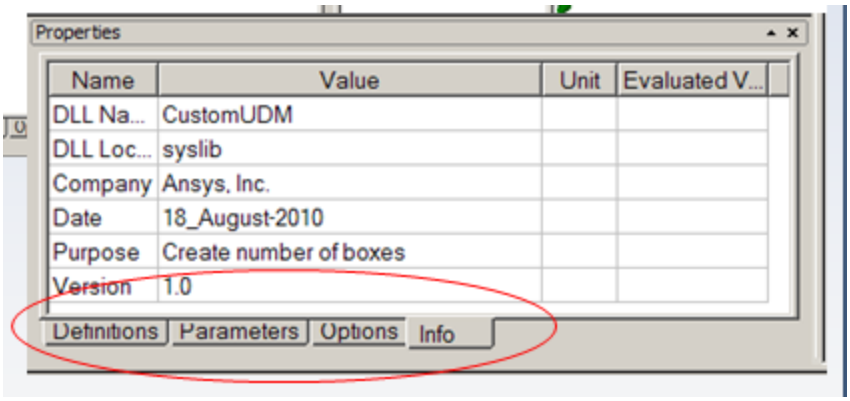
To insert a UDM into a design, use the User Defined Model command on the Draw menu for the Modeler window.



UDM Properties

UDM Properties have four tabs - Definitions, Parameters, Options and Info.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Definition tab has

- UDM name
- Coordinate system used to position UDM
- May have external reference to file

Info tab has:

- UDM dll or .py name, dll or .py location, version etc

Option tab:

- may have options if any

UDM Parameters

UDM geometry can be manipulated through its parameters.

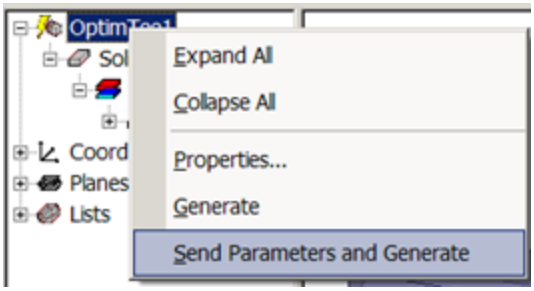
- Can be mapped to design or project variable for animation, parametric analysis.
- IDs are persisted (allowing to retain boundaries) during parameter edits.

  UDM geometry in HFSS invoked through Workbench is not 'dynamic updated' upon parameter edits.

- In such cases, UDM shows a lightning bolt icon by the model name when parameters are edited.

- You must run the **Send Parameters and Generate** command to synchronize parameters with geometry

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



UDM Part Edits

Several modeling operations are allowed on UDM parts

- Operations will be part of history tree and retained during model refresh

The following operations are not allowed

- Non history tree operations like healing, defeature.
- Operations which use UDM parts as tool, such as sweep or boolean (but allowed when clone tool option is selected)

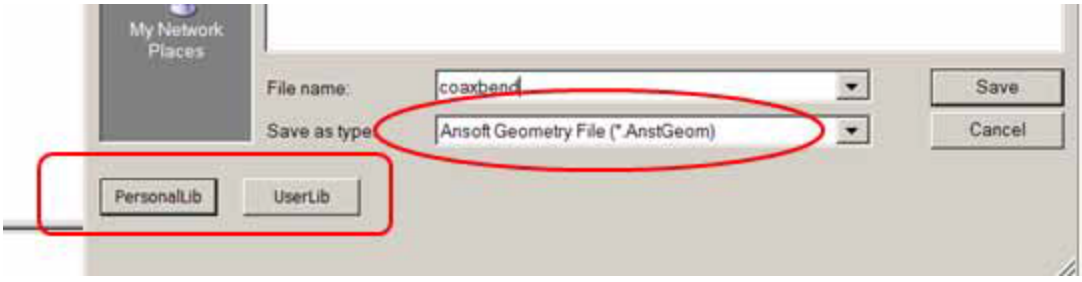
Following part attributes can be modified for UDM parts

- Model/Non Model flag
- Solve Inside flag
- Part orientation

Library of Models for CAD Integration

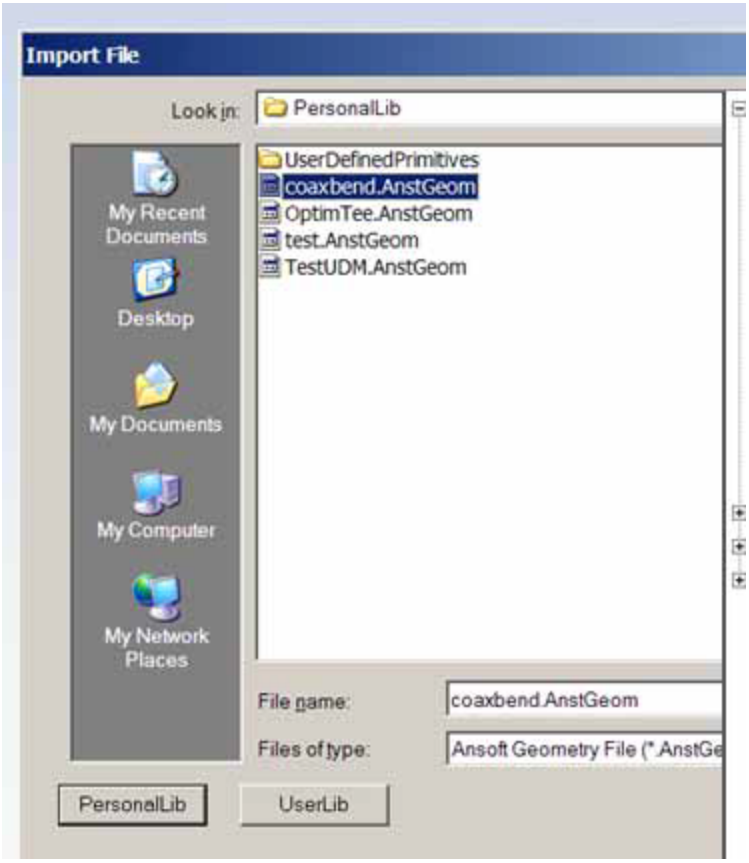
UDM technology allows a library of models

- Any Ansys EM model can be exported as 'Ansoft Geometry File'

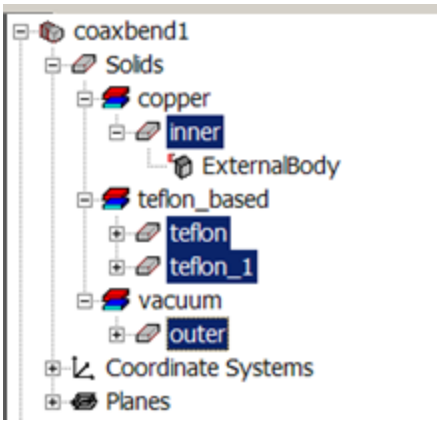


- An Ansoft Geometry File can be imported back as a UDM.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Any Design/project variables associated with model are brought in as UDM parameters.



Geometry computation for UDM (and also UDP) can be specified in the Modeler options as either done on.

- Engine side (default):

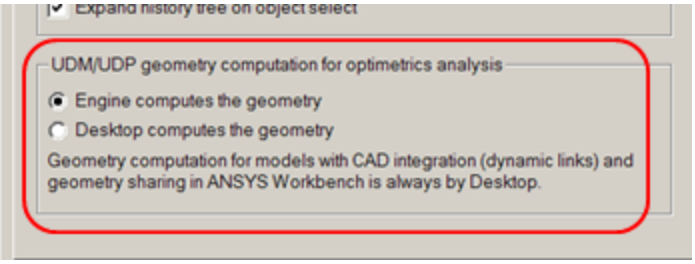
  Requires deployment of UDM on each node

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Desktop side:

  UDM need not be deployed on each engine

  Desktop will be busy during parametric analysis



3D Modeler Discovery Link

The 3D Modeler Discovery Link allows you to create a User-Defined Model (UDM) from the geometry model in an Ansys Discovery project. Discovery Link can be used only if Discovery with the same version as Ansys Electronics Desktop is installed on the same machine. Discovery Link is only available on Windows.

Discovery Link provides following functionality:

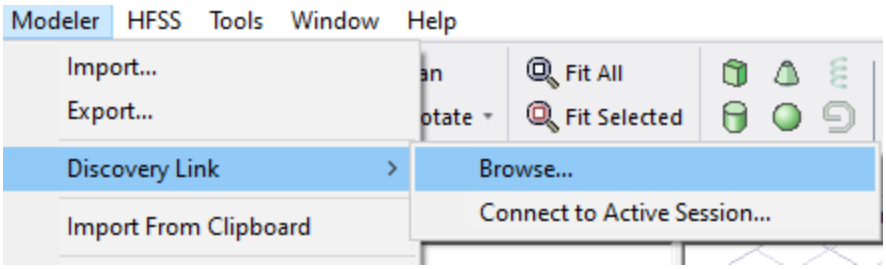
- Browse and select Discovery project files (*.dsco) to import geometry directly.
- Import geometry by connecting to an active Discovery session.
- The Discovery geometry is imported as a UDM along with geometry parameters with units, rendering attributes and material assignments. Object, material, and parameter names with non-ASCII characters are not supported.
- The Discovery geometries can be imported as faceted bodies if the design supports lightweight geometries.
- A Discovery document referenced by a Discovery link is included in the project archive automatically.
- You can break a Discovery link. In this case, all the parts in the UDM are converted to imported objects.

Importing Discovery Geometry into Ansys Electronics Desktop

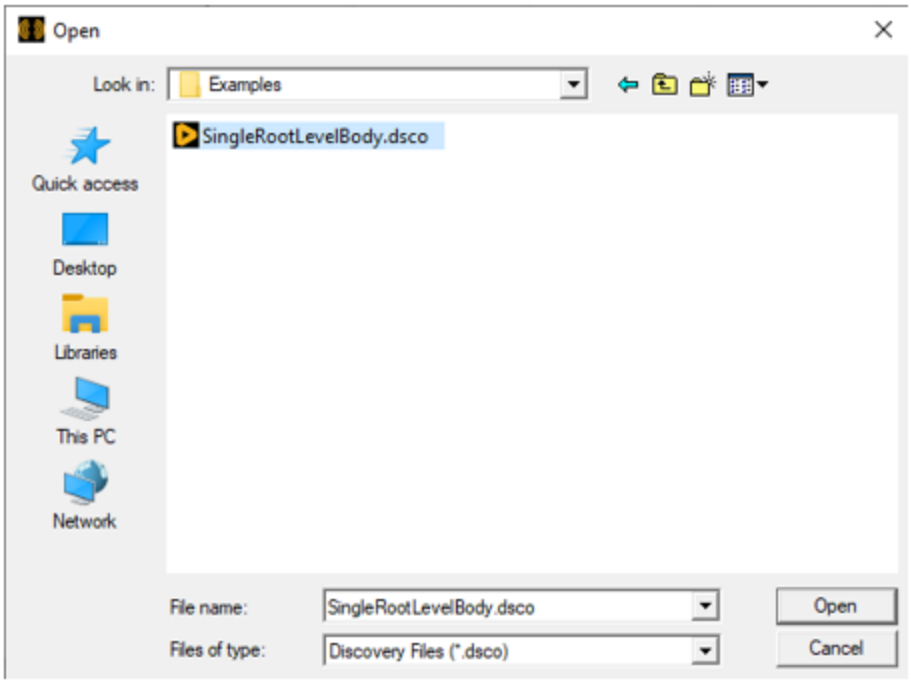
To select a Discovery file:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select **Modeler** > **Discovery Link** > **Browse** or **Browse Discovery** in the **Model** ribbon.



2. Navigate to and select a *.dsco project to load into the Modeler window.

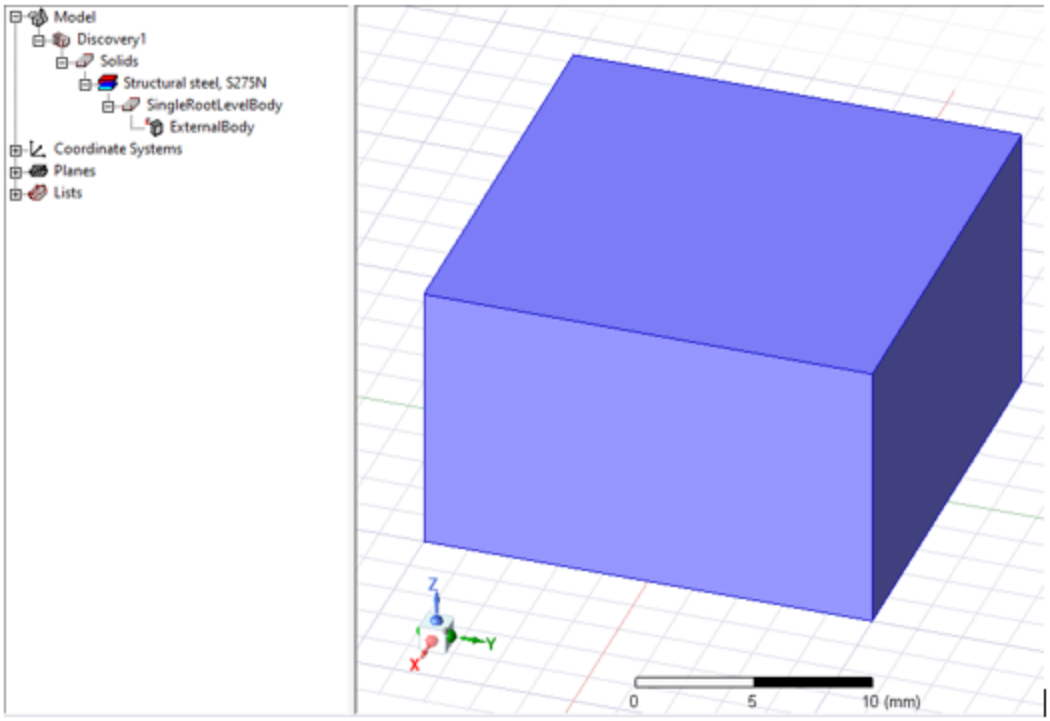


3. Click **Open**.

   The progress bar displays import progress. When the model has loaded, it appears in the Modeler window, and in the history tree as a User-Defined Model (UDM) listed under

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discovery.



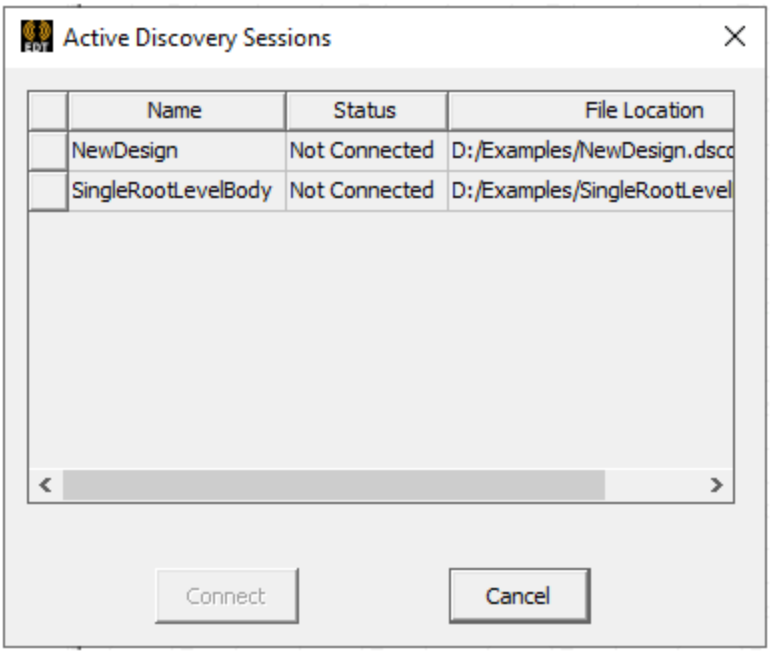
If there is an active Discovery session which is using the selected document, the created UDM will be linked to the active Discovery session. If there is not one, a new non-graphical Discovery session will be used, which makes the creating and refreshing slow. Afterwards, if an active Discovery session is launched and opens the document, the UDM will be linked to the active session.

Connecting to an Active Discovery Session

If one or more Discovery sessions are running on the machine, Discovery link can connect to them to import geometry.

When you select **Modeler** > **Discovery Link** > **Connect to Active Session** or **Connect to Discovery** in the Model ribbon, the Active Discovery Sessions dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



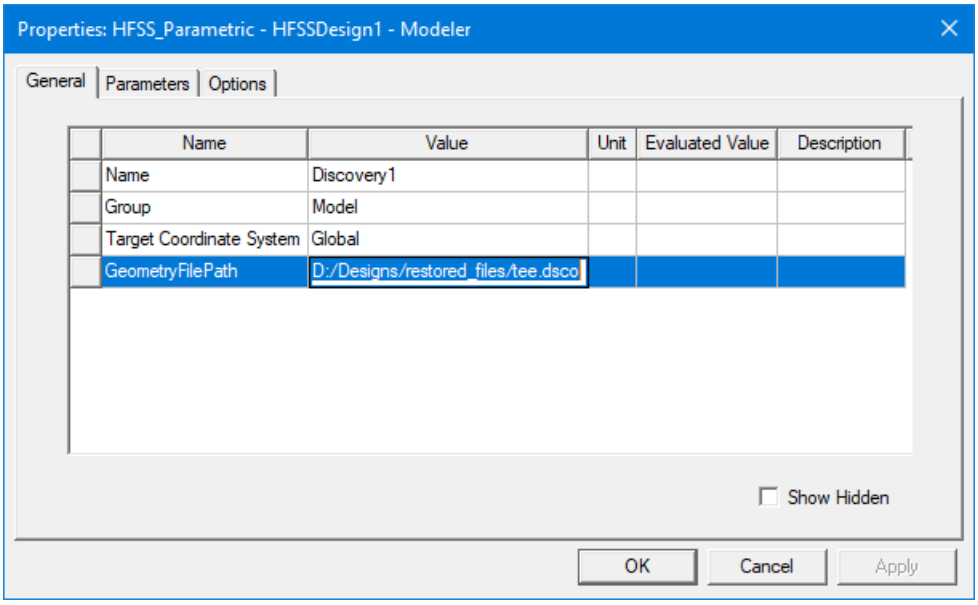
It shows all active Discovery sessions with their document names and connection status. Sessions which are not using a saved document are not included. You can select a session and click **Connect** to import geometry from that session. Once connected, that session will be used to regenerate the corresponding Discovery link UDM.

Property Window of Discovery Link UDM

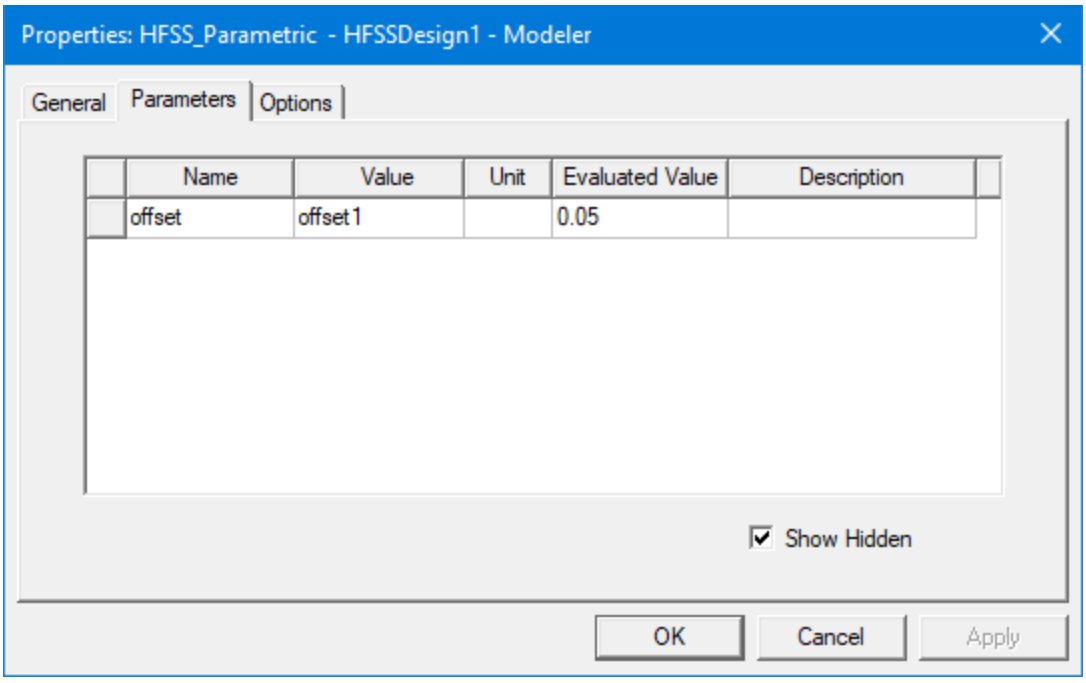
The Property window of a Discovery link UDM has three tabs:

- The **General** tab shows general information such as Discovery document path. Note that the Geometry File Path is editable; users can update the path if the original file is moved.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- The **Parameters** tab shows all parameters.

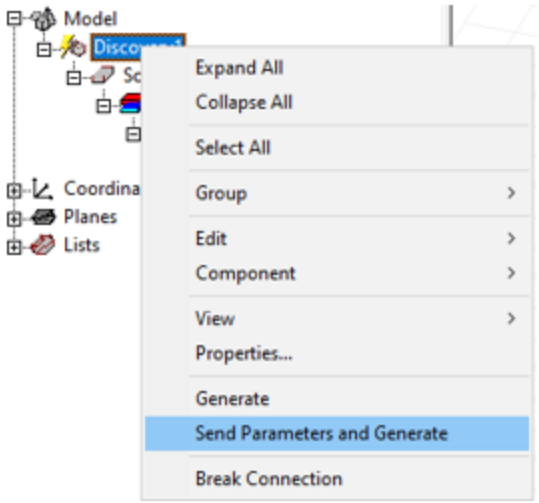


You can edit the parameters and send them back to Discovery to get the geometry updated accordingly.

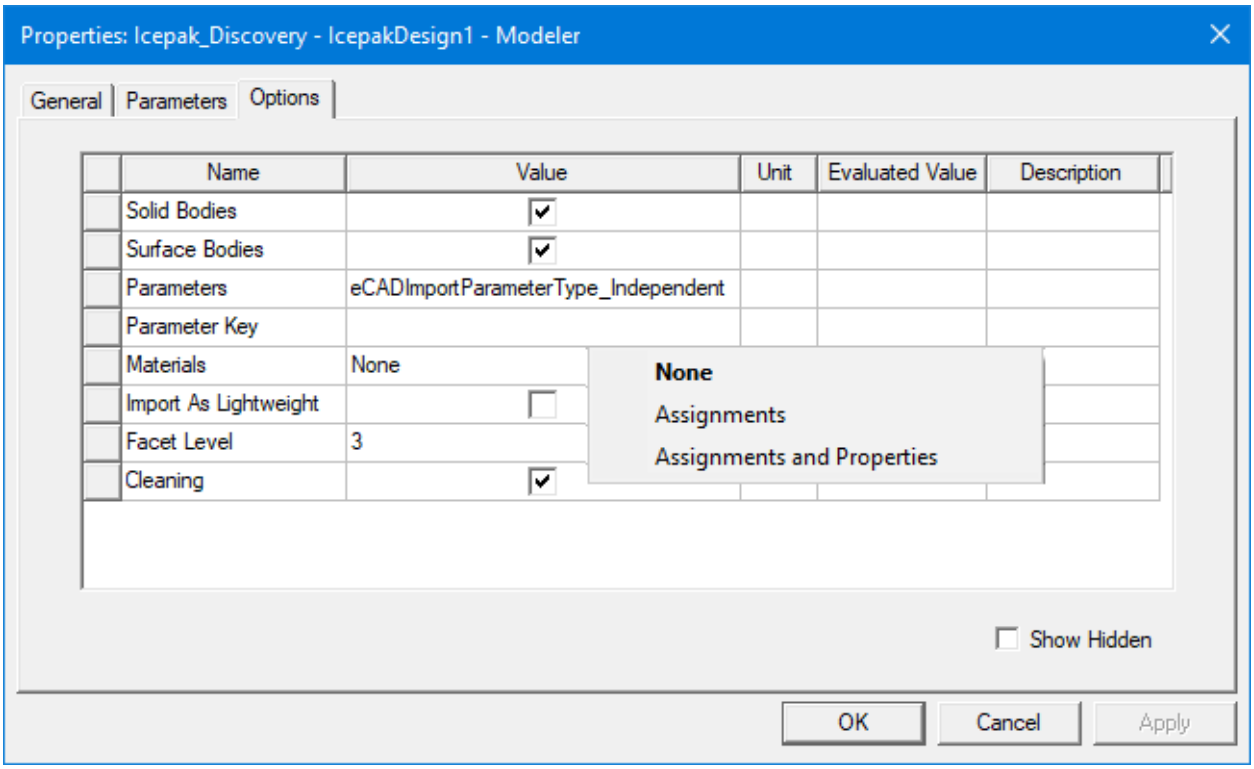
After editing the parameter, you will need to use **Send Parameters and Generate** in the context menu of the Discovery link UDM. If there is an active Discovery session linked, its

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

model will be updated but not saved. If there is not an active Discovery session linked, the Discovery document will not be updated.



- The **Options** tab shows supported options. Note that Materials**Assignments and Properties** is only available for Icepak and Mechanical designs. See Discovery Link Options for details on the options in this window.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Archive a Project Containing Discovery Link

If you want to share a project with Discovery link with other users, you need to archive the project, because the linked Discovery file is not included in the project file. When archiving, the Discovery documents corresponding to the links will be included in the project archive automatically. Once the archive file is unarchived, referenced Discovery documents will be put in restored_files subfolder in the project directory. All definition paths are updated to refer to the reference files in restored_files folder.

Break Connection with Discovery Link

A **Break Connection** command is available in the context menu of the Discovery link UDM.

- **Break Connection** converts the UDM bodies to imported objects. The UDM will be removed.
- Any operations that you have performed on UDM bodies are preserved.
- Parameters of the UDM are lost. You lose the ability to create parametric variations with Discovery parameters.
- Attributes such as rendering attributes, materials, etc., are preserved.

SpaceClaim Integration

As of 2025R1, the direct link between the Ansys Electronics Desktop software and the SpaceClaim application has been discontinued. Users are encouraged to convert SpaceClaim documents to the Discovery application file format and to use the Discovery link (**Modeler** > **Discovery Link**) to import geometry into AEDT.

Legacy AEDT projects containing SpaceClaim links can be opened. Any regeneration of SpaceClaim geometry will open the SpaceClaim file in background session of the Discovery application. We recommend that users convert source *.scdoc* files to *.disco* files by opening and saving them in the Discovery application. Users can then modify the source file path under the UDM link to point to the Discovery file.

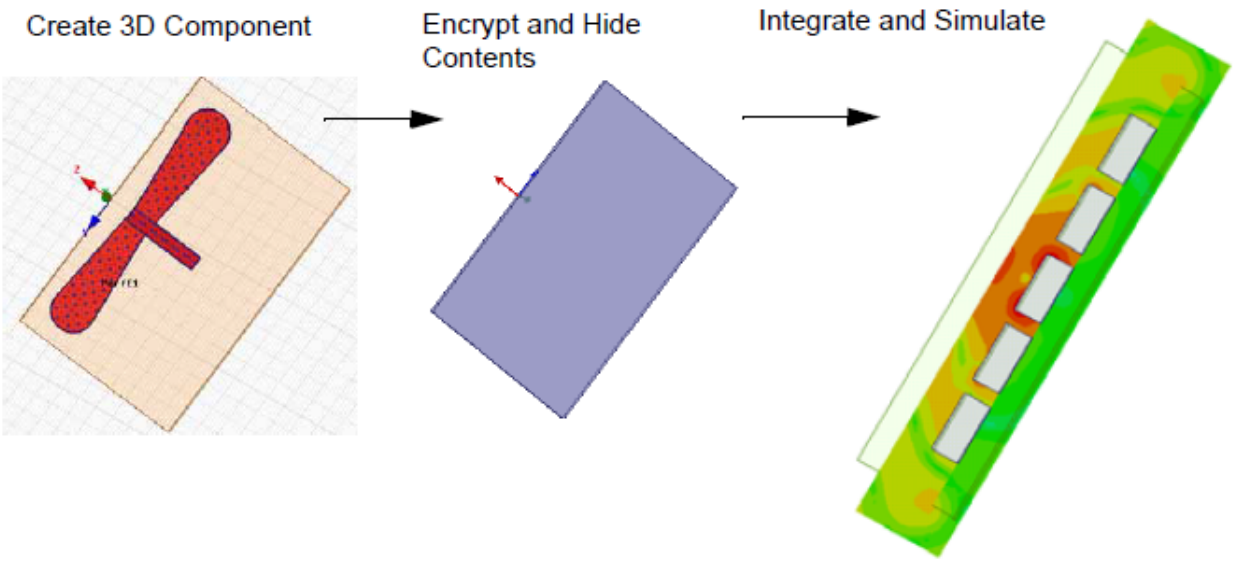
3D Component Library

The 3D Modeler lets you access predefined 3D component libraries and to create 3D components using:

- Geometry (objects and coordinate systems)
- Design data (boundaries, excitations and mesh operations)
- Material assignment and definition for components
- Component parameter, including variables used by geometries, design data, and material properties

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Optional logo or icon image to display with components in the modeler window
- Optional encryption with password definition or internal key as a password



3D Component Libraries may be included as part of Electronics Desktop, downloaded from third-party vendors, or created from scratch.

The ability to create components from models lets you easily share and reuse components. You can add components to a library and share components with other users. Once you insert a component into the target design, you can directly manipulate whole components using **Arrange** operations such as **Move**, **Rotate** and **Mirror** or **Duplicate** operations like **Duplicate Along Line**, **Duplicate Around Axis** and **Duplicate Mirror**.

The documentation includes the following sections:

Creating a 3D Component from an Existing Model

Replace Selection with 3D Component

Layout Component in HFSS 3D (Beta)

Inserting a Component into a Design

Using the Component Libraries Window

Viewing Properties of 3D ComponentsViewing

3D Components in the Project Manager Window

3D Components in the History Tree

Editing 3D Component Properties

Editing a 3D Component Definition

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Update 3D Component Definition

Component Priority

3D Component Agents

Mesh Assembly for Modal FE-BI and metallic IE Region Components

Additional **Show**, **Show Only**, **Hide** and **Fit** features, helpful in working with 3D components are described here:

Fit All Objects in a View Window.

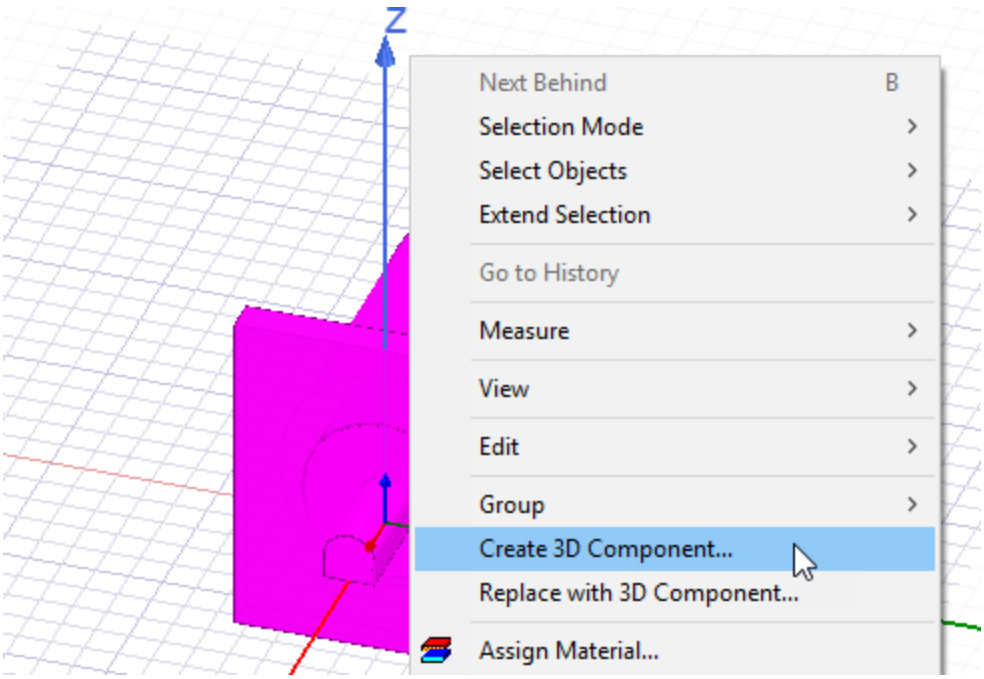
Fit Selected objects in a View Window.

Showing Only Selected Objects in All or Active Views

Hiding Objects from View

Creating a 3D Component from an Existing Model

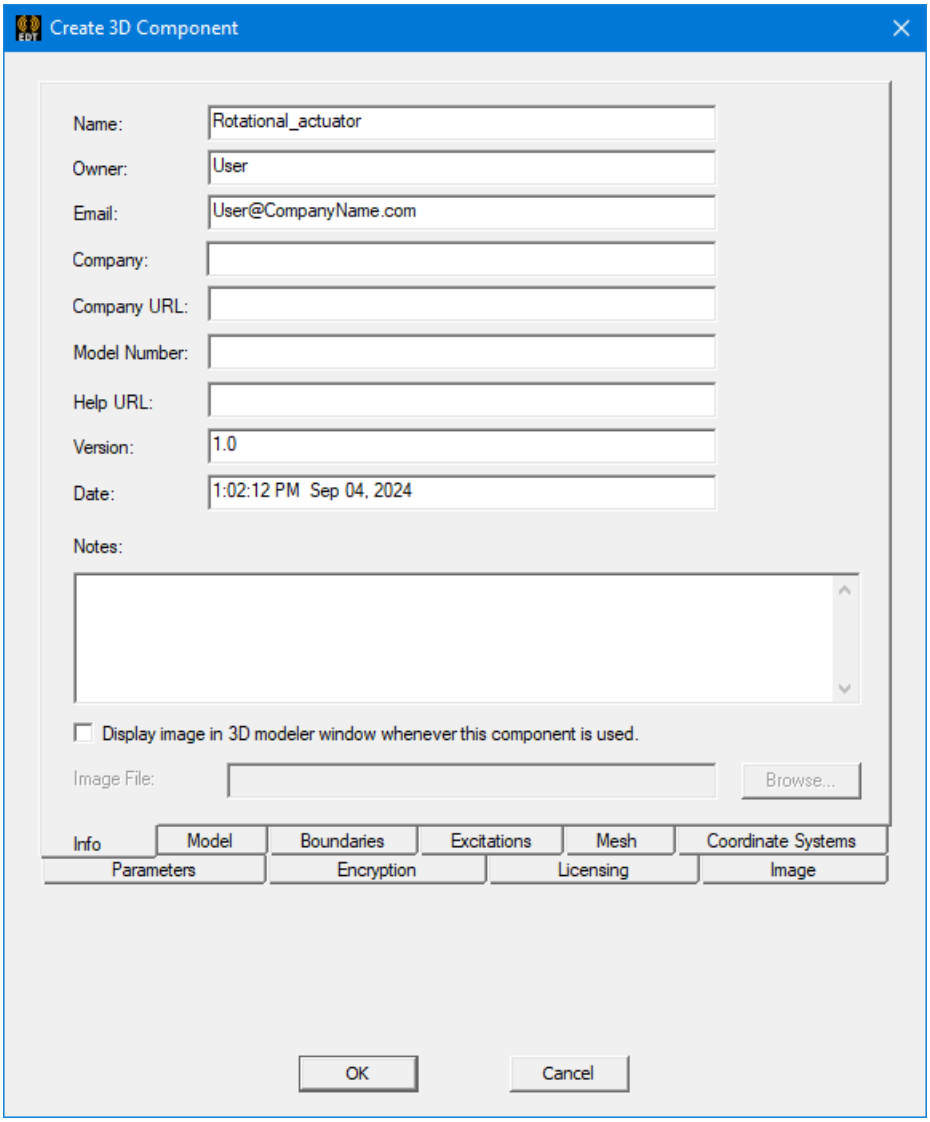
To create a 3D component, select an existing object or model, right-click, and select **Create 3D Component**. You can also click **Draw** > **3D Component Library** > **Create 3D Component**. You can also go to the **Model** tab of the ribbon and select **Create 3D Component**.



This opens the *Create 3D Component* dialog box.

You cannot create components from a design with an array setup. However, you can create 3D component from a unit cell and use that to create a Multi-Unit Cell array.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The tabs let you view the component features organized according to geometry, design data, and settings:

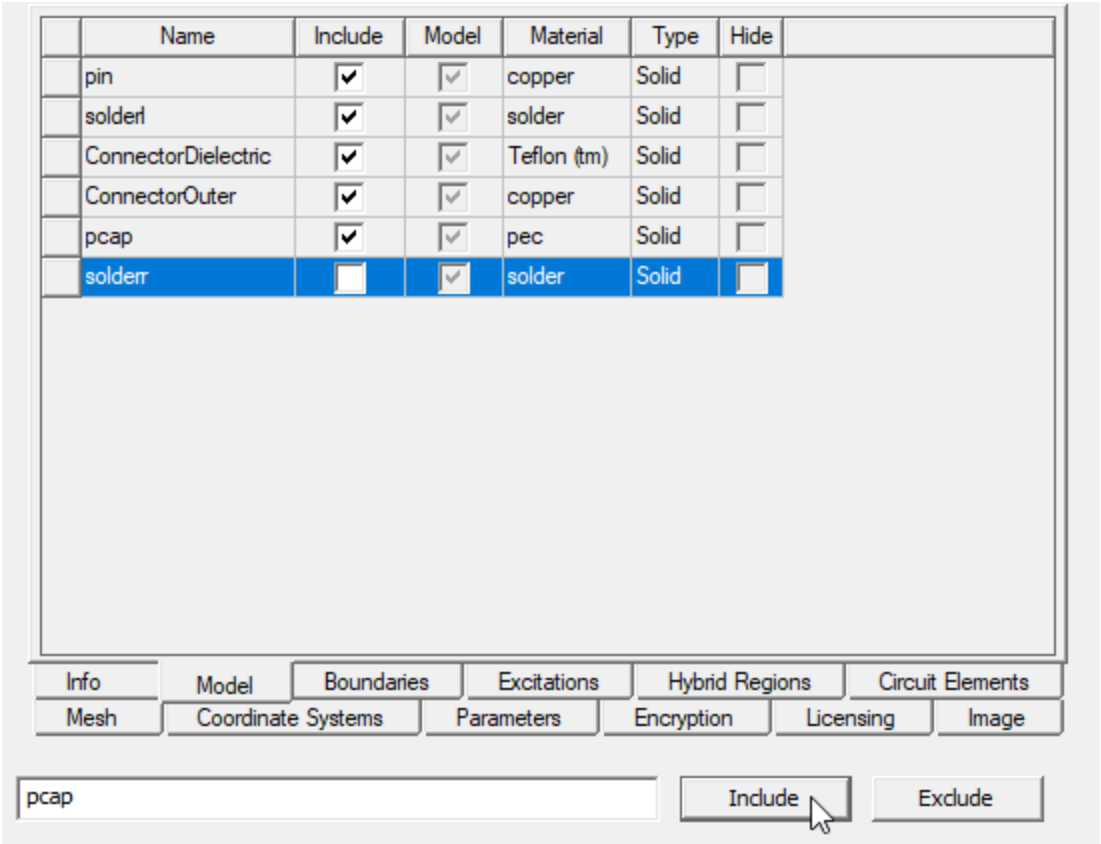
- Info tab
- Model tab
- Boundaries tab
- Excitations tab
- Circuit Elements tab
- Hybrid Regions tab
- Mesh tab
- Coordinate Systems tab

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Parameters tab
- Encryption tab
- Licensing Tab
- Image tab

Include or Exclude Feature from Component

You can view the various tabs and select which features to **Include** or **Exclude** in creating the component. You can use the check boxes in the Include column for this purpose. You can also use the text field. Type a feature Name which you can then use the command buttons to Include or Exclude.



Object selection for **Include** or **Exclude** impacts selection in other tabs. For example, if you exclude an object from the component creation, the design data, parameters, and coordinate systems corresponding to that object are also excluded.

Sorting Columns Using the Headers

The features listed for each tab include columns for the Name, check boxes for whether to include, and properties. You can click on the column headers to sort by Name, Model, or other

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

feature listed for each tab. Click the column header to selects the column to sort. Click again to invert the column.

You can sort lists by using all columns except Include.

Design Data Notes:

- DC thickness on selected objects is always included in components. It is not listed in **Boundaries** tab.
- The **Create 3D Component** dialog does not list design data without an assignment (for example, a far/near field incident wave in HFSS).
- The parent of any included boundary/excitation is included, as long as the parent does not require assignment.
- Design settings like material overrides are included.

Once you have made the Include and Exclude settings, and have specified the Image and Info you click **OK** to Save 3D Component File. When you click OK, the component is validated for everything that is included in the component. For example, you cannot include boundaries if the object on which the boundaries have been created is not included. After validation, you see a dialog for saving the component to a location.

Create 3D Component: Model Tab

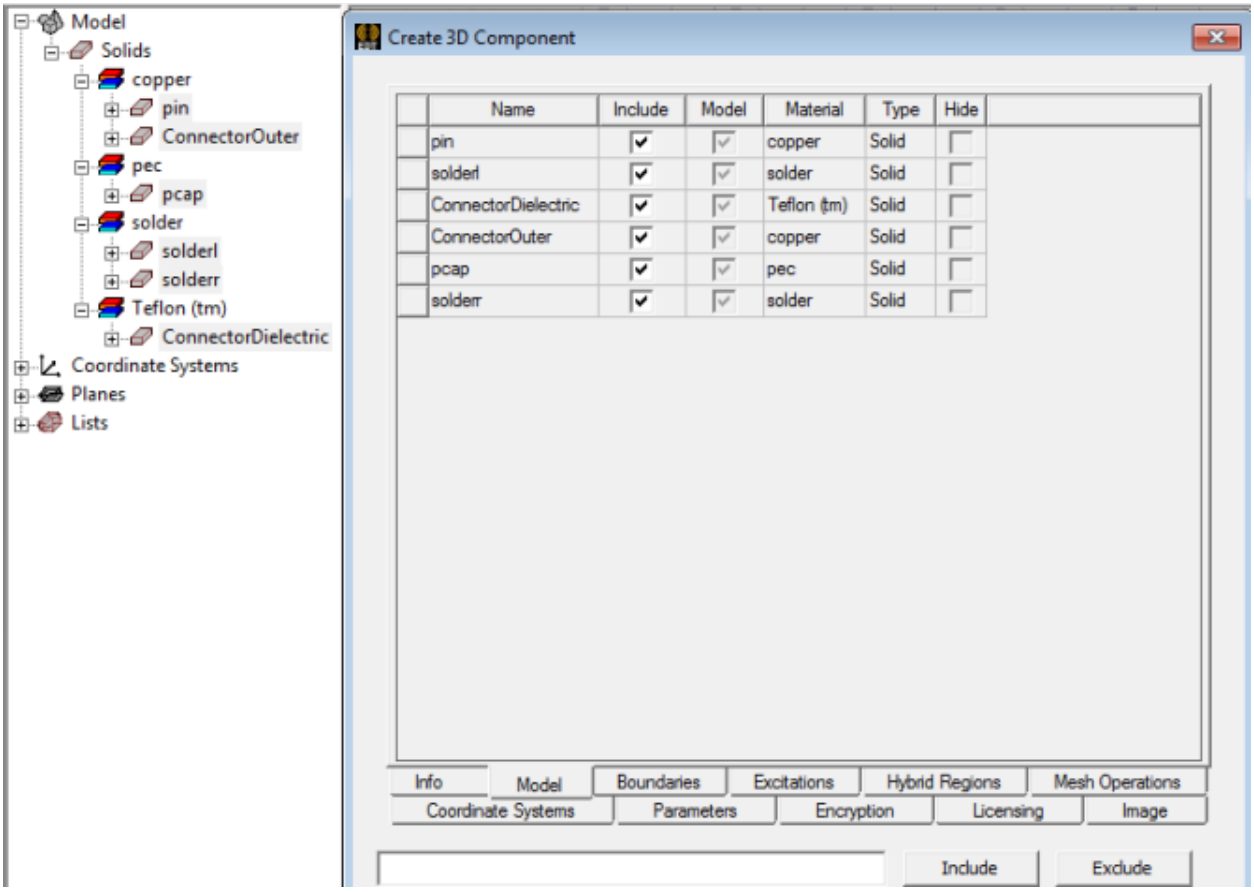
The **Model** tab lists all of objects selected before you invoked **Create 3D Component**. These correspond to the selected objects listed in the history tree.

- Use the **Include** check box to include objects in the 3D component (selected), or to Exclude them from the component (cleared).

  Alternatively, you can type an object **Name** into the text box and click the **Include** or **Exclude** button. Wildcards (? and *) are supported when you use this method, so you can change the inclusion state of multiple objects with similar names in a single operation.

- The **Material** and **Type** columns are read only.
- For Encrypted components, you can also chose which objects to hide.
- The **Hide** column is only enabled if you have the **Hide Selected Contents** option selected on **Encryption** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create 3D Component Boundaries Tab

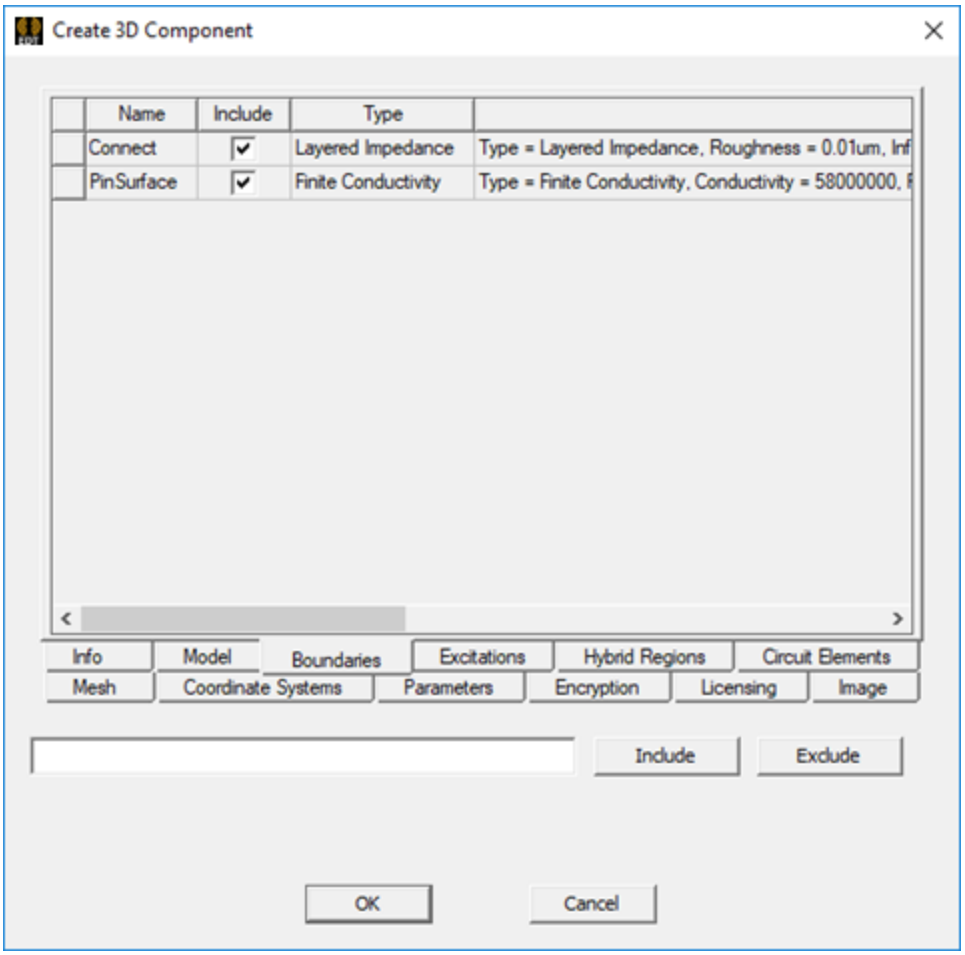
The list is populated with all the boundaries on selected objects.

- Use the **Include** check boxes to include (selected) or exclude (cleared) individual boundaries for the 3D component.

  Alternatively, you can type a boundary *Name* into the text box and click the **Include** or **Exclude** button. Wildcards (? and *) are supported when you use this method, so you can change the inclusion state of multiple boundaries with similar names in a single operation.

- The *Type* and *Description* columns are read only.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create 3D Component Excitations Tab

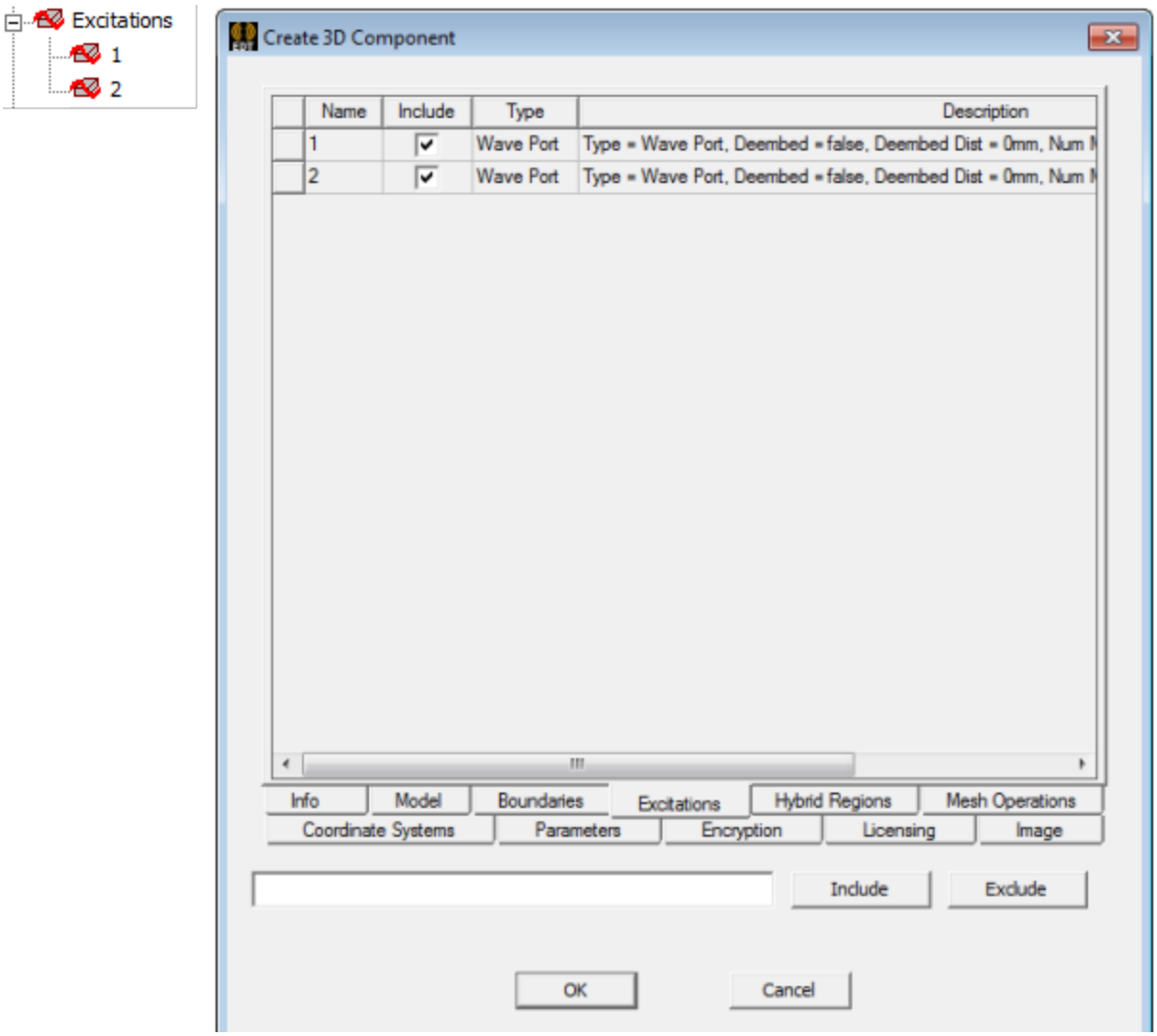
The list is populated with all the excitations on selected objects.

- Use the **Include** check box to include excitations in the 3D component (selected), or to Exclude them from the component (cleared).

  Alternatively, you can type an excitation *Name* into the text box and click the **Include** or **Exclude** button. Wildcards (? and *) are supported when you use this method, so you can change the inclusion state of multiple excitations with similar names in a single operation.

- *Type* and *Description* columns are read only.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

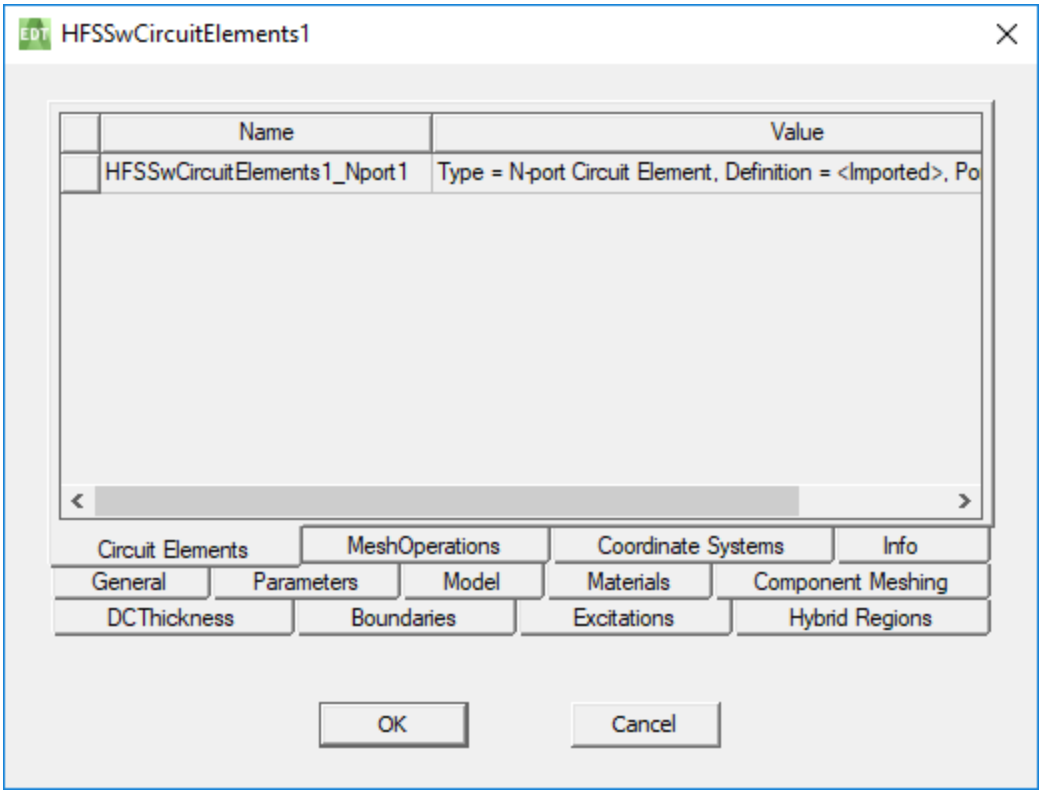


Create 3D Component Circuit Elements Tab

The list is populated with all the Circuit Elements on selected objects.

- **Name** and **Value** columns are read only.

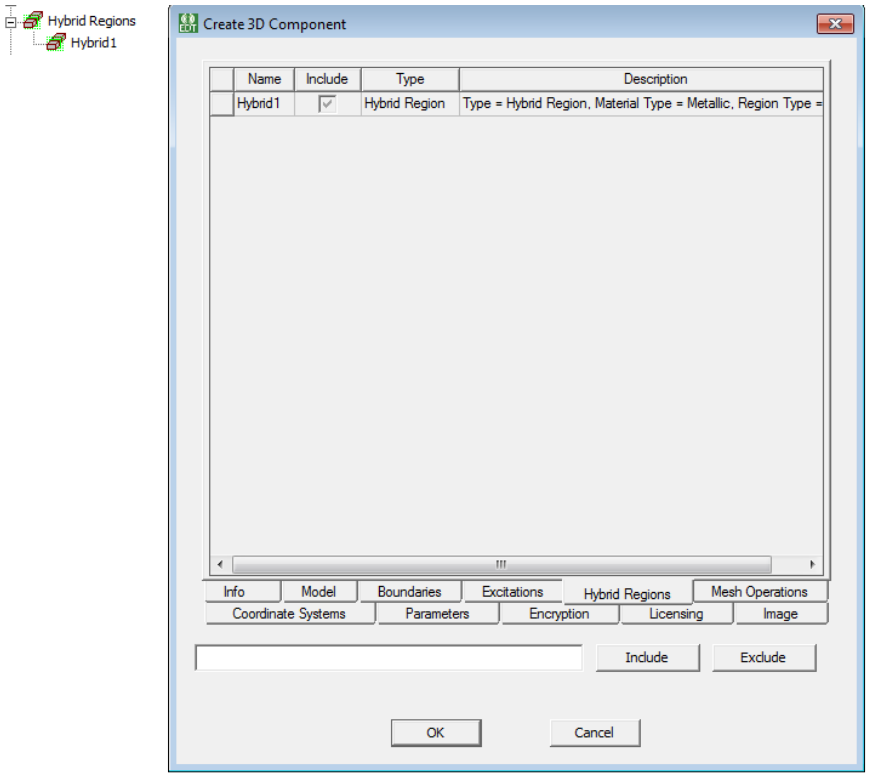
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create 3D Component Hybrid Regions Tab

If you have included Hybrid Regions for the model they are listed in the **Hybrid Regions** tab. You can use the Include check box to include them in the created model. However, if you create a 3D Component from an SBR+ Solution type design, it will not contain a **Hybrid Regions** tab and can only be used in an SBR+ Solution type design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create 3D Component Mesh Tab

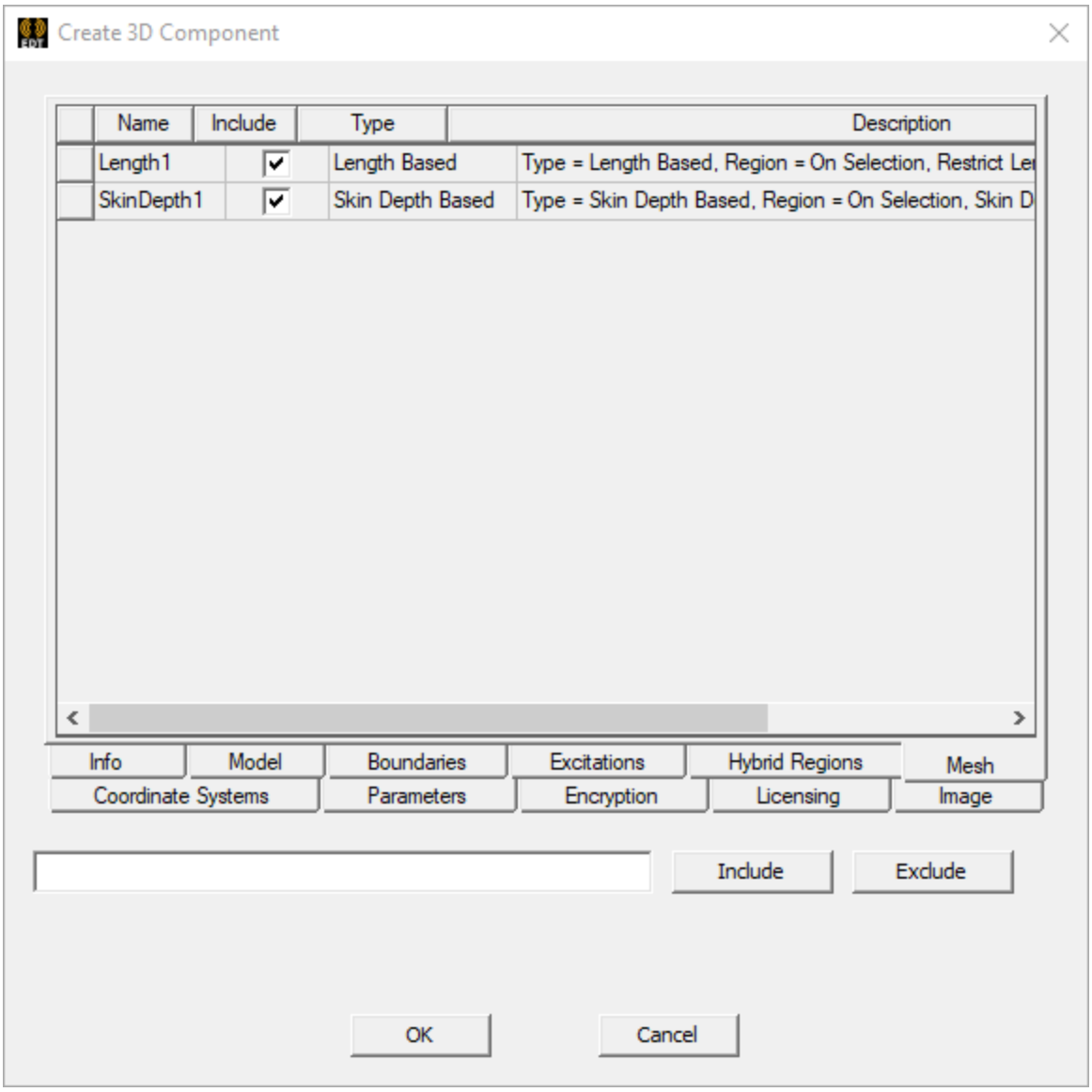
The list is populated with all the mesh operations on selected objects.

- Use the **Include** check box to include mesh operations in the 3D component (selected), or to Exclude them from the component (cleared).

  Alternatively, you can type a mesh operation *Name* into the text box and click the **Include** or **Exclude** button. Wildcards (? and *) are supported when you use this method, so you can change the inclusion state of multiple mesh operations with similar names in a single operation.

- *Type* and *Description* columns are read only.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create 3D Component Coordinate Systems Tab

The list is populated with all the Coordinate Systems on selected objects. By default, only the Coordinate Systems used to define the object orientations are included. You can include Coordinate Systems created on specific model parts.

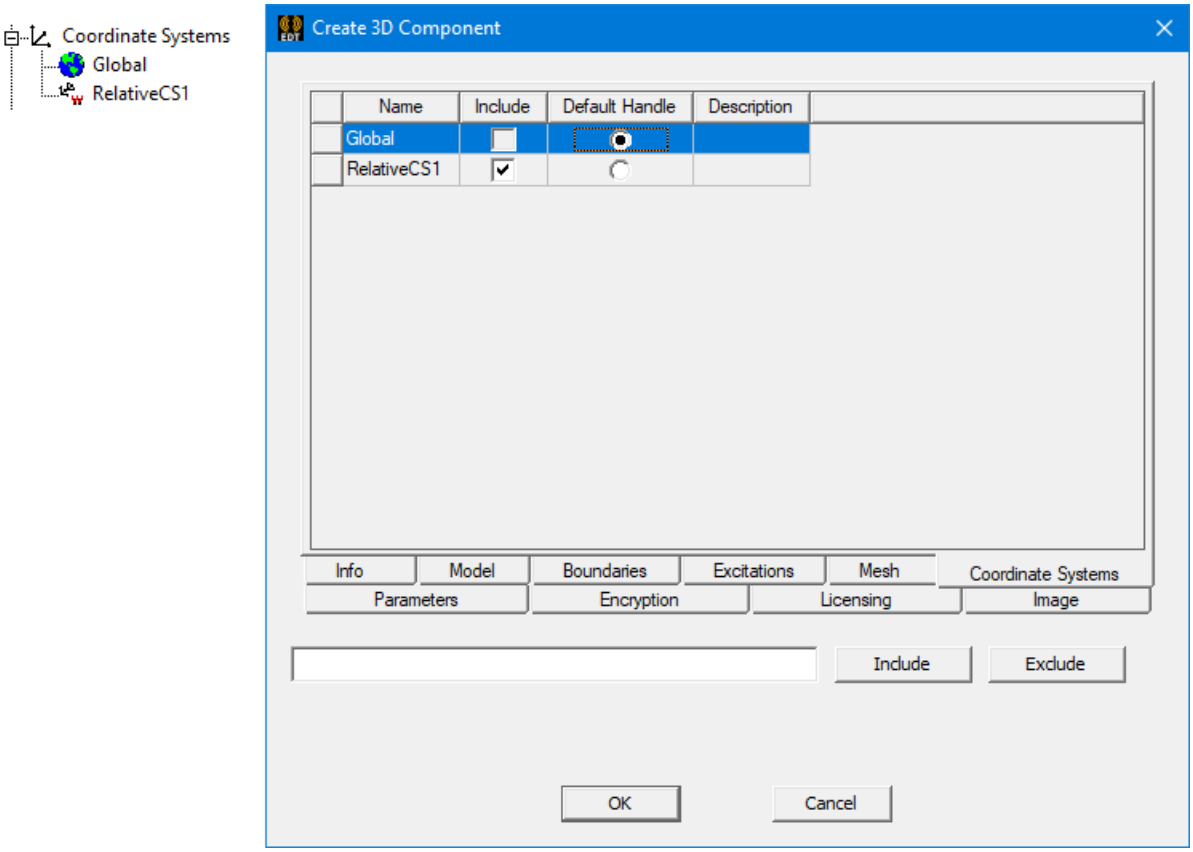
- Use the **Include** check box to include coordinate systems in the 3D component (selected), or to Exclude them from the component (cleared).

  Alternatively, you can type a coordinate system *Name* into the text box and click the **Include** or **Exclude** button. Wildcards (? and *) are supported when you use this method,

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

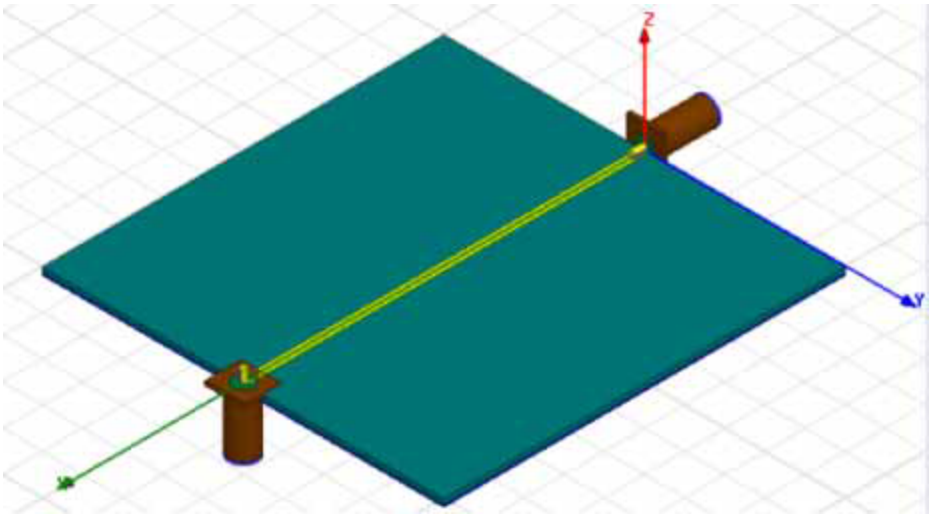
so you can change the inclusion state of multiple coordinate systems with similar names in a single operation.

- You can select any of the CS as the default handle. When this component is placed in assembly design, the handle CS is aligned to target CS in assembly design. Note that the handle CS can be changed after import; see Editing 3D Component Properties for more details.
- By default, the current working CS is designated as the default handle CS.
- The default handle CS must be included in the model (except for global).



The specified default handle affects the orientation of the component upon insertion:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create 3D Component Parameters Tab

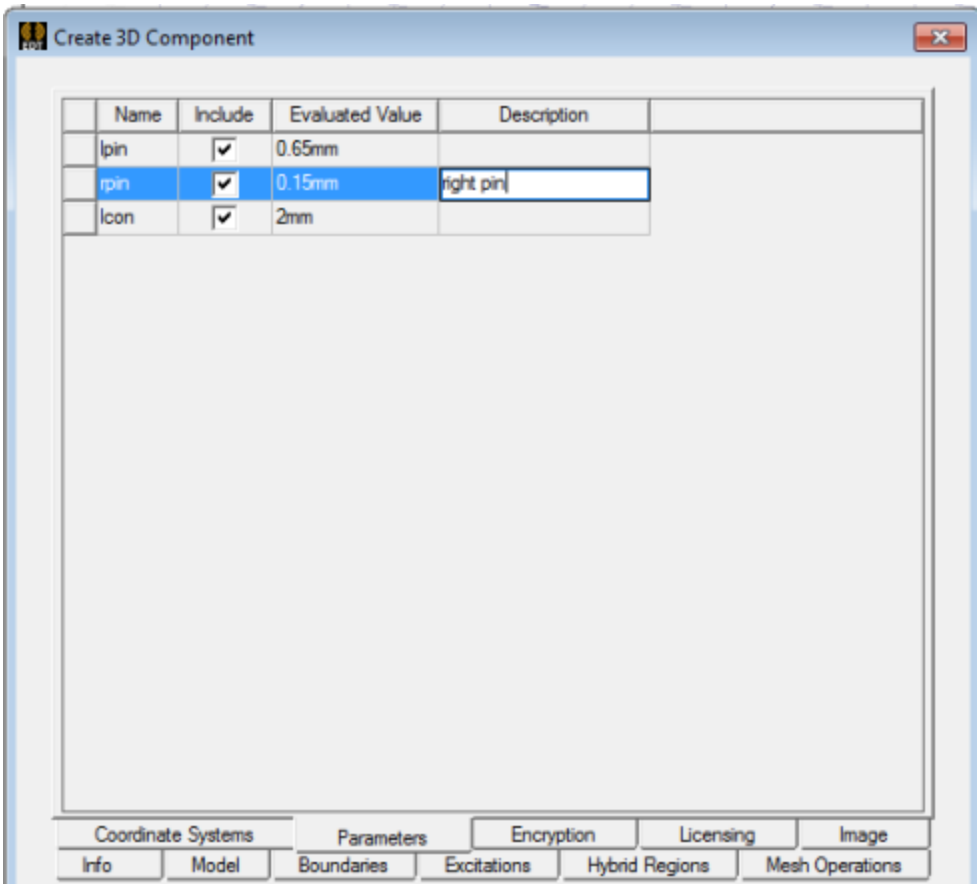
Component properties can be parameterized by assigning variables. The parameters list is populated with all the variables used by objects and coordinate systems, design data, and material properties included in the component.

- Use the **Include** check box to include parameters in the 3D component (selected), or to exclude them from the component (cleared).

  Alternatively, you can type a parameter **Name** into the text box and click the **Include** or **Exclude** button. Wildcards (? and *) are supported when you use this method, so you can change the inclusion state of multiple parameters with similar names in a single operation.

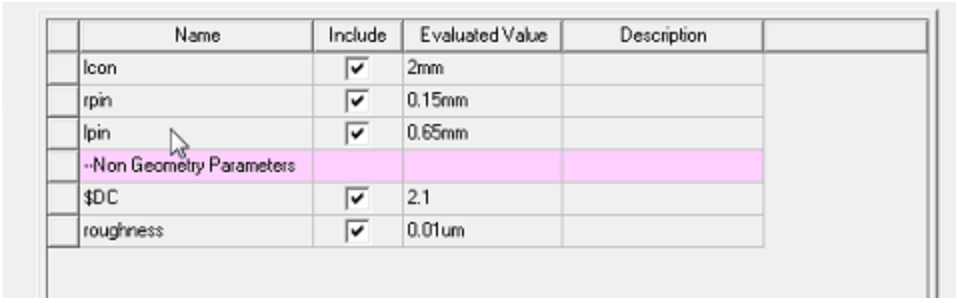
- You can fill in the **Description** field, if desired.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Non-Geometry Variables in 3D Components

If you create non-geometry variables materials or boundaries, these are listed separately:

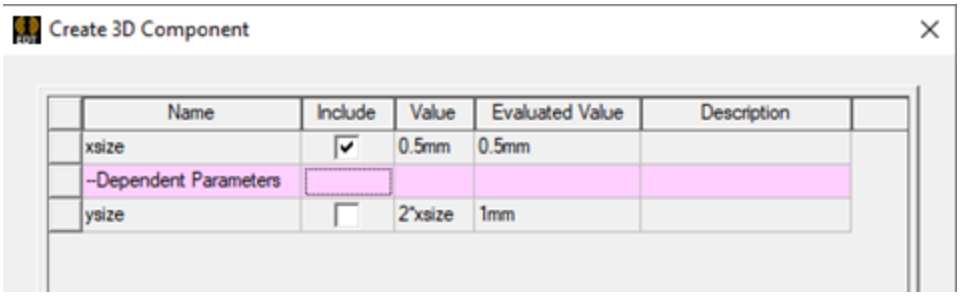


If you exclude any geometry, boundary, or material associated with a variable, the associated variable is automatically excluded on the **Parameters** tab.

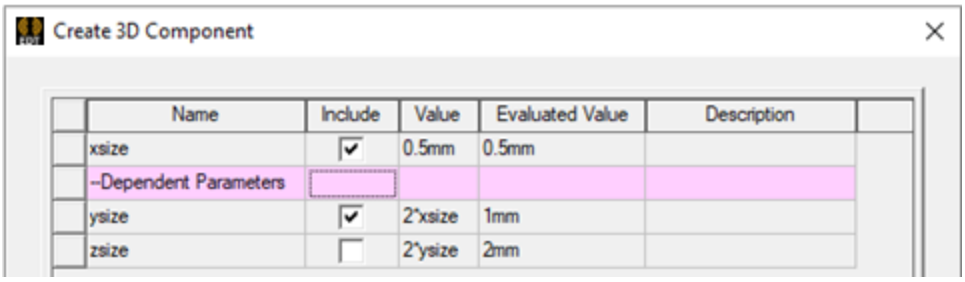
Dependent Variables in 3D Components

If you create dependent variables, you can choose to display the values.

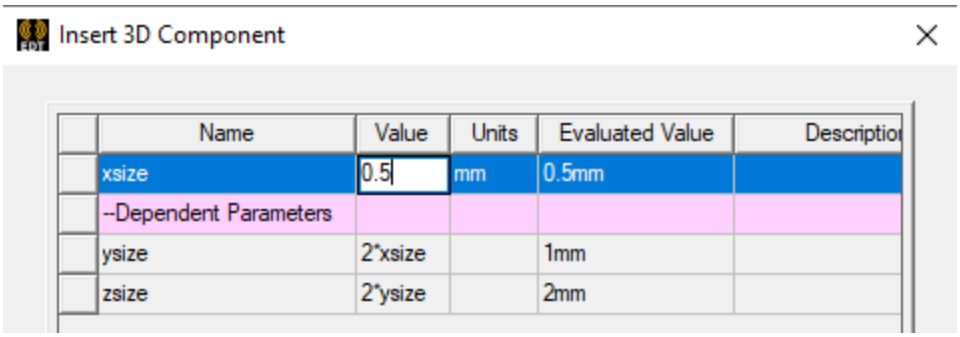
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If more variable dependency occurs for an Included variable, such variables are also displayed with the option to include.



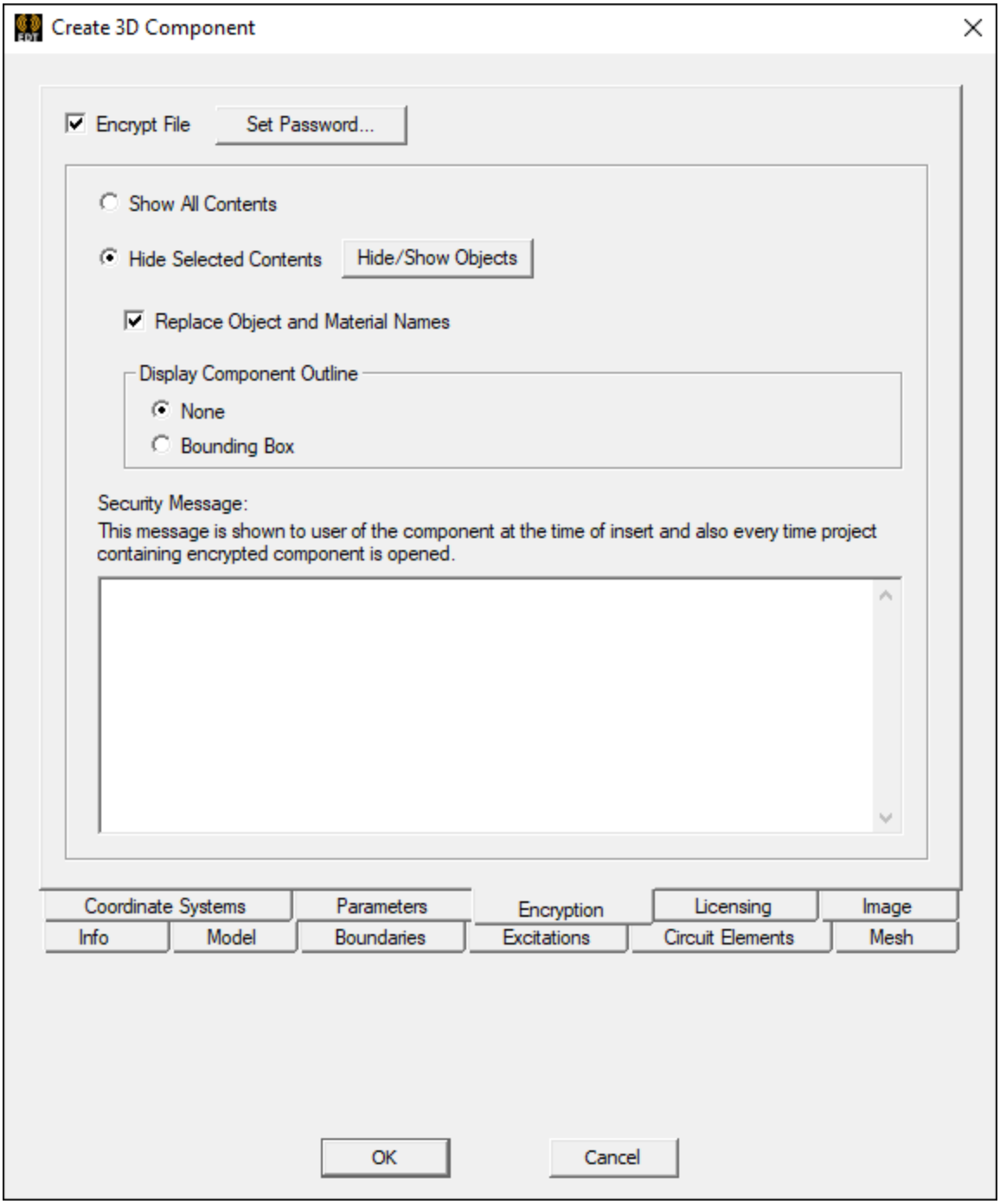
When you Insert a 3D Component with dependent Parameters included, you can edit the Independent variables, but can only view the evaluated values of the Dependent Parameters.



Create 3D Component Encryption Tab

The **Encryption** tab contains options for 3D component encryption.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selecting the **Encrypt File** check box enables all options and automatically launches the **Set Passwords** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Encryption is a common technique for encoding information in a manner that is very difficult for unauthorized users to decode. When this option is enabled for 3D components, the contents of the component file are encrypted to protect them from external viewing or editing.

- 3D components use the Advanced Encryption Standard (AES) with a 256-bit key and password-based key derivation.
- If the encrypted component is created with an **Internal Key**, it can be used without additional authorization.
- If the encrypted component is created with a **User Password**, that password must be entered when the component is used in a design.
- It is not possible to edit the definition of an encrypted component within Electronics Desktop unless **Allow Component Edit with User Password** was selected when defining the component.
- If the **Show All Contents** option is selected, the geometry and other contents will be shown when the component is used. Geometry export and links will operate as for general geometry.
- The **Hide Selected Contents** option can be used to further protect access to an encrypted component's details. When this option is selected, hidden contents will not appear in the user interface.
  - All geometry will be hidden, except for objects that the user specifies as visible during component creation.
  - Hidden geometry will not be visible in the model window or included in the history tree.
  - Geometry export and links will not be allowed.
  - Materials, boundaries, and other setup data will also be hidden, with limited exceptions such as excitation settings that must be accessed in order to control the simulation and post-processing of the design.
  - For components with hidden contents, all geometry will be excluded from field calculations and from plots of fields and meshes. Plots in surrounding volumes will extend to, and can include, the outer surfaces of the component, but not the containing volume.

    > **Important:**
    >
    > Such plots may indirectly reveal component content by providing a view of the surfaces where the plots terminate. To prevent such exposure, include additional modeled objects that surround the hidden contents of the component and have material of the surrounding material environment (e.g., air, vacuum).

    See: Hiding Component Content.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

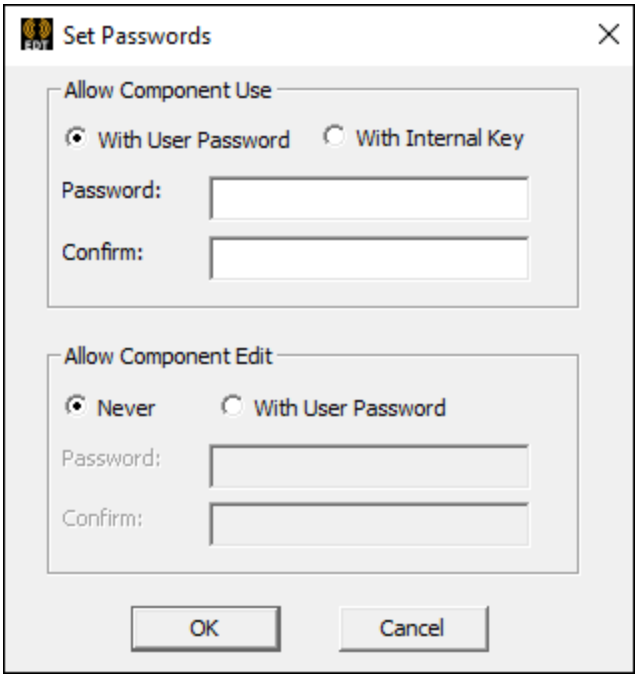
- When using **Hide Selected Contents**, the names of objects, materials, boundaries, and other setup data may appear in progress updates, messages, and other informational text. The **Replace Object and Material Names** option can be used to automatically convert object and material names to generic names in the exported component. Other sensitive names should be changed before exporting.
- The **Security Message** field specifies text that is displayed at component insertion and when the project containing the component is opened.
- When a project contains one or more encrypted 3D components, the project file is encrypted in order to protect its contents. This encryption uses an internal key, so that the file can be opened without entering a password.

> **Important:**
>
> There are some additional considerations with respect to projects using encrypted components:
>
> - The component itself will always be stored in encrypted form.
> - When an HFSS design contains an encrypted component with hidden contents, it will also protect solver inputs and other project files containing data that could be used to extract component information. However, note that solver inputs associated with SBR+ regions are not protected, even when encrypted components are in use.
> - For a hybrid design that contains a combination of SBR+, FEM, and IE regions: the FEM and IE regions will be fully protected, while the SBR+ regions will not.
> - Other design types, such as Maxwell, Q3D, or Icepak, do not include extended protection of solver inputs and other project files.

Set Passwords

Clicking **Set Password** launches a window containing fields for setting component passwords. This is the same window that automatically launches upon enabling **Encrypt File**.

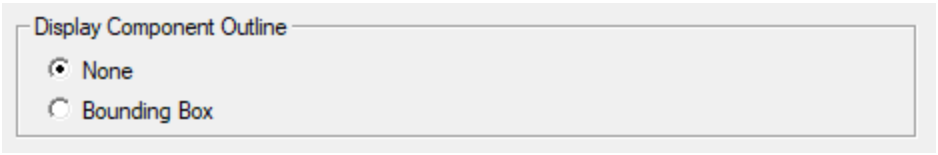
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Allow Component Use** – allows you to select between encrypting with a user password or an internal key (which requires no additional authorization). If setting a password, enter it in both the **Password** and **Confirm** fields and click **OK**.
- **Allow Component Edit** – allows you to choose between never allowing edits to components or allowing edit with a user password. If selecting the latter, enter a password in both the **Password** and **Confirm** fields and **click** OK.
- When enabling a password for both component use and component edit, the password does not have to be the same.

Hiding Component Content

The **Hide Selected Contents** option allows selected parts of the component to be displayed either as a **Bounding Box** or without an outline (**None**).

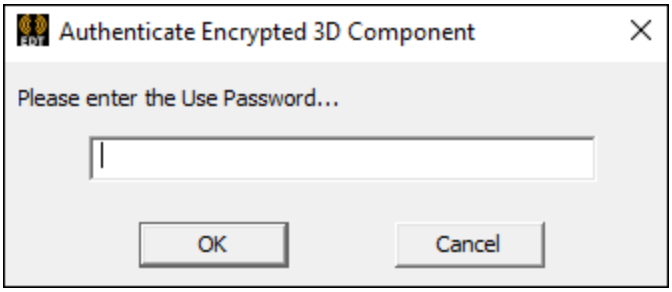


Once encryption is set, visibility can be set on a per-object basis using the **Model** tab of the **Create 3D Component** window. Preview the component using the**Image** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

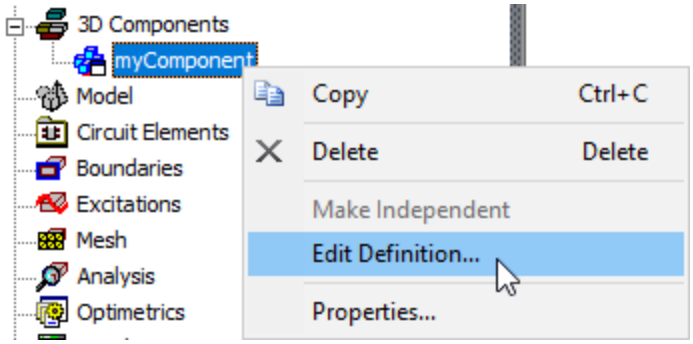
Inserting Encrypted Components

When an encrypted 3D component is added to a design, a prompt will appear asking for the password.

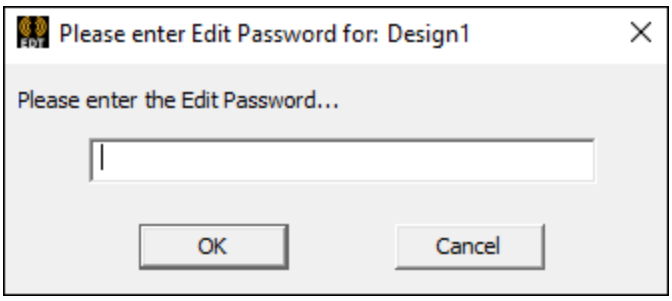


Editing Encrypted Components

3D component definitions can be edited from the **Project Manager**:

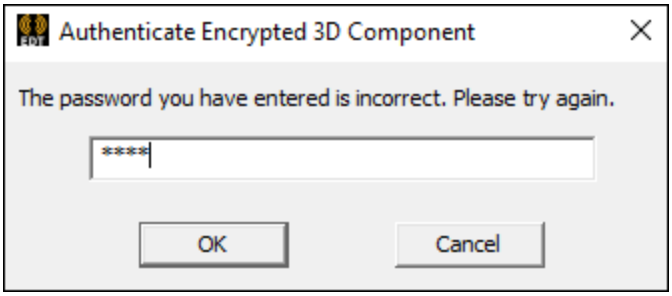


A prompt appears asking for the password:

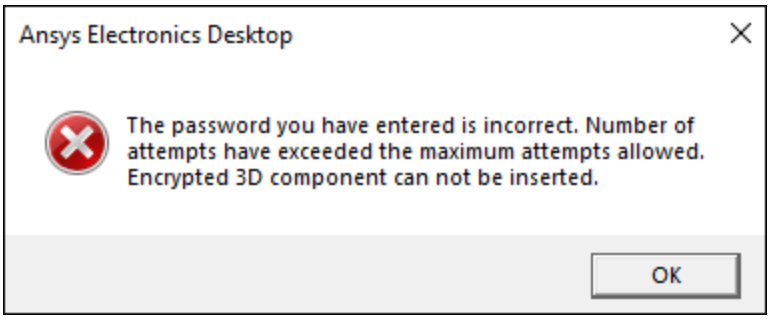


If the password is incorrect, a warning appears:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

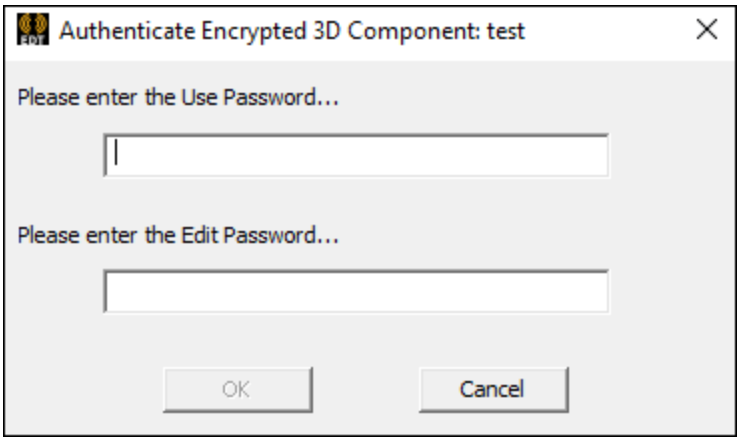


There is a three attempt maximum, after which a warning will appear and no further attempts may be made:



Opening a 3D Component File

When opening an encrypted 3D component file using **File** > **Open**, both the use password and edit password must be entered:



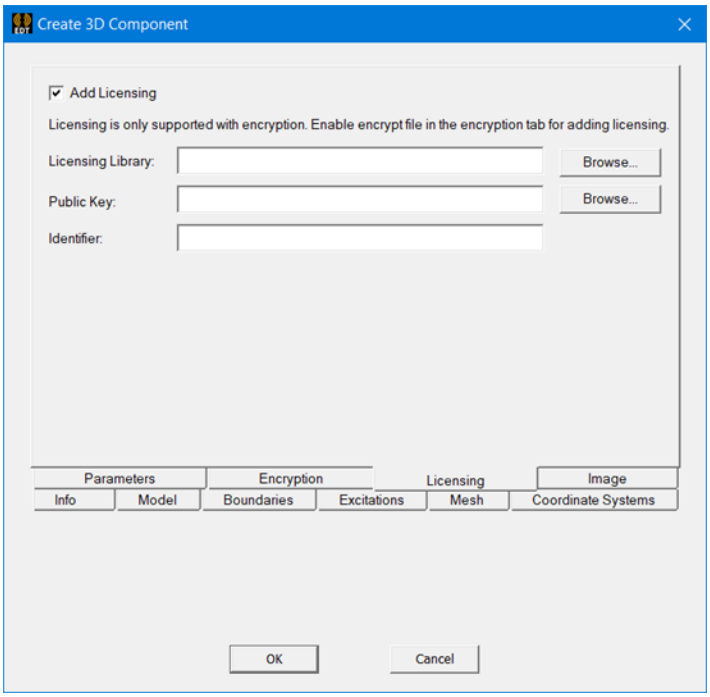
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copying and Pasting Encrypted Components

Encrypted components may be Copied and Pasted. They will be remain encrypted.

Create 3D Component Licensing Tab

A 3D Component can be licensed using an external licensing library. This feature is for those who want to license and distribute their 3D Component models. On the **Licensing** tab, you specify a 3D Component Licensing dll, a public key and an optional component identifier. You can license a 3D Component only if it is also encrypted.

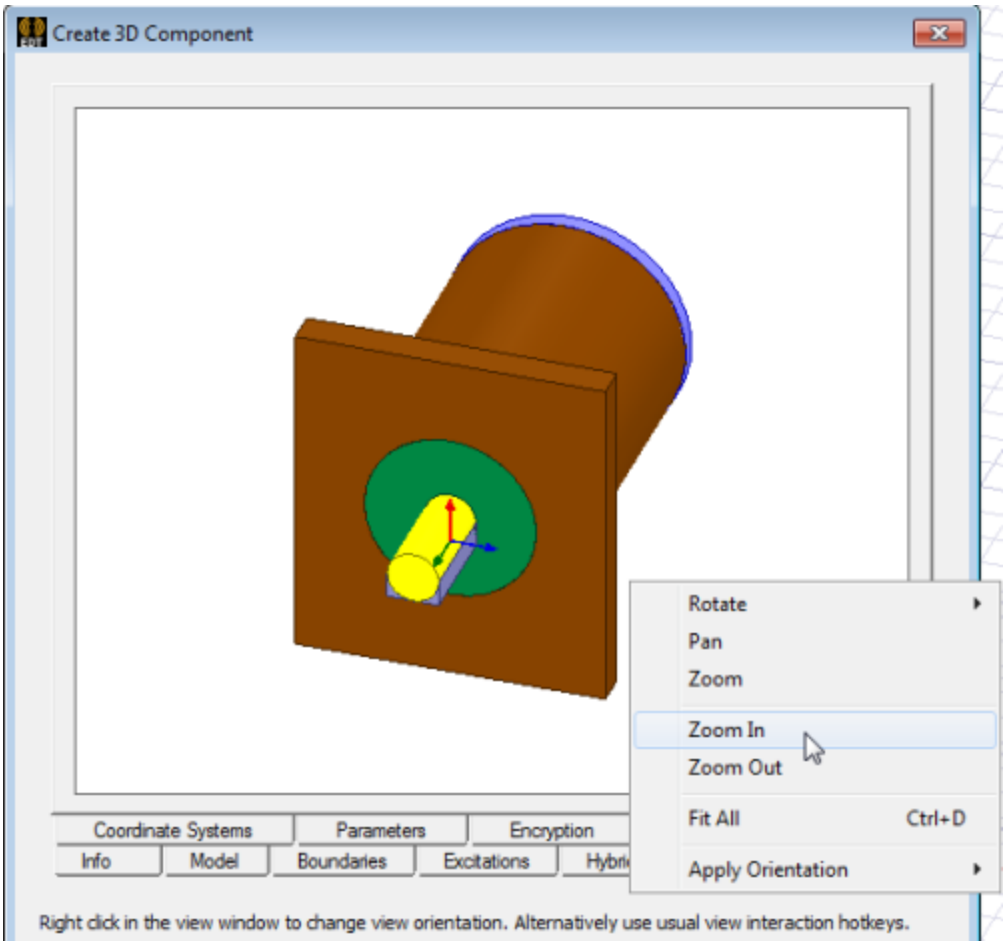


Create 3D Component Image Tab

The **Image** tab shows the preview image for the component, based on the reference Coordinate System. The image also responds when you **Include** or **Exclude** objects from the **Model** tab.

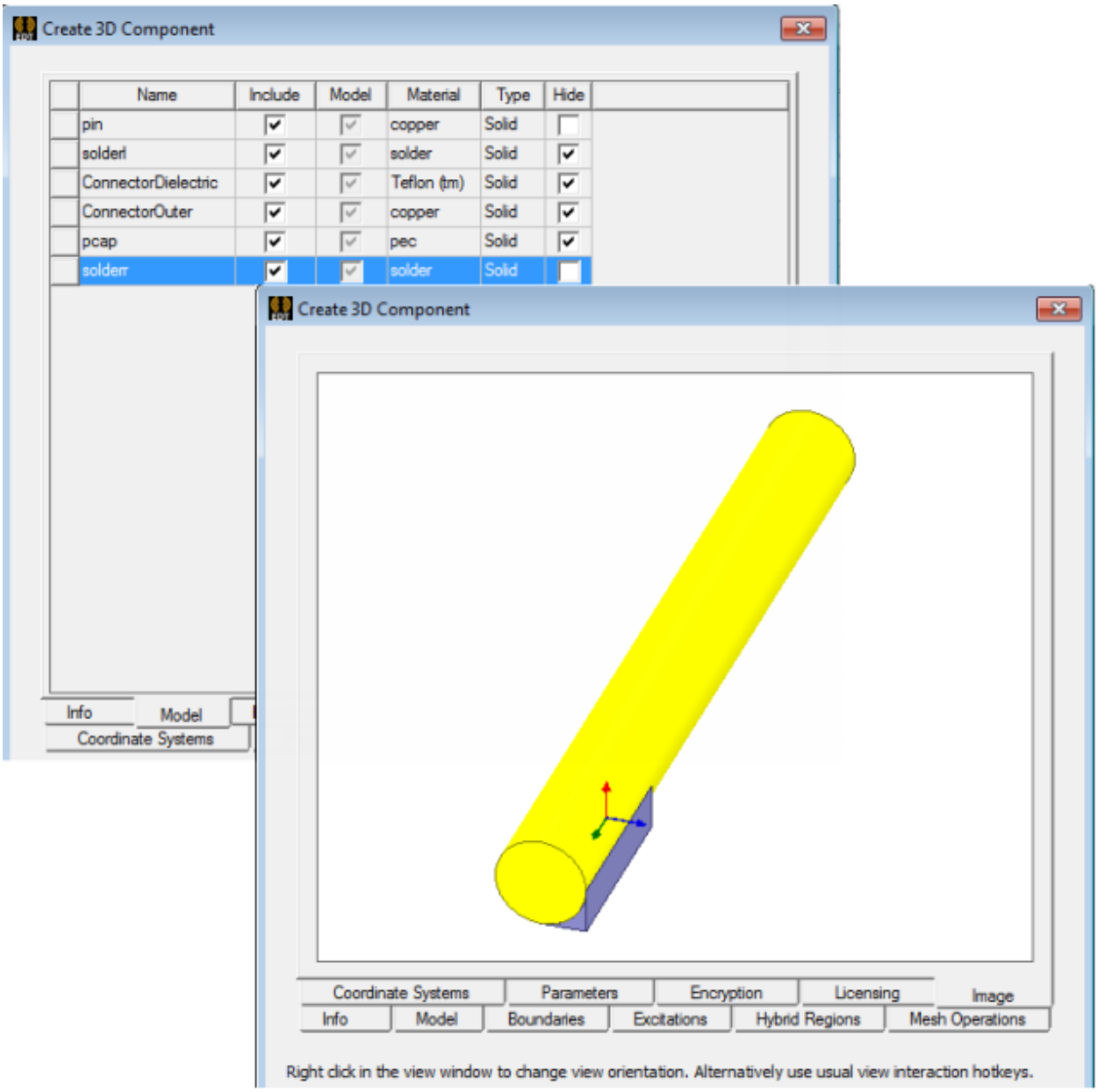
- You can right-click for a shortcut menu to change the view of the preview image.
- Changing the preview orientation does not affect the modeler window view.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



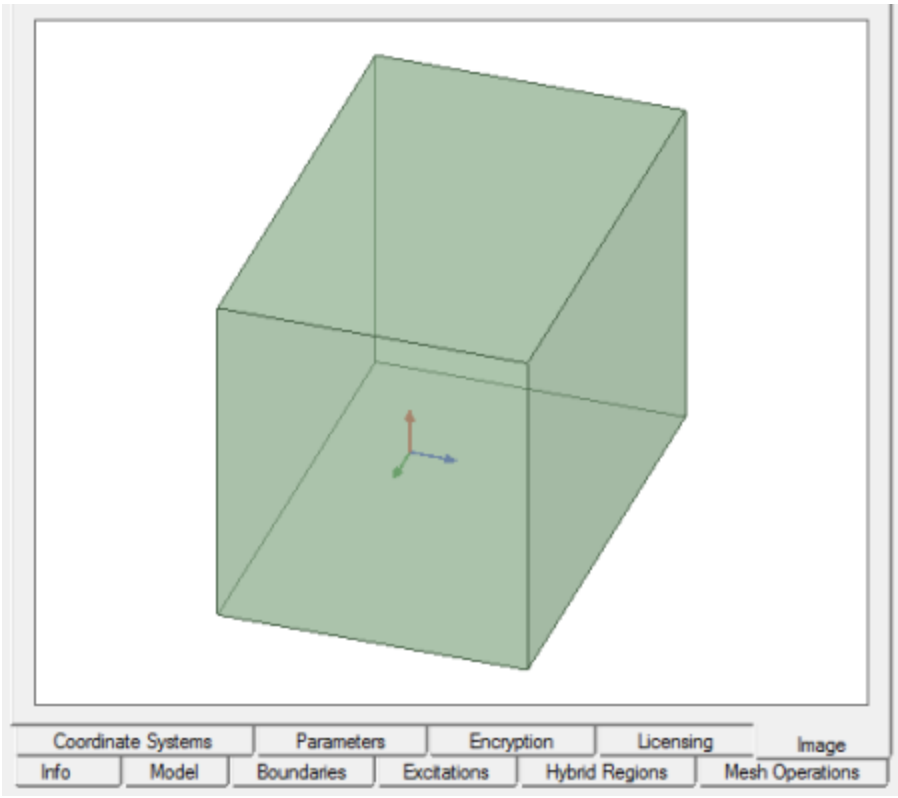
If you use the **Encryption** tab and **Model** tab settings to hide/show the model or specified model objects, the **Image** tab responds. You must use encryption to enable the Hide check boxes on the **Model** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you use the **Encryption** tab to display only an object outline, the **Image** tab responds.

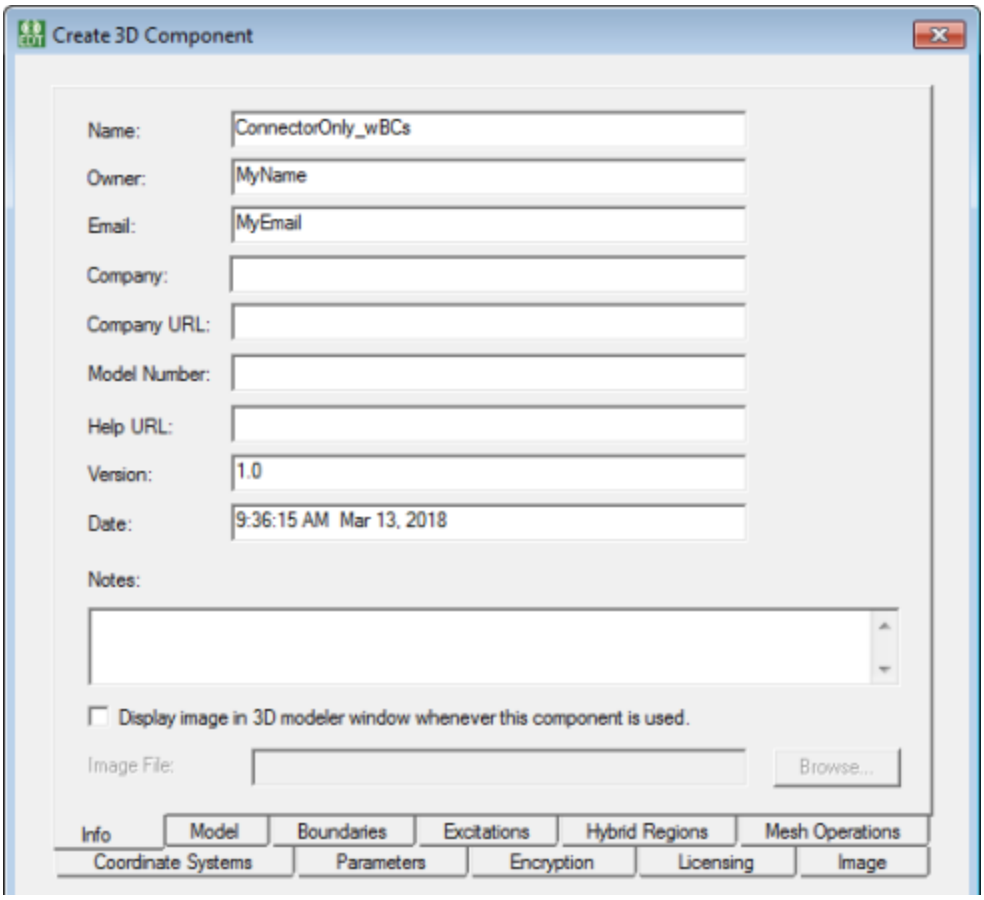
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



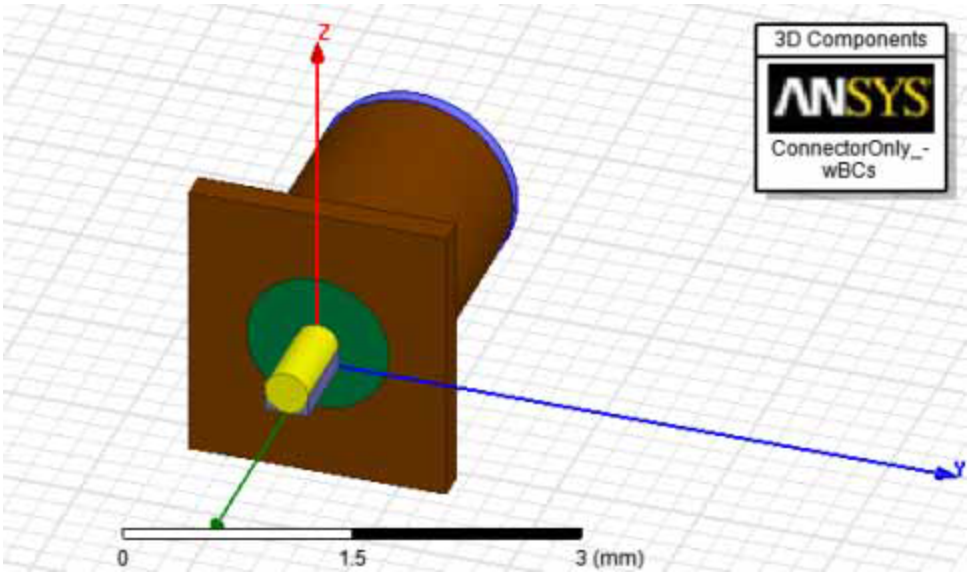
Create 3D Component Info Tab

On the **Info** tab you can view and edit component information, as well as add notes. The *Name* in the first field is used when the component is inserted into a design. Other fields include *Owner, email, Company, Company URL, Model Number, Help URL version, Date,* and *Notes*.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

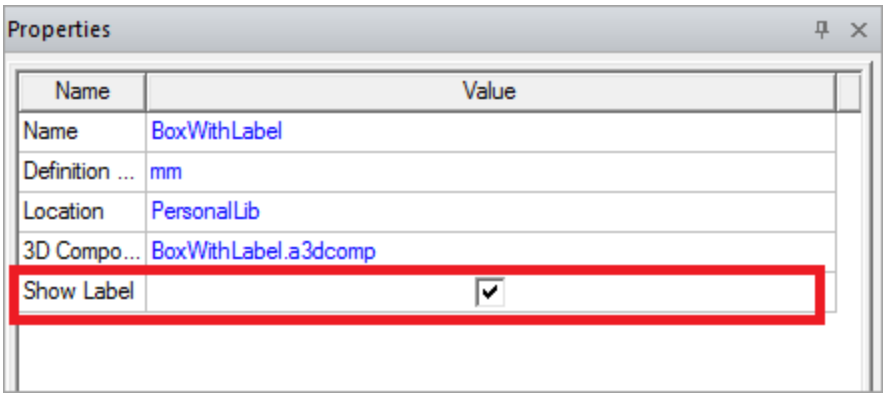


You can also included a .bmp format image file to display in the upper right area of the 3D modeler whenever the component is used. The image file typically contains a company logo. Check the **Display image** box to enable the **Image File** field, and click **Browse** to open a window to navigate your file system select the image.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

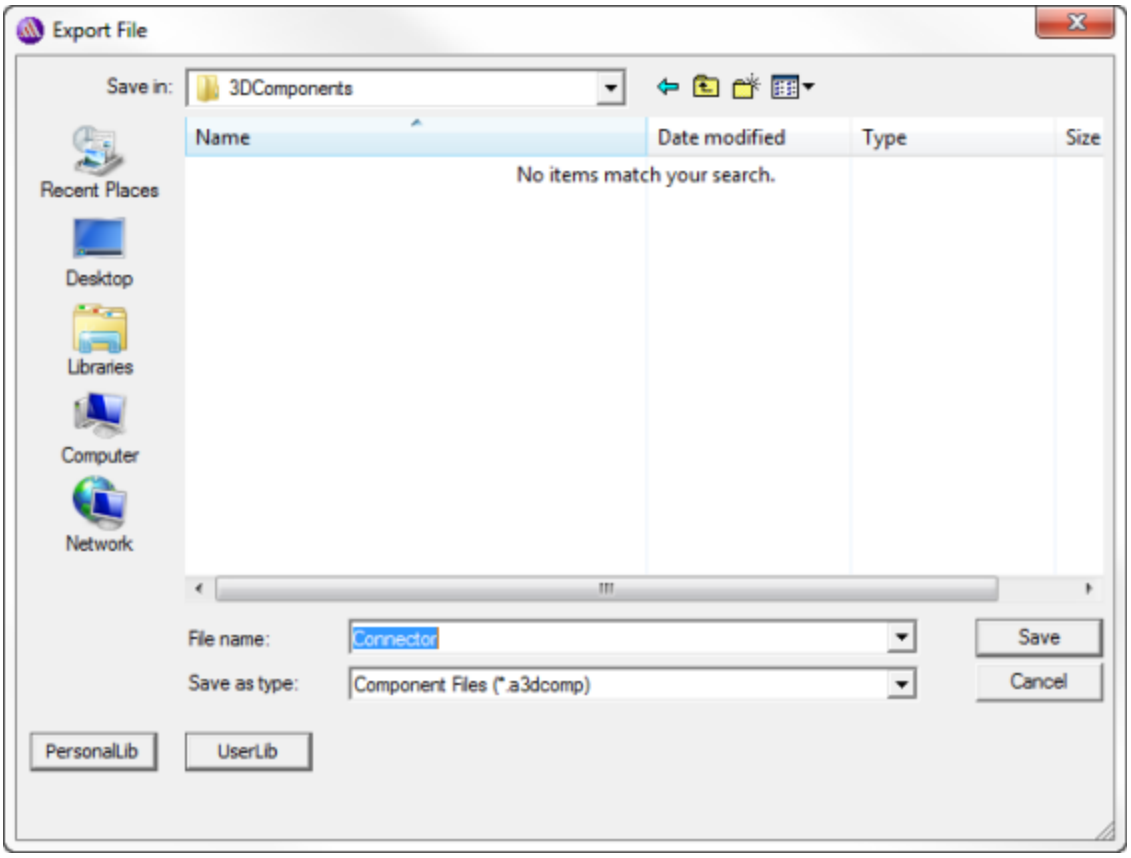
You can move the image in the modeling workspace by clicking the "3D Components" area of the box and dragging it. You can hide the image by editing the 3D component and clearing the **Display image...** check box, or by deselecting the Show Label check box in the Properties window:



Save 3D Component File

When you **OK** the *Create 3D Component* dialog box, an *Export File* dialog box displays. By default the **Save in** field shows the model source folder. Click **PersonalLib** or **UserLib** to display a 3D Components folder.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

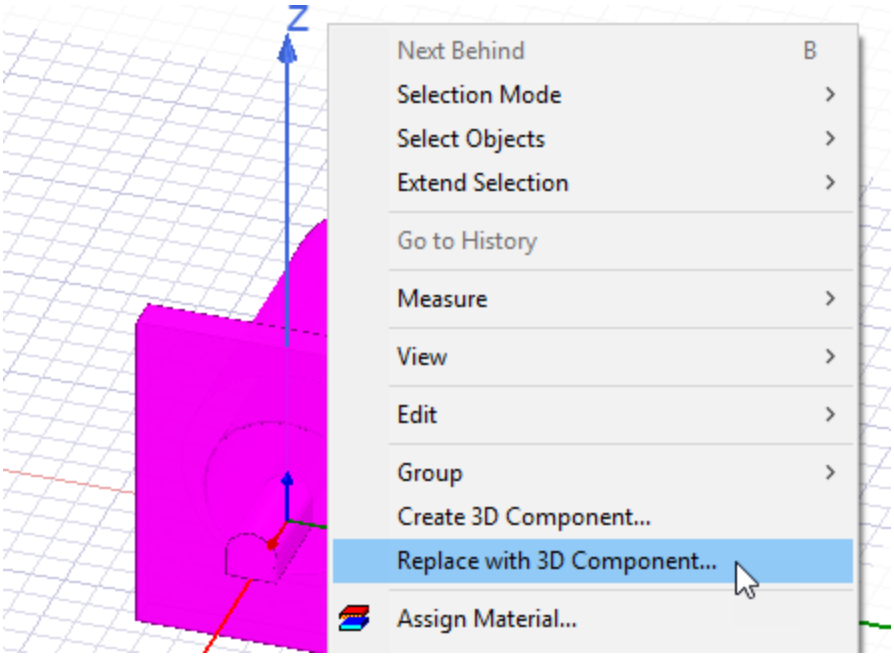


The default **File name** is the component *Name* specified in the *Info* tab.

Replace Selection by 3D Component

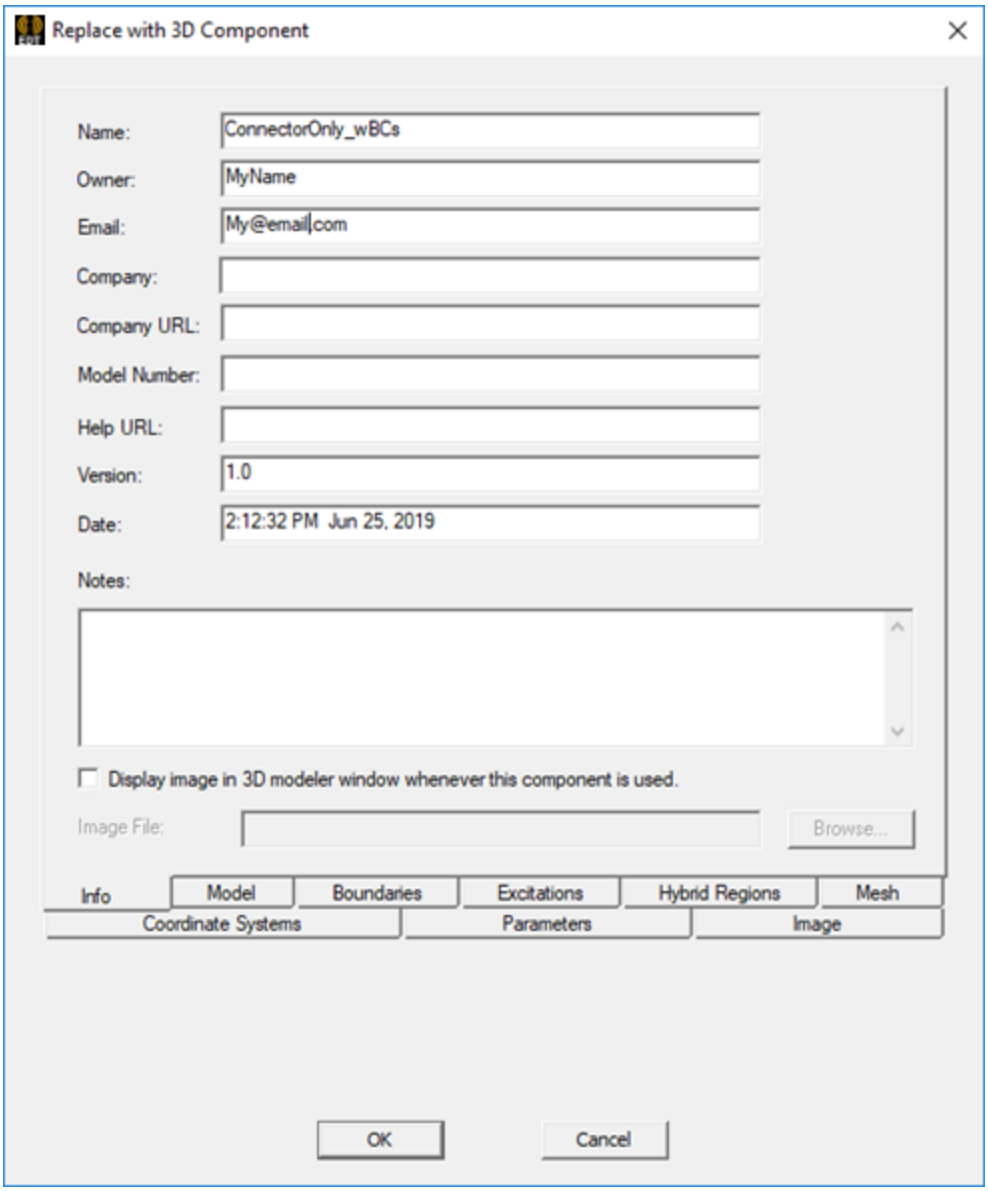
To replace a selection in the 3D modeler window with a 3D component version of the selection, select an existing object or model, right-click, and select **Replace with 3D Component**. You can also click **Draw> 3D Component Library> Replace with 3D Component**. You cannot replace components from a design with an array setup. However, you create a 3D component from a unit cell and use that to create a Multi-Unit Cell array. When you replace a selection, object names are retained, port names are retained, component parameters map to existing design variables and report, field plot, and edit source are all preserved.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This command opens the **Replace with 3D Component** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The tabs let you view the component features organized according to geometry, design data, and settings. The **Replace with 3D Component** dialog box does not offer tabs for encryption or licensing, compared to Creating a 3D Component. You can use these tabs to change inclusion of objects, boundaries, ports, coordinate systems, or parameters for the 3D component. When you click **OK**, no *Save* dialog box is shown.

- Info tab
- Model tab
- Boundaries tab
- Excitations tab
- Hybrid Regions tab

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

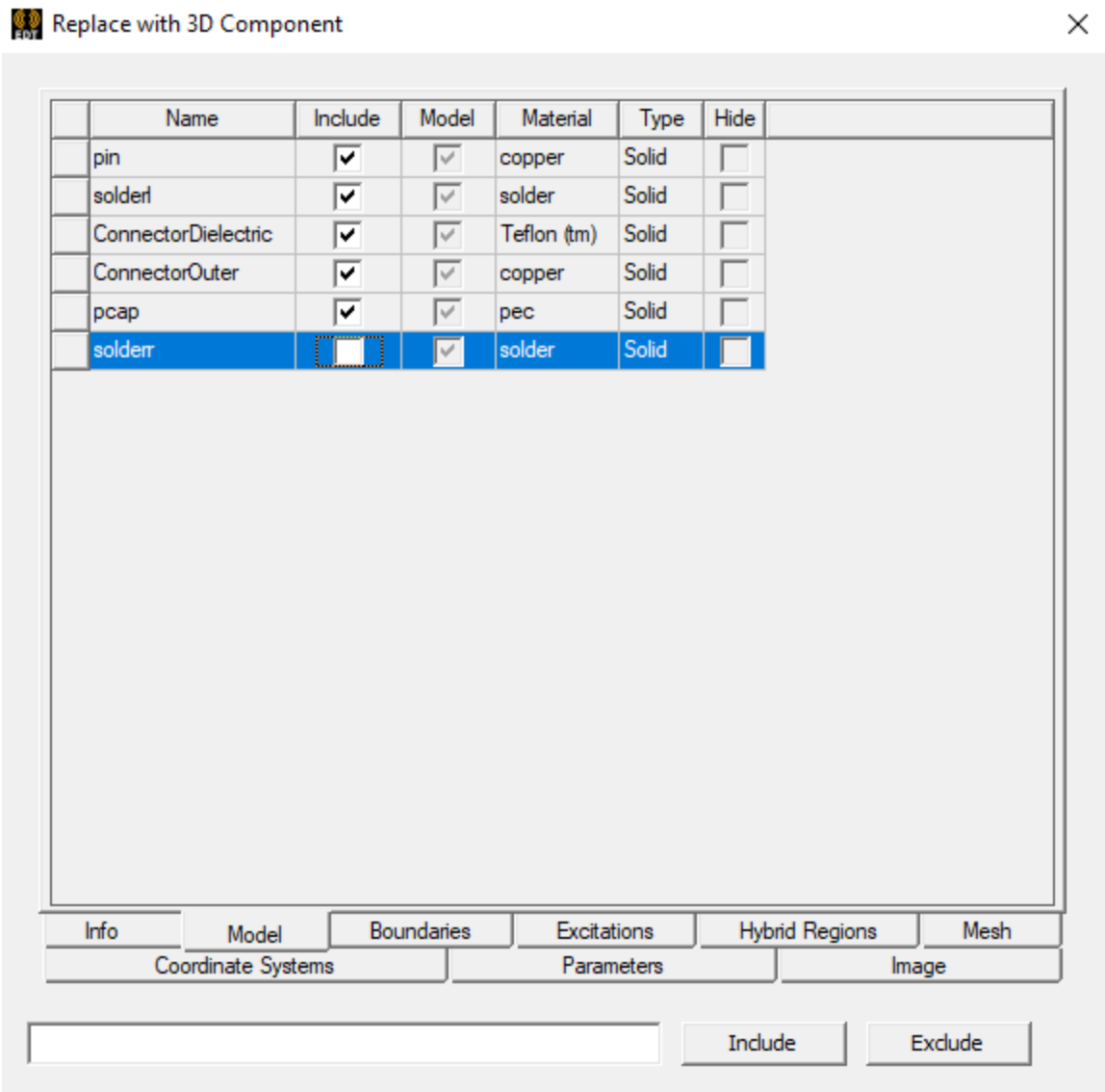
- Mesh tab
- Coordinate Systems tab
- Parameters tab
- Image tab

When you click OK, no save dialog is shown. The 3D component is saved to a temporary location. Insertion of instances uses the temporary file, but works just as regular 3D component insertion. The Project tree shows the Component under 3D Components, the boundaries are no longer listed under the Boundaries icon. Excitations, Reports and Analysis Setups remain the same. History Tree no longer shows detailed model edits. You can **Undo** and **Redo**.

Include or Exclude Feature from Component

You can view the various tabs and select which features to include or exclude while replacing the selection with a component. You can use the check boxes in the **Include** column for this purpose. You can also type a feature *Name* in the text box and use the **Include** or **Exclude** button. The text field supports wildcard characters (* and ?), so you can toggle the inclusion state for multiple objects with similar names in a single operation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



An object's **Include** option under the *Model* tab impacts the inclusion state in other tabs. For example, if you exclude an object from the component creation, the design data, parameters, and coordinate systems corresponding to that object are also excluded.

Sorting Columns Using the Headers

The features listed for each tab include columns for the Name, check boxes for whether to include, and properties. You can click on the column headers to sort by Name, Model, or other feature listed for each tab. Click the column header to selects the column to sort. Click again to invert the column.

You can sort lists by using all columns except Include.

Design Data Notes:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

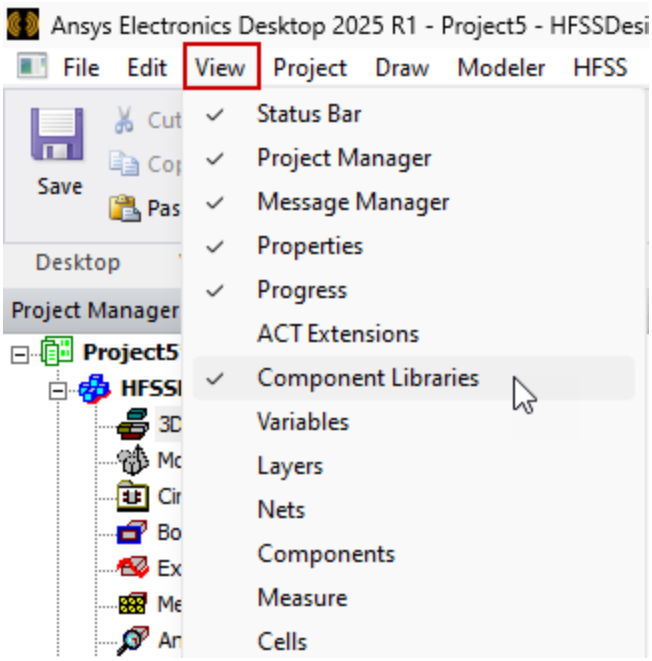
- DC thickness on selected objects is always included in components. It is not listed in the **Boundaries** tab.
- The **Create 3D Component** dialog does not list design data without an assignment (for example, a far/near field incident wave in HFSS).
- The parent of any included boundary/excitation is included, as long as the parent does not require assignment.
- Design settings like material overrides are included.

Once you have made the Include and Exclude settings, and have specified the Image and Info you click **OK** to Replace with 3D Component. When you click OK, the component is validated for everything that is included in the component. For example, you cannot include boundaries if the object on which the boundaries have been created is not included.

Using the Component Libraries Window

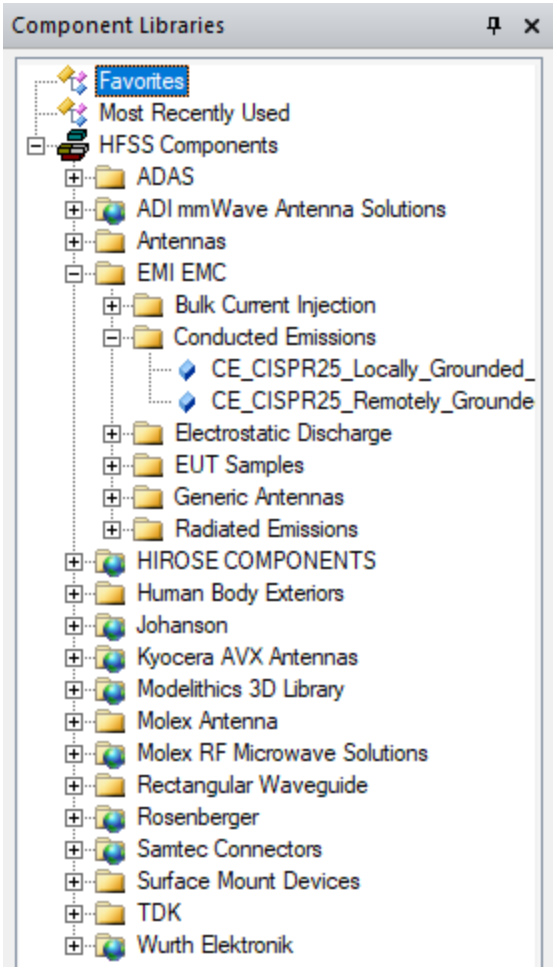
Once you save one or more components to a library, and create target coordinate systems in the design where you intend to place the component, you can right-click the 3D component icon ( ) in the Project Manager or, using the menu bar, click one of the **Draw** > **3D Component Library** commands to browse your folders or libraries.

Toggle the visibility of the docked **Component Libraries** window from the menu bar by clicking **View** > **Component Libraries** and navigate the installed libraries from there. Visibility of the **Component Libraries** window is remembered for each design type so that, if context switches between different design types, the window is shown or hidden based on setting for new design type.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

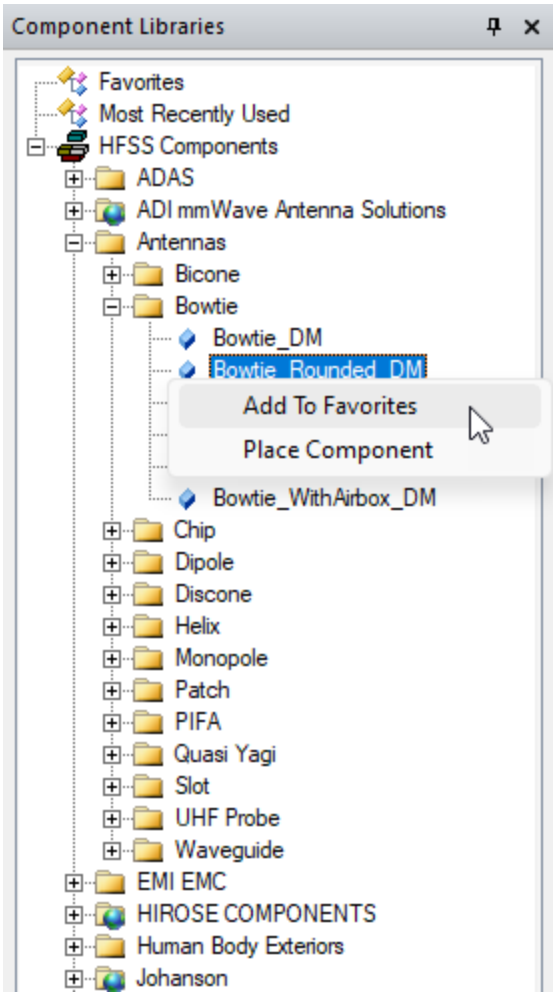
From the **Component Libraries** window, you can navigate the directory of installed components, as well as any in your Personal Library. The **Component Libraries** window lists all the 3D components available in syslib, UserLib and PersonalLib. It shows a folder corresponding to a library only if there are components for the active design type in that library folder. There are also additional optional folders for managing Favorites and Most Recently Used components. **Component Libraries** window contents, including Most Recently Used and Favorites correspond to the active design. When context is switched between different designs, the *Component Libraries* window will be refreshed so that it is relevant to the active design.



You can control whether to show the Favorites and Most Recently Used folders. Access the *Options* dialog box from the menu bar by clicking **Tools** > **Options** > **General Options**. The folder visibility options are on the General > Component Libraries Options page.

Right-click a component to view the **Add to Favorites** command for including frequently used components in the Favorites folder. Once a model is included in the Favorites folder, you can also right-click it to **Remove from Favorites**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When you have added a component to a design, it also appears in the **Most Recently Used** folder. You can also right-click model there to view the **Add to Favorites** and **Place Component** commands.

You can select any component from the library, and drag and drop to insert it in the design. If you insert a component by the drag and drop method, the component is immediately inserted, skipping the **Insert Component** dialog box. The component is inserted with default component parameter values and is placed in the currently active coordinate system.

You can also select **Place Component** from the shortcut menu or double-click any model in the component libraries to view the *Insert 3D Component Instance* dialog box opened to the **Parameters** tab. Using the **Place Component** menu command allows you to review component parameter values, change target coordinate system, and so forth, using the *Insert Component* dialog box. You can also view the **Image** and **Info** tabs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component file organization on the disk

3D Component files are organized in three libraries (that is, *syslib, UserLib,* and *PersonalLib*). 3D component files must be in a **3DComponents** folder under the appropriate library location. Files must be further organized by design type in the **3DComponents** folder. For example, component files for HFSS should be organized under 3DComponents\HFSS, as done for syslib. The 3D Component Libraries window looks for components under a folder corresponding to the active design type, and it ignores component files under folders corresponding to another design type. For example, HFSS will ignore component files placed directly under the Maxwell subfolder.

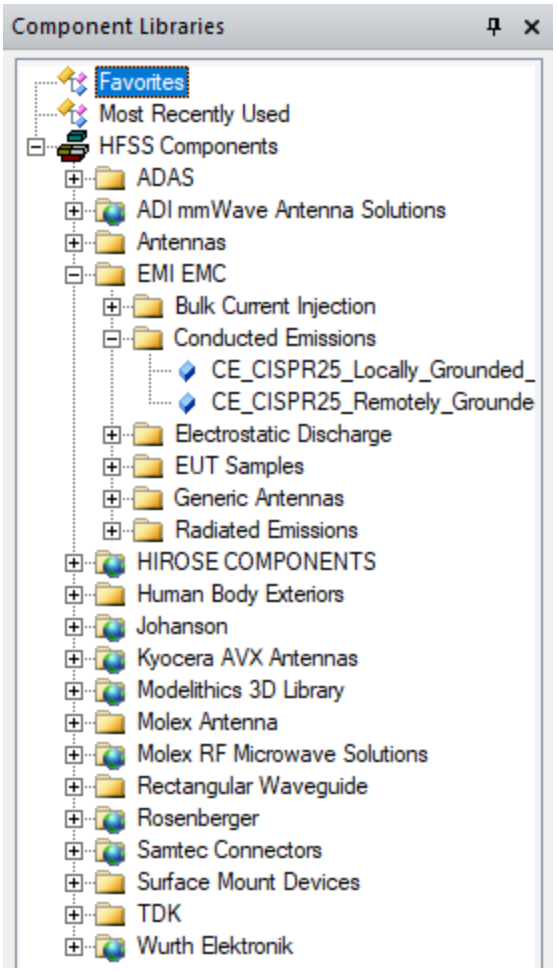
Inserting a 3D Component into a Design

Once you save one or more components to a library, and create target coordinate systems in the design where you intend to place the component, you can right-click the 3D Component icon in the Project Manager or use the **Draw** > **3D Component Library** commands to browse your folders or libraries, or use the **View** > **Component Libraries** to display a Component Libraries window to navigate installed libraries. Alternatively, you can insert an instance of a component by right-clicking the 3D Component icon in the Project Manager and selecting **Insert Instance**.

From the Component Libraries window, you can navigate the directory of installed components, as well as any in your Personal Library. The hierarchical tree display includes **Favorites** and

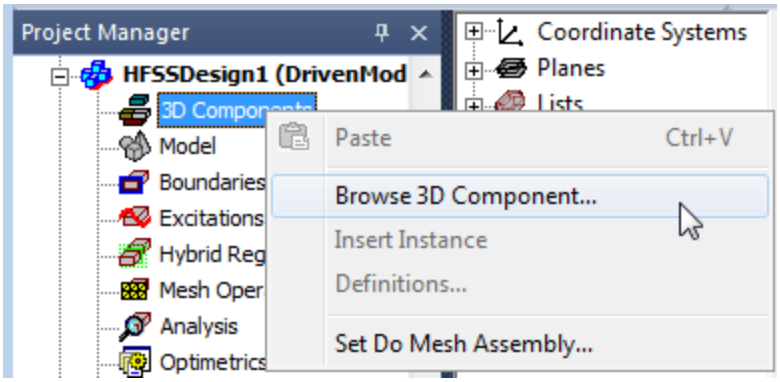
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Most Recently Used branches.



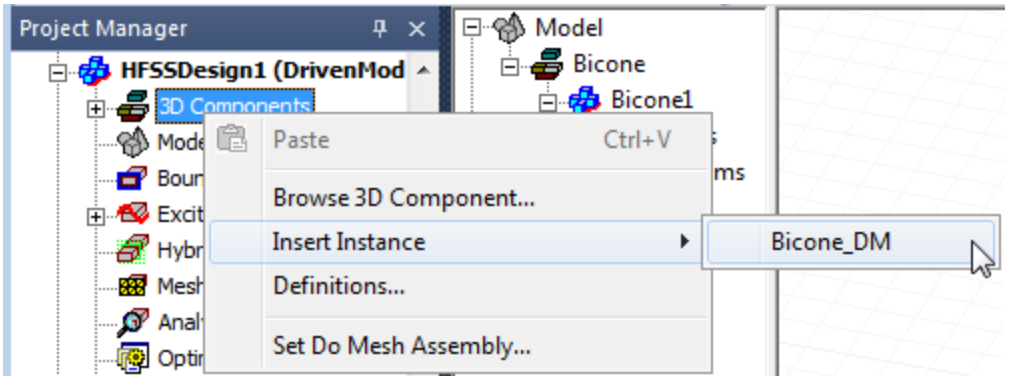
You can select any component from the library, and drag and drop to insert it in the design. Double-click any model in the component libraries to view the **Insert 3D Component Instance** dialog opened to the **Parameters** tab. You can also view the **Image** and **Info** tabs, and select the target coordinate system.

Selecting **Browse 3D Components** lets you navigate directories via a browser window.

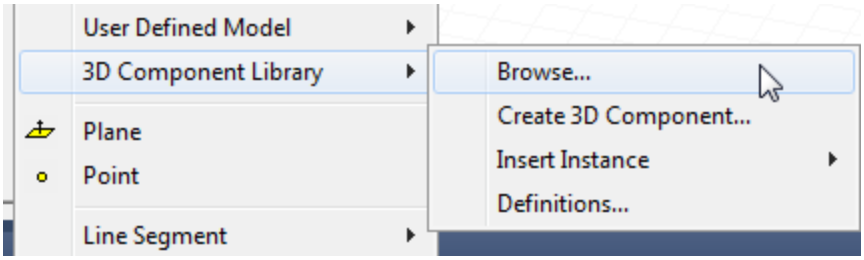


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

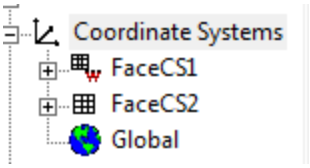
If you have previously inserted a 3D Component, the **Insert Instance** command is enabled on the menu. You can use this command to select from 3D Models in the design to easily insert another instance.



You can also click **Draw> 3D Component Library** to access the **Browse** and **Insert Instance** menus **Antennas, Rectangular Waveguides**, and **PersonalLib** to select a component to insert into a design.

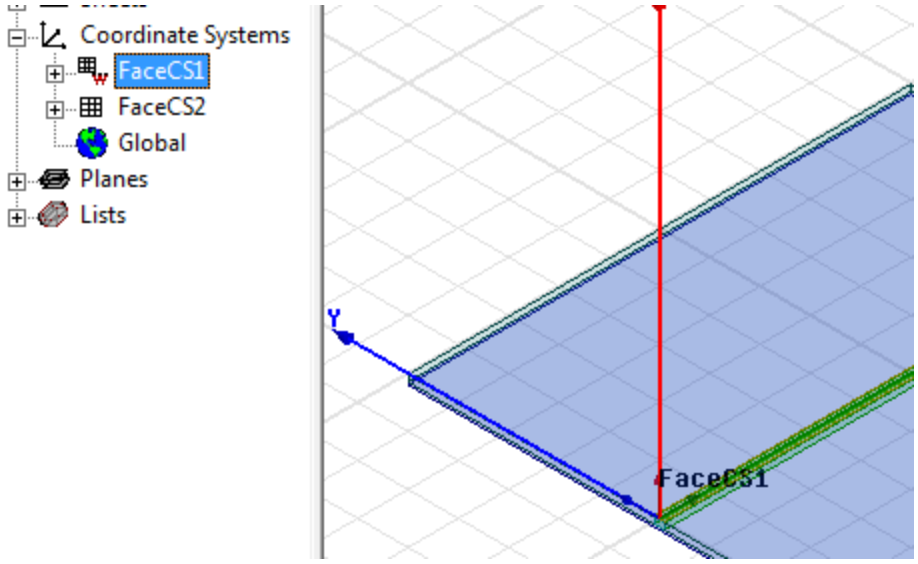


A target coordinate system provides a location for a component. For example, a design includes two additional coordinate systems:



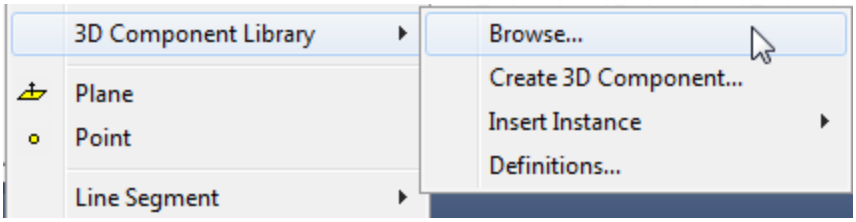
Selection of a coordinate system in the History tree displays a potential location for inserting a component.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



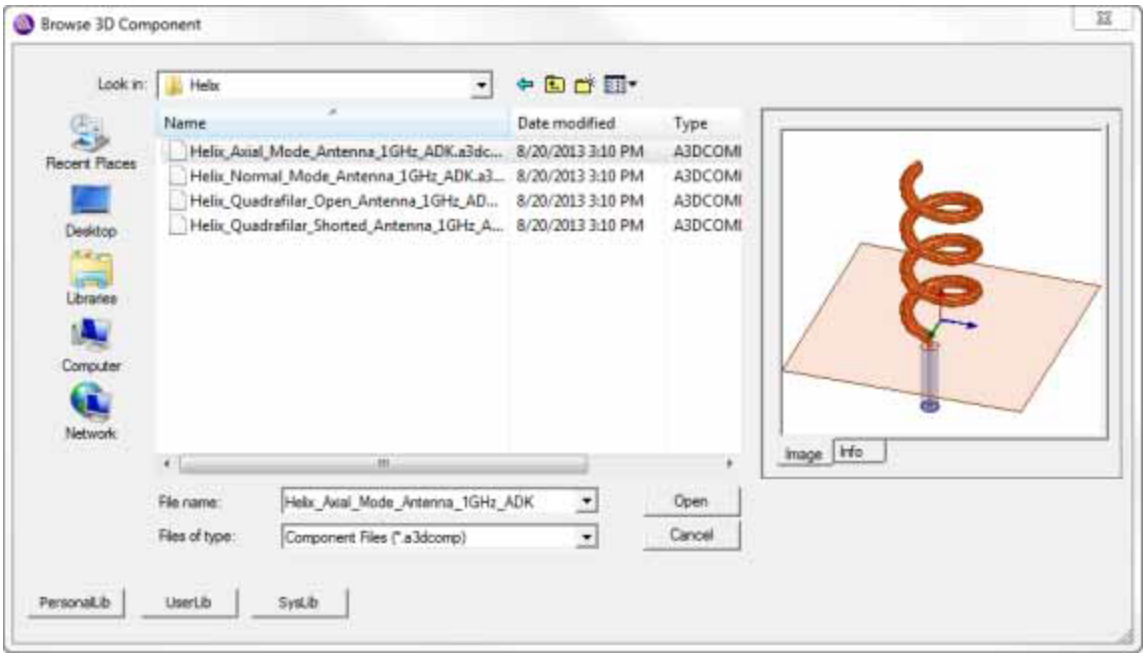
To Insert a 3D Component from the Menus

1. Right-click the **3D Component** icon in the **Project tree** to open shortcut menu. You can select **Browse** to use a browser window navigate the file system to the component.

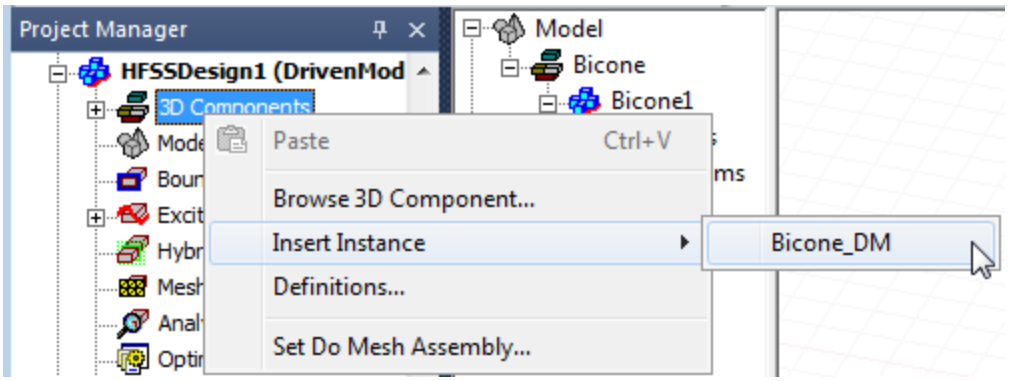


   Selecting a 3D component file causes a display of the component image and the File name.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

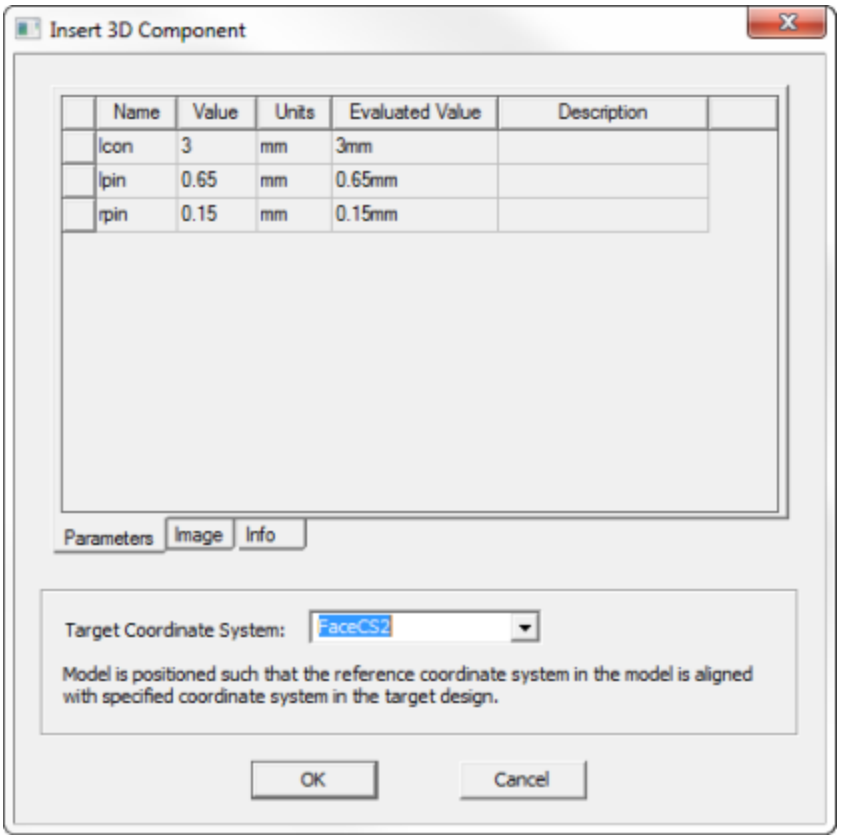


If you have previously inserted a component into a Project, the shortcut menu for the 3D Component displays that component so that you can easily insert another instance of the same component.



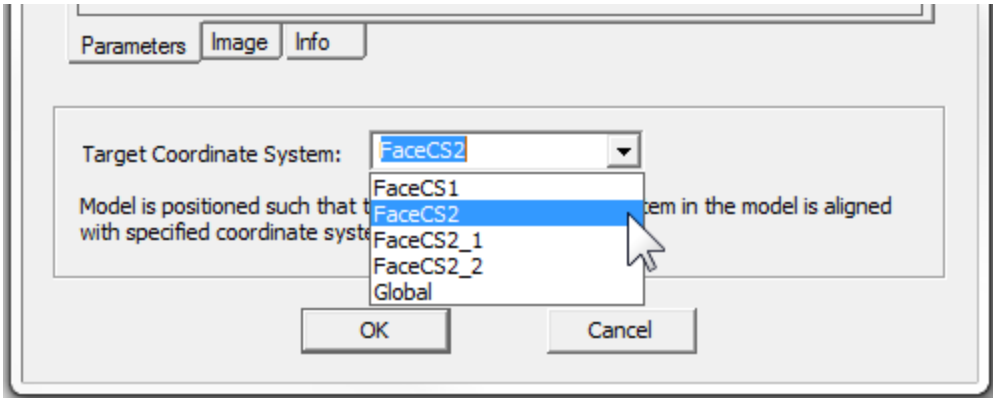
You see the **Insert 3D Component** dialog opened on the Parameters tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



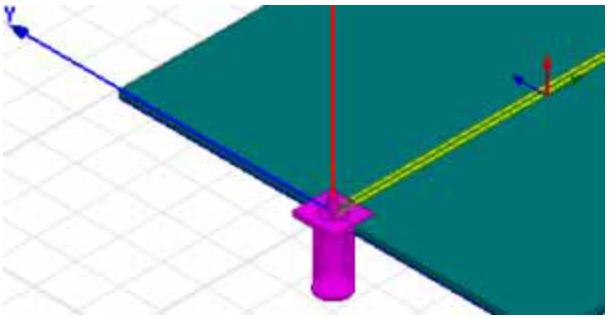
Tabs let you view the Parameters, Image, and Info. You can edit parameter values, and assign variables or expressions for parameters

2. Use the menu to select the Target Coordinate System if any have been defined in addition to the Global coordinate system. The target coordinate system that you select is highlighted in the modeler window.

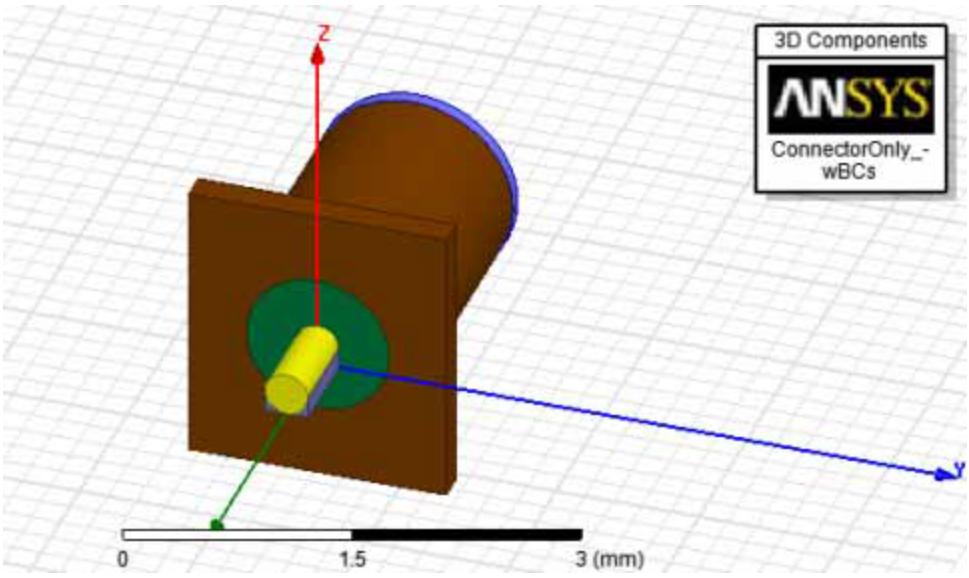


3. When you click **OK** the component is placed at the coordinate system you selected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

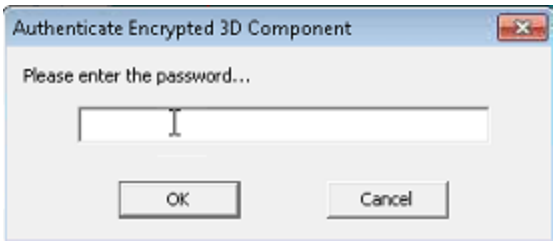


4. If the component has a logo defined, the image is always displayed in the upper right of the modeler window.



5. If you have used Browse, select a component and click **Open**. If you use the Insert Instance menu, select the component name.

If the component has been encrypted, you may be prompted to enter a password. If you insert another instance of the same component, you do not need to enter the password again for that component.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You have three tries to enter the correct password. Upon successfully entering the password, the Insert 3D Component dialog opens with the Encryption tab displayed.

> **Note:** There are some additional considerations with respect to projects using encrypted components. When a project contains encrypted components, the project file and other related files are also encrypted. Encrypted component contents may also be hidden from view in the design where the component is used. For more details, see Create 3D Component Encryption Tab.

Making Component Instances Independent

When inserting an instance of a component, it is listed under the original component and assigned the same definition. To create a unique component from a component instance, use the **Make Independent** option in the Project Manager or History Tree. In HFSS designs, **Make independent** is available for 3D components and parametric antennas. **Make Independent** is only available if multiple instances of the component exist.

> **Note:**
>
> Right-click a component in the history tree to access the **History Tree Layout** menu. You can organize components in the history tree by definition.

Project Manager

1. In the **Project Manager** under **3D Components**, right-click on a component instance.
2. Select **Make Independent**. The history tree displays the independent component.

History Tree

> **Note:**
>
> From the history tree, you can select multiple instances and make them independent. If you select all instances of a definition, **Make Independent** is unavailable.

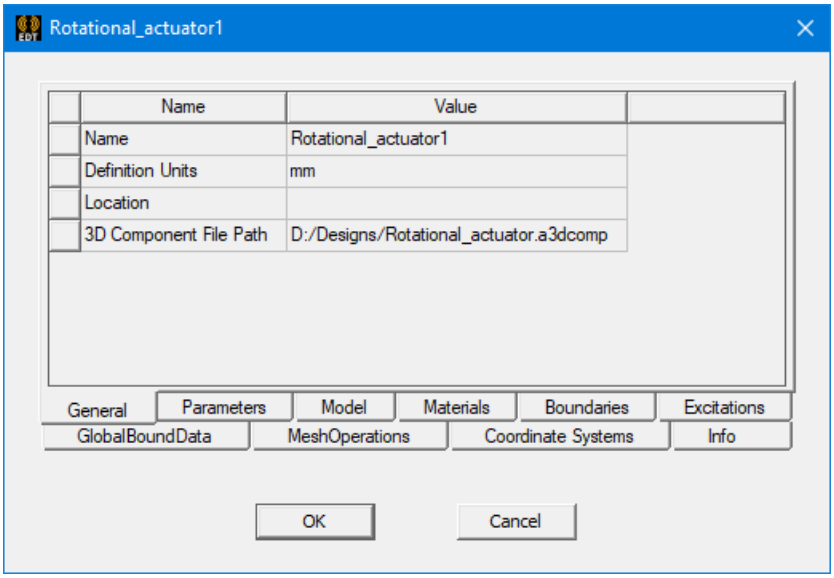
1. In the history tree, right-click on a component instance.
2. Select **Component** > **Make Independent**.

   In the history tree, the selected component appears independent of the original component and can be assigned a unique definition.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

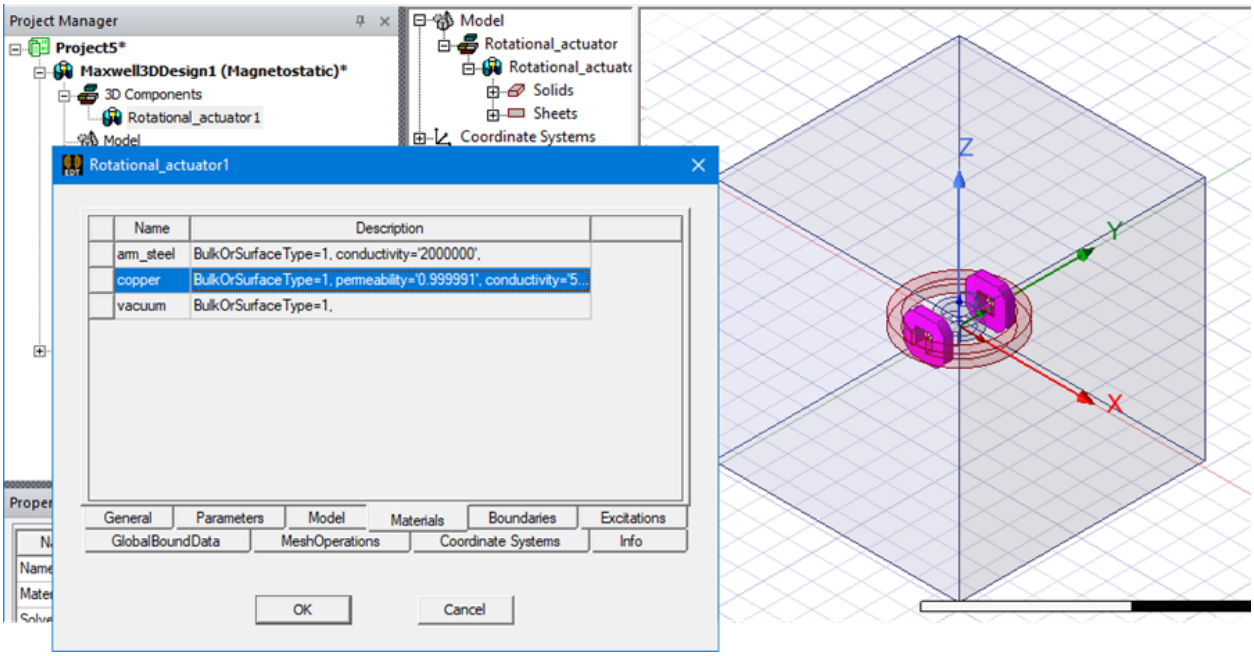
Viewing Properties of 3D Components

The Properties displayed for a selected 3D component depend on how you select that component. For a complete display of Properties, right-click the selected component (Project Manager, History Tree, or Modeler window) and click **Properties** on the shortcut menu. Alternatively, click **Edit** > **Properties** using the menu bar. This action displays a window with tabs for all component properties.

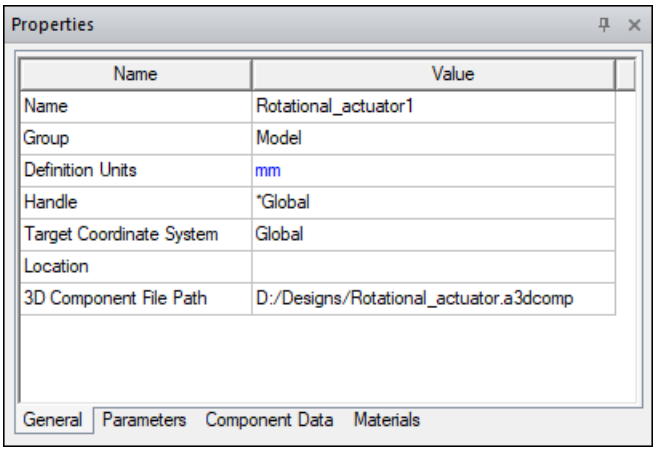


If you choose the *Materials* tab, you can then select any listed material, and the parts of the component made of that material are highlighted in the Modeler window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



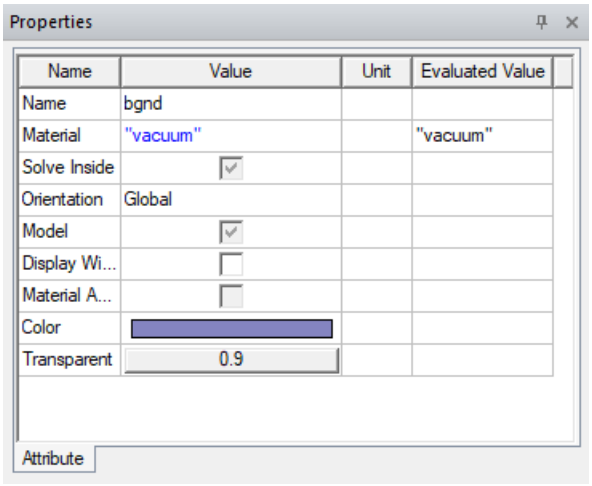
Selecting a 3D component in the Project Manager or the history tree displays tabs for *General, Materials,* and *Component Data* in the docked *Properties* window.



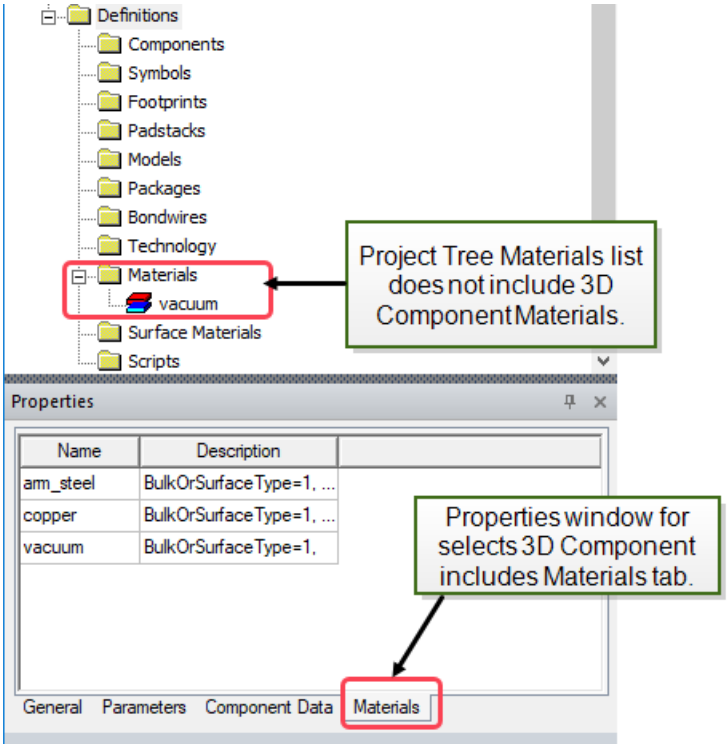
Note that the Handle property lists the handle coordinate system that was selected as reference when the 3D component was created; from this window, it can be changed to any coordinate system in the 3D component.

Selecting a 3D component object in the Modeler window displays the Attributes tab in the docked Properties window:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



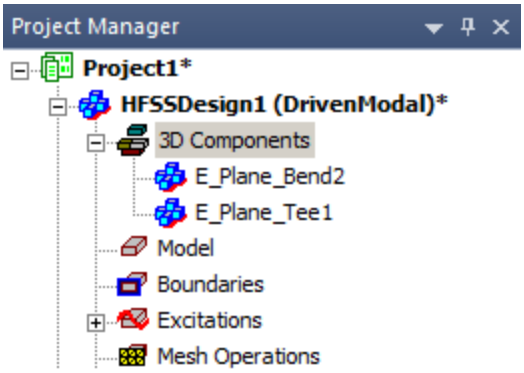
Component materials do not appear in the Project Manager's Definitions list. If you select the icon for an inserted component, the docked *Properties* window contains a *Materials* tab that shows the materials defined for the selected component.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

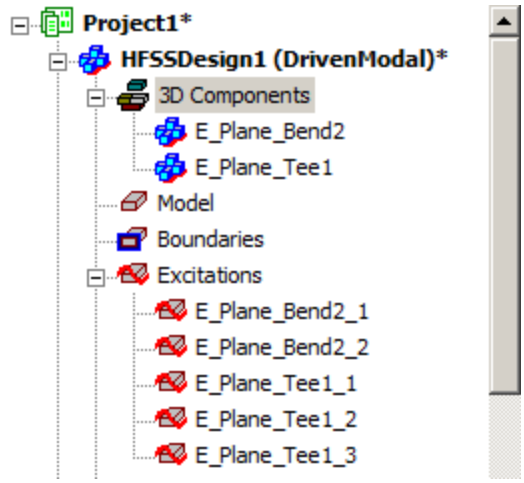
3D Components in the Project Manager Window

The **Project Manager** window organizes any 3D components that you add to a design under the 3D Components icon in the tree.



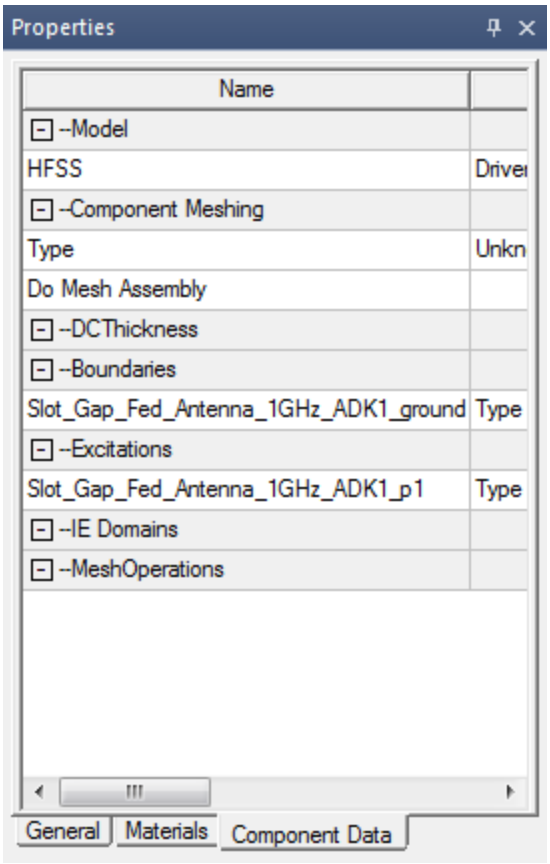
Selecting these inserted components populates the docked **Properties** window, which has tabs that display the instance attributes.

Excitations included in the 3D Components appear in the Project Manager. This gives you the ability to edit Post Processing settings, such as the de-embed distance, renormalization, and power settings.

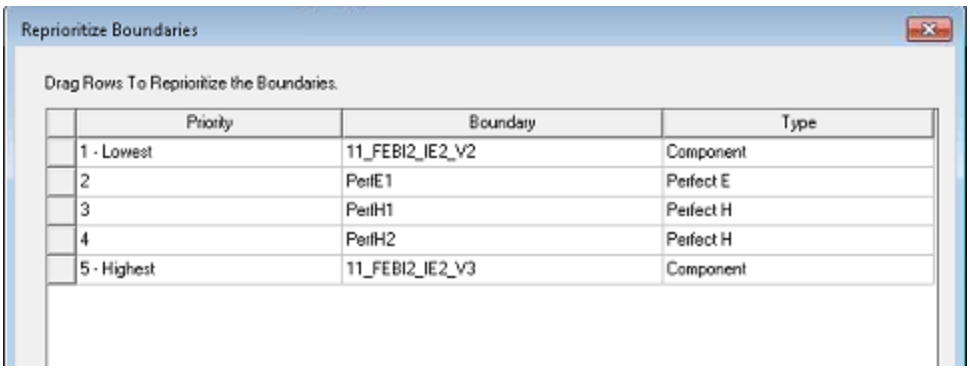


Boundaries defined inside the component do not appear in the Project Manager. However, boundary properties can be seen when you select the component in the Project Manager and view the component properties under the *Component Data* tab of the docked *Properties* window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



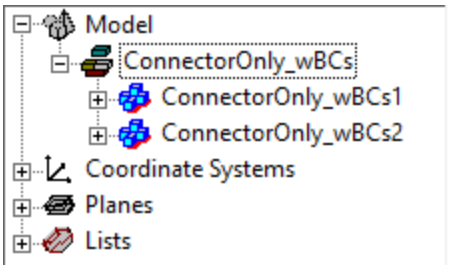
However, you can use **HFSS>Boundaries>Reprioritize** to set the priorities of all boundaries relative to non-component boundaries.



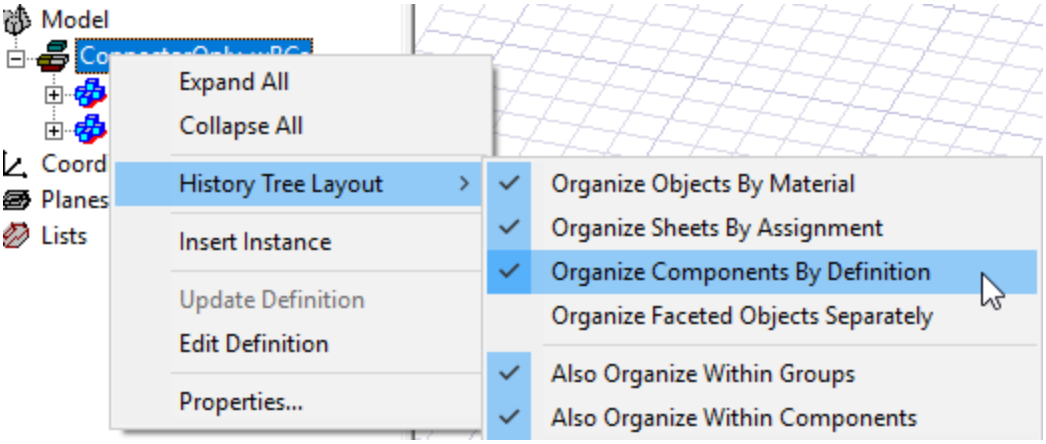
3D Components in the History Tree

Although Group Commands do not work with 3D components, you can group all instance of a components by the component definition in the History tree.

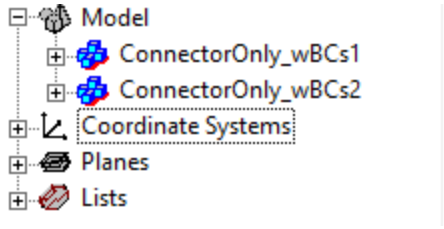
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To disable this grouping by instance, right-click an instance, and unselect **History Tree Layout>Organize Components by Definition**.

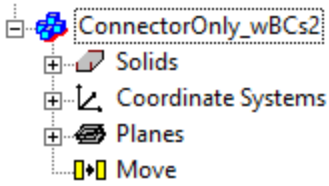


This causes the display to show numbered instances listed without the icon for the primary component.



Component Operations in the History Tree

- Component operations appear under that component folder toward the end.



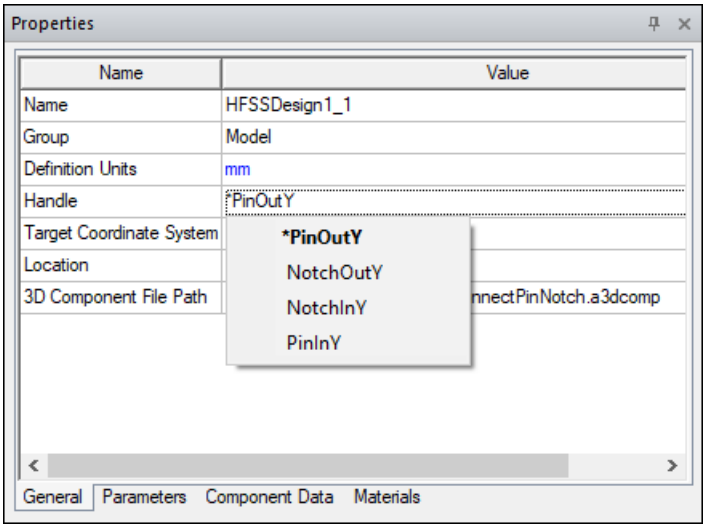
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Component operations acts on parts as well as CS.
- Boundaries, mesh operations are duplicated with duplicate and copy/paste operation.

Editing 3D Component Properties

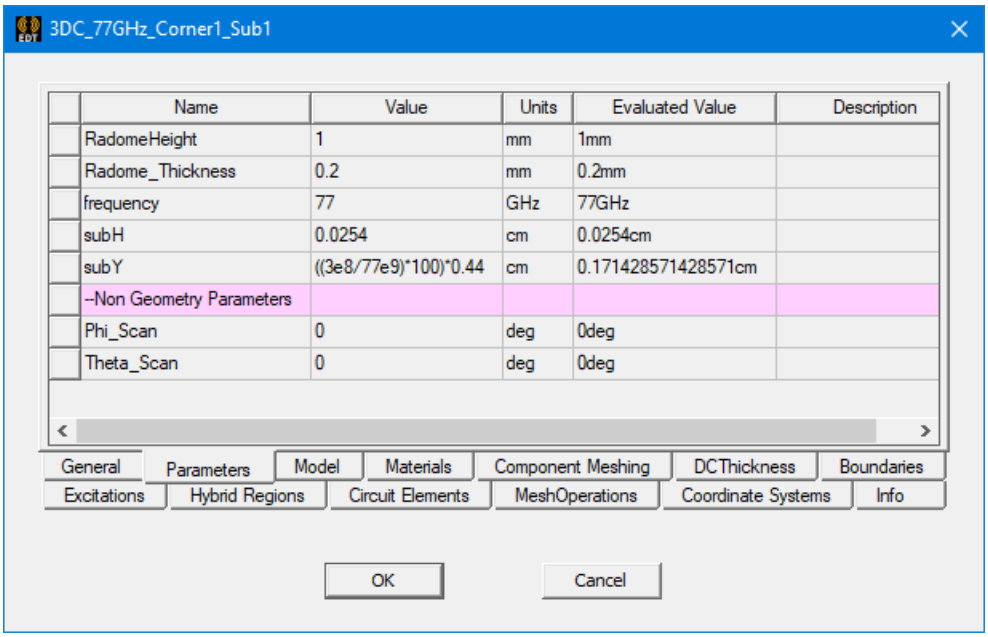
You can change and manipulate 3D components in several ways:

- Edit component handle and target CS in the General tab of component instance properties. All CS that are included in the component are available to be used as the handle. All coordinate systems in the assembly design that are eligible for component placement are available as target CS. Note that the default handle is shown with an asterisk.

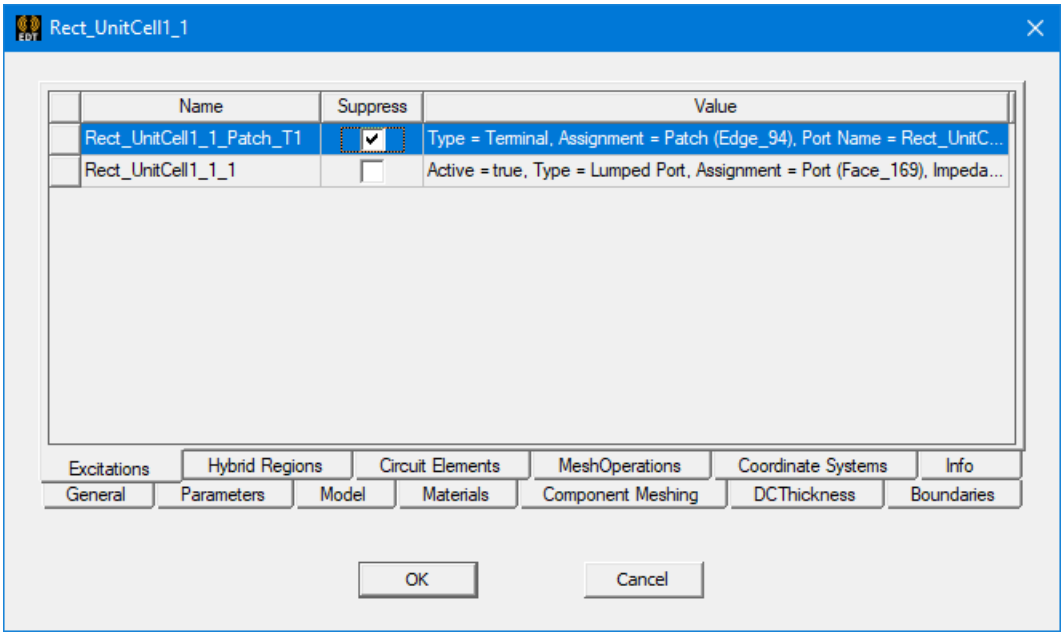


- Edit component parameters, if defined for that component. Editing the parameters of one component instance affects only that instance. If a component includes non-Geometry parameters, these are grouped separately on the Parameters tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



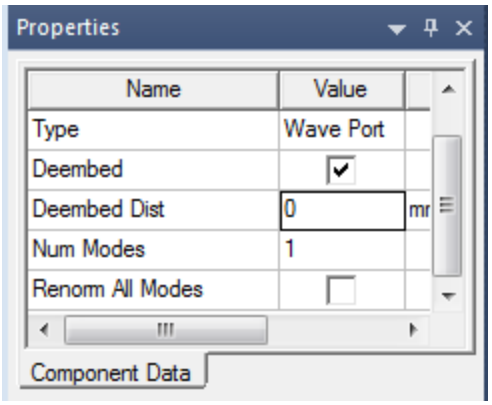
- Edit excitations by suppressing their use by means of the **Excitations** tab in the **Properties** window.



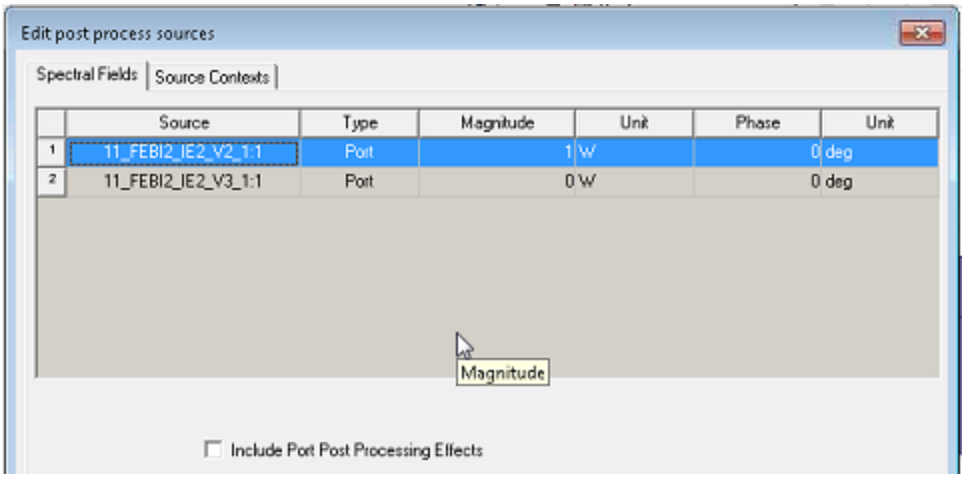
Checking Suppress in the Properties window removes the port on the selected instance from the Excitations list on the Project tree. Unchecking Suppress activates that port for the model and causes the port to appear in the Excitations list.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Edit excitation Post Processing such as de-embed distance, renormalization.



- Edit Sources for Post Processing



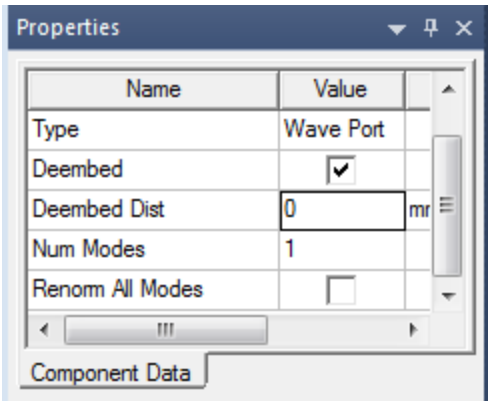
- Edit Port Impedance Multiplier value
- Edit Mesh operations coming from component

> **Note:**
>
> Operations on individual parts of a component are not allowed.

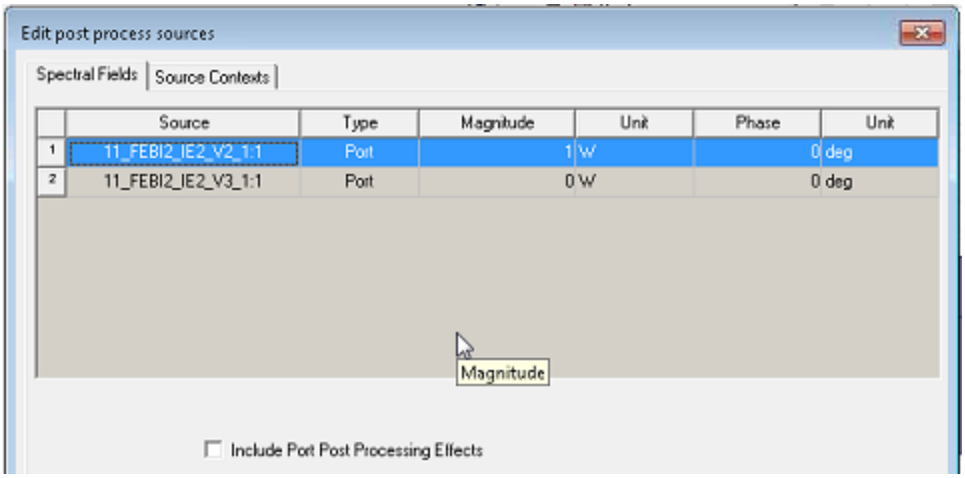
- Copy/Paste component
- Delete component
- Arrange (Move, Rotate, Mirror) component
- Duplicate component (Along Line, Around Axis, or Mirror)
- Edit Attributes for Display Wire frame, Color, and Transparency.
- Use **Edit** > **Select** > **Submodel** to quickly select component

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Edit excitation Post Processing such as de-embed distance, renormalization.



- Edit Sources for Post Processing



- Edit Port Impedance Multiplier value
- Edit Mesh operations coming from component

> **Note:**
>
> Operations on individual parts of component are not allowed.

- Copy/Paste component
- Delete component
- Arrange component: Move, Rotate, and Mirror
- Duplicate component: Along Line, Around Axis, and Mirror
- Edit Attributes for Display Wire frame, Color, and Transparency
- Use **Edit** > **Select** > **Submodel** to quickly select component

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

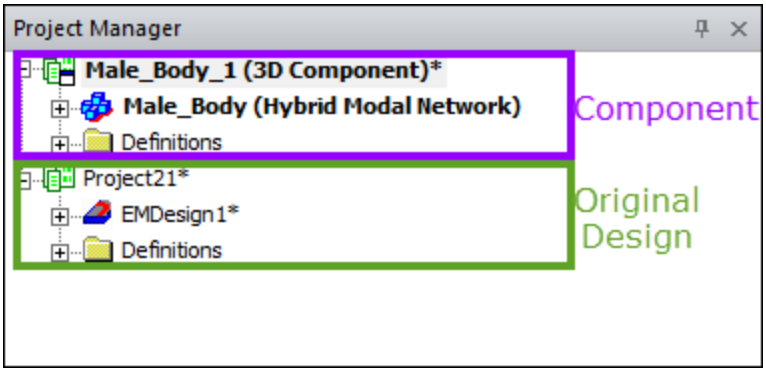
Editing a 3D Component Definition

> **Note:**
>
> If you reached this page looking for help on editing the definition of a **Layout Component**, as opposed to a *3D Component*, go here instead.

The **Edit Definition** feature allows the user to create new component definition using an existing, unencrypted 3D component file. This speeds up the task of creating new versions of components without having to start from the beginning. When **Edit Definition** is initiated, **Electronics Desktop** automatically creates a new project containing all 3D component data from the file in 'exploded' form. Component materials are added at the project level; variables are added at the project and design level; a detailed history is created for all objects in the component; and design data is added to the appropriate folders in the Project Manager.

> **Important:**
>
> Only 3D components that are **not encrypted** can be edited as described in this section.

The **Edit Definition** command creates a new project, named for the component, in the **Project Manager** window.

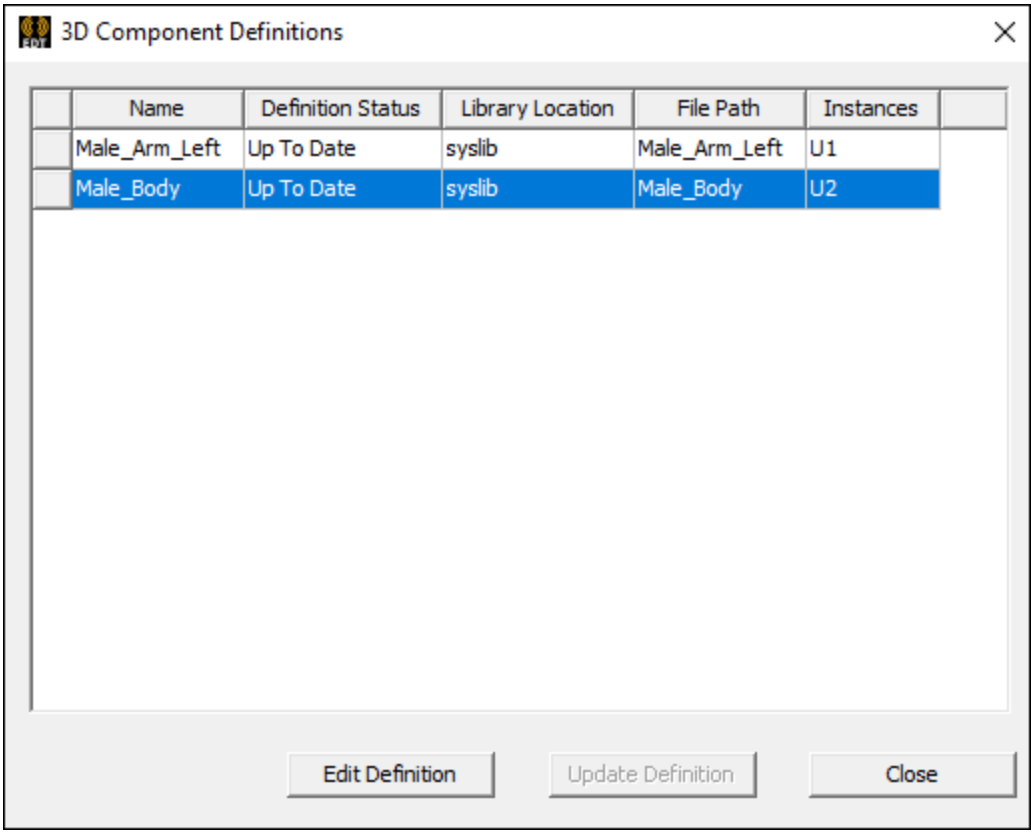


When the edited component is saved, an **Update 3D Component** window appears. From the **Update 3D Component** window, choose whether to save the edited component as a local instance or to the original library. To edit a 3D component definition:

1. Navigate to the **Edit Definition** window by following one of the following paths:
   - From the **History Tree**, right-click the component and select **Edit Definition**.
   - From the **History Tree**, right-click an instance under the component and select **Components** > **Edit Definition**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

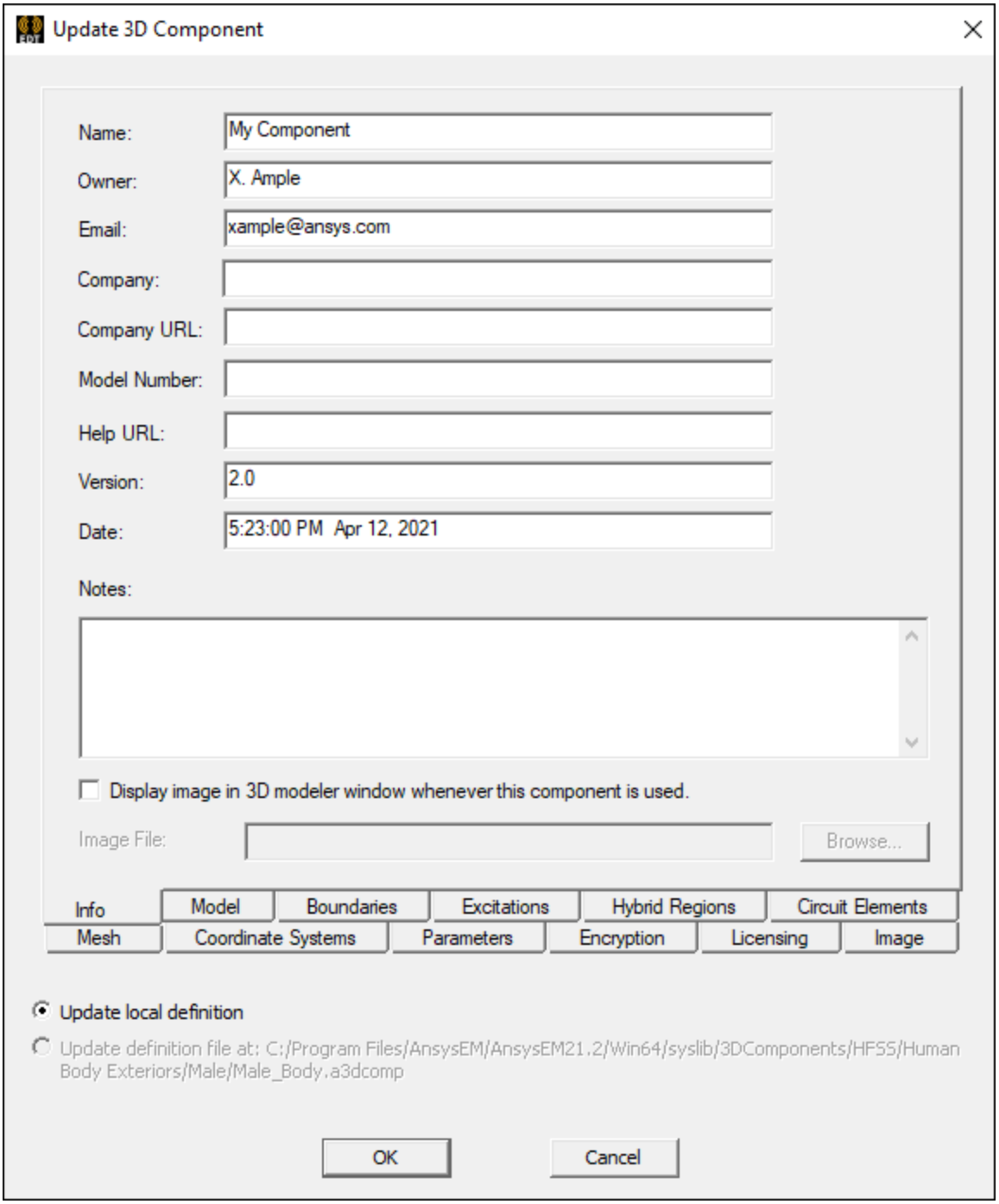
- From the **Project Manager**, right-click **3D Components** and select **Definitions** to open the **3D Component Definitions** window. From the **3D Component Definitions** window, select a component and click **Edit Definition**.



Regardless of which path is taken, a copy of the 3D component opens in the **Layout Editor**, and a new project folder appears in the **Project Manager** window, containing a copy of the original design.

2. Edit the component as desired.
3. To save the component as a new component (*.*a3dcomp*) or as its own **Electronics Desktop** (*.*aedt*) file, click **File** > **Save As**. Specify a file name and destination and click **Save**.
4. To save the component in the current project or for all projects, click **File** > **Save** to open the **Update 3D Component** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This window is populated with all entities (objects, boundaries, excitations, etc.) in the component design.

5. Use the radio buttons to select one of the following options:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Update Local Definition** – updates the component definition for use in the original design only.
- **Update Definition File At <Path>** – updates the component in the component library, affecting all other instances.

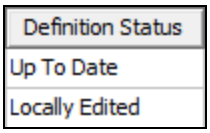
6. Click **OK**.

Verifying Updates to Components

To verify that a component has been edited:

1. Access the **3D Component Definitions** window:
   - From the **Project Manager**, right-click **3D Components** and select **Definitions**.

   The **Definition Status** column shows "Locally Edited" for any altered components.



2. Click **Close** to exit the window.

Updating Definitions

Once the updated component is saved, return to the original design and update all instances of that component. See: Updating 3D Component Definitions.

> **Note:**
>
> Definition update is not automatic even if **Edit Component** was initiated from that design. Instead, when an assembly design becomes active, a dialog box appears, prompting the user to update the 3D component definition. Then choose to update the definition from the prompt.

To update a component's definition:

1. Access the **3D Component Definitions** window:
   - From the **Project Manager**, right-click **3D Components** and select **Definitions**.

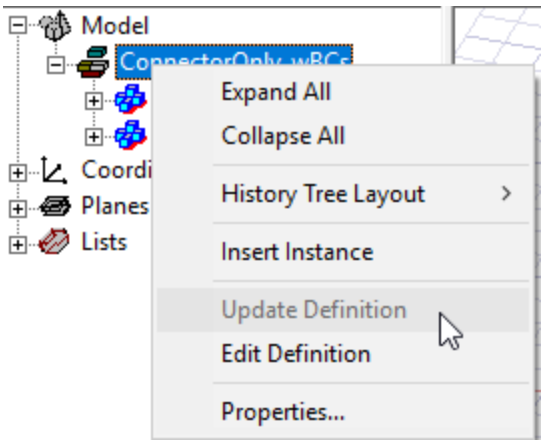
   Components with updates available display "Update Available" in the **Definition Status** column.

2. Select the desired component and click **Update Definition**.
3. Click **Close** to exit the window.

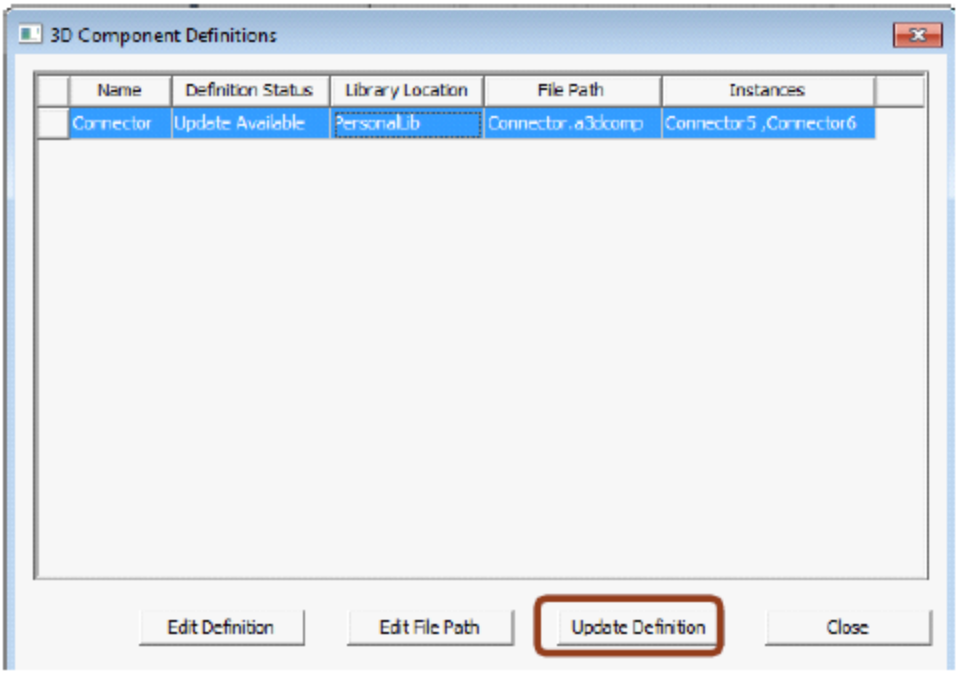
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Updating a 3D Component Definition

If you create or obtain a new version of a component, and then open a project that contained an earlier version of that component, you can right-click the component in the History tree and select **Update Definition** to replace the older version of the component with the newer one This updates all the instances of the component using that definition. Update Definition is unavailable if a newer definition is unavailable.



You can also accomplish this by right-clicking on the 3D Component icon in the Project Tree and selecting **Definitions...** from the shortcut menu. This displays a dialog listing available component definitions. The Definition status column lets you know if an undated definition is available.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All instances and component operations in the design are updated:

- Parameter values are preserved.
- New parts are generated in all instances.
- Deleted parts are deleted from all instances.
- Edited parts are changed in all instances.
- Design data, materials, coordinate systems can also be updated.
- Undo is supported; command is scriptable.
- The Definitions dialog box provides complete list of definitions in the design.
- Definitions can be updated from the dialog box.

Legacy Projects with 3D Components

Components created in earlier projects do not have the same degree of encapsulization and are not compatible with the current version and Ansys Electronics Desktop.

Legacy Projects: Do not encapsulate components

Layout Components [Beta] – Topic Index by Product

Support for layout components is a beta feature available in HFSS, Maxwell, and Mechanical designs. It is also a native feature for modeling printed circuit boards (PCBs) in Icepak.

One or more *Layout Components* can be included in a design, each of which is based on a source *HFSS 3D Layout* of a PCB. Availability of the feature may be limited to specific solution types, depending on the product.

Product-specific *Layout Component* topics are provided in the help. Click the appropriate link below for the product and topic of interest:

- **HFSS:**
  - Layout Component in HFSS 3D (Introduction)
  - Layout Component in HFSS 3D Process Flow
  - Using Layout Custom CS for Placement in HFSS 3D
  - Editing a Layout Components Definition
  - Replacing or Editing Layout Components
  - Visualization for Layout Components in HFSS 3D Modeler
  - Flex PCB Component Support in HFSS 3D
- **Icepak:**

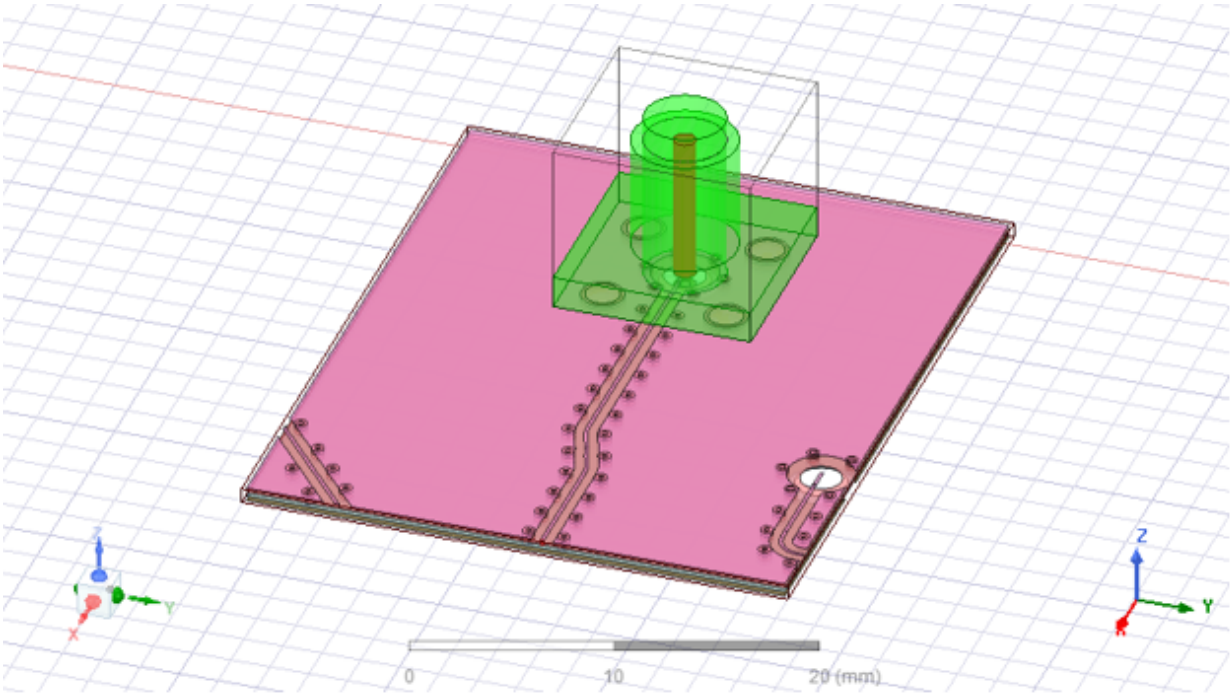
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - Layout Components in Icepak (Introduction)
    - Adding a Layout Component
    - Inserting a Layout Component Instance
    - Editing a Layout Component Definition
    - Replacing or Editing Layout Components
    - Layout Component Properties
- **Maxwell:**
    - Using a 3D Layout Component
    - Using Layout Custom CS for Placement
    - Editing a Layout Component Definition
    - Replacing or Editing Layout Components
    - Visualization for Layout Components in 3D Modeler
    - Flex PCB Component Support
- **Mechanical:**
    - Layout Components in Mechanical (Introduction)
    - Adding a Layout Component
    - Inserting a Layout Component Instance
    - Editing a Layout Component Definition
    - Replacing or Editing Layout Components
    - Layout Component Properties
    - Layout Component Properties: Visualization Tab

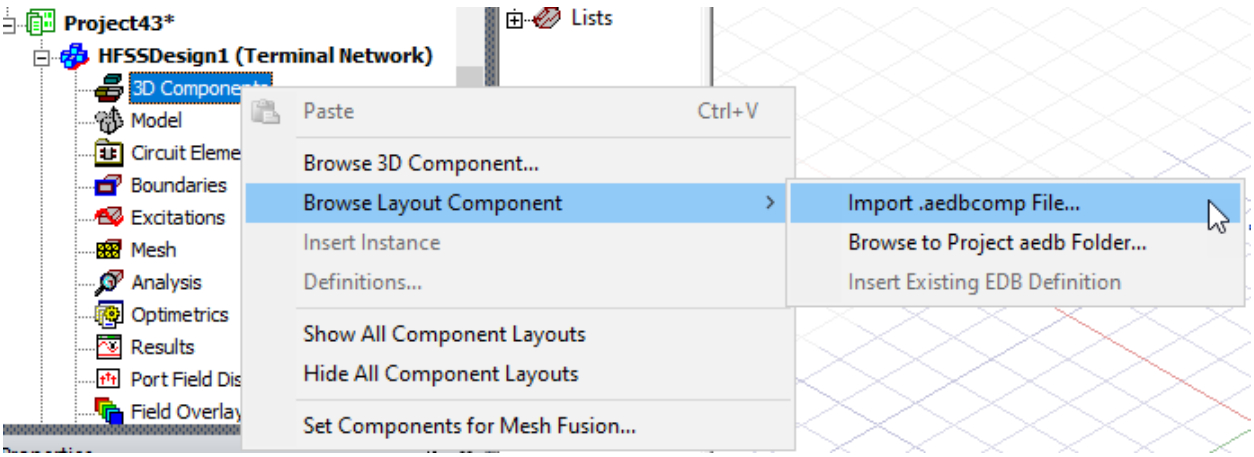
Layout Component in HFSS 3D (Beta)

A Beta feature lets you import HFSS 3D Layout designs as components into HFSS 3D environment. You can have a complex layout component interact with the surrounding 3D geometries without the burden of using a 3D geometry exported from HFSS 3D Layout. This feature allows great scalability in having a complex layout geometry sit in an EMI/EMC 3D environment.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The layout component can be imported from an aedbcomp file, Project aedb folder, or EDB definition to an HFSS 3D design with terminal solution type.



Layout Component in HFSS supports relative CS placement, move, rotate, etc. Once imported, its ports will be created in the project tree The bounding box and ports have actual HFSS 3D geometry while others are not selectable and are only for visualization.

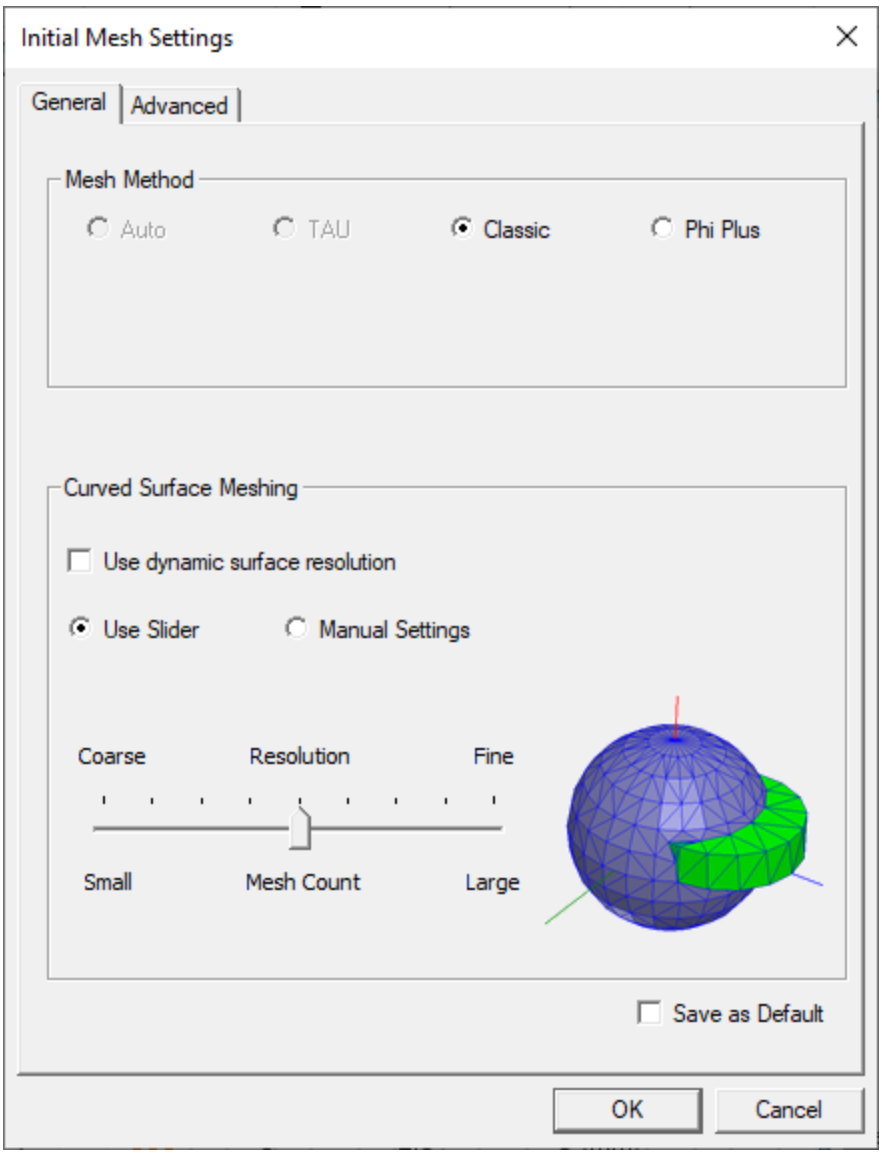
If the original layout design includes additional CS, these are included as properties to the layout component in HFSS 3D. Then as you select the original layout design to import to HFSS you choose a custom coordinate system as the reference CS. After creation in HFSS, you choose a

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

target CS for the layout component, and the component will be moved to a place where its reference CS snaps to the target CS. This can simplify placement in HFSS.

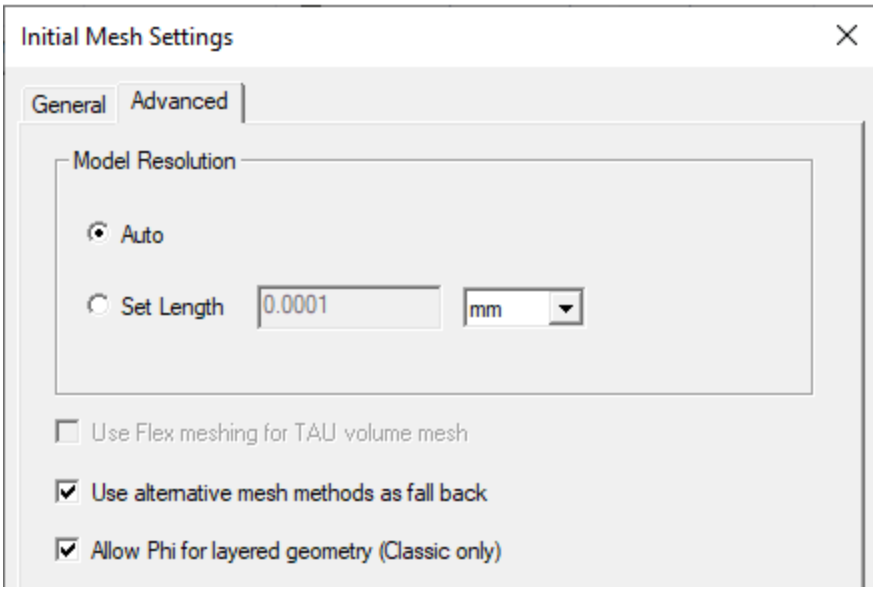
Phi mesher is optimized for speed by generating the mesh in a layered structure. When a layout component marked as Mesh Fusion is meshed using Phi mesher, it usually has this layered structure on its four sides. The simulation currently cannot handle the case where another mesh fusion component is in contact with such a layout component on its sides.

If the HFSS design has layout components meshed in native (not marked as mesh fusion), its Initial Mesh Setting dialog is modified. The dialog will disable the Auto and TAU mesh method and add an additional mesh method called Phi Plus.



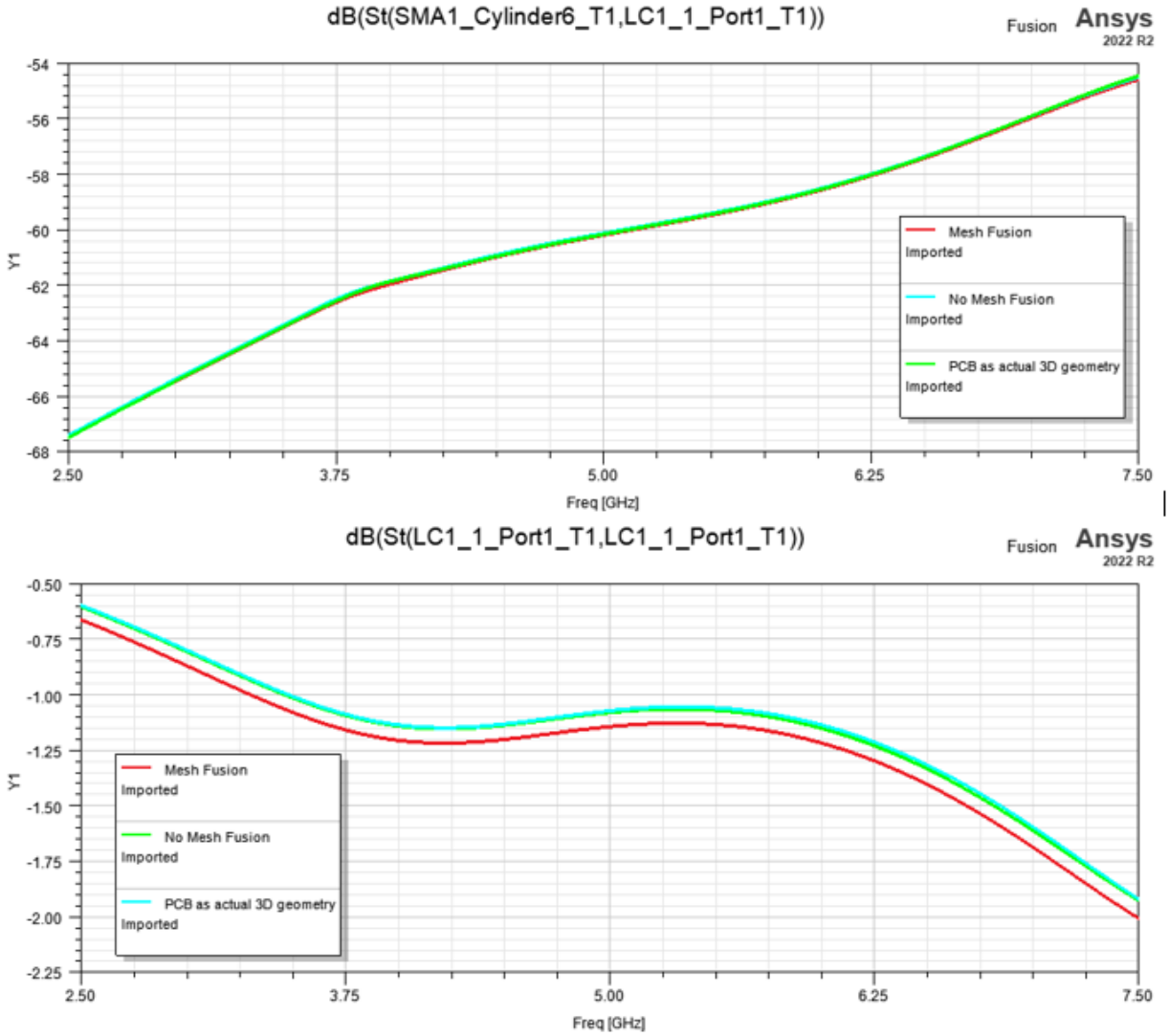
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The options related to TAU in the advanced setting page are disabled.



A comparison of an S-Parameter plot is shown below among 3 different cases: 1. Mesh each component using mesh fusion. 2. Mesh everything together. 3. Import the actual geometry of the layout design to HFSS directly instead of importing it as a layout component.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Supported functionalities for Layout Component in HFSS 3D include:

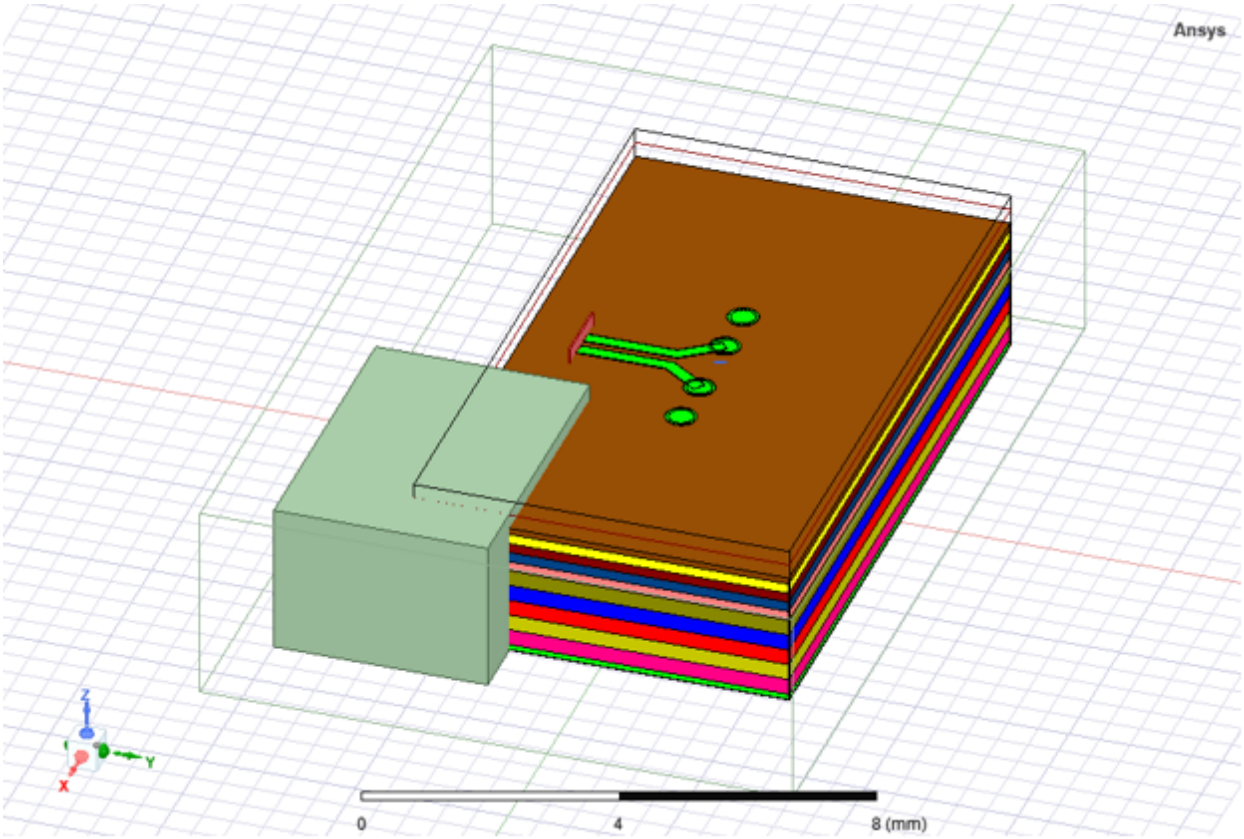
- Normal simulation workflow: Run simulation and view matrix and field overlay
- Layout geometry visualization and option to change object visualization attributes.
- Flex PCB Component support for HFSS 3D.
- Visualize RLC elements in layout components the same way as they do in 3D layout, including RLC boundaries and components that are categorized as circuit elements.
- Parametric Analysis: Using design variable to control component
- Import of custom coordinate systems from layout component design to HFSS 3D.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Intersection Validation
    - Between MCAD geometry and layout component
    - Between different layout components
- Show Nets for Layout Components in HFSS 3D
    - Only MCAD nets are shown now, which may connect through ECAD conductors
    - Net attributes can be view through component object attributes
- Port Display
- Mesh Link
- FEBI/Hybrid Regions
- Can use non-model sheet or cut plane to plot in layout component
- Q3D DC Solve. You also choose Q3D solver to solve designs with layout components at low frequency.
- Support simulation with encrypted 3D component. Field and mesh plots are hidden inside encrypted 3D components. Any intersection or containment between encrypted and unencrypted components results in an error message.
- Full Mesh Fusion support
    - Full interaction with other layout components, 3D components and native geometry
    - Special mesh method option (classic and phi) specific to layout component.
    - Enable layout components to be marked as priority if they are also marked as mesh fusion
    - Record imprint steps to a script file to reproduce the same mesh in the future, similar to that in mesh fusion for 3D components
    - Support parallel meshing
- Priority Component. Layout components marked as Mesh Fusion also support the priority option. The priority option provides a way to resolve component intersection conflicts with native objects. If such intersection appears, the intersecting portion will be subtracted from the native object, so that the components marked as priority stay intact and the simulation could continue.

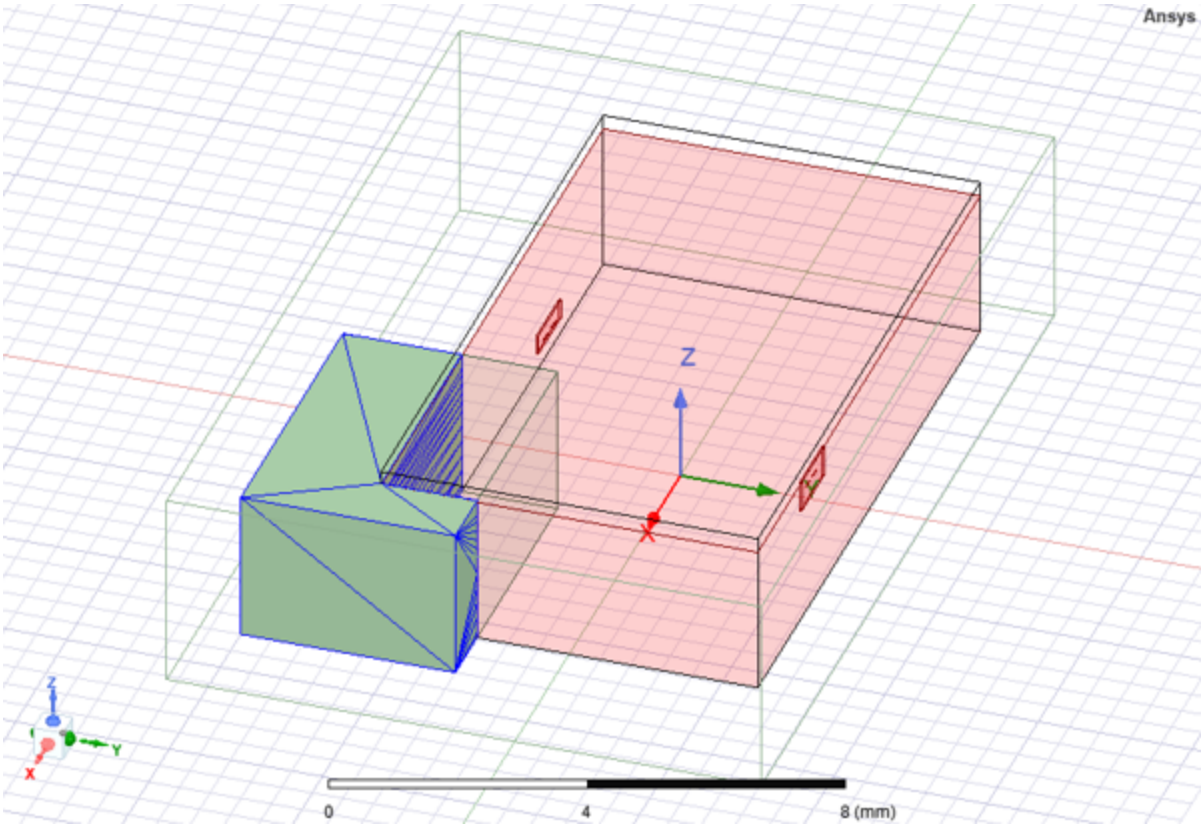
  For example, consider a design with layout component being intersected by a native box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



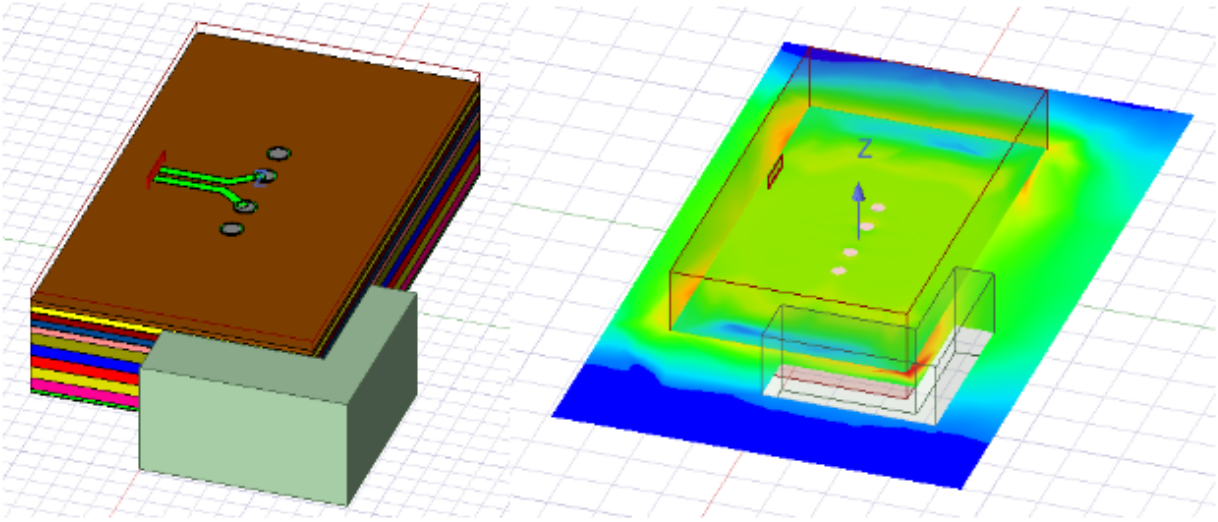
If the priority option is not marked for this layout component, the simulation reports an error. If the component is marked as priority, then the intersecting portion is subtracted from the native object, and the simulation could complete. You can plot mesh on the native objects to confirm that the subtraction is applied, and the mesh for the layout component marked as priority is intact.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- During post processing, when generating field or mesh plots, anything on the interior of encrypted components will not be shown to maintain data security. The figures below show an example project with both layout components and encrypted 3D component (left) and its resultant field plot (right).

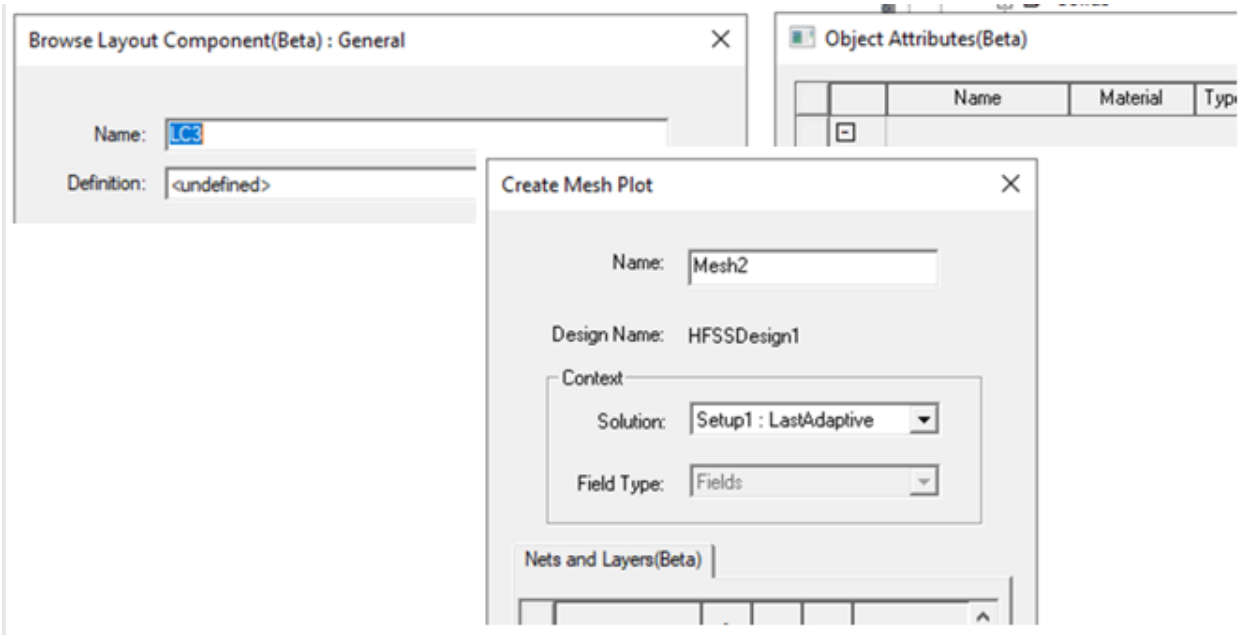
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Limitations:

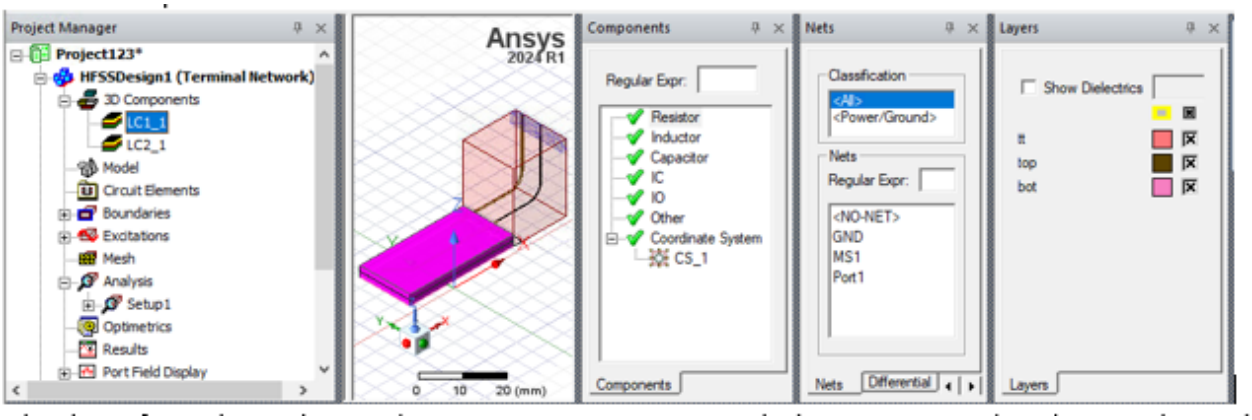
- Single top design EDB only
- No Derivative support

Because the layout component feature in HFSS 3D is changed to exposed beta, all dialogs related to layout component feature Beta text in the title. This includes the Browse Layout Component dialog, Object Attributes dialog, and Nets and Layers selection sub-dialog.

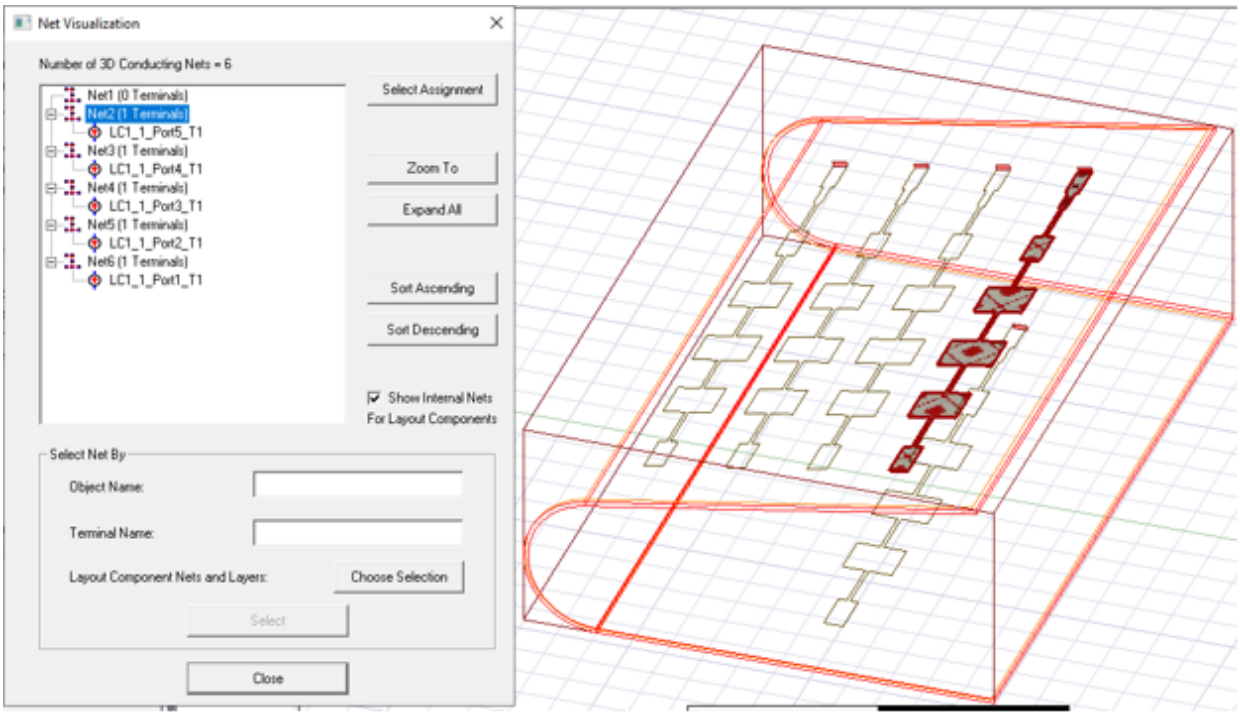


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When a layout component is selected in the project tree or history tree, the Components/Nets/Layers tabs will display the information about the component in the same way as they do in a 3D layout design for that component.



Finally, in the **Net Visualization** dialog, edge ports are correctly identified as part of the nets formed by ECAD and MCAD geometries.



Related Topics

Using Layout Component in HFSS 3D Process Flow

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Visualization for Layout Components in HFSS 3D

Visualization for RLC Boundaries and Component Groups Designated as Circuit Elements

Replacing or Editing Layout Components

Using Layout Custom CS for Placement in HFSS 3D

Flex PCB Components in HFSS 3D

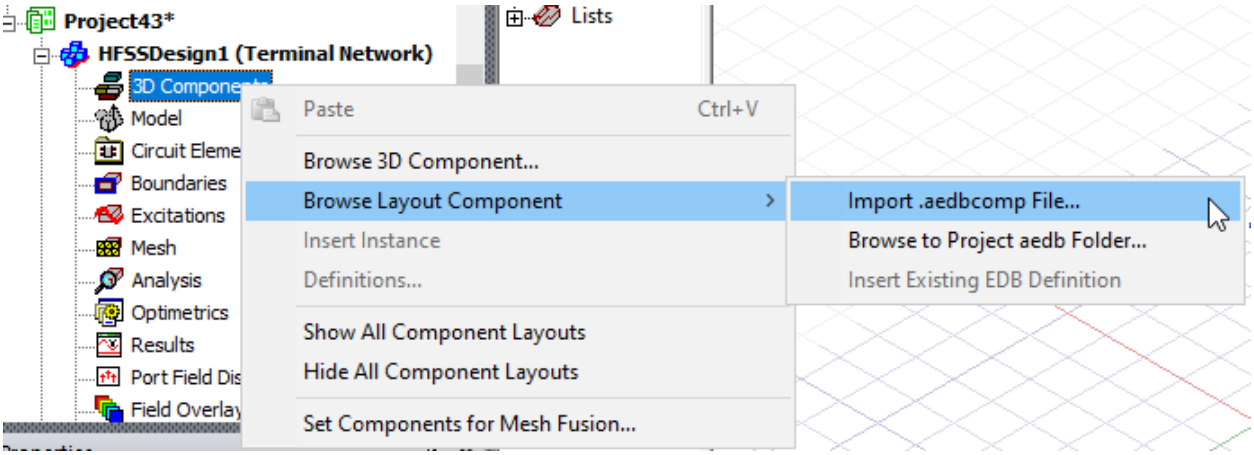
Field Plots on Layout Component's Layer or Net

Using the Fields Calculator with Layout Component in HFSS 3D

Plotting the Mesh for Layout Components

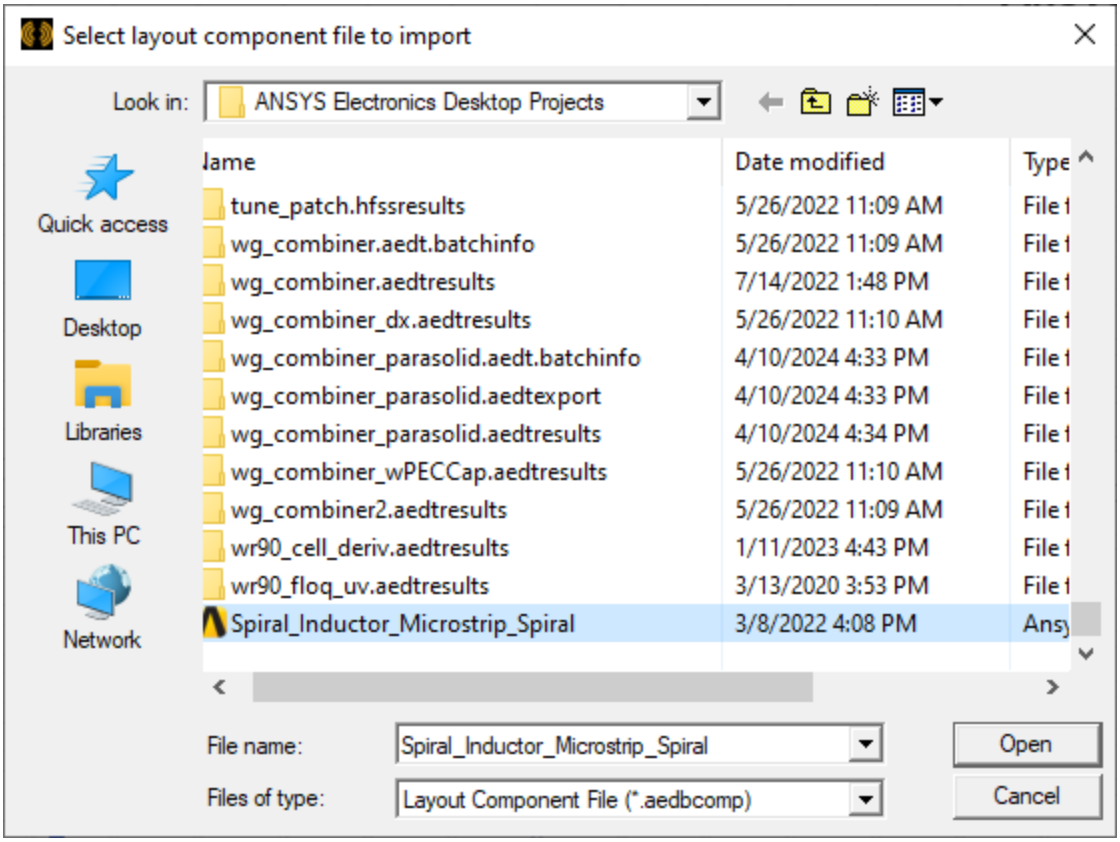
Layout Component in HFSS 3D Process Flow

The layout component can be imported to HFSS 3D from an .aedbcomp file, Project aedb folder, or EDB definition to an HFSS 3D design with terminal solution type.



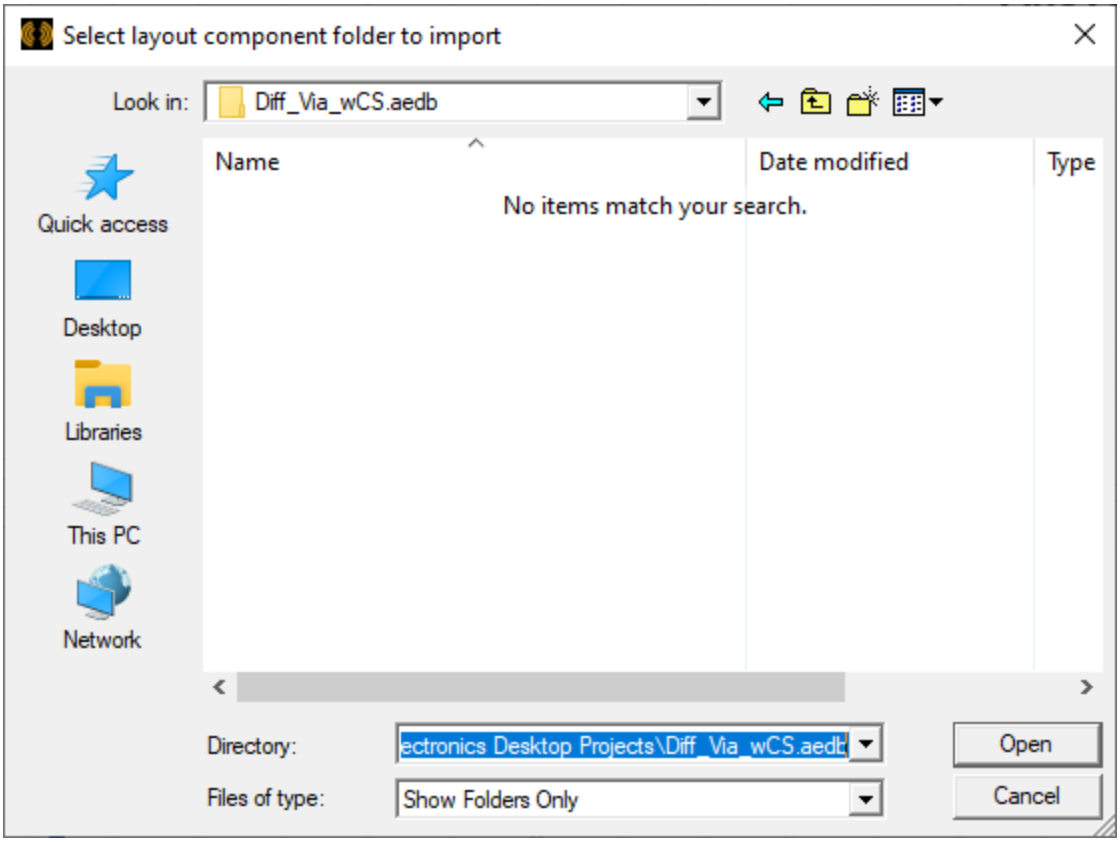
Import .aedbcomp File… opens a browser window allowing you to select an .aedbcomp file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

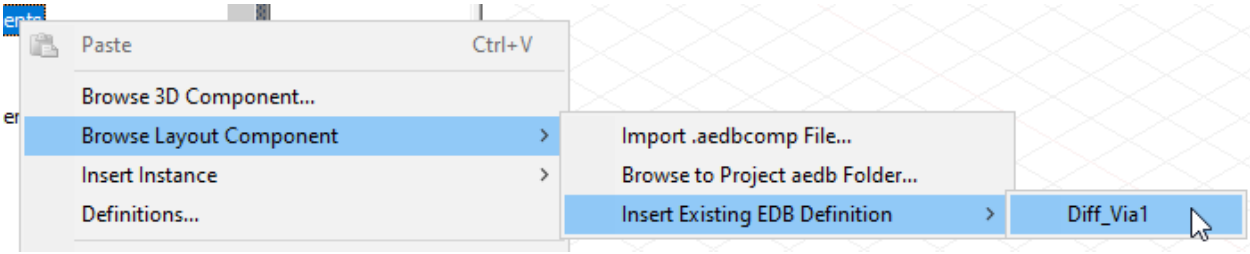


Browse to Project aedb Folder… opens a browse window allowing you to select an .aedb folder. Usually the *.aedb folder to import layout component should be a separate location that is independent of the target Ansys Electronics Desktop project file location. As the *.aedb is the location to store layout component data, this component may come from any where. If you attempt to import an encrypted component from the hierarchy of the current project, you will receive an error message. An encrypted component will not be in the first .aedb folder, but the one of the next two subfolders.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

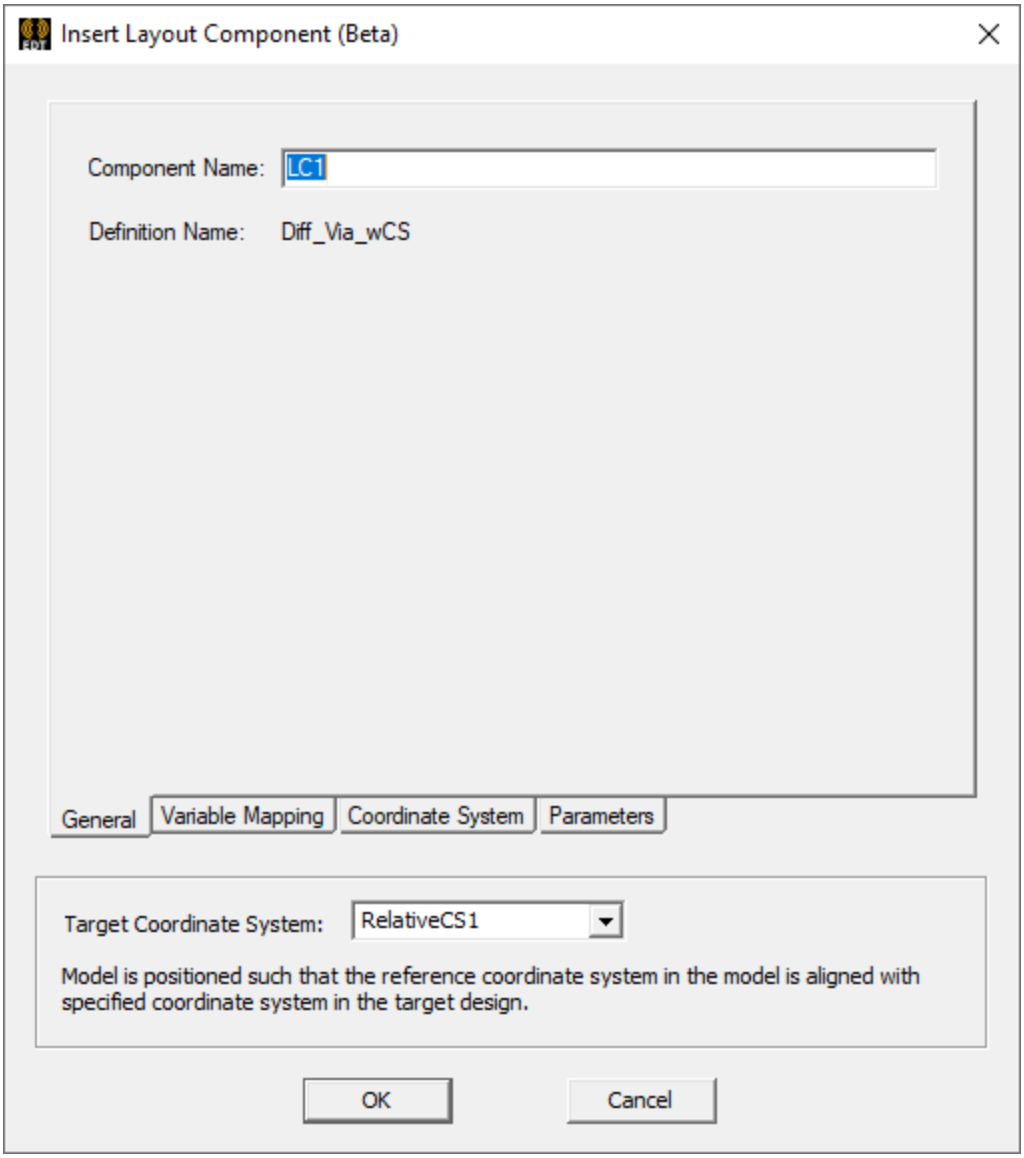


Insert Existing EDB Definition shows a list of existing EDB definitions available within the project.



Selecting an .aedbcomp file, .aedb folder, or an existing EDB definition opens an import layout Component dialog.

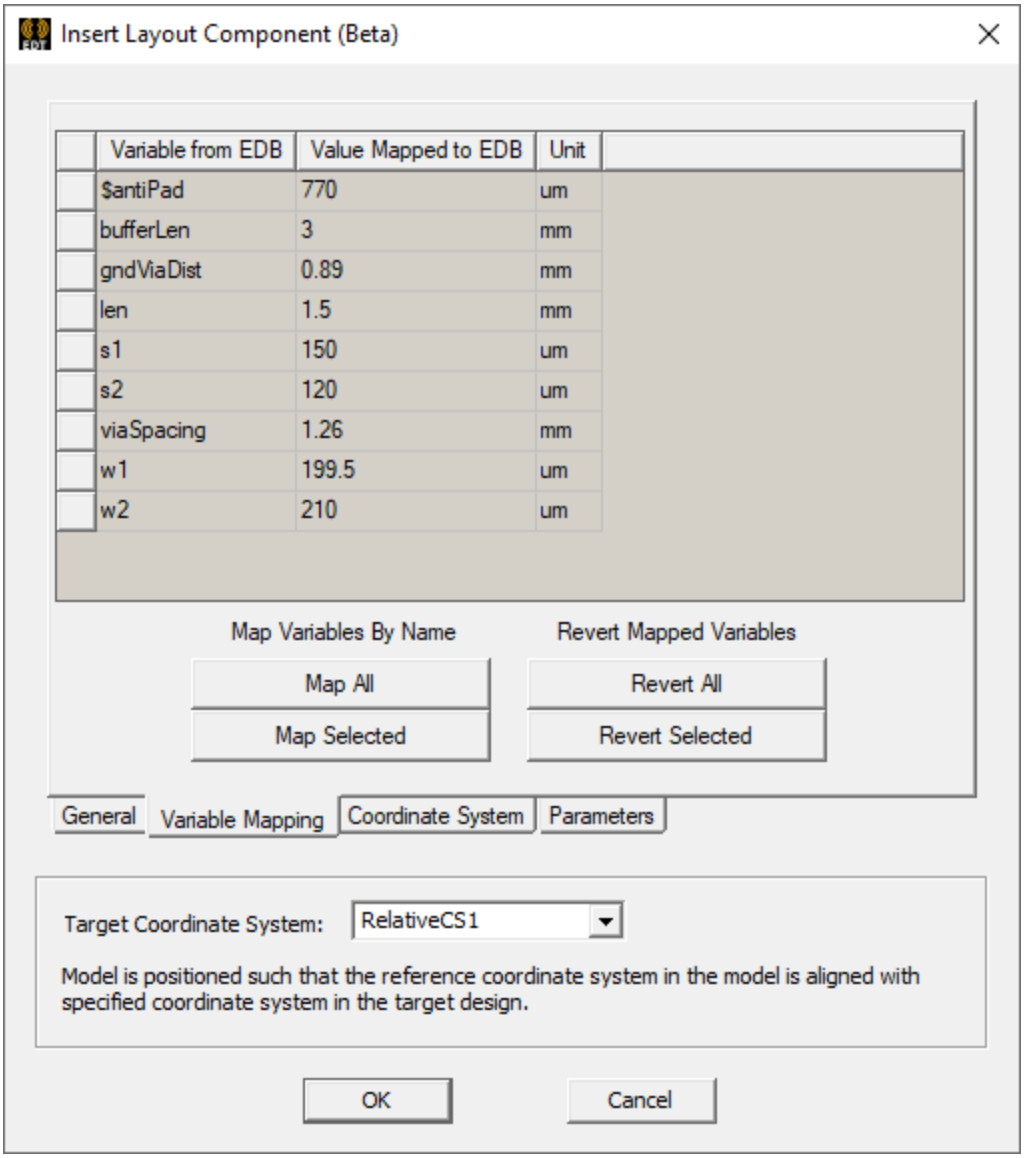
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **General** tab page allows you to specify a Target Coordinate system. If you have defined multiple coordinate systems, you can select from the drop down men.

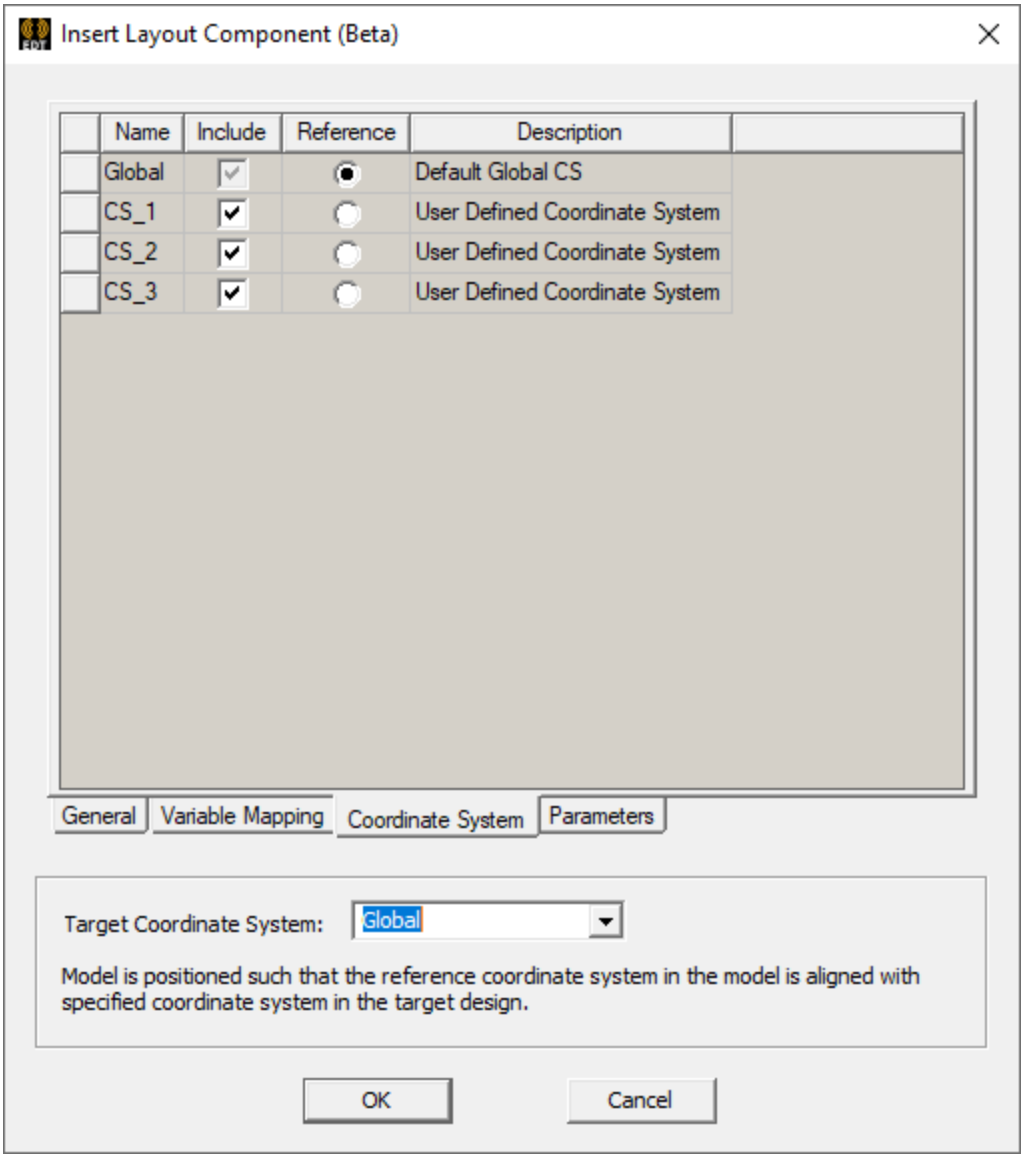
The **Variable Mapping** tab allows you do map any defined variables from the EDB to the instance being created.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



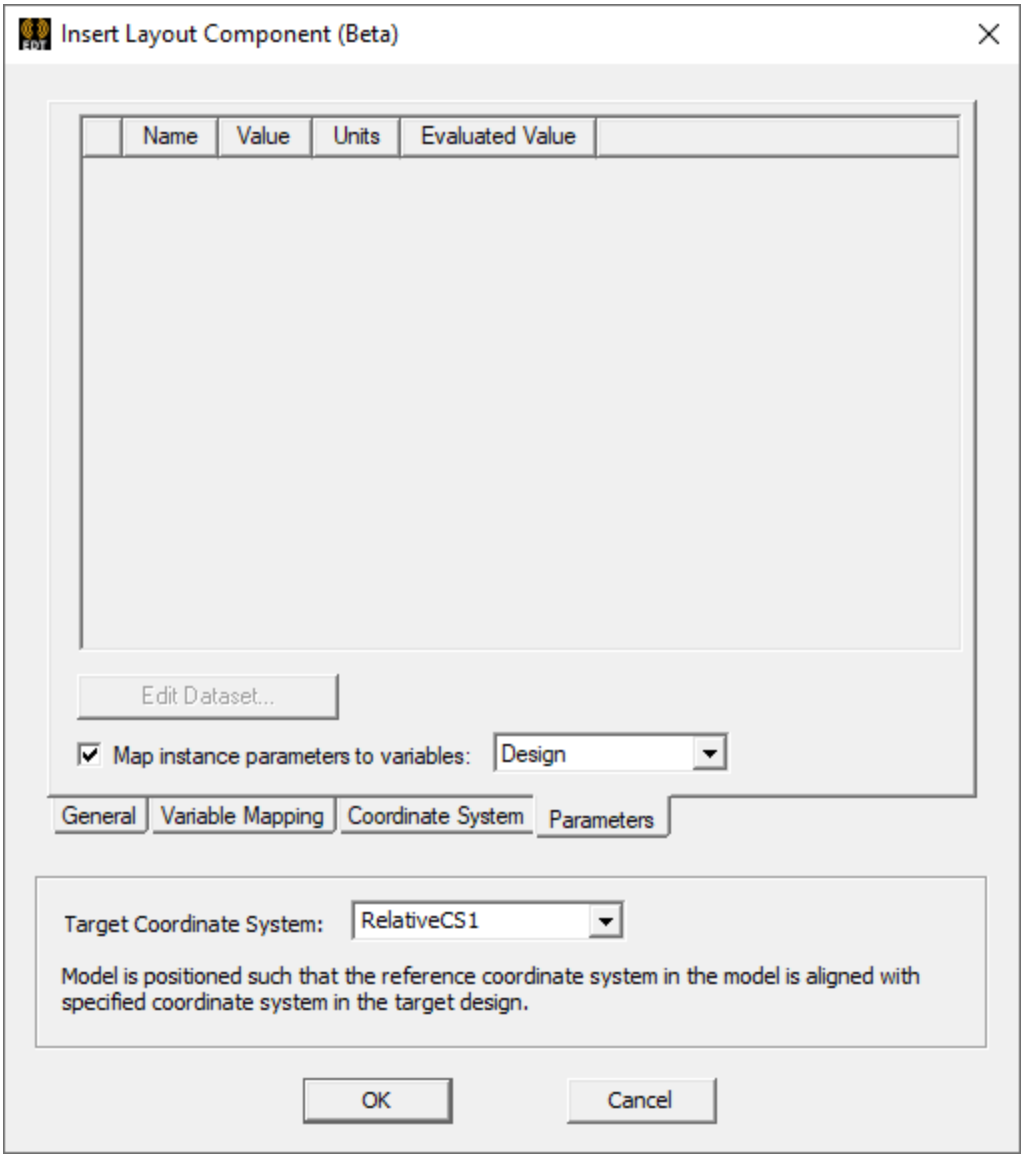
The **Coordinate System** tab allows you to consider descriptions for CS systems to include and reference.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Parameters** tab allows you to select and edit any existing layout component parameters and map them to any existing Design or Project variables.

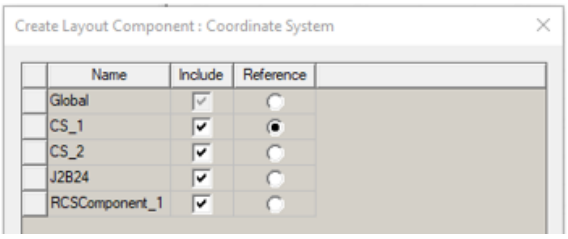
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



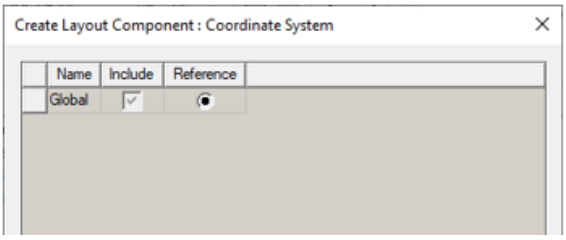
Layout Component in HFSS supports relative CS placement, move, rotate, etc. Once imported, its ports will be created in the project tree The bounding box and ports have actual HFSS 3D geometry while others are not selectable and are only for visualization.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

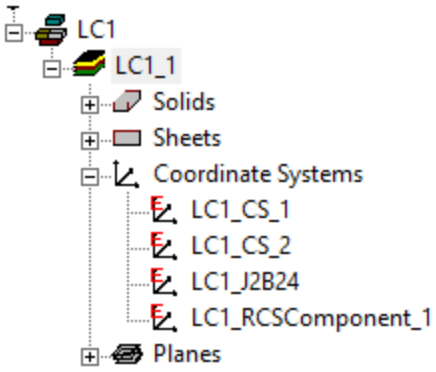
Layout Component with Custom CS



Layout Component without Custom CS

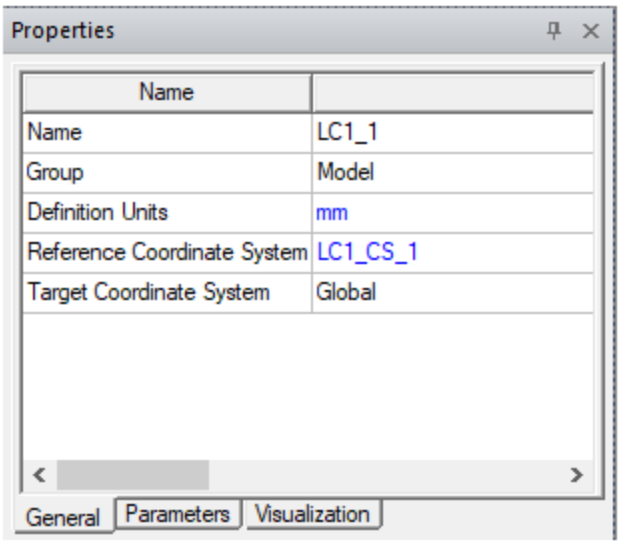


Because you can use custom coordinate systems defined in original Layout design to simplify placement in HFSS, you should consider creating them in Layout for this purpose. All coordinate systems of the selected component are extracted from the 3D Layout design and displayed in the page. The global CS is always displayed. You can select one or more CS in the “Include” column to import them to the HFSS design. The global CS and the CS that is selected as the reference CS are always included. Once a CS is selected in the “Include” column, each component instance of that definition will create such CS in its modeler tree, which is shown below.



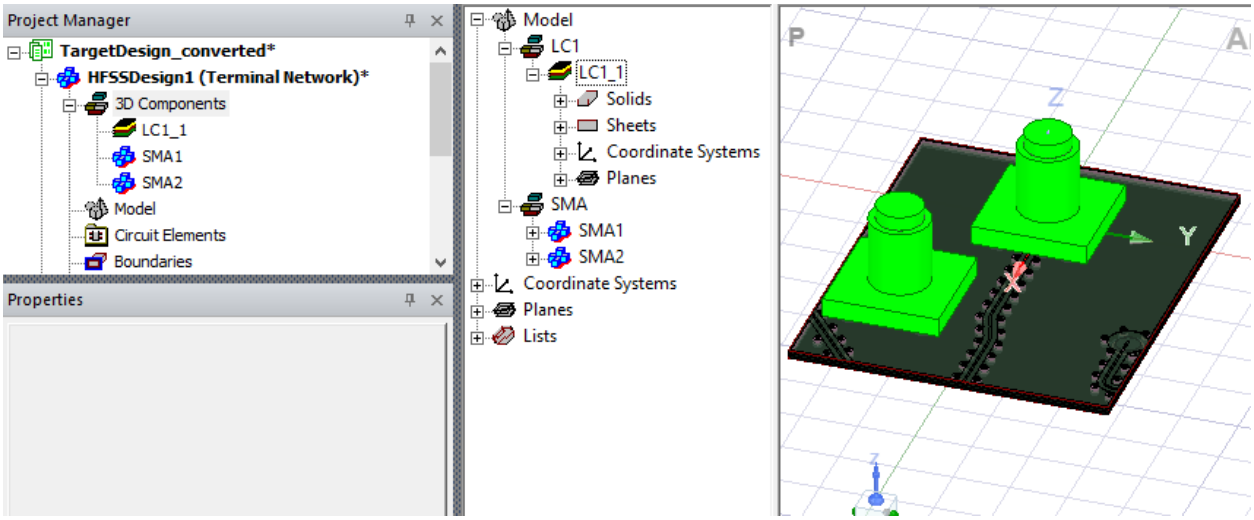
You can also select one CS from the table to be the reference CS by using the “Reference” column. By default, the CS closest to the center of the layout component is marked as the reference. Once the layout component creation/Editing is finished, the selected CS is displayed in the component’s general property window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



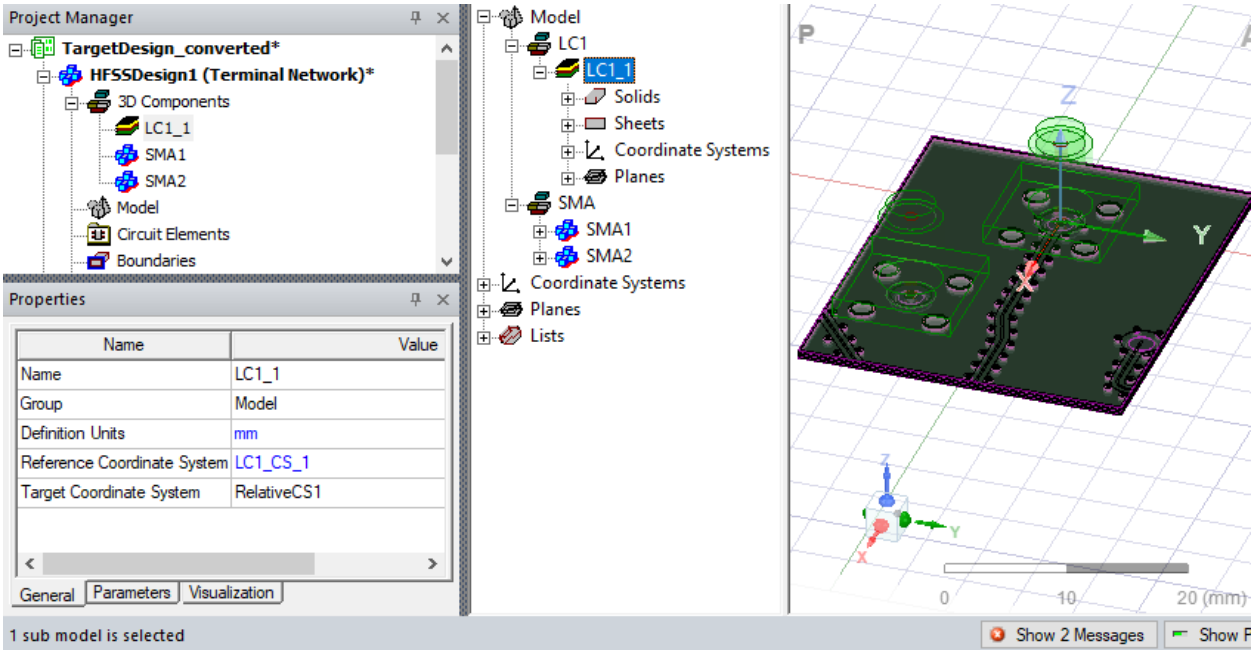
The "Layout Component in HFSS 3D Process Flow" on page 6-226 section provides an example use of this feature.

When you have added a Layout Component instance, it appears in the Project tree, the History Tree, and the Model window.



Selecting the component highlights it in the Model window, and displays its Properties window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Properties** window includes **General** tab, **Parameters** tab where you can edit instance values for variables you have created, and **Visualization**.

Related Topics

Layout Component in HFSS 3D

Visualization for Layout Components in HFSS 3D

Visualization for RLC Boundaries and Component Groups Designated as Circuit Elements

Show Nets for Layout Components in HFSS 3D Geometries

Replacing or Editing Layout Components

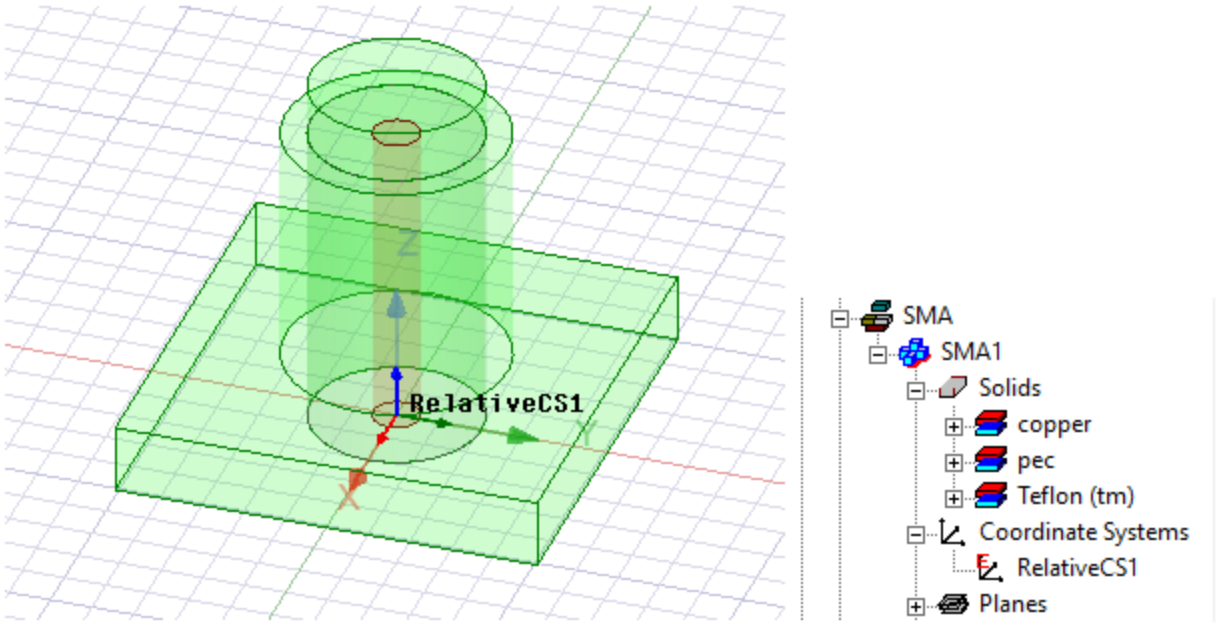
Using Layout Custom CS for Placement in HFSS 3D

Flex PCB Components in HFSS 3D

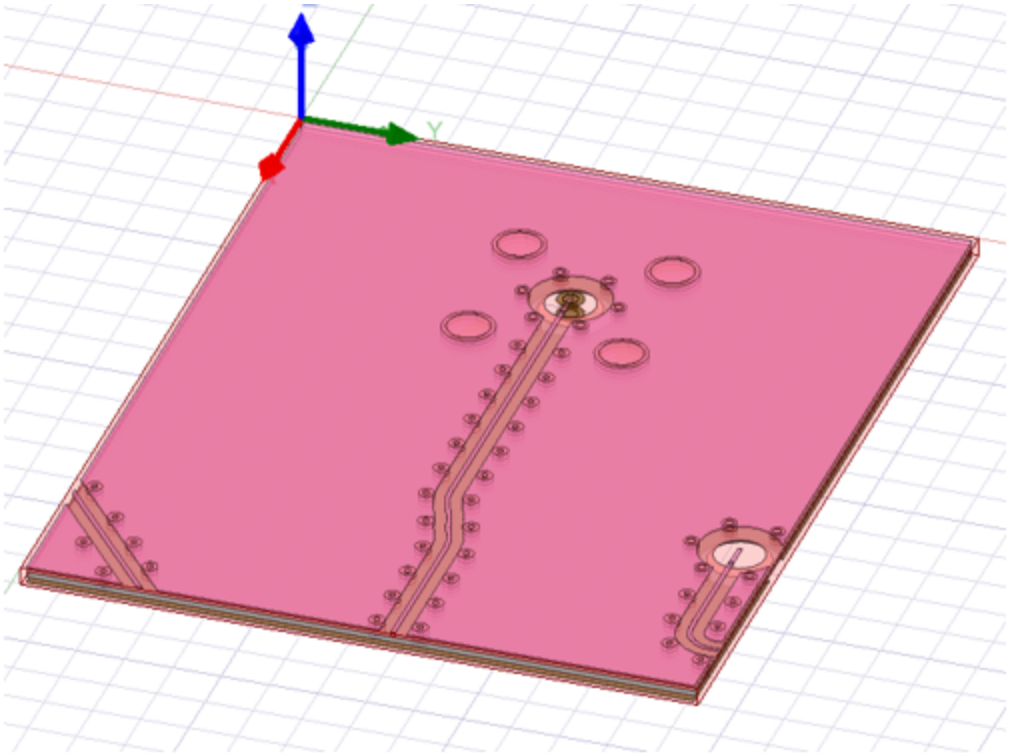
Using Layout Custom CS for Placement in HFSS 3D

The ability to define coordinate systems in layout facilitates placing a layout component on a desired location relative to other geometry. In this example illustrates a 3D component (shown in green) that comes with a CS called “RelativeCS1”. This CS is located at the bottom of the component and at the center of the internal cylinder structure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

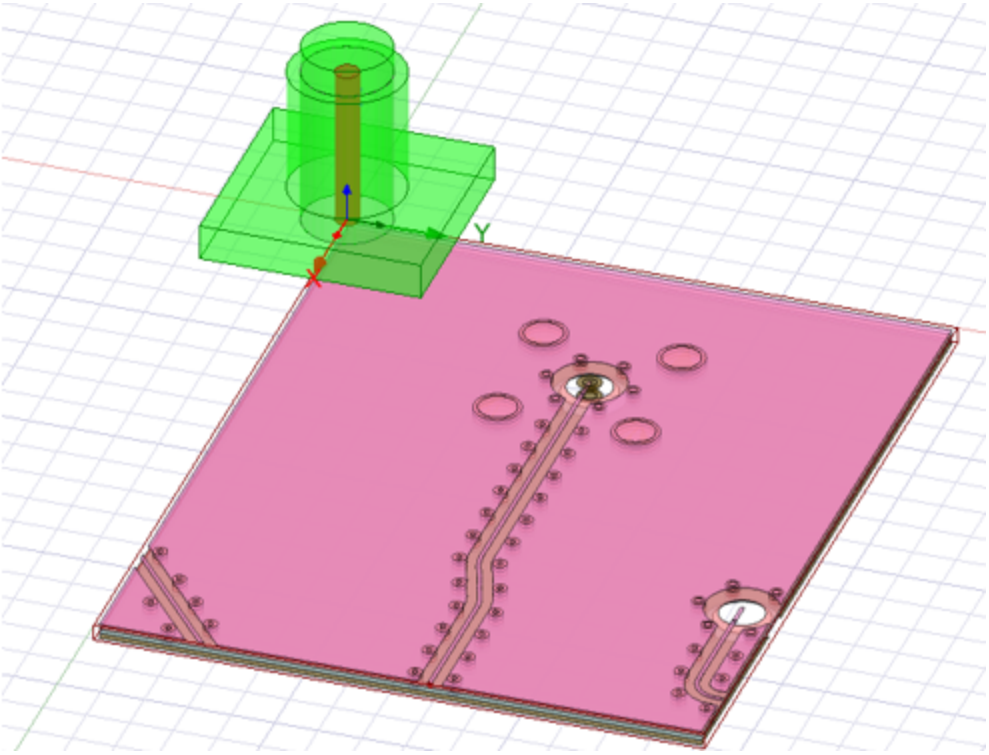


We also have a layout component shown in red. In the original layout design of this component, the global CS is located at one of the centers of the entire structure. Without this feature, when you create such a layout component in HFSS, its global CS will be used as the reference CS.



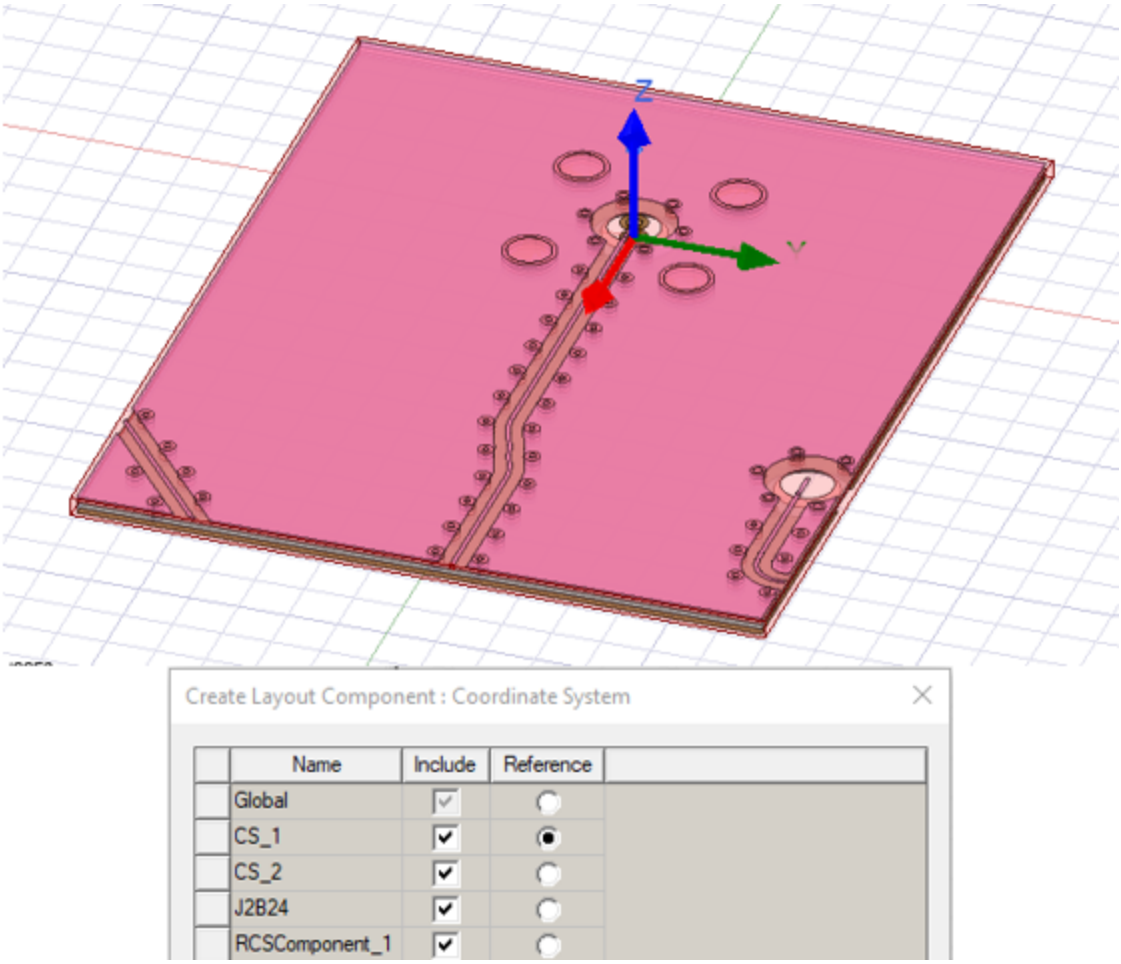
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this example, you want to place the layout component under the 3D component so that its holes (in the middle of the component) are aligned with the hole from the 3D component. When you import the layout component in Maxwell using its global CS as the reference CS and “RelativeCS1” as the layout component’s target CS, it snaps the layout component’s global CS to the target CS, which will look like the figure below.



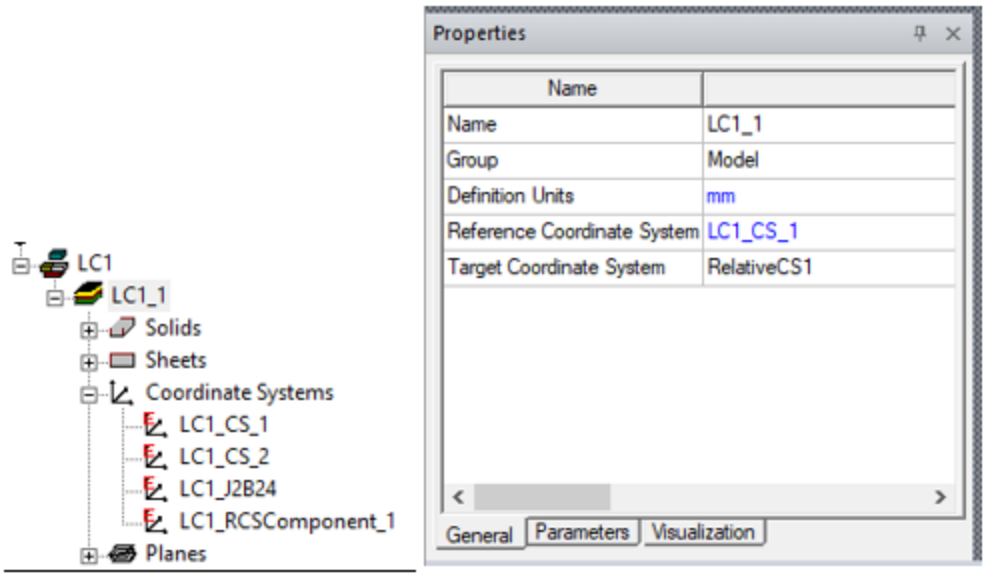
In the original layout design, there is another CS at the center of the holes called “CS_1." When you import the layout component, select this new CS as the reference CS.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

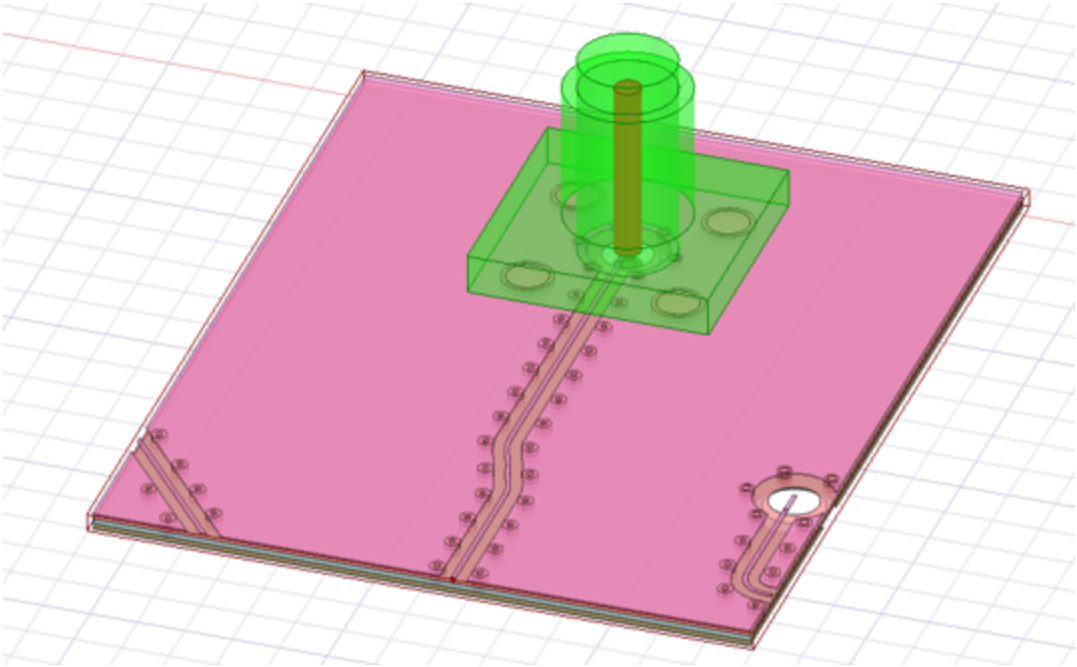


Using this new reference CS, the select the target CS as “RelativeCS1”.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The layout component is now in the right place where its reference CS “CS_1” snaps to the target CS “RelativeCS1.”.



Note: After import, you can edit the placement of a layout component; see Editing Placement of a Layout Component for details.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Related Topics

Using Layout Component in HFSS 3D Process Flow

Visualization for Layout Components in HFSS 3D

Visualization for RLC Boundaries and Component Groups Designated as Circuit Elements

Replacing or Editing Layout Components

Using Layout Custom CS for Placement in HFSS 3D

Flex PCB Components in HFSS 3D

Field Plots on Layout Component's Layer or Net

Using the Fields Calculator with Layout Component in HFSS 3D

Plotting the Mesh for Layout Components

Editing the Placement of a Layout Component

Once you have imported a Layout Component, you can move it using standard modeler operations. Modeler operations like **Move**, **Rotate**, and **Mirror** also apply to Layout Components. During these operations, snapping points can be helpful to set the start and end points of the move. Any visible object (layout component geometry and 3D geometry) will have snapping locations when in snap mode or when creating a CS. Note the following about layout component snapping:

- The snapping location of a layout component behaves the same way as that for a 3D component.
- All snapping types are supported, which include vertex, face center, edge center, edge quadrant, and arc center.
- If an object from a layout component is visible, its snapping locations will be turned on. Once the object becomes invisible, its snapping locations will be removed.
- Snapping locations are restricted to the top and bottom of layers.

To move a layout component, select one of its vertices snapping point as the start location and another 3D object's vertex as the end location. The layout component will be moved from the start to the end location.

To demonstrate, we will have the example shown below. The green component is a 3D component, and the pink component is an imported layout component. In this example, you want to place the layout component under the 3D component so that its holes (in the middle of the component) are aligned with the hole from the 3D component.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right click on the Layout Component, and select **Edit** > **Arrange** > **Move**.

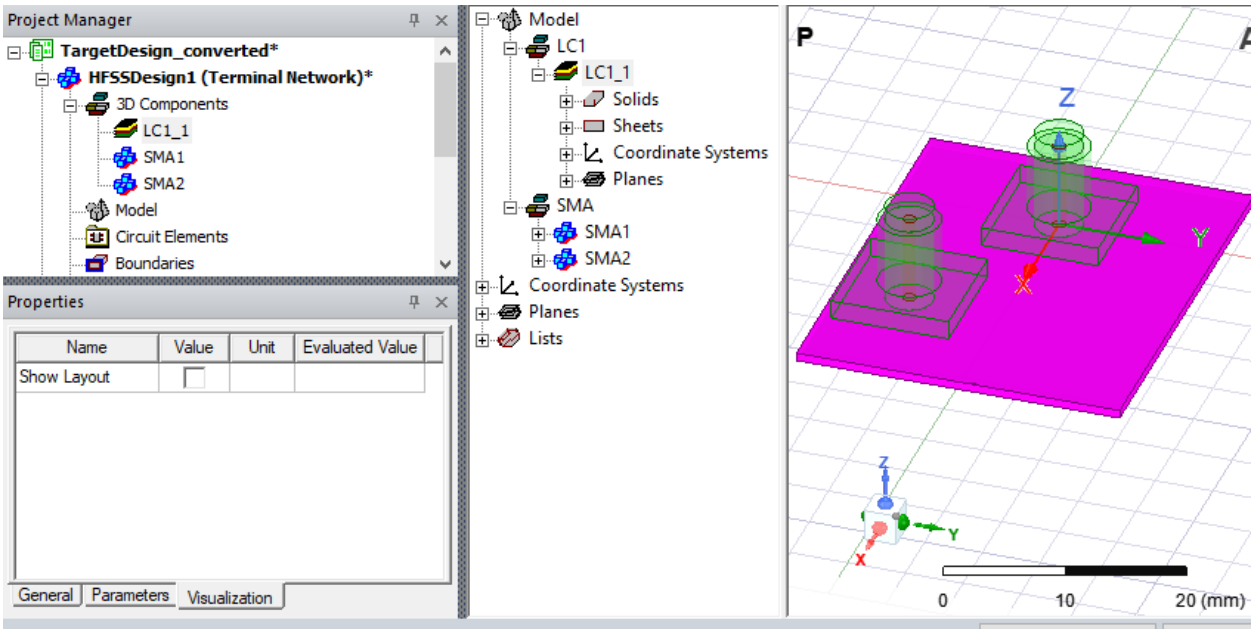
2. Zoom in on the center hole of the Layout Component, and a snapping point will appear. Click to select.

3. Drag your cursor to the snap location where you want to move the layout component, and then click.

The layout component is now located underneath the 3D component.

Visualization for Layout Components in HFSS 3D Modeler

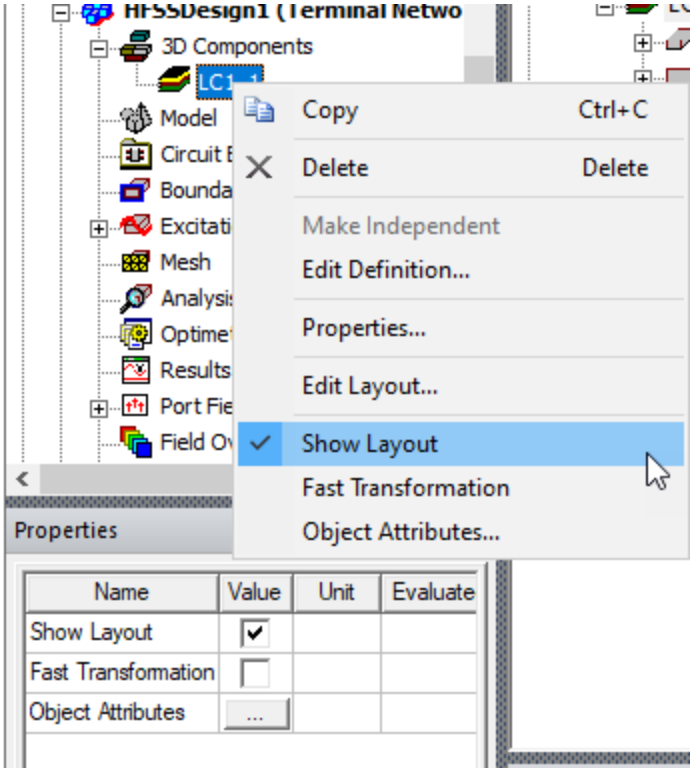
You can show each layout component in the 3D editor and specify the visualization attributes of each layer, net, or object in the layout component. This can be done in each layout component's property window through the **Visualization** tab or by checking Show Layout in the right-click shortcut menu on the component. If Show Layout is not checked, the layout component is displayed as a bounding box. You can also visualize RLC boundaries assigned on an edge or across layers, and component groups that are categorized as circuit elements.

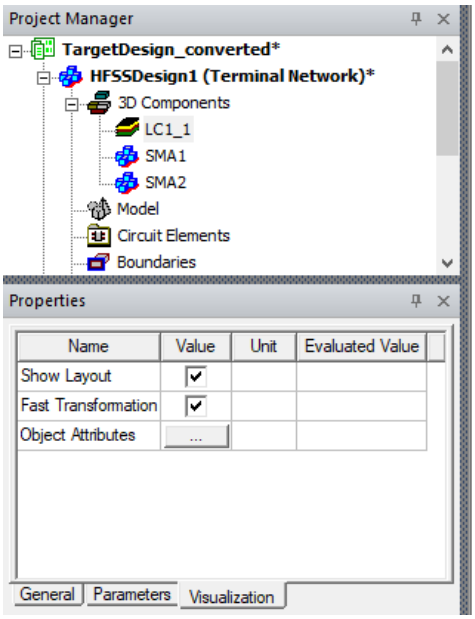


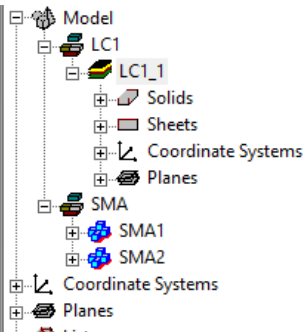
Using Show Layout for Layout Components

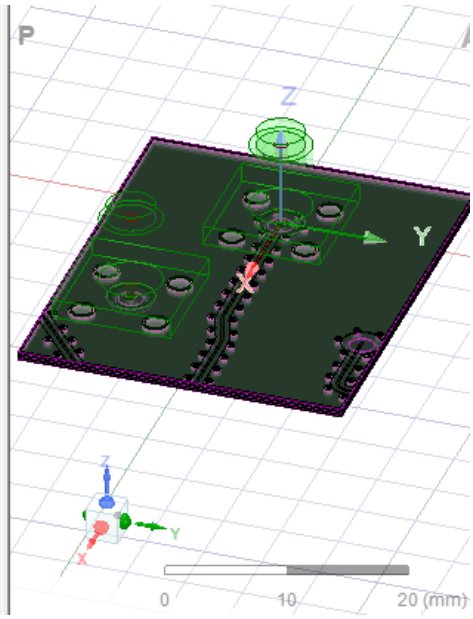
Checking **Show Layout** on the shortcut menu or the **Visualization** tab displays the layout.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





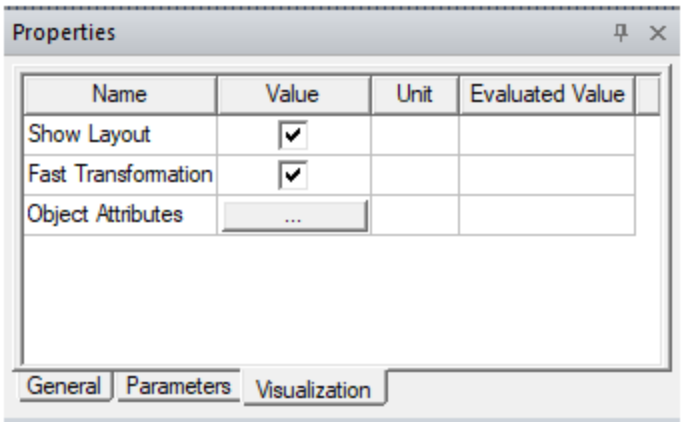




Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

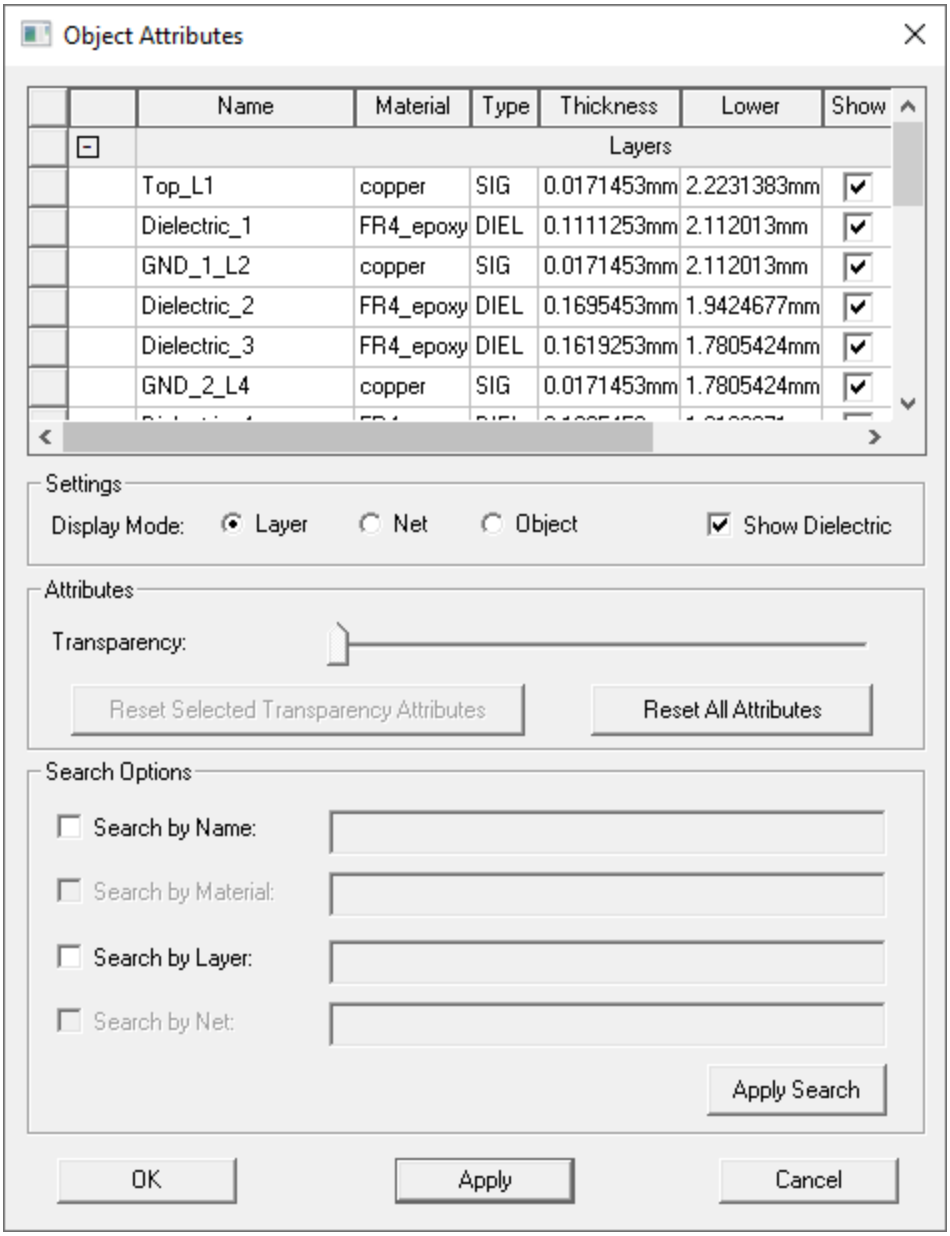
Object Attributes Control of Visualization for Layout Components

If **Show Layout** is checked, the **Object Attributes** [**...**] button provides access to the visualization attributes of the component in a separate dialog box. The **Fast Transformation** option toggles whether layout components show outlines of objects across layers during view changes. This improves the visualization performance.



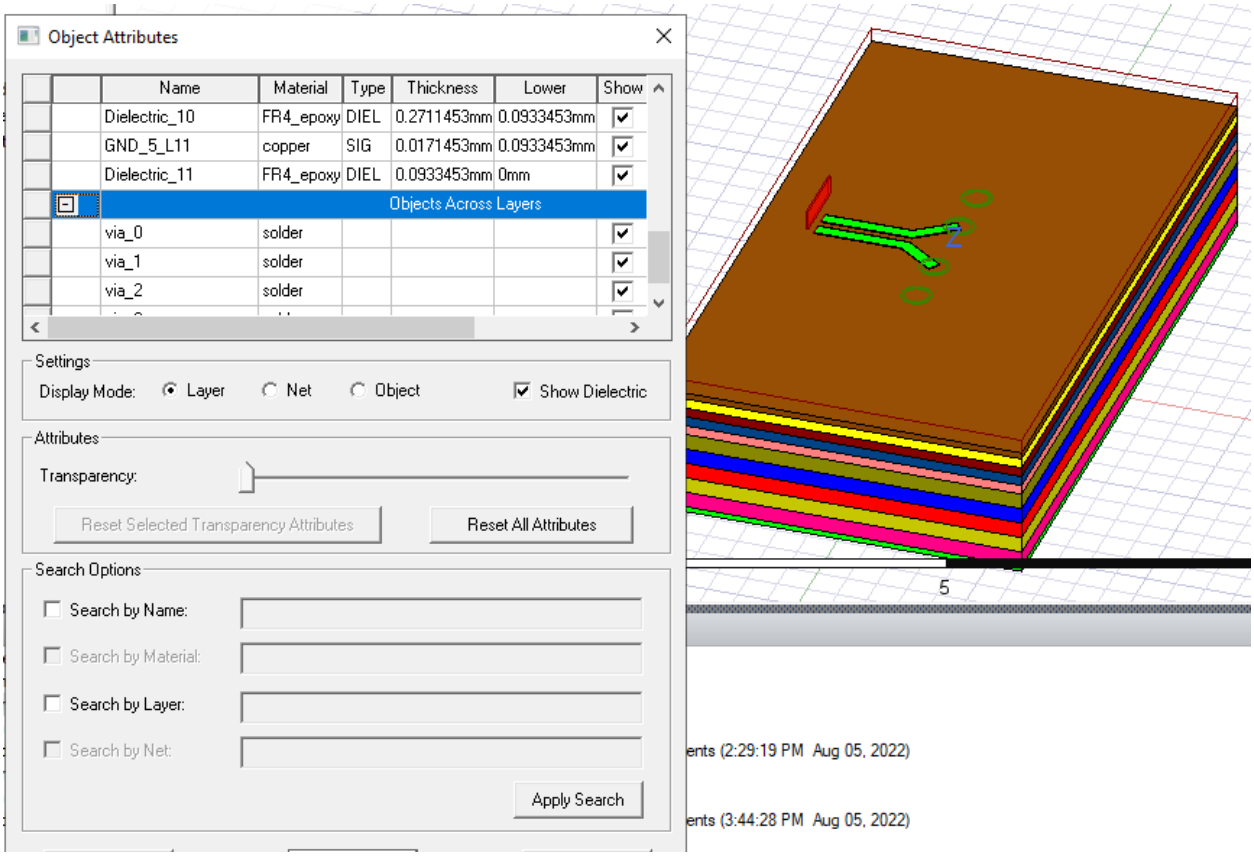
This is the **Object Attributes** dialog box. It provides fine control over how the Layout 3D Component is displayed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

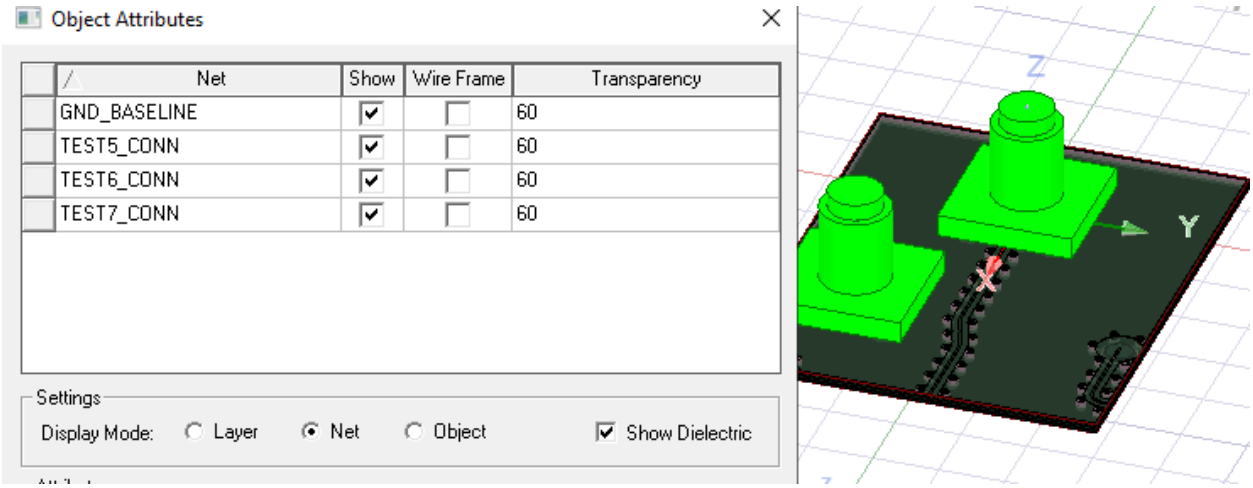


In the Settings group box, you can set the display mode as **Layer**, **Net**, or **Object**, and whether you want to **Show Dielectric** objects in the component. In the layer mode, this dialog displays information about each layer like the material, type, and thickness. Each layer is sorted by height instead of alphabetically. A button named **Reset All Attributes** is added to the dialog box to quickly reset the attributes of all items in this mode. The **Layer** mode displays every layer and every other object across layers separated by a separator. For Layer mode only, the table includes grouping buttons.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



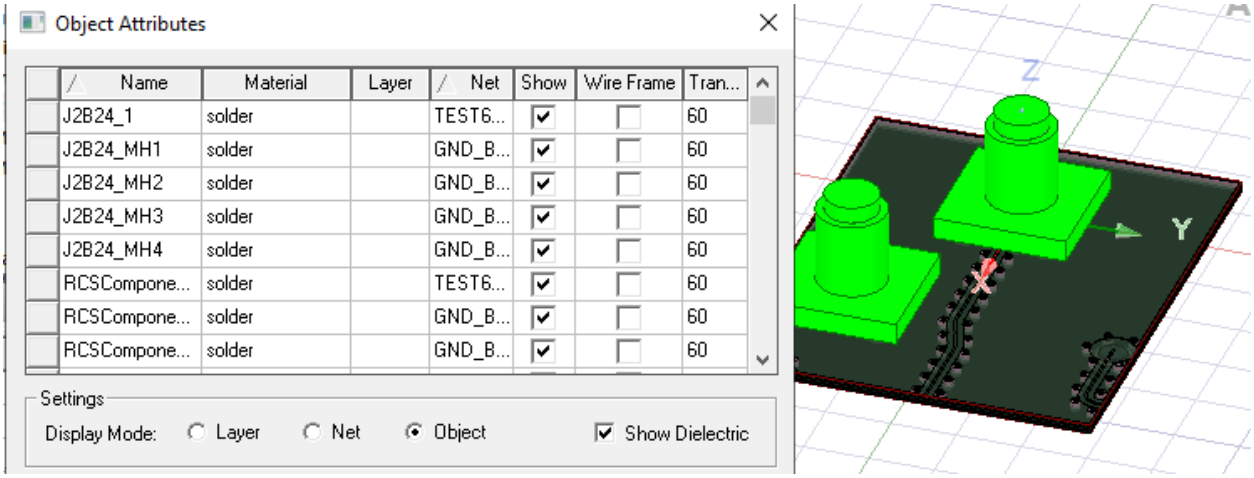
The **Net** mode displays component nets and ignore others objects not belonging to a net.



The **Object** mode displays every object from the component.

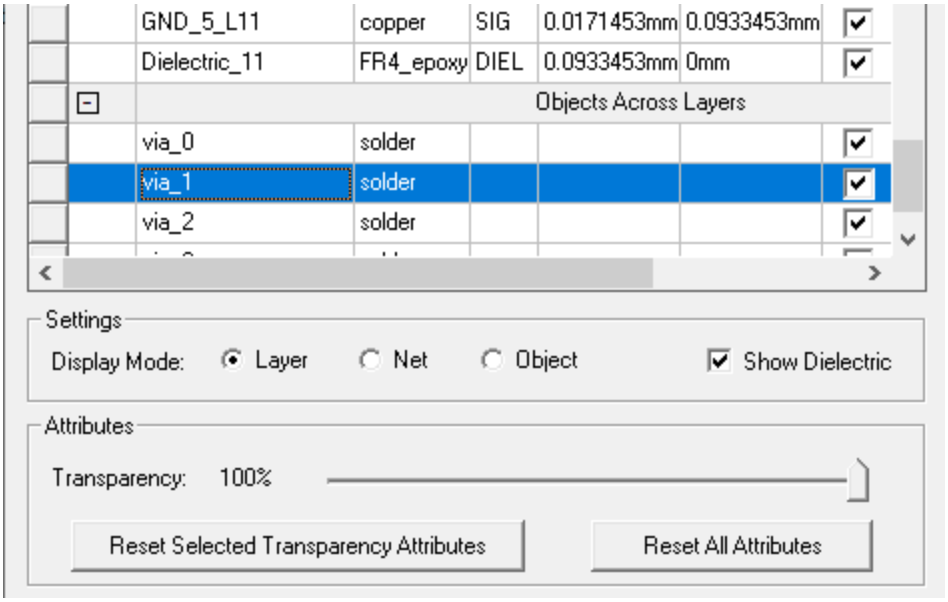
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** For large layout components with many objects, the performance in object mode could be slow.



For each item listed in the table, you can choose how you want to show it by selecting options in the **Show**, **Wire Frame**, and **Transparency** columns. You can check/uncheck the entire **Show** or **Wire Frame** column by clicking the respective column header. Click the **Transparency** column header to select/deselect all rows. Click other column headers to sort the table according to names from that column.

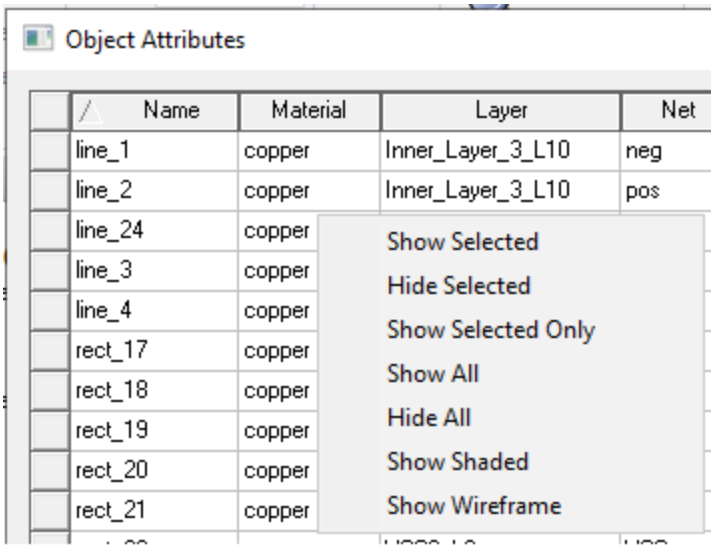
The dialog box supports multi-select functionality. When you select rows, the **Attributes** options become available.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

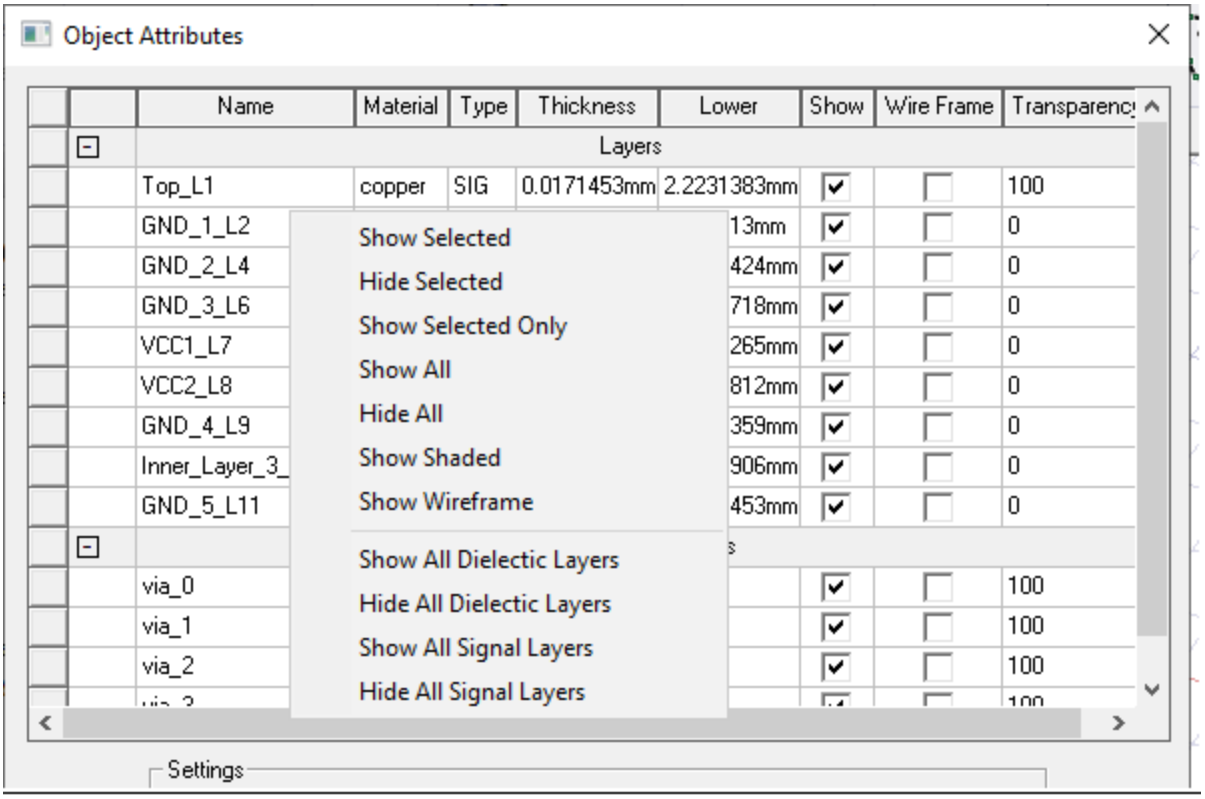
You can drag the **Transparency** slider bar to adjust the transparency value of all selected rows. Finally, the **Reset Selected Transparency Attributes** button sets the transparency value of all selected rows back to the default values when the component was imported. The default transparency of a layer or a net is calculated as the average transparency of all objects belonging to it.

In the **Layout Component Object Attributes** dialog box, if you right-click anywhere inside the table, a pop-up menu will appear to let you quickly turn on/off visualization of objects.

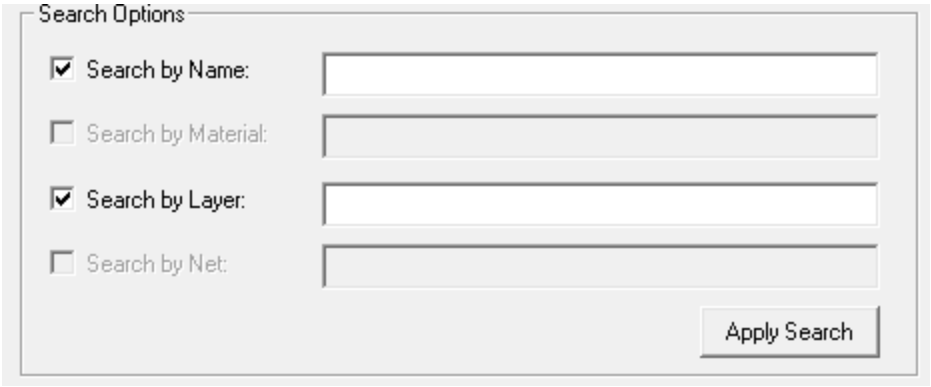


If the dialog box's display mode is layer mode, and you right-click the region containing layers, the pop-up menu contains additional menu items for quickly showing/hiding certain layers.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The dialog box also has search functionality to search for specific items. You can enter their search criteria in the **Search Options** group box.



By clicking **Apply Search**, the table refreshes to only show items that match the search criteria.

Related Topics

Using Layout Component in HFSS 3D Process Flow

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Visualization for Layout Components in 3D Modeler

Visualization for RLC Boundaries and Component Groups Designated as Circuit Elements

Replacing or Editing Layout Components

Editing Placement of a Layout Component

Flex PCB Components Support

Field Plots on Layout Component's Layer or Net

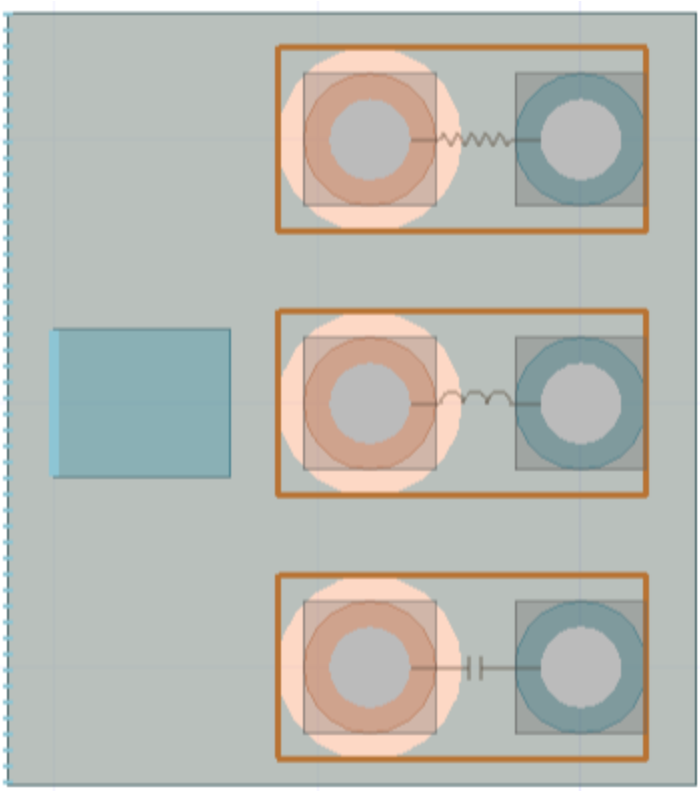
Visualization for RLC boundaries and Component Groups Categorized as Circuit Elements

You can visualize RLC elements for layout components in HFSS in the same way as they appear in HFSS 3D Layout Components

- The types of circuit elements that can be visualized include RLC boundaries assigned on an edge or across layers, and component groups that are categorized as circuit elements. Other forms of circuit elements currently cannot be included in a layout component and thus are not supported.
- Display each circuit element’s name on top of the visualization.

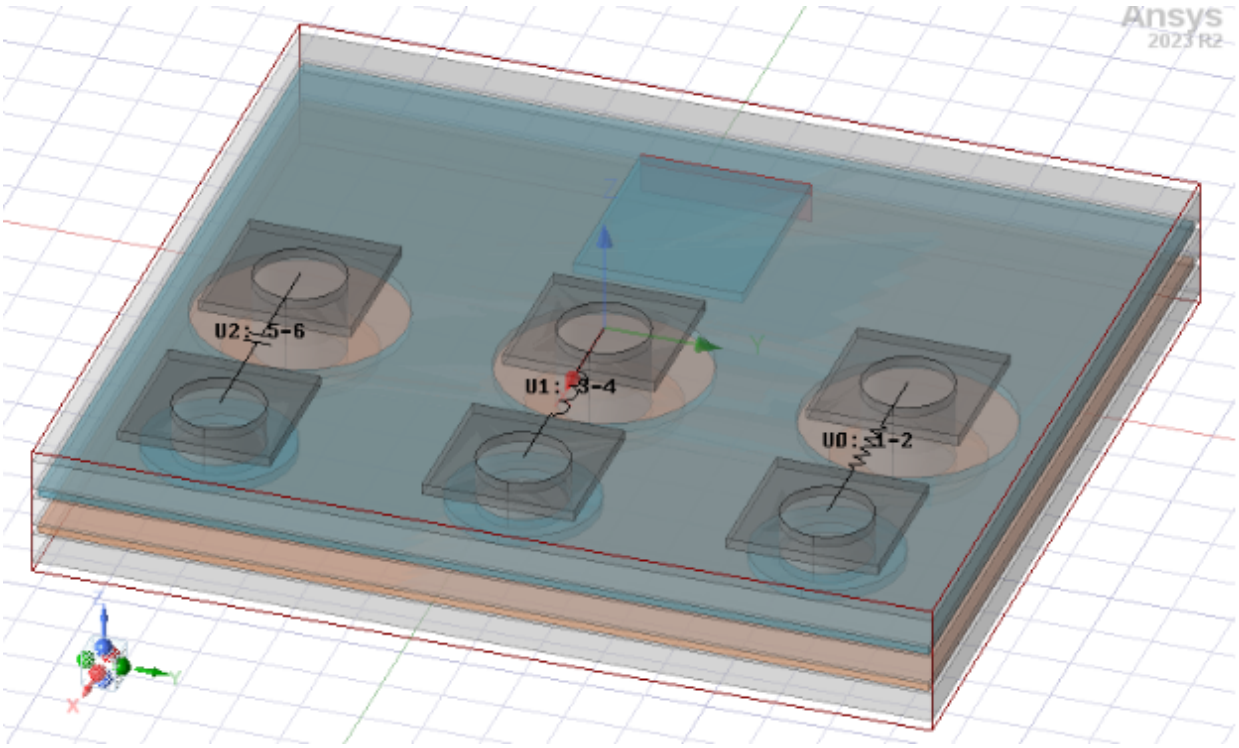
You can toggle these RLC visualizations with the **Show Layout** option. The intent is to display each layout component’s component data in its property window similar to that of 3D components. For example, given a 3D layout design with circuit elements, the visualization of circuit elements in the HFSS 3D Layout modeler window can appear as shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When imported as a Layout Component into HFSS, the layout components would appear as shown below.

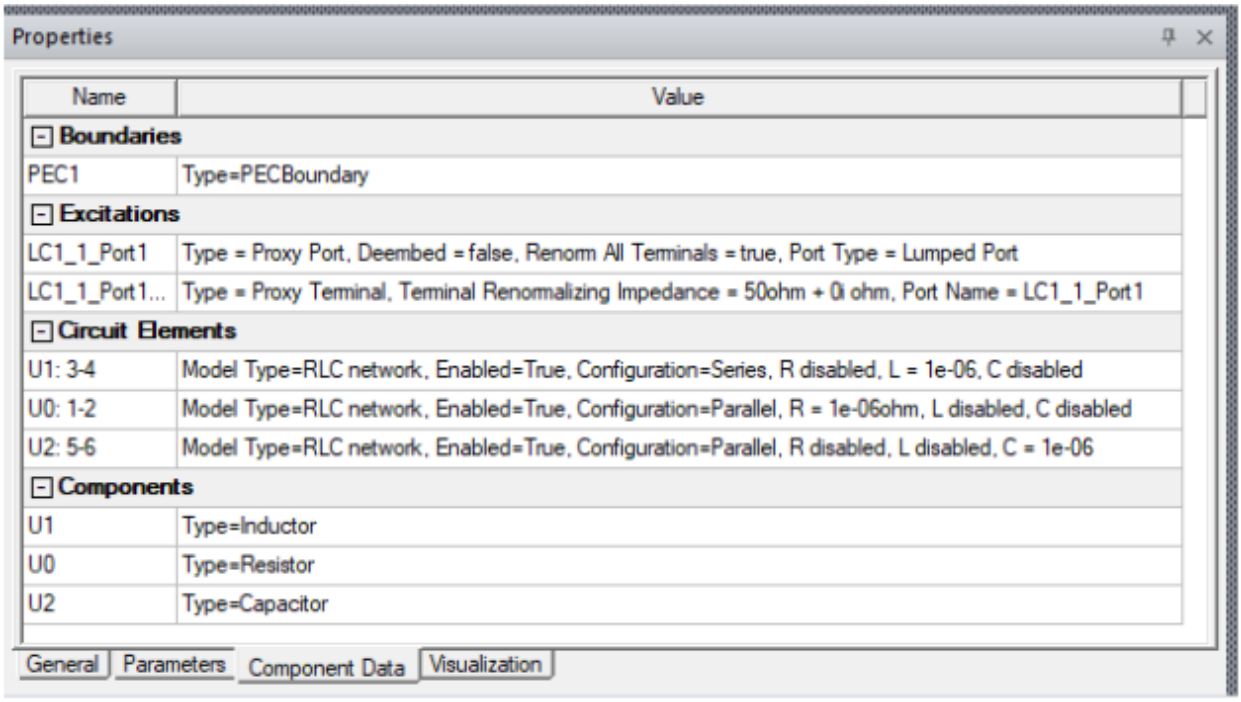
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The names of the circuit elements are also displayed on top of the visualization. When the circuit elements become too small to display due to zooming out, these names disappear to improve the clarity of the window. This visualization of RLC circuits with their names can be turned on/off by toggling the layout component's **Show Layout** option.

This property page will display the information about each boundary, excitation, circuit element, and component group of the layout component.

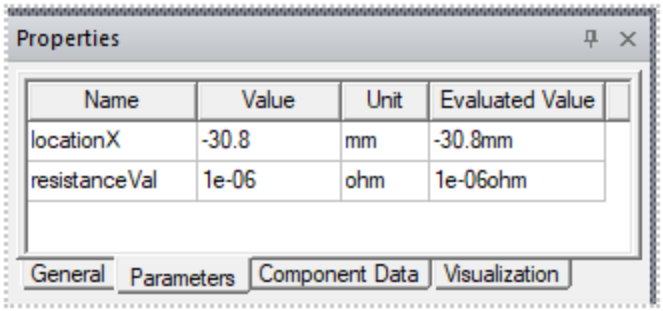
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Parameter Support for Layout Components

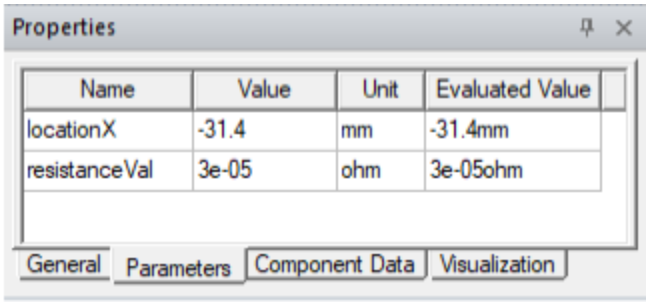
The circuit element visualization and component data property page supports parametrization. If the location of a circuit element or a part of the description in the component data is controlled by a parameter, changing this parameter will cause the visualization or the property page to update and reflect the change.

For example, in the above example, the layout component has two variables as shown below.

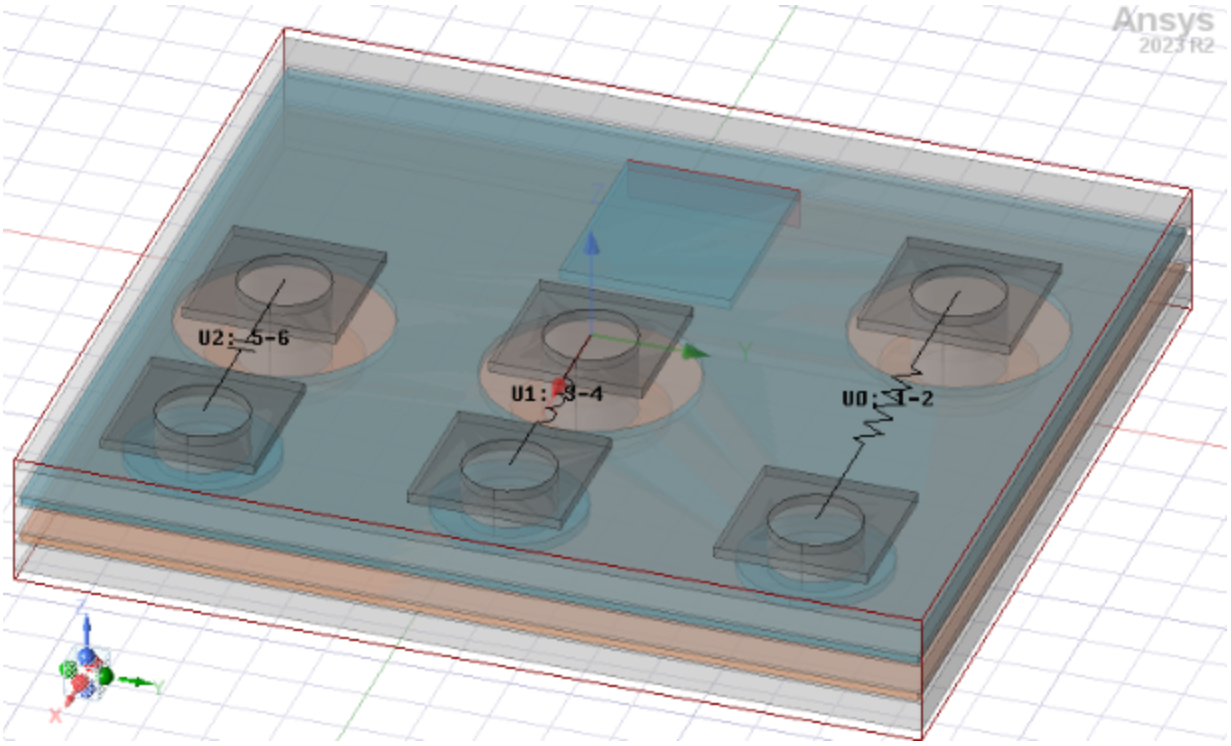


The **locationX** variable controls the x-location of one of the pins in circuit element U0: 1-2. The **resistanceVal** variable controls the electrical resistance in circuit element U0: 1-2. You could change the value of these two variables shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

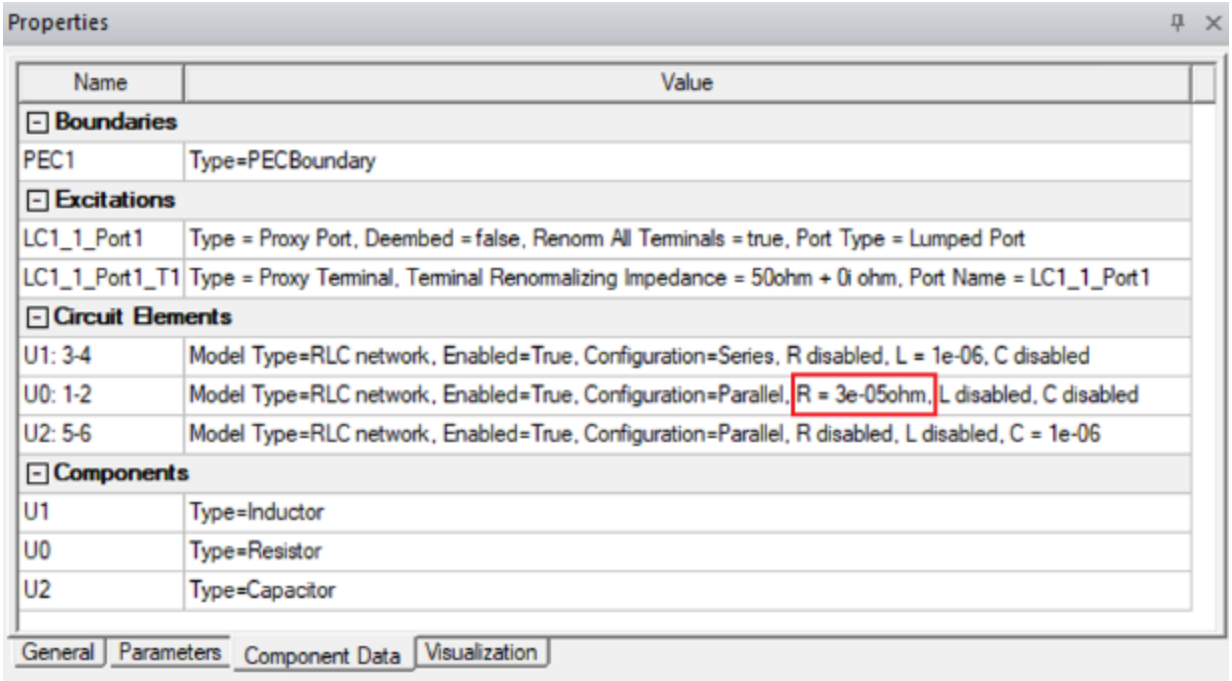


With this variable change, the component's circuit element visualization updates to reflect this change.



The Properties page for affected Circuit Elements also updates.

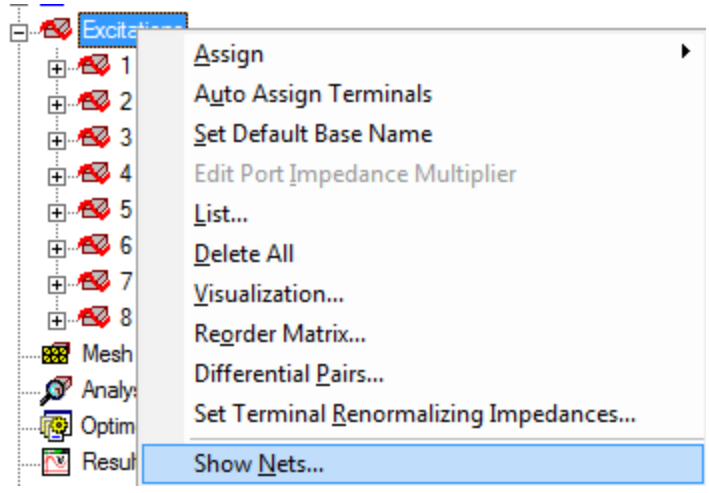
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Show Nets for Layout Components in HFSS 3D Geometries

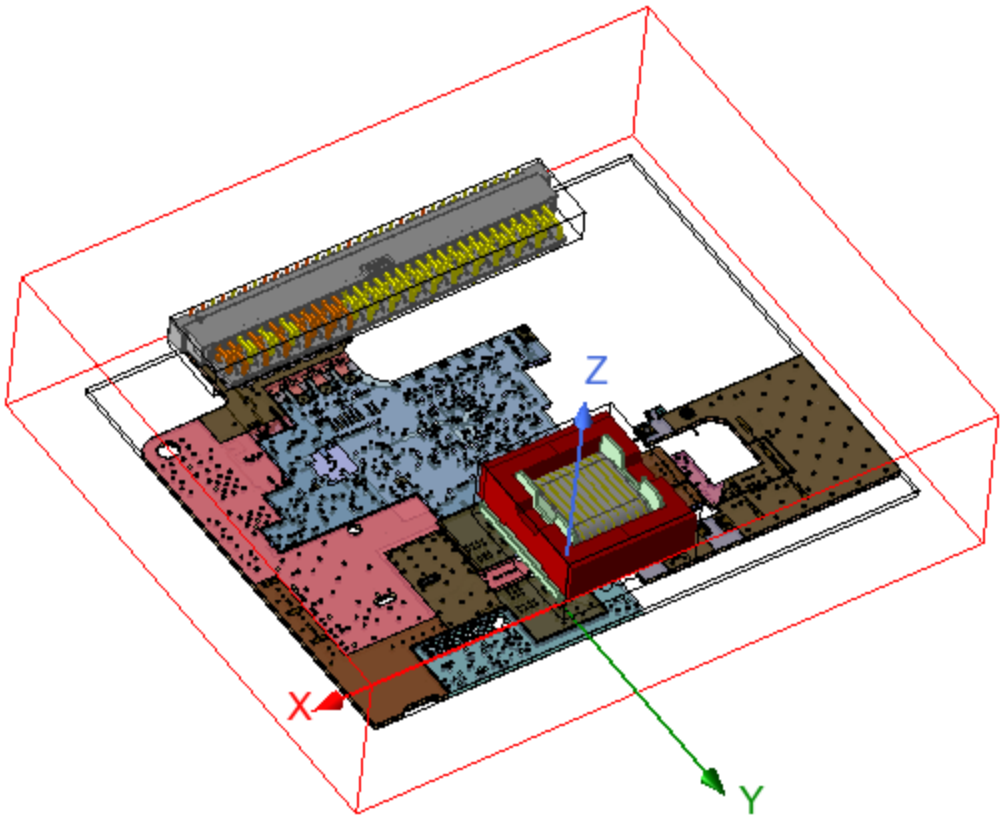
For Terminal Solution types, HFSS designs that include Layout Components, you can use the **Show Nets** command to visualize conducting nets and terminal associations formed by 3D geometries and layout components in a HFSS design.

The **Show Nets** command appears in the **HFSS** > **Excitations** menu and by right-clicking on **Excitations** in the Project tree.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting **Show Nets...** displays the **Net Visualization** dialog box that lists the nets and associated terminals and includes any included Layout Components. You can use the dialog box to select any net or terminal. You can continue to work in the Modeler window with the dialog open. Consider the following example design.



Note:

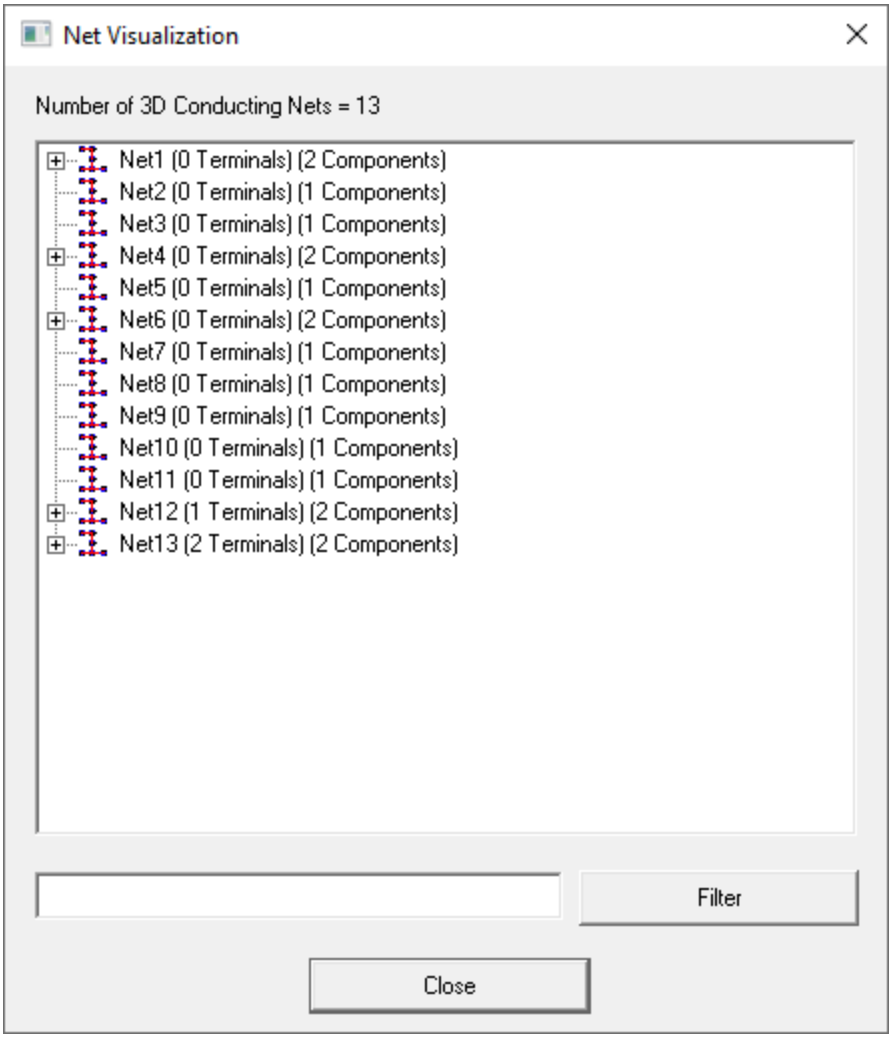
For a net to appear in the **Net Visualization** dialog box, the conductivity of the assigned material must be equal to or greater than 20,000 siemens/m, or the **Resistance** must be less than 3.142 ohms. Conductivity is set by material assignment (in the **3D Modeler** window or a boundary condition), and **Resistance** is set by boundary conditions (e.g., **Impedance** or **LumpedRLC**).

In the **Net Visualization** dialog box, you can sort the nets in ascending or descending order, relative to the number of terminals. This helps you locate GND nets, which have no terminals.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Text at the top of list box provides the total number 3D conducting nets. Naming of nets is automatic and follows the convention: "Net1 (0 Terminals)", "Net2 (1 Terminals)"…"Net<N> (<T> Terminals)". Net Names are not editable.

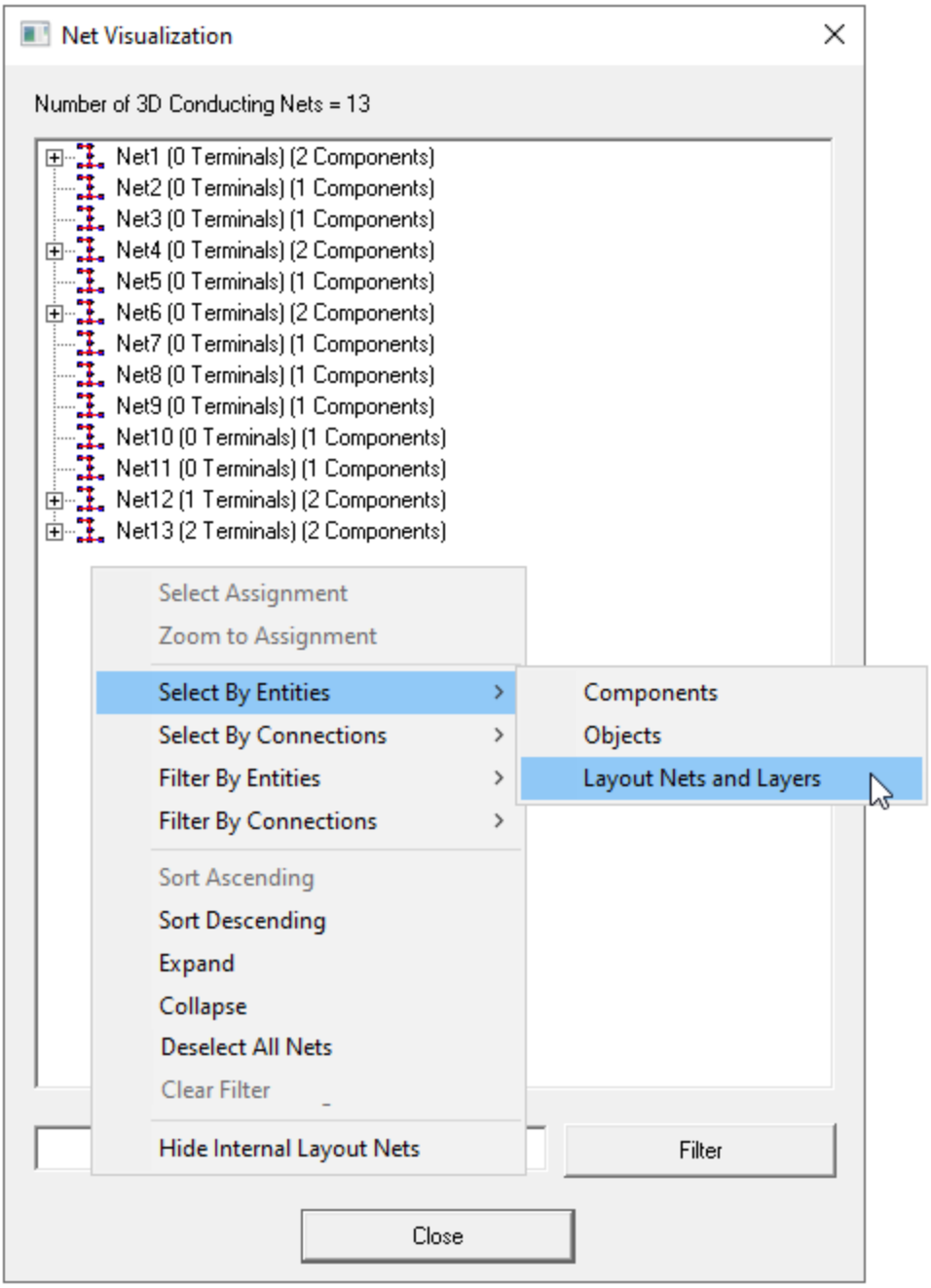
The **Net Visualization** dialog box is shown below.



For designs with many nets, you can enter text and wild cards in the field, and click the **Filter** button.

Right-click in the **Net Visualization** dialog box to view the short-cut menu. The menu is context sensitive, depending on whether you click on a particular net object, or in the white area.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Select Assignment**: If you right-click over selection in the tree, selects entities present in the selection. If the right-click menu is invoked on the surrounding blank space, this menu item is disabled as there is no selection.
- **Zoom to Assignment**: If you right-click over a selection in the tree, zooms to the entities present in the selection. If you right-click on the surrounding blank space, this menu item is disabled as there is no selection.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Select by Entities**: Invokes a dialog box containing object, component, or net and layer names present in the selection if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, all object or component names present in the project are displayed. Selecting one or multiple of these object or component names selects all the nets that contain any of the selected entities.
- **Select by Connections**: If you right-click over some selection in the tree, this produces a dialog box containing object or component names present in the selection. If you right-click in the surrounding blank space, a dialog box with all object or component names present in the project displays. Selecting a pair of these objects or component names selects all the nets that contain this pair of entities, signifying a direct or indirect connection between these entities.
- **Filter by Entities**: A new dialog box is invoked containing object, component, or net and layer names present in the selection if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, all object or component names present in the project are displayed. Selecting one or multiple of these objects or component names filters all the nets that contain any of the selected entities.
- **Filter by Connections**: A new dialog box is invoked containing object or component names present in the selection if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, all object or component names present in the project are displayed. Selecting a pair of these objects or component names filters all the nets that contain this pair of entities, signifying a direct or indirect connection between these entities.
- **Sort Ascending**: Sorts the nets in ascending order of their names.
- **Sort Descending**: Sorts the nets in descending order of their names.
- **Expand**: Expands the selection in the tree if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, it expands all the nets.
- **Collapse**: Collapses the selection in the tree if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, it expands all the nets.
- **Deselect All Nets**: Deselects all the entities present in the project. Clears the selection of all the previously selected entities.
- **Clear Filter**: Clears all the filtering applied on nets and shows all the nets.
- **Hide Internal Layout Nets**: Hides all the nets that are internal to layout component

Along with the “filter by entities” and the “filter by connections” menu items, you can also filter nets using text input. The text box for this functionality is shown below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

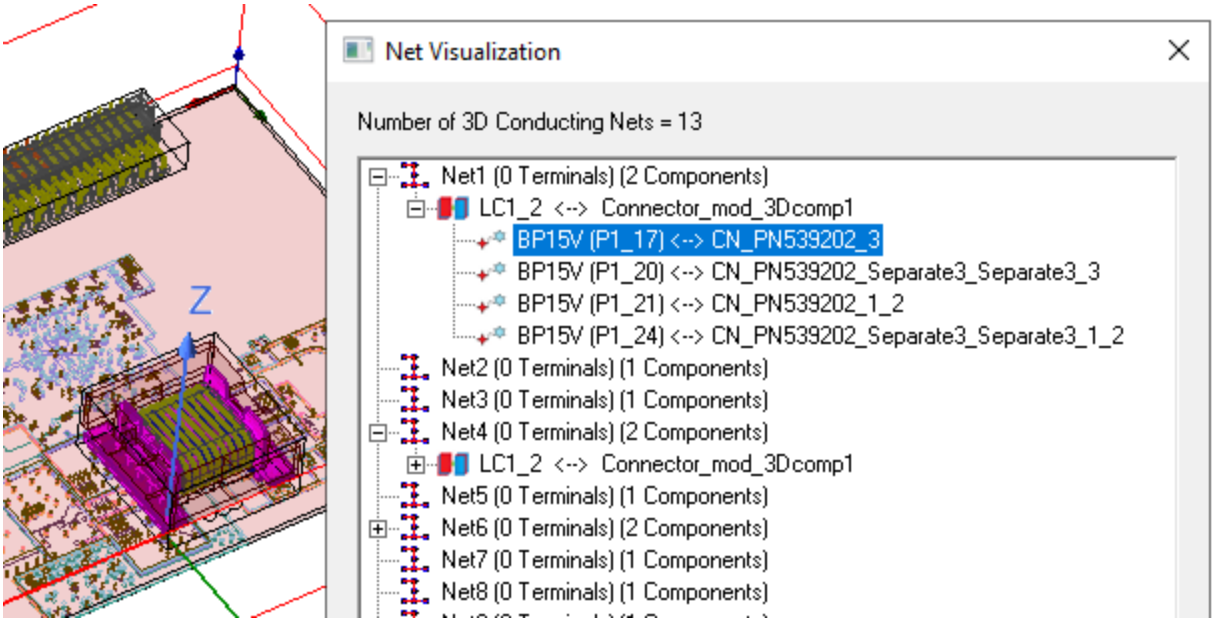
Features of this text box are

- **Select Assignment**: If you right-click over selection in the tree, selects entities present in the selection. If the right-click menu is invoked on the surrounding blank space, this menu item is disabled as there is no selection.
- **Zoom to Assignment**: If you right-click over a selection in the tree, zooms to the entities present in the selection. If you right-click on the surrounding blank space, this menu item is disabled as there is no selection.
- **Select by Entities**: Invokes a dialog box containing object, component, or net and layer names present in the selection if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, all object or component names present in the project are displayed. Selecting one or multiple of these object or component names selects all the nets that contain any of the selected entities.
- **Select by Connections**: If you right-click over some selection in the tree, this produces a dialog box containing object or component names present in the selection. If you right-click in the surrounding blank space, a dialog box with all object or component names present in the project displays. Selecting a pair of these objects or component names selects all the nets that contain this pair of entities, signifying a direct or indirect connection between these entities.
- **Filter by Entities**: A new dialog box is invoked containing object, component, or net and layer names present in the selection if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, all object or component names present in the project are displayed. Selecting one or multiple of these objects or component names filters all the nets that contain any of the selected entities.
- **Filter by Connections**: A new dialog box is invoked containing object or component names present in the selection if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, all object or component names present in the project are displayed. Selecting a pair of these objects or component names filters all the nets that contain this pair of entities, signifying a direct or indirect connection between these entities.
- **Sort Ascending**: Sorts the nets in ascending order of their names.
- **Sort Descending**: Sorts the nets in descending order of their names.
- **Expand**: Expands the selection in the tree if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, it expands all the nets.
- **Collapse**: Collapses the selection in the tree if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, it expands all the nets.
- **Deselect All Nets**: Deselects all the entities present in the project. Clears the selection of all the previously selected entities.

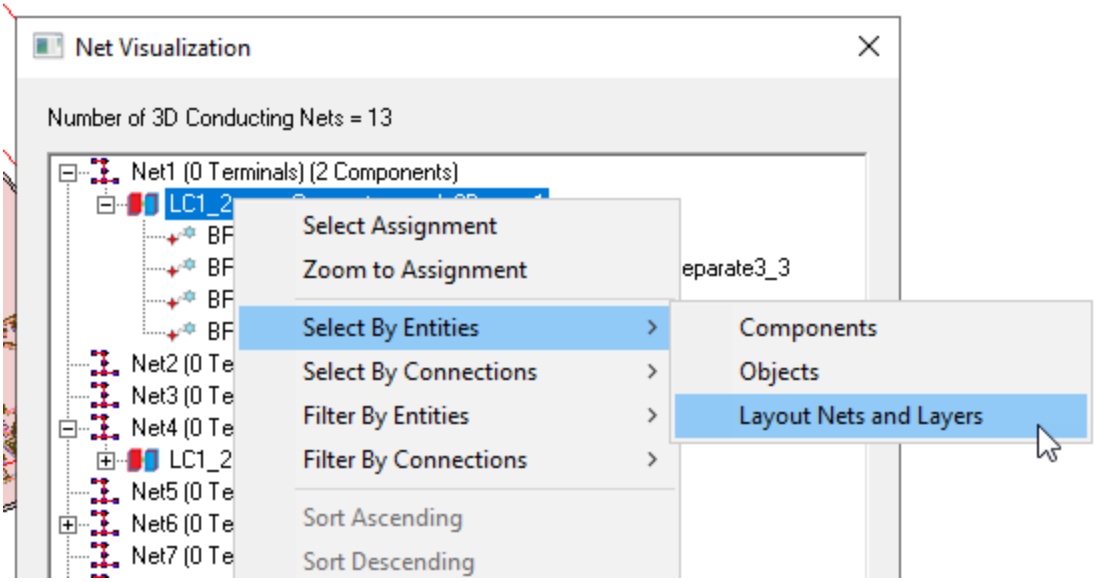
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Clear Filter**: Clears all the filtering applied on nets and shows all the nets.
- **Hide Internal Layout Nets**: Hides all the nets that are internal to layout component

Selecting any component or net in the **Net Visualization** list highlights it. For example, the following figure shows a Layout Component selected in the expanded tree list and highlighted in the **Modeler** window.

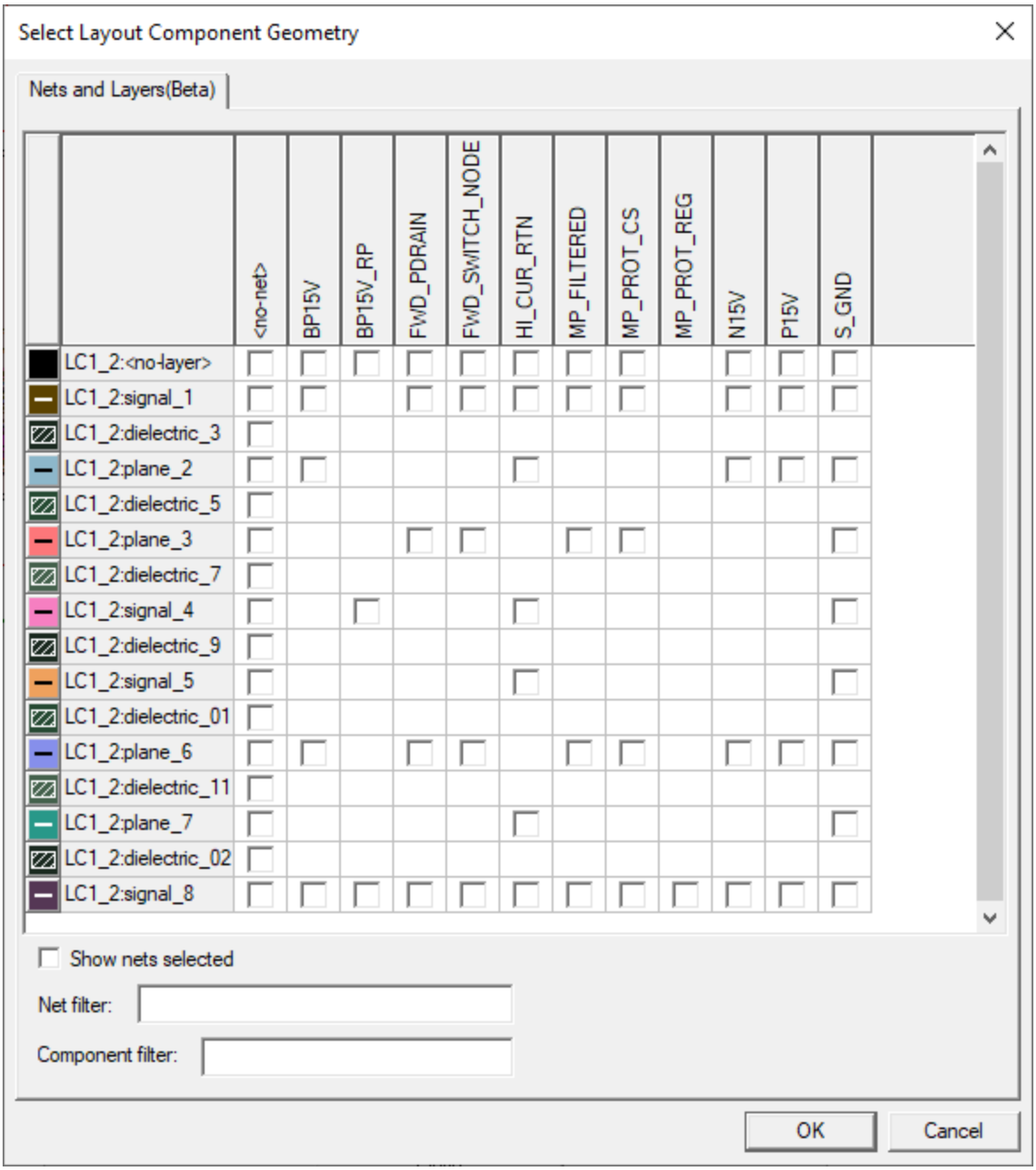


For example, right-click on the Layout Component name in the **Net Visualization** dialog box to display the menu and select **Select by Entities** > **Layout Nets and Layers**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This displays the **Select Layout Component Geometry** dialog box.



With the **Select Layout Component Geometry** dialog box, you can select objects from layout components' local nets and layers and search for nets in the overall design that contain these objects. This dialog displays all the layers and local nets of every layout component in the design. If there are objects in the layout component that belong to a specific layer and net, a checkbox will become available for you to select. Once you make selections and click **Apply**,

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

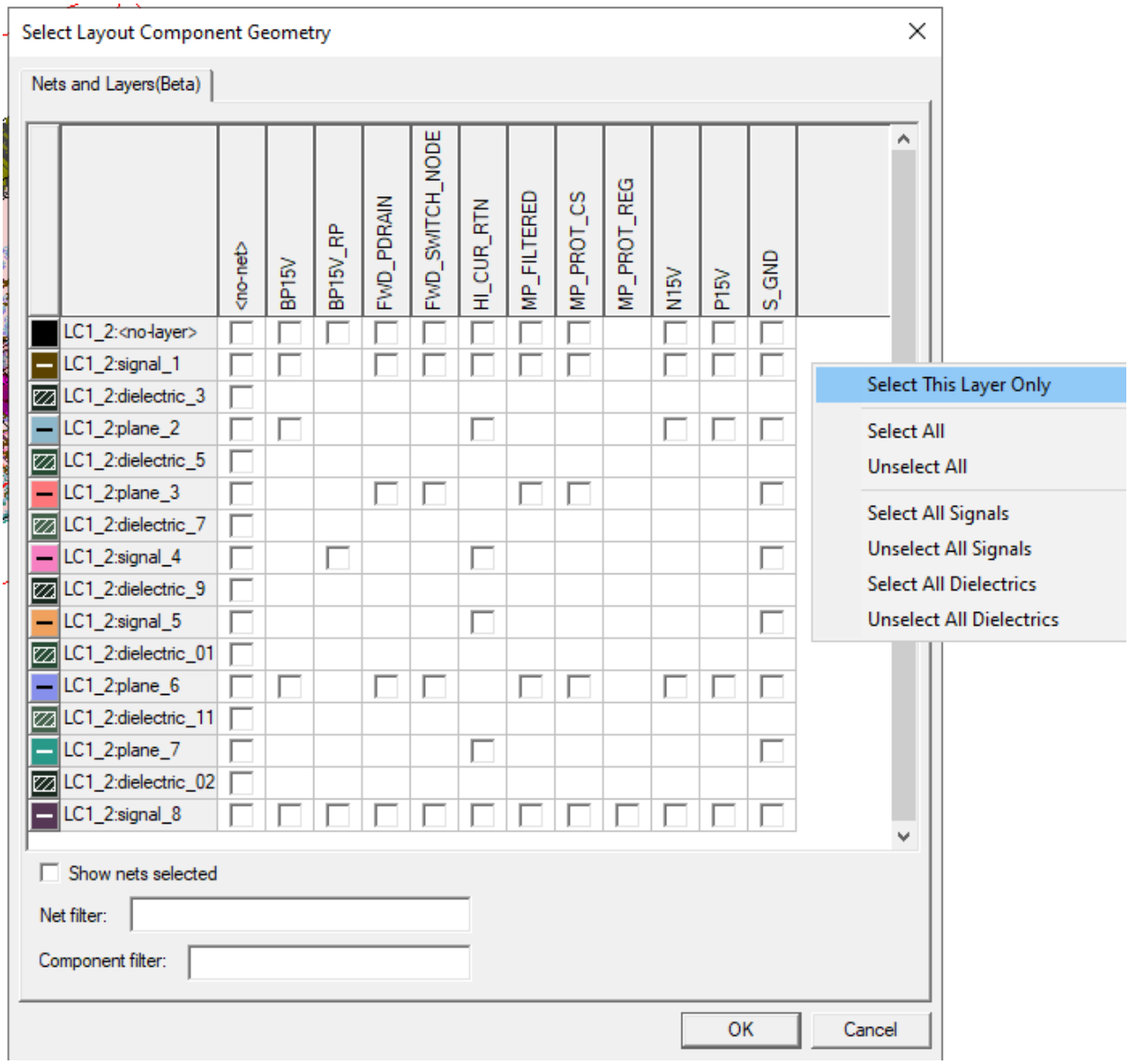
you can go back to the parent **Net Visualization** dialog box and click **Select** button, which selects all the nets in the **Net Visualization** dialog that contain these objects.

The **Select Layout Component Geometry** dialog is modeless, that means you can work in the 3D Modeler while keeping this dialog box open. This way, you can use the modeler window to better find the layout component objects you want to search for.

By clicking the column headers, you can select or deselect checkmboxes in the entire column. The **Show nets selected** check box can be toggled to move all the selected nets to the left side of the table. Use the Net filter and Component filter to filter fields to designate specific nets and components. These filters support '*' and '?' wildcards and can delimit multiple filters with comma. The table is refreshed each time users update the filter text.

Additionally, the **Select Layout Component Geometry** dialog box has a right-click menu as shown below.

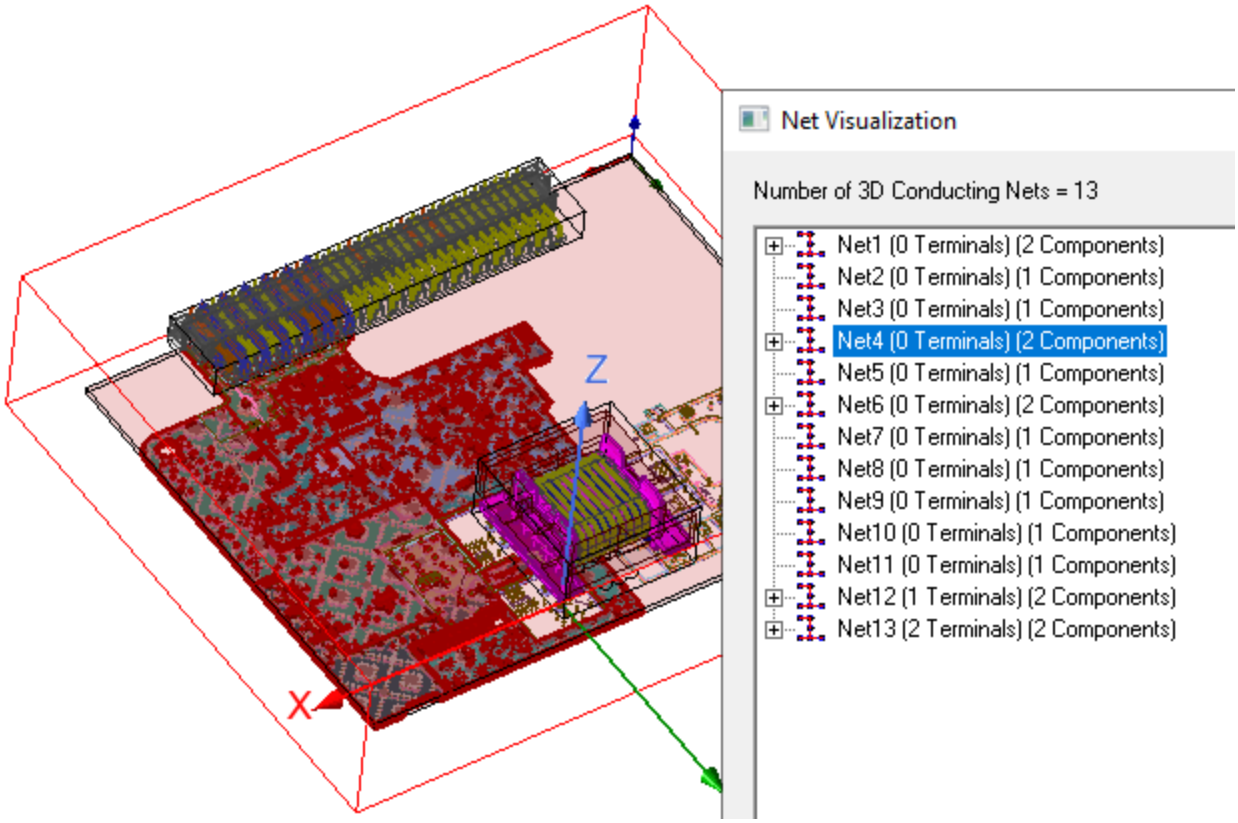
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can use the dialog to select any net or terminal. You can continue to work in the Modeler window with the dialog box open.

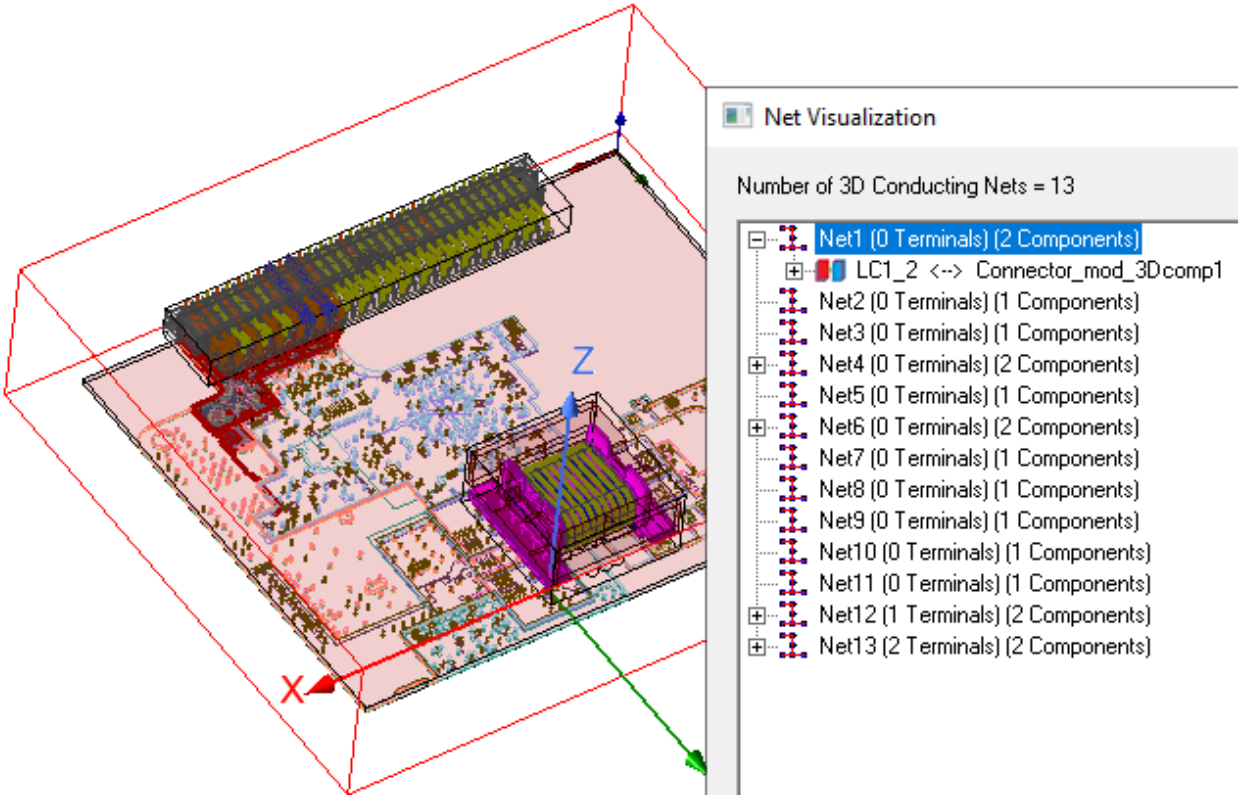
The **Net Visualization** dialog box identifies all nets and will include both 3D geometries and layout components with correct terminal associations. As shown in the figure below, any net you select in the Net Visualization list highlights the objects in the net in the modeler window. By default, 3D geometries are highlighted with blue texture, and objects from layout components will be highlighted with red texture. The rest of objects in the layout component not belonging to the selected net will be shown as wireframes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The next figure shows a different selection from the list of 3D conducting nets to illustrate the changed highlighting.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

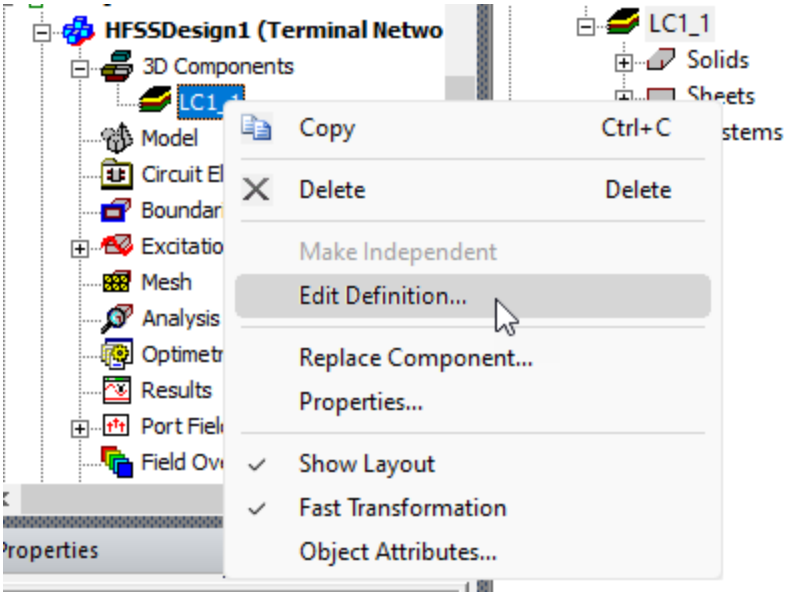


All the functionalities in the Net Visualization dialog that works for regular nets are also supported for nets containing layout component objects, such as **Zoom To**. Because layout components do not have any actual geometries in the modeler window, the **Select Assignment** will only select 3D geometries in each net.

Editing a Layout Component Definition

All layout components in HFSS designs have a command in the right-click menu called **Edit Definition**.

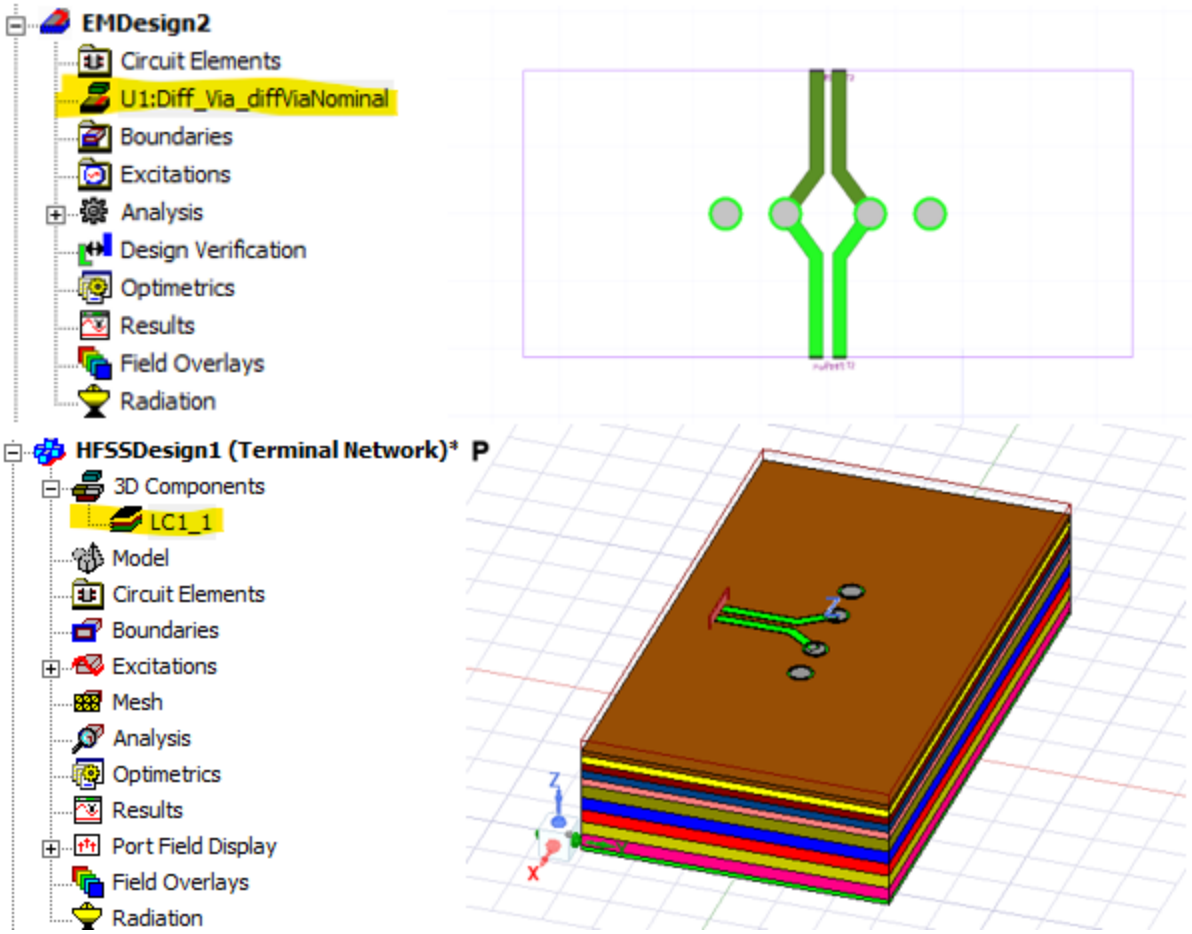
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



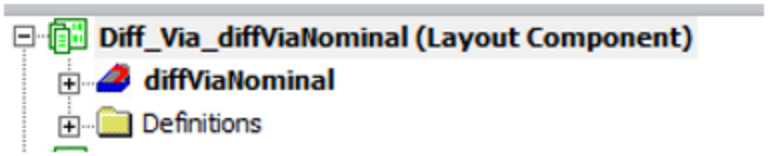
Once clicked, it opens the design containing the EDB definition of the layout component in a separate project to allow for any editing. Once a layout component is edited, either through a parent 3D layout design or through this command, all designs of different types that use this component update it accordingly.

For example, in an Ansys Electronics Desktop project, multiple designs of different types can have layout components of the same definition. Below is an example project where both an HFSS 3D Layout design and an HFSS design have a layout component with the definition “Diff_Via_diffViaNominal”.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

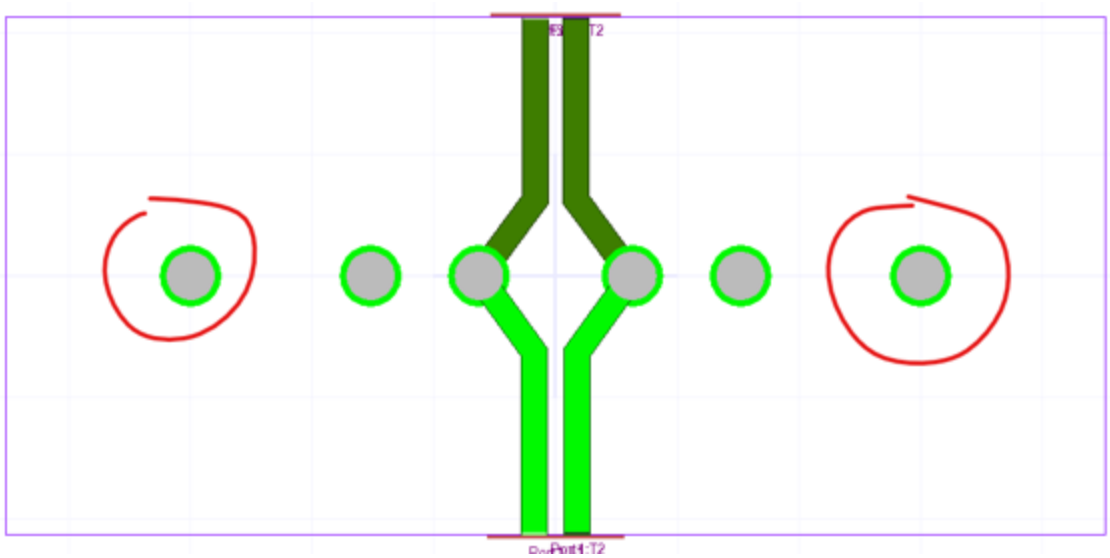


In addition to editing the source layout design using the **Edit Definition** command, you can edit the layout component's local settings, or replace it with a different component, using the **Replace Component** command.



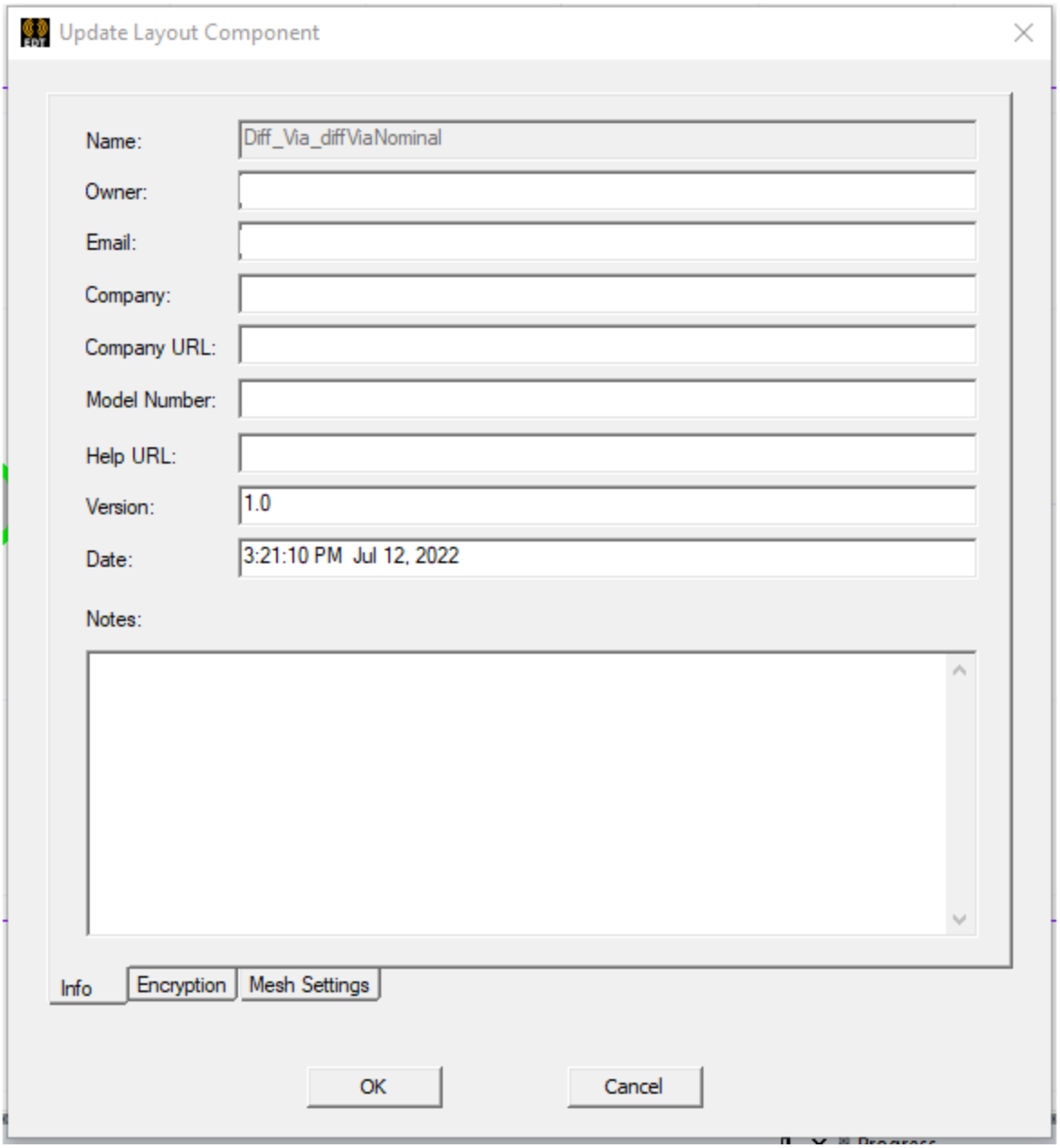
In the source design, you can edit the layout component however you want. This example adds two more vias on the left and right side as shown in the figure below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



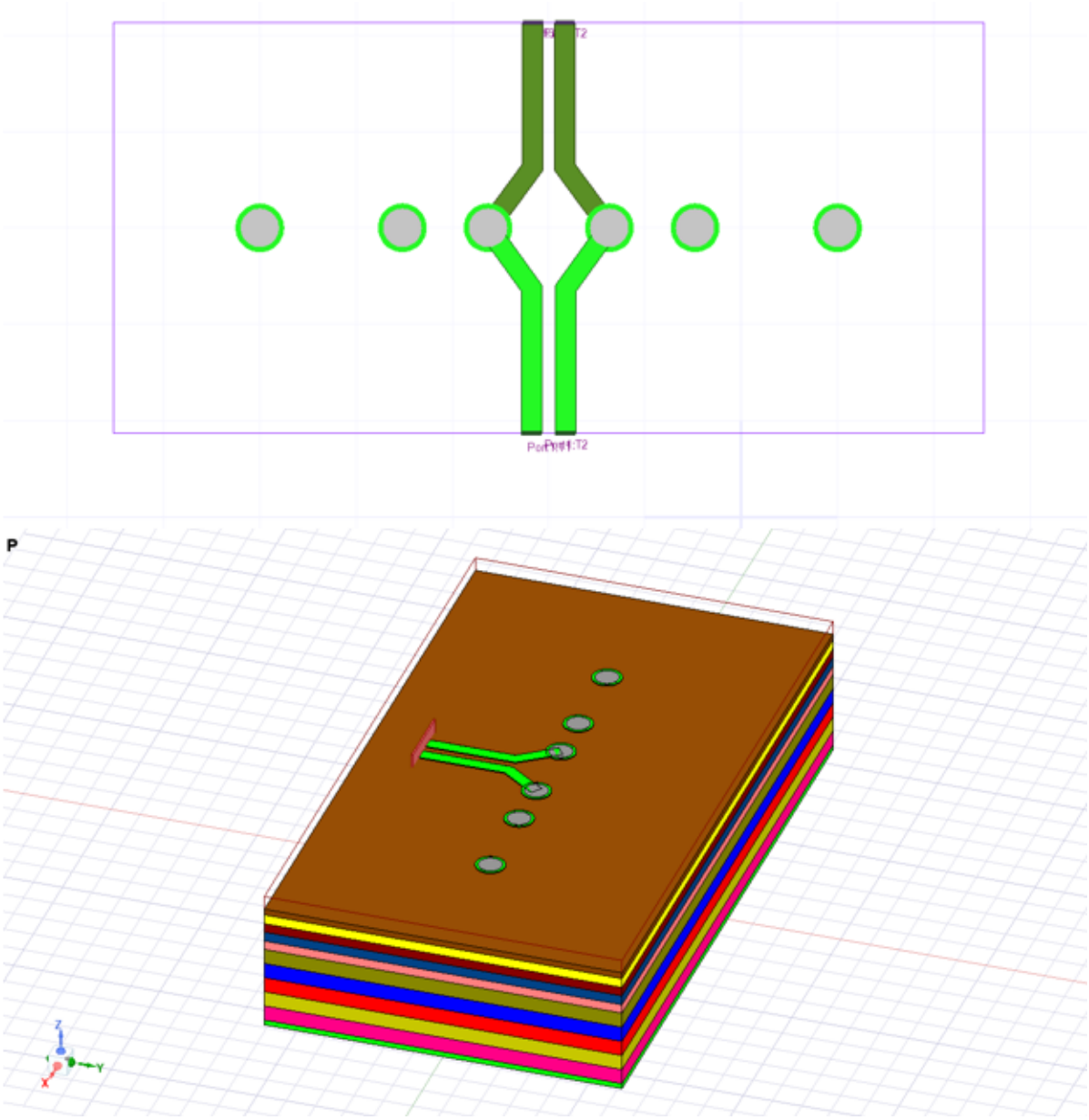
After editing, you can save the source design, and a dialog box opens for you to confirm the changes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once the changes are confirmed, both the HFSS 3D Layout design and the HFSS design containing this layout component update to reflect the changes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

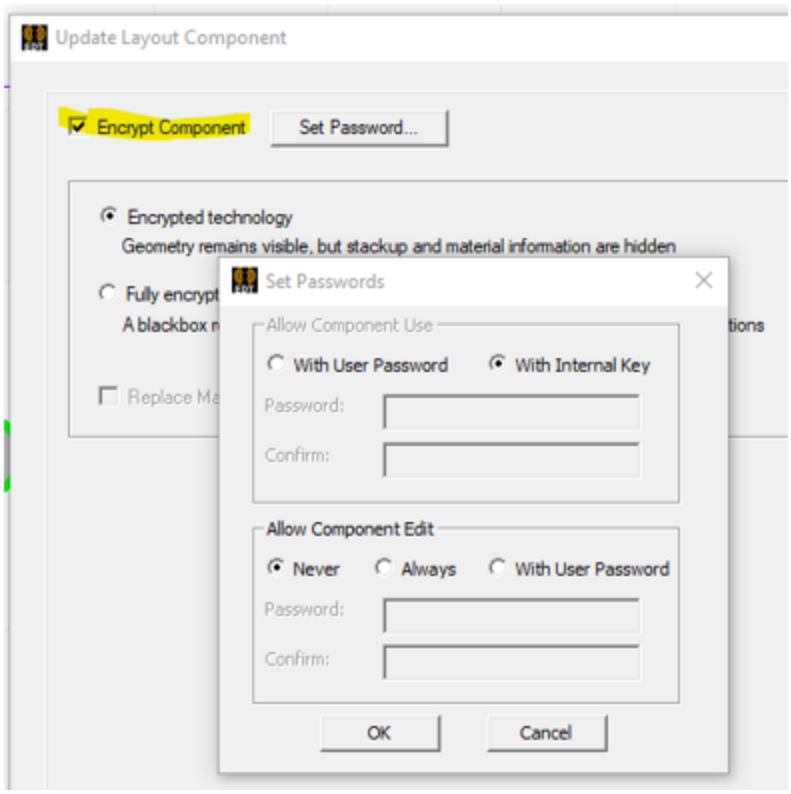


Note:

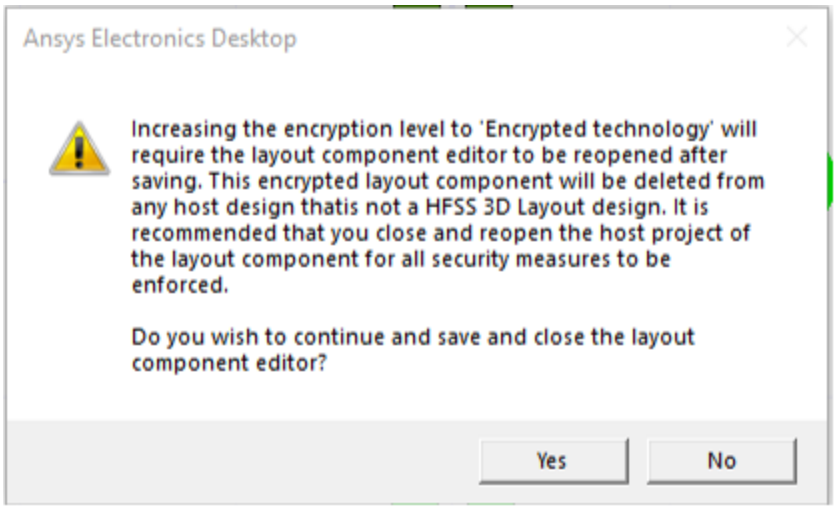
The editing is not undoable. After saving the source model, the target project is automatically saved, and the undo stack is cleared.

While editing, you may choose to encrypt the layout component.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

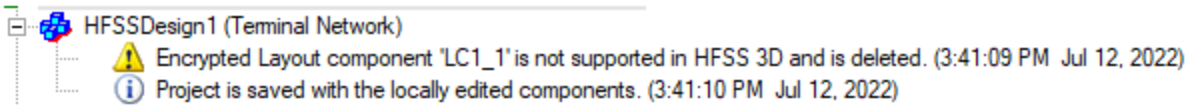


If the layout component is encrypted, a dialog box with warning messages alerts you about the potential impact to the designs containing this layout component.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After editing, all encrypted layout components are deleted from any target designs as encrypted layout components are not currently supported. A warning message appears for each deleted layout component.



Related Topics

Using Layout Component in HFSS 3D Process Flow

Visualization for Layout Components in HFSS 3D

Visualization for RLC Boundaries and Component Groups Designated as Circuit Elements

Using Layout Custom CS for Placement in HFSS 3D

Flex PCB Components in HFSS 3D

Replacing or Editing Layout Components

Field Plots on Layout Component's Layer or Net

Plotting the Mesh for Layout Components

Replacing or Editing Layout Components

Use the command, **Replace Component** to edit the properties that were specified when a layout component was inserted into your design or to replace the component with a different one. This command is located in the shortcut menu that appears when you right-click a layout component entry in the Project Manager. A multi-tab dialog box appears, *Edit Layout Component*, which is very similar to the *Insert Layout Component* dialog box discussed in the topic, *Adding a Layout Component*.

Insert *"Replace Component"* instructions consistent with HFSS/Maxwell behavior here and delete this comment.

Edit/Replace Procedure

To edit an existing layout component in your HFSS design, or to replace it with a different component, do the following:

1. Under *3D Components* in the Project Manager, right click the ***<Layout_Component_Name>*** and choose **Replace Component**.

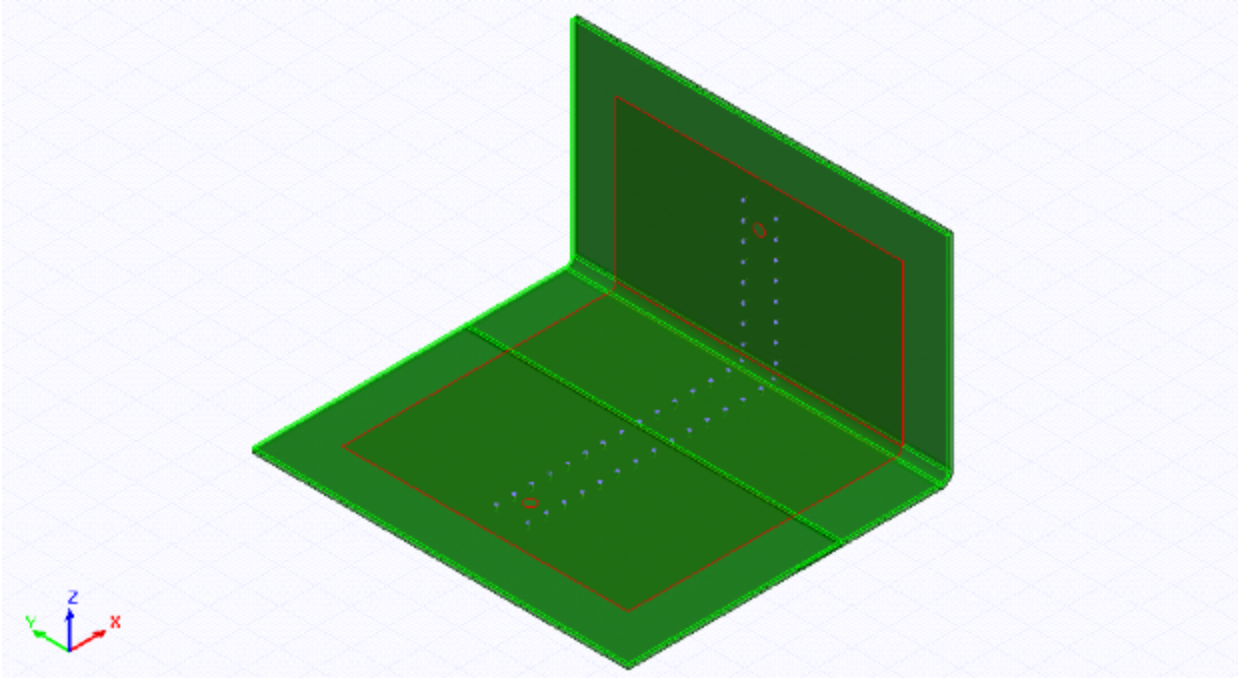
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The *Edit Layout Component* dialog box appears with the **General** tab initially selected:

2. **Insert the procedure steps and images that are consistent with HFSS/Maxwell behavior here and delete this comment.**

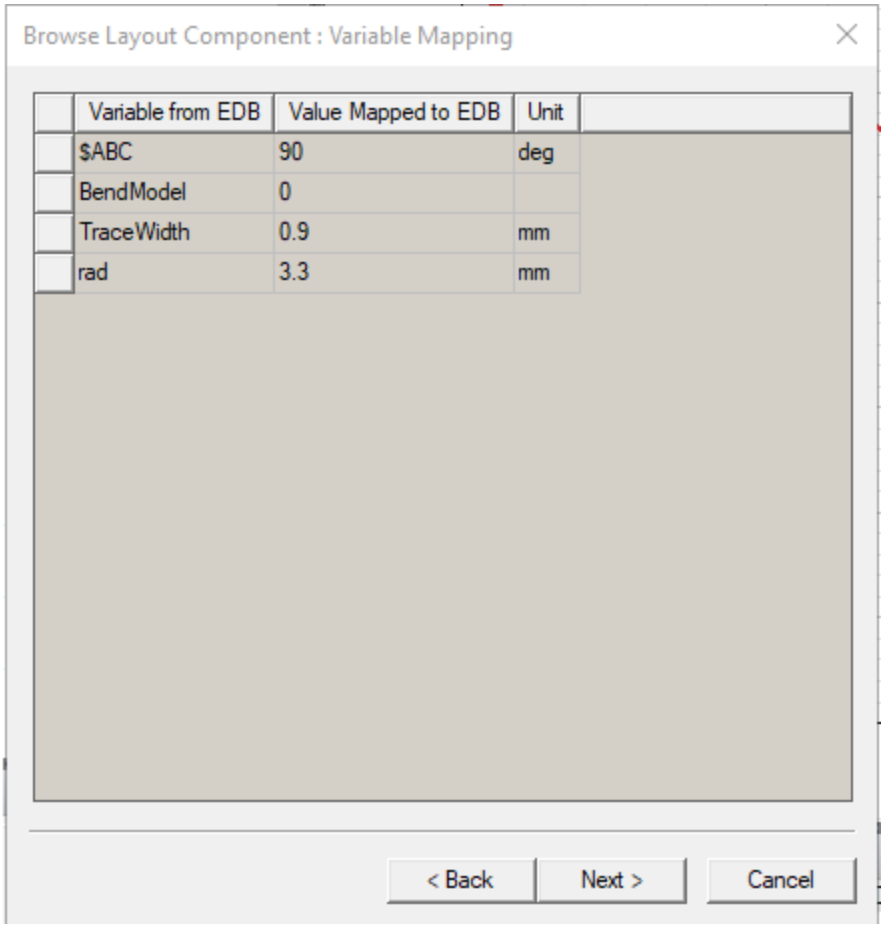
Flex PCB Component Support in HFSS 3D

Users can now export Flex PCBs designs (i.e., components with rigid-flex bends) from HFSS 3D Layout as components and import them to HFSS 3D. All regular component functionality is supported for components with rigid-flex bends (e.g., visualization attributes, validation checks, component coordinate systems, circuit elements visualization, net identification, mesh fusion, postprocessing, and so forth).



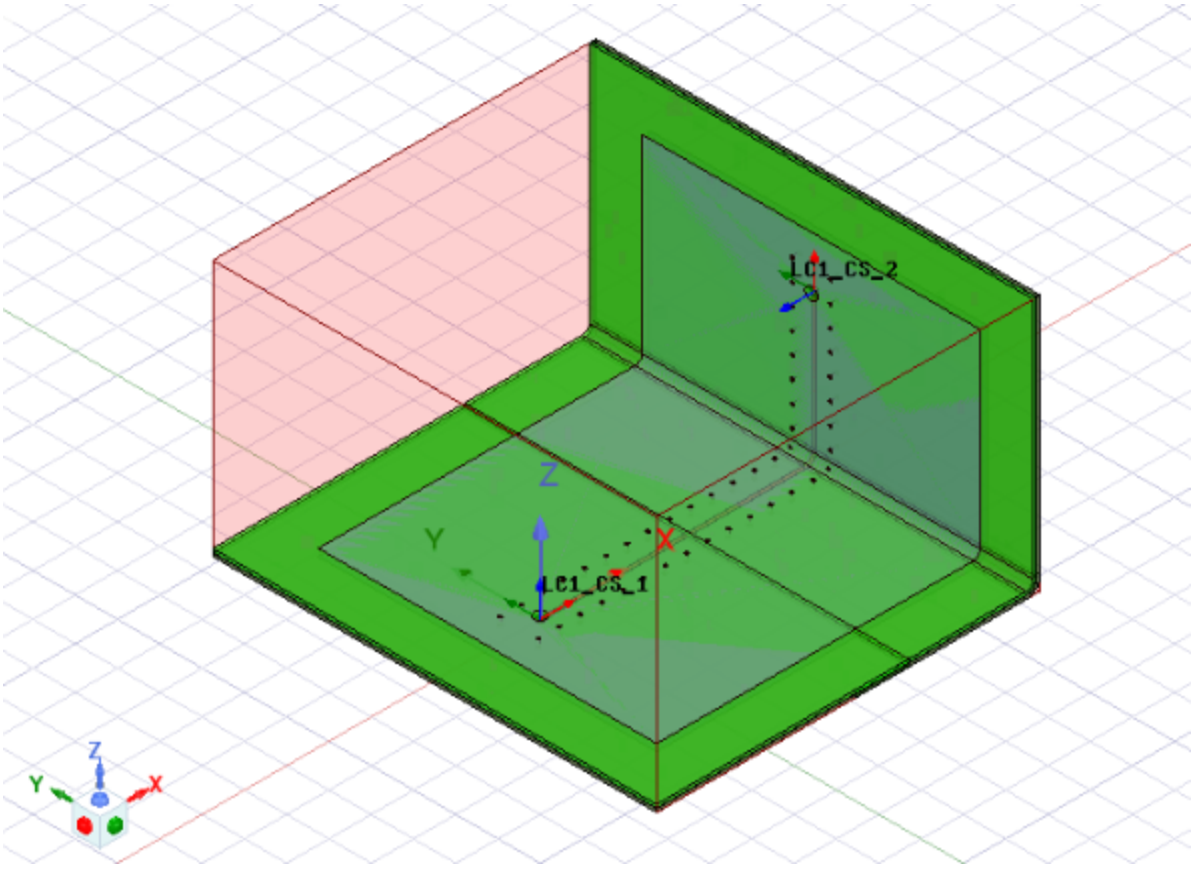
When components with rigid-flex bends are imported, the **BendModel** variable appears in the **Browse Layout Component : Variable Mapping** window, in addition to other variable types shared by other components in the definition design. When the value of **BendModel** is 0, all rigid-flex bends on the components are inactive. Otherwise, the bends on the components are active.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



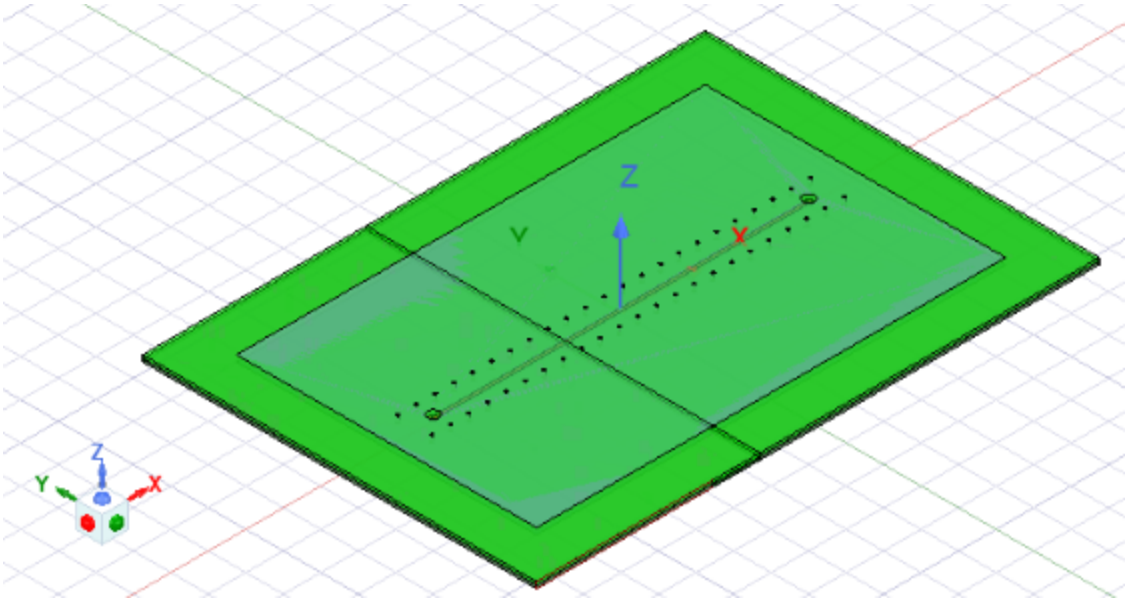
When rigid-flex bends are active, all ports, circuit elements, and component coordinate systems on the bent part of the components are transformed to accommodate the bends.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

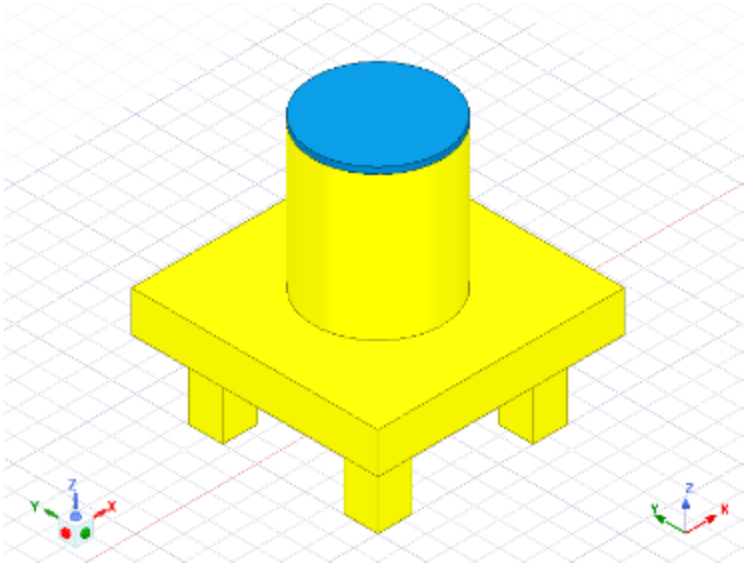


If bends are inactive (i.e., **BendModel** = **0**), all bends in components are inactive, and the components appear in their unbent states.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

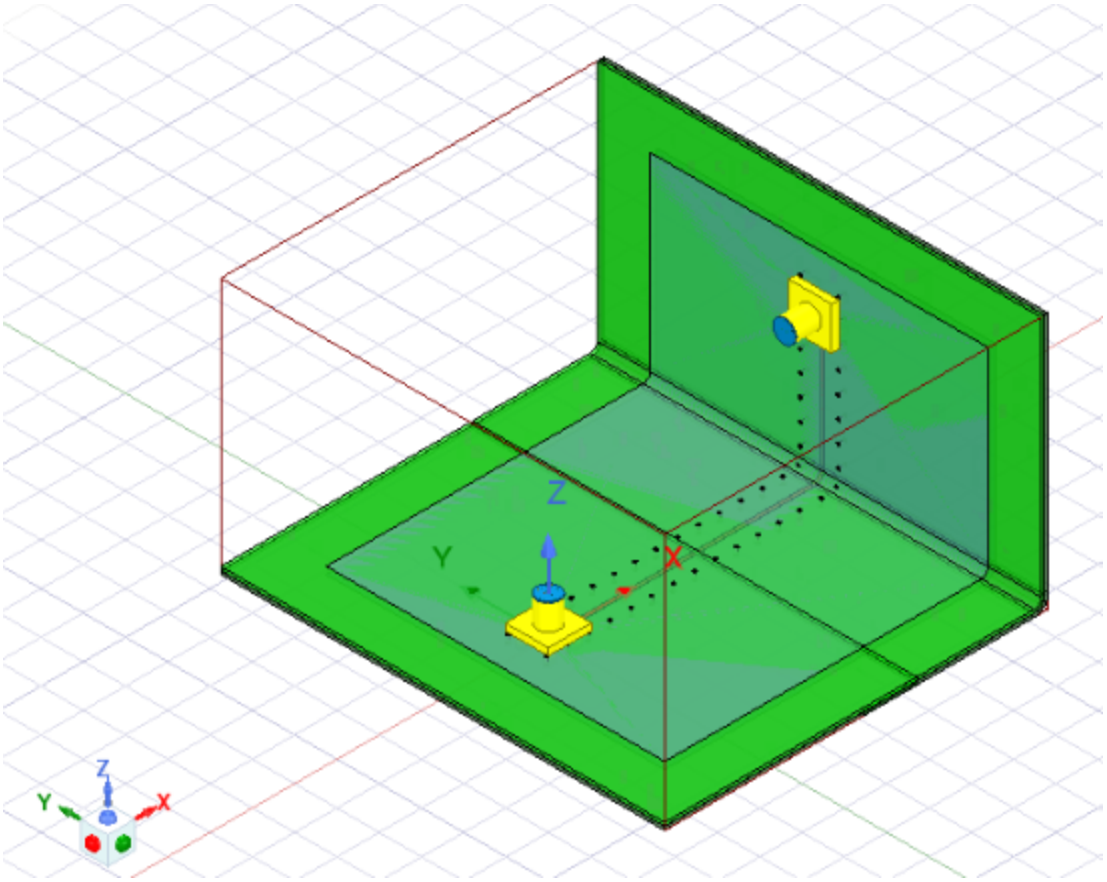


With bends re-enabled, other 3D components can be imported and assembled with bent components.



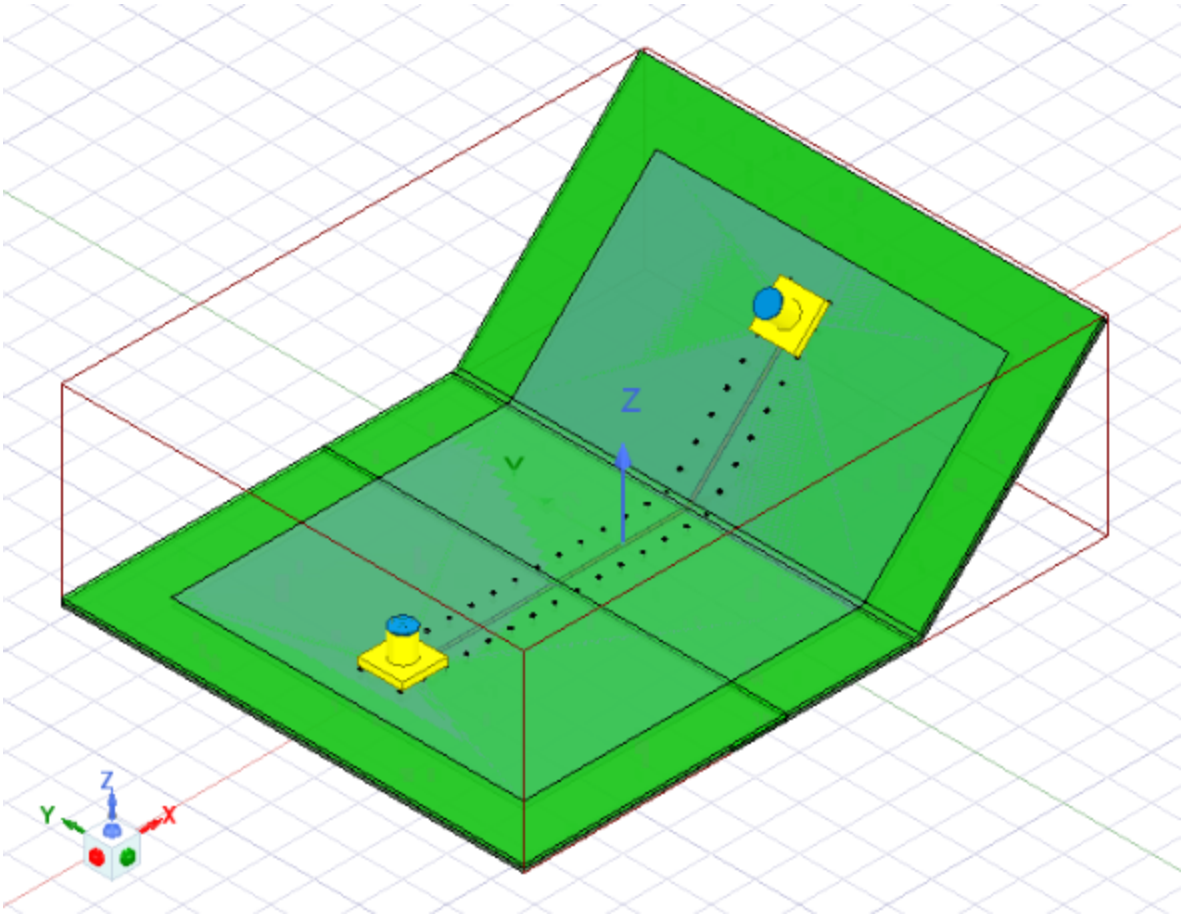
The final assembled system may match the following example (i.e., two 3D components placed on the two coordinate systems of the component).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



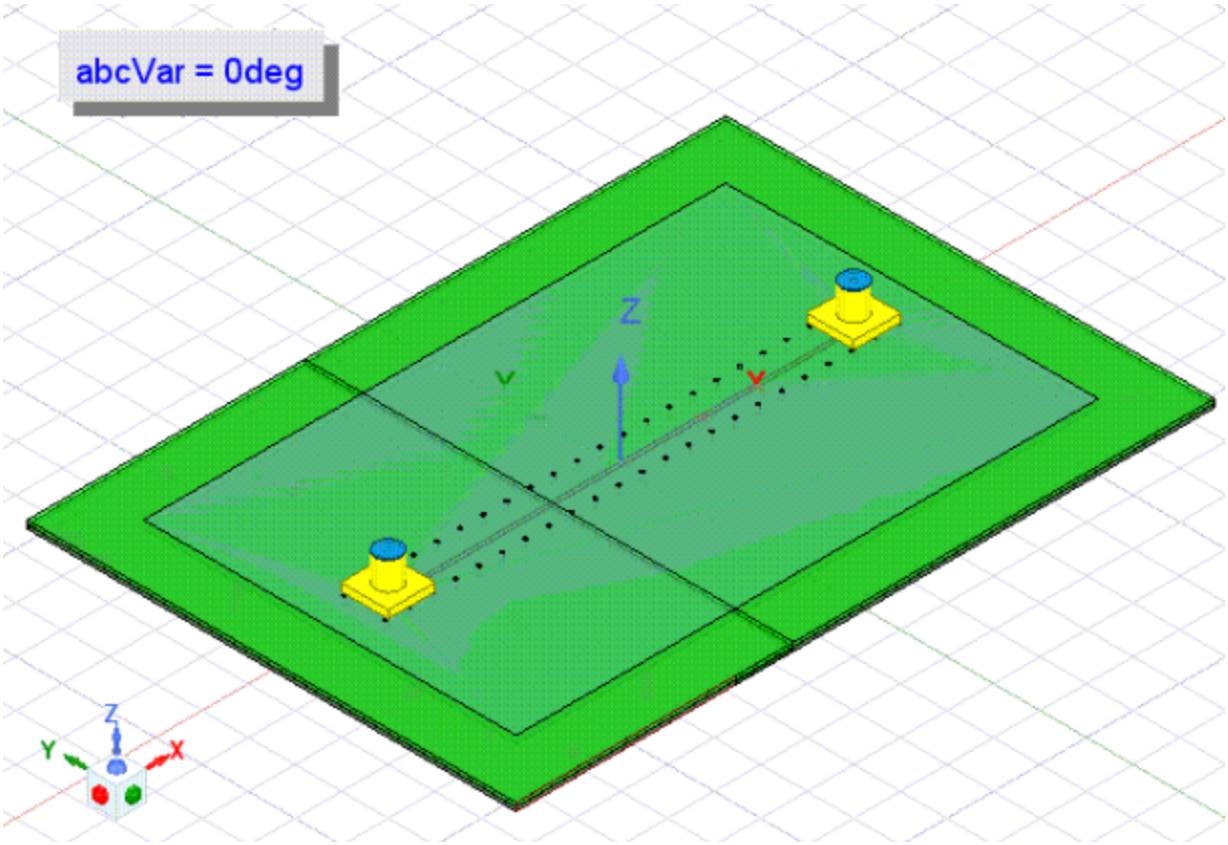
In the following example, the angle of the bend is parametrized with a variable. When components are imported to HFSS, the parametrized variable can be used to control their bends. If the value is changed, the bend angle of the components update to reflect the change. Other parts of the design that depend on the bend angle are also updated.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



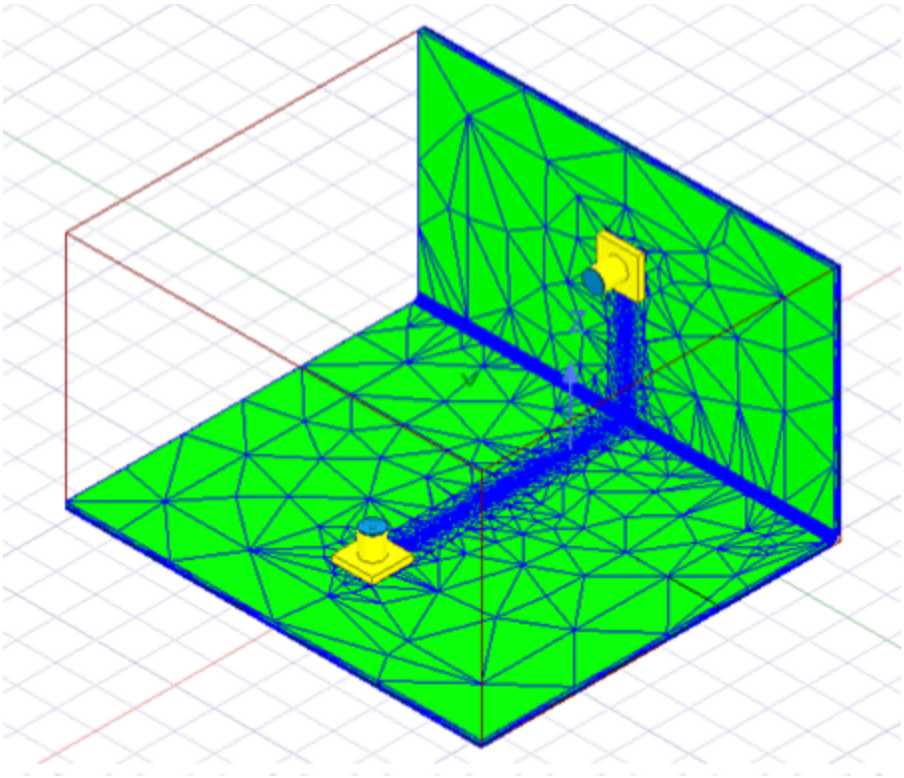
Users can also generate an animation based on the different values of the bend angle.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



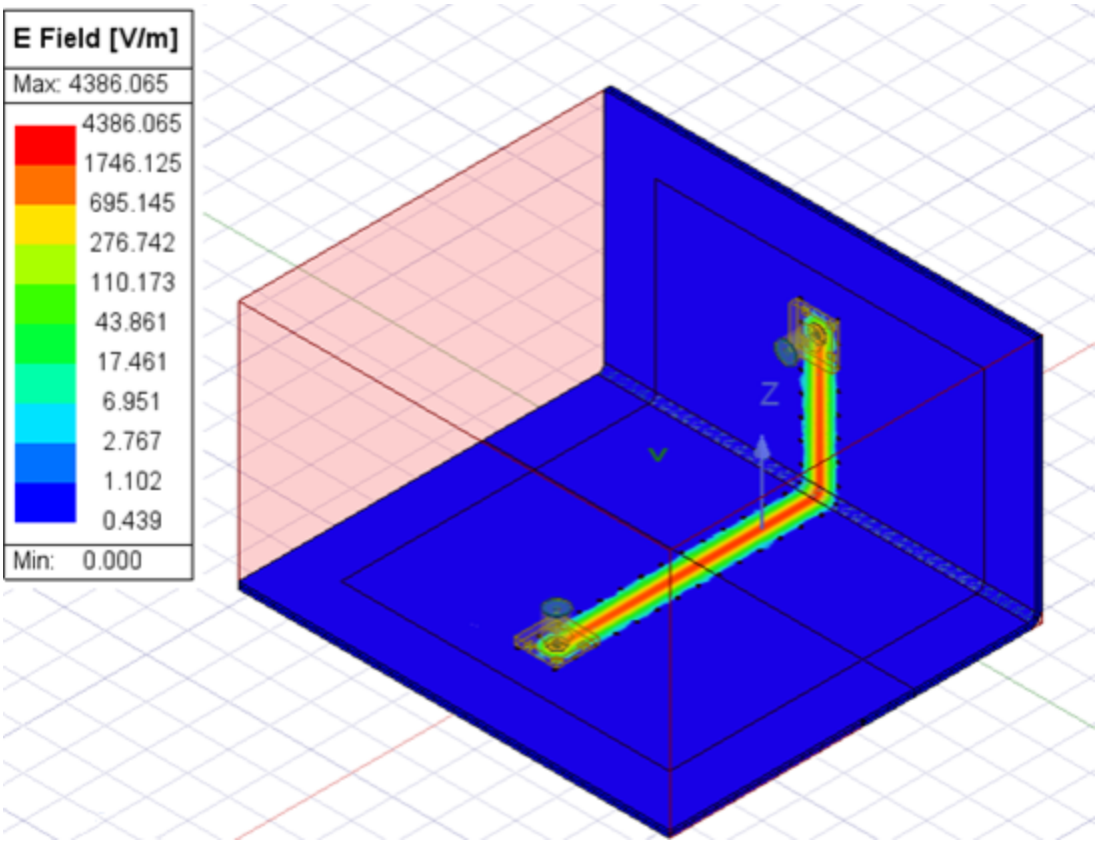
Components with rigid-flex bends can also be meshed and simulated. All the simulation and post-processing functionality for regular components are supported for components with rigid-flex bends and the workflows are identical; plot the mesh-selected nets and layers of the components, as appropriate.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



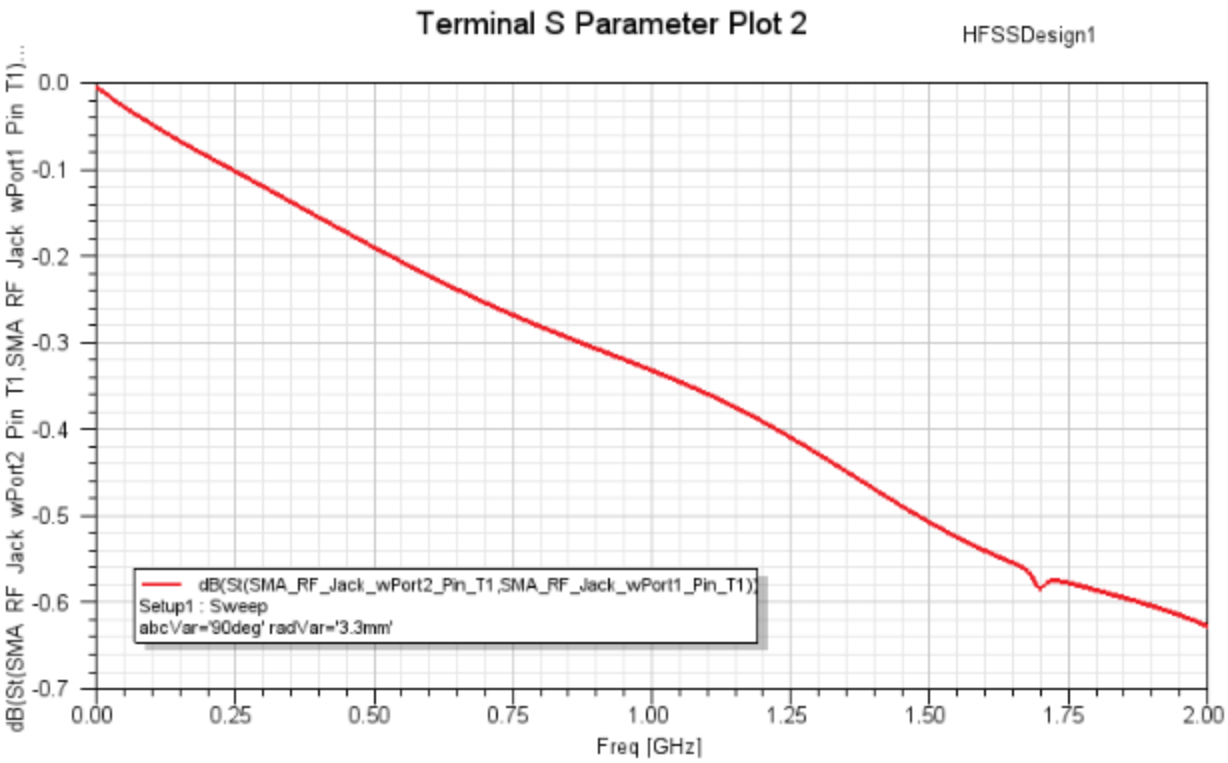
Or plot the fields-selected nets and layers of the of the component, as appropriate.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Finally, create plots (e.g. S Parameter plots).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Field Plots on Layout Components Layer or Net

If an HFSS design includes Layout Components, you can plot field quantities on layout components' layers and nets. This functionality works seamlessly with the capabilities of plotting fields on 3D geometries. Additionally, the Fields Calculator can calculate field quantities on certain objects inside a layout component that is only drawn in the modeler window without actual geometry.

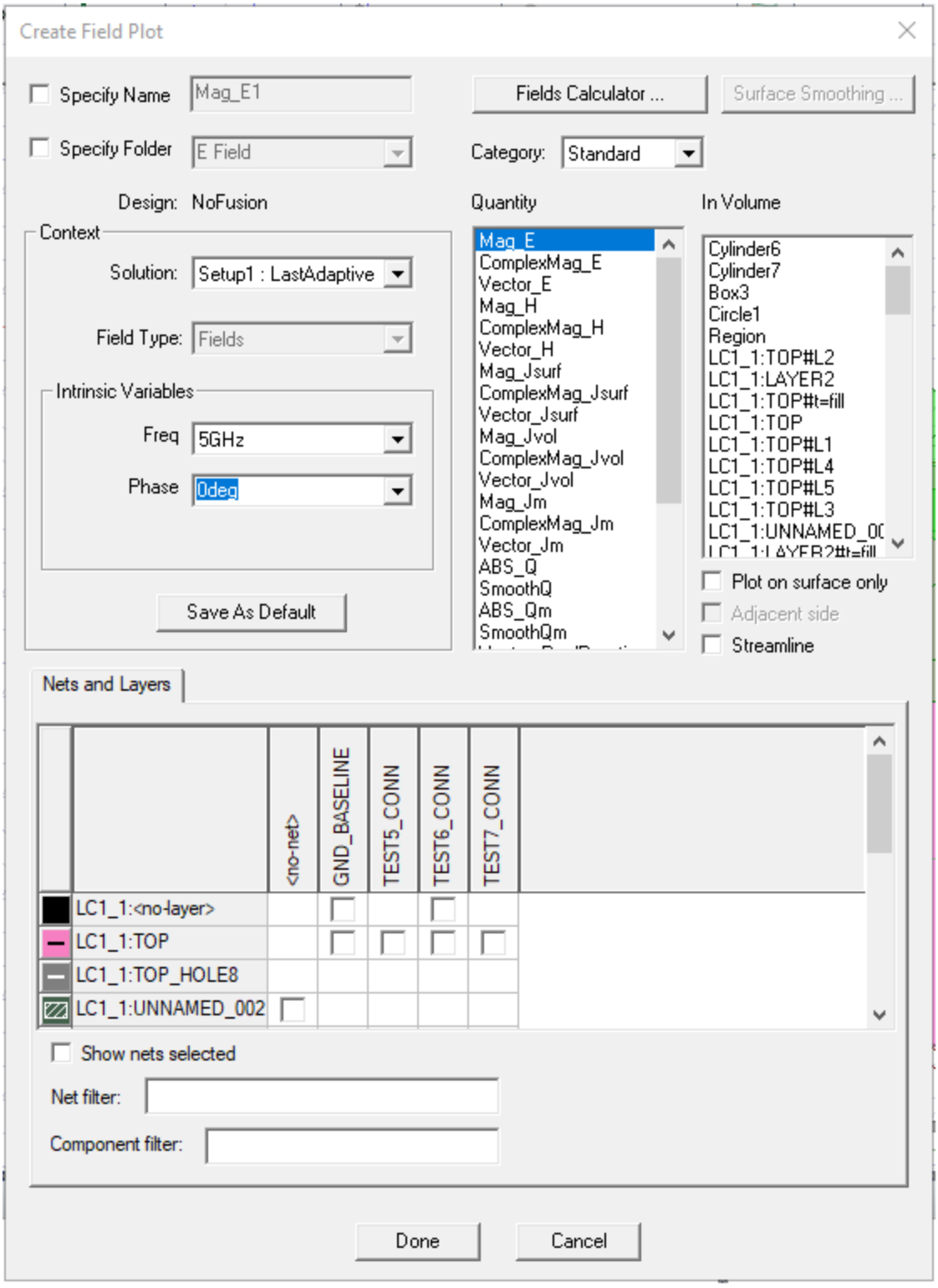
To Plot Fields for a Design Including Layout Components

First, analyze the design. During simulation setup, ensure the Save Fields option is selected. Designs analyzed before release 2023R2 must be analyzed again to extract the necessary information for plotting.

Whereas for designs without layout components, you must select a geometry for the plot before you select the **Plot Fields** command, for designs with layout component, you make selection both before and after selecting **Plots Fields** to open the **Create Field Plot** dialog box.

For designs that include Layout Components, the **Create Field Plot** dialog box includes a Nets and Layers panel that provides additional functionality, showing the layers and nets of every layout component in the design. The **In Volume** field also allows selection of geometries from the layout components.

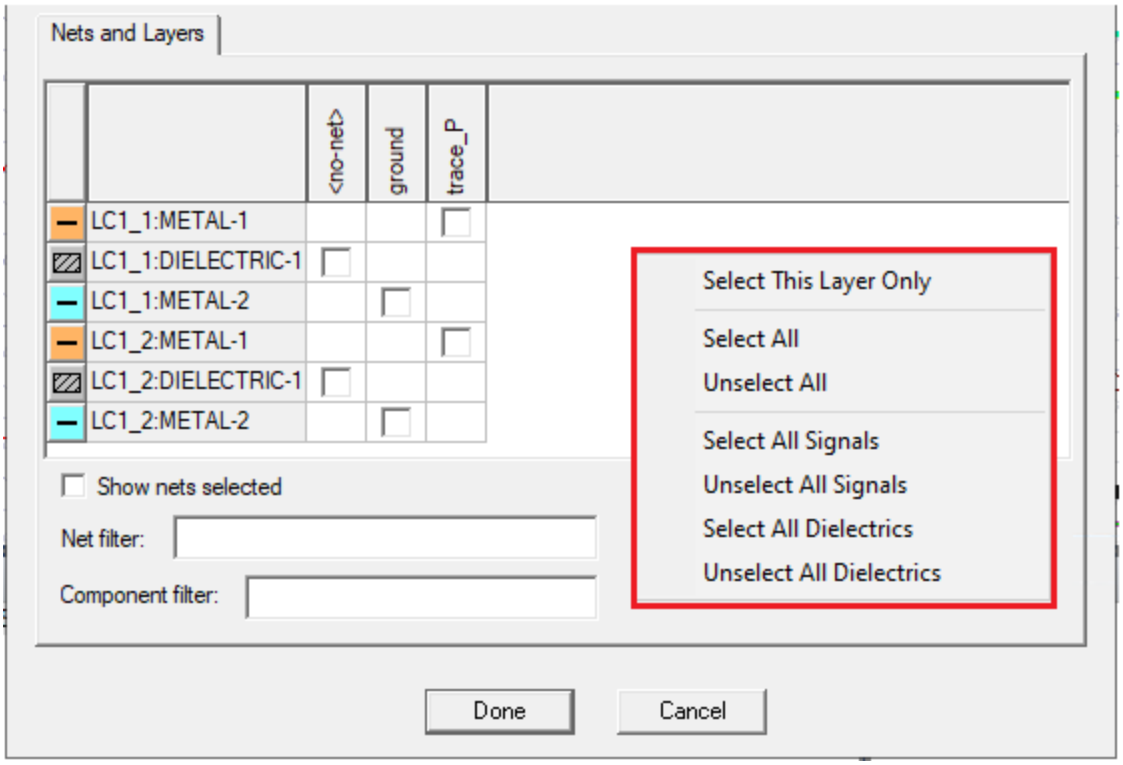
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This **Nets and Layers** section displays all the layers and nets of every layout component in the design. Upon opening, it adjusts its size to try to show all the rows and columns for the layout components. You can resize the parent dialog box. For objects in the component belonging to a specific layer and net, you can select a checkbox. By clicking the column headers, you can select or unselect check boxes for an entire column. You can toggle the “Show nets selected” checkbox to move all the selected nets to the left side of the table. You can use the **Net filter** and **Component filter** fields to filter out specific nets and components. The net and component filters support ‘*’ and ‘?’ wildcards and you can delimit multiple filters with commas. The Nets and Layers table is refreshed each time you update the filter text.

If you right click on the Nets and Layers panel, a shortcut menu appears that lets you **Select This Layer Only**, **Select** or **Unselect All**, **Select** or **Unselect All Signals**, or **Select** and **Unselect All Dielectrics**.

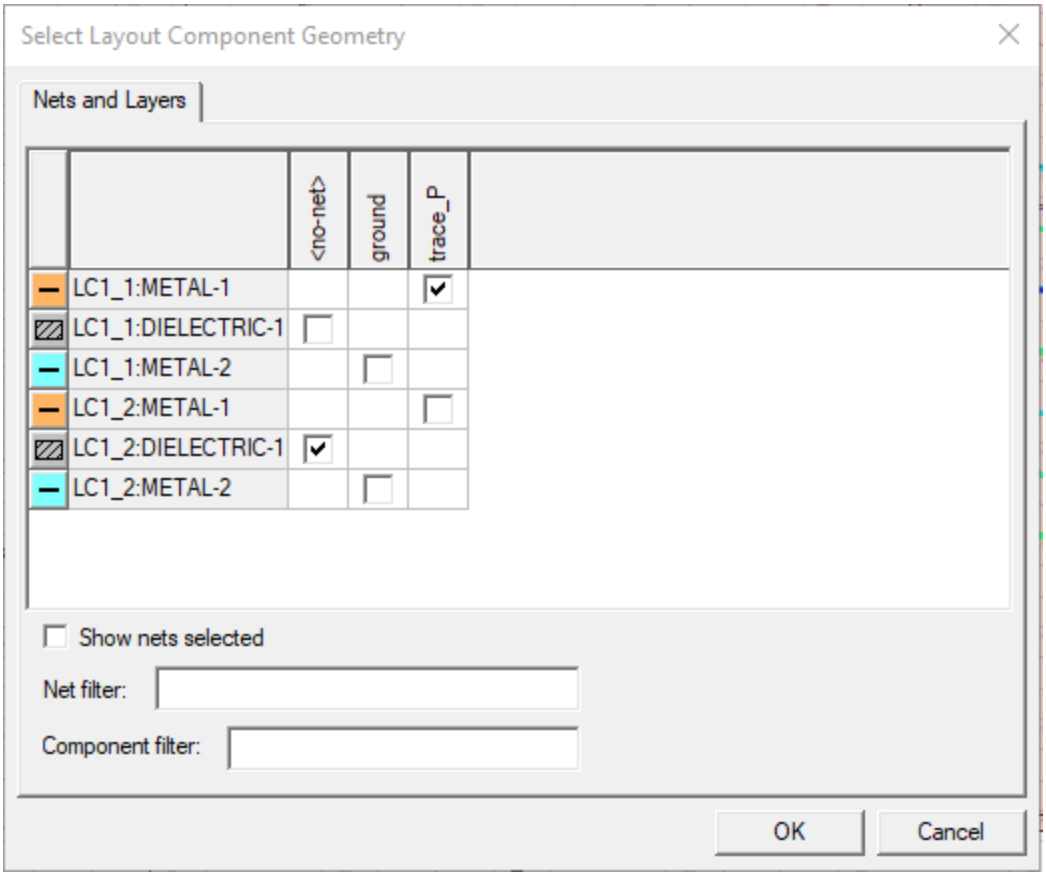


You select Context, Quantity and In Volume information as usual for Plotting Field Overlays.

Once you click **Done** to apply and close the dialog box, HFSS plots the fields on all objects belonging to the selected layers and nets, in addition to any 3D geometries selected from the modeler window.

If there are layout components in the design, and you want to reassign an existing field plot or mesh plot, a new dialog box will pop up as shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This dialog box allows you to reassign the existing plot on layout components' nets and layers. As with the **Nets and Layers** sub dialog for the **Create Field Plot** dialog box, this dialog box shows layers and nets of every layout component in the design, with identical behavior and usage, including the short-cut menu that lets you Select This Layer Only, Select or Unselect All, Select or Unselect All Signals, or Select and Unselect All Dielectrics.

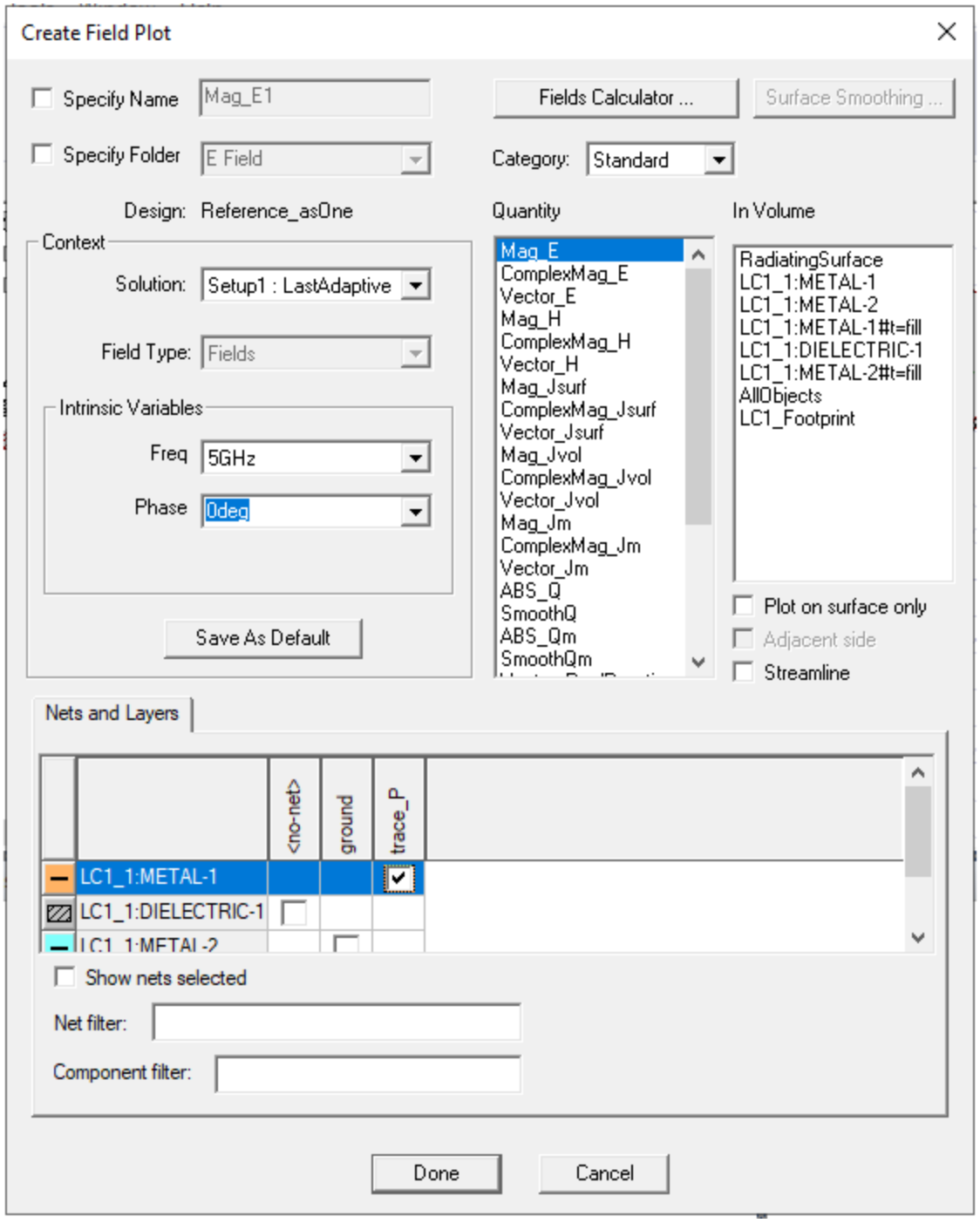
Once you click **OK**, HFSS reassigns the field/mesh plot on all objects belonging to the selected layers and nets, in addition to any 3D geometries previously selected from the modeler window. If you click **Cancel**, HFSS will only reassign the field/mesh plot on 3D geometries previously selected. If there are no previously selected 3D geometries, the program issues an error message.

If there are no layout components, this dialog box is not shown, and the workflow for reassigning field/mesh plot is unchanged.

Example with Metal Trace from Layout Component

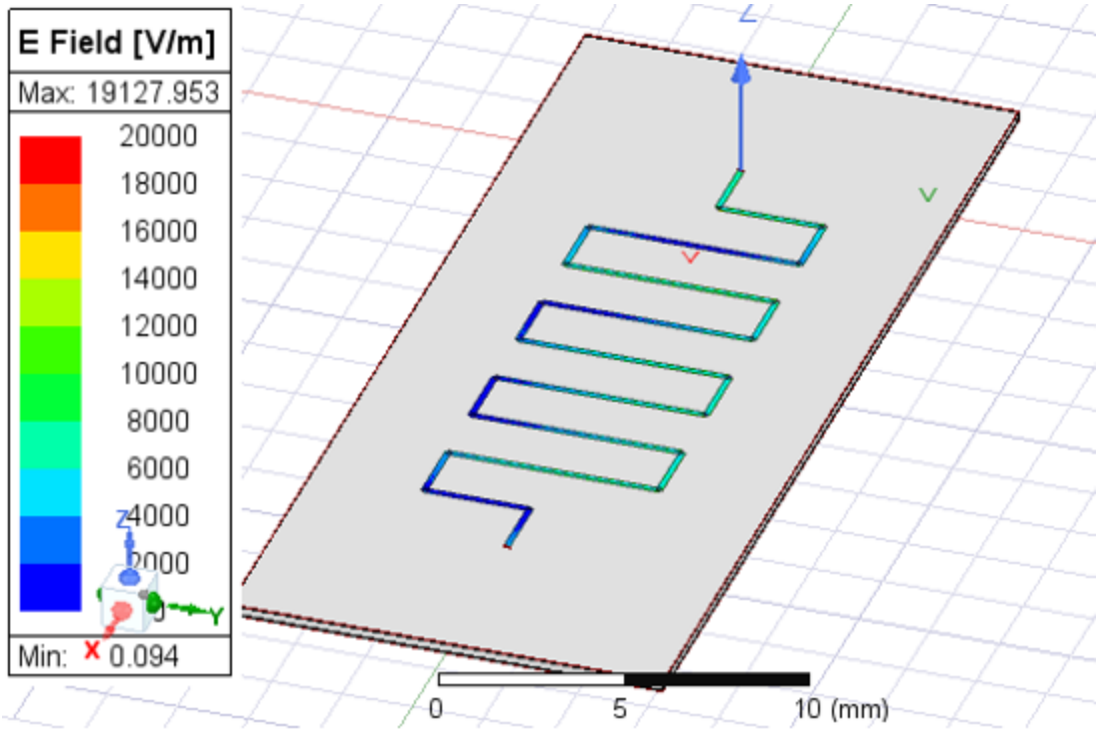
For example, you could select a metal trace from a layout component in the Nets and Layers table as shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



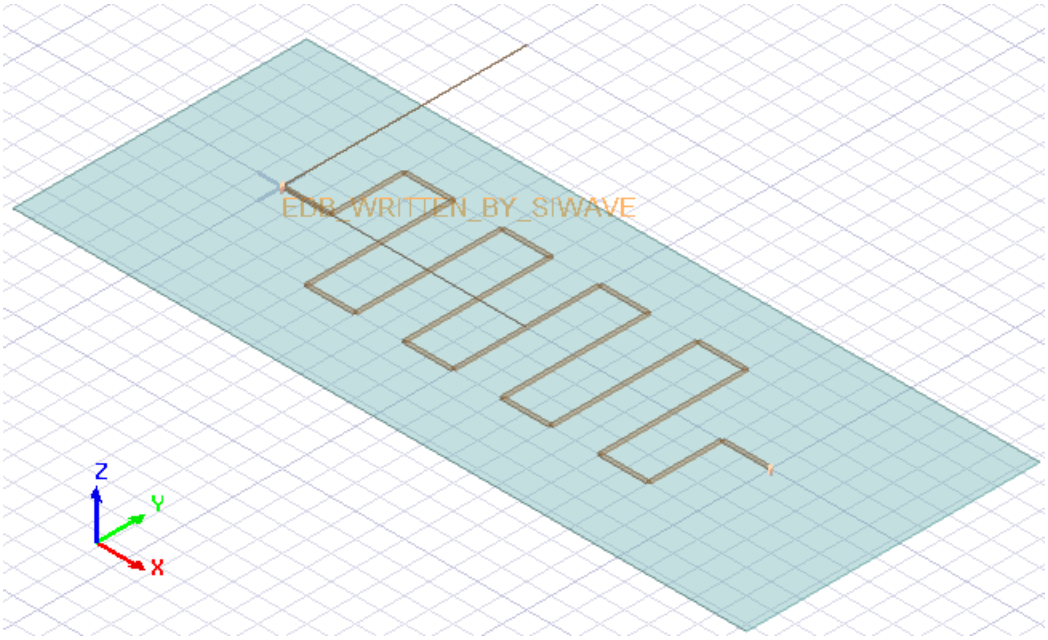
The Model window will show the field plot as follows.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Using Fields Calculator with Layout Component in a Design

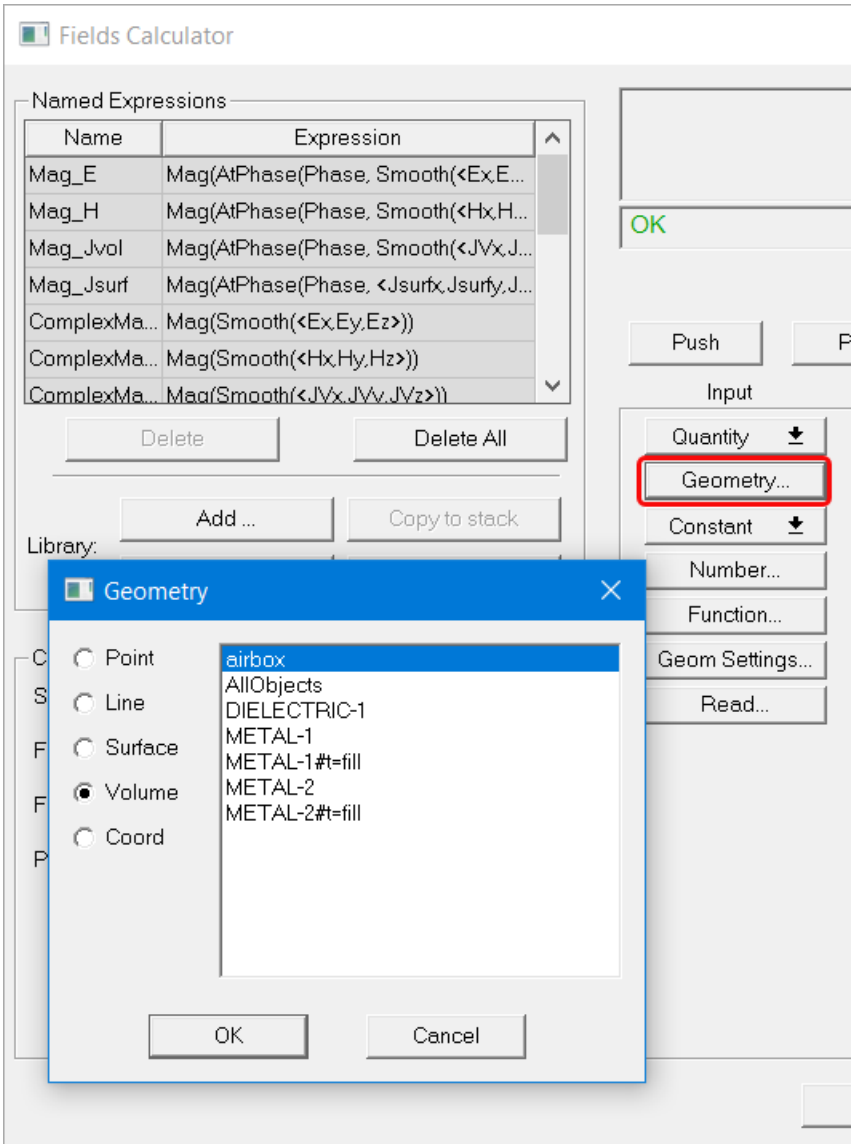
If there are layout components in the design, users can create volume or surface geometry operators in Fields Calculator using any objects from layout components.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

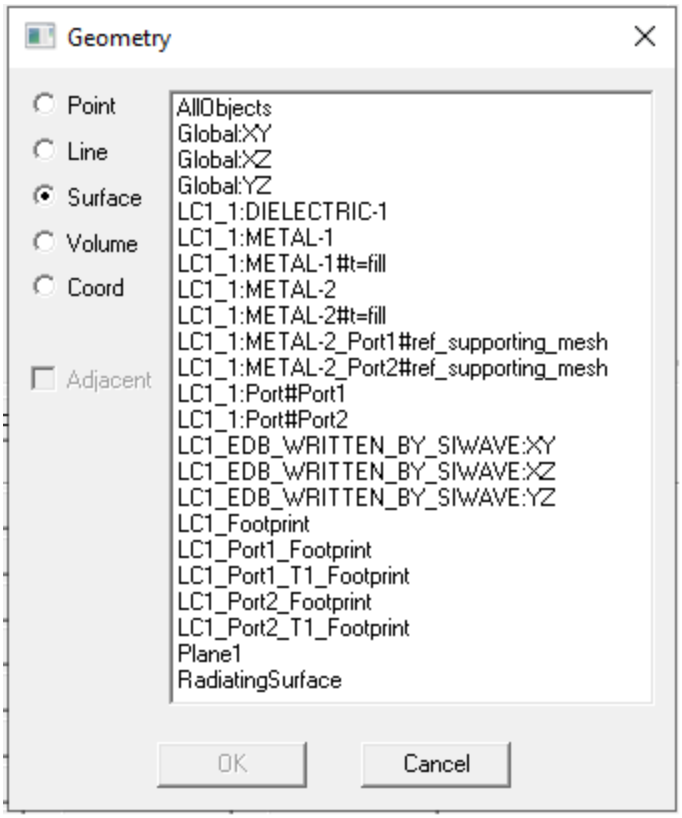
First, analyze the design. Designs analyzed before release 2023R2 must be analyzed again to extract the necessary mesh and fields information for the Fields Calculator.

Open the Fields Calculator and create a Geometry operator. The following figure shows the **Geometry** dialog with Volume selected, showing Layout Component objects.



The following **Geometry** dialog shows layout components added for the Surface section.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the modeler window, these Layout Component objects are drawn as visualizations without actual geometry but can now be selected and used to calculate field quantities as shown below.

These geometry operators can be used to perform computations just like one constructed from regular 3D geometry. Remember, these Layout Component objects are generated only after running a simulation, so they won't show up if there is no solution.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Related Topics

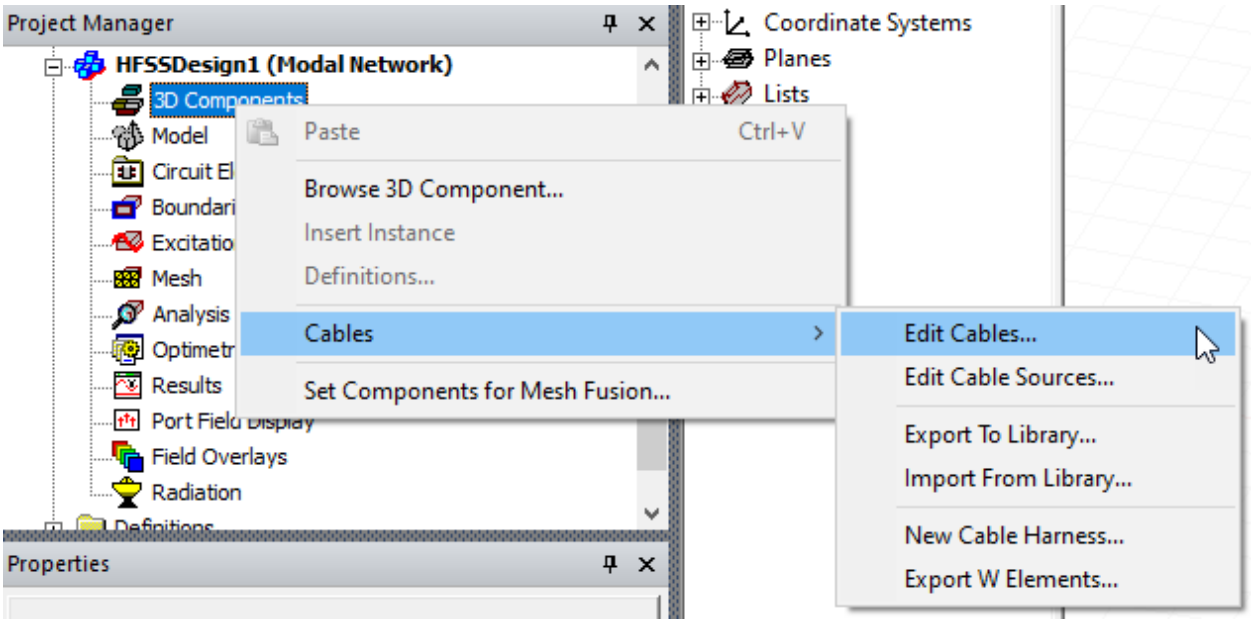
Layout Component in HFSS 3D

Plotting Field Overlays

Using the Fields Calculator

Cable Modeling in HFSS (Beta)

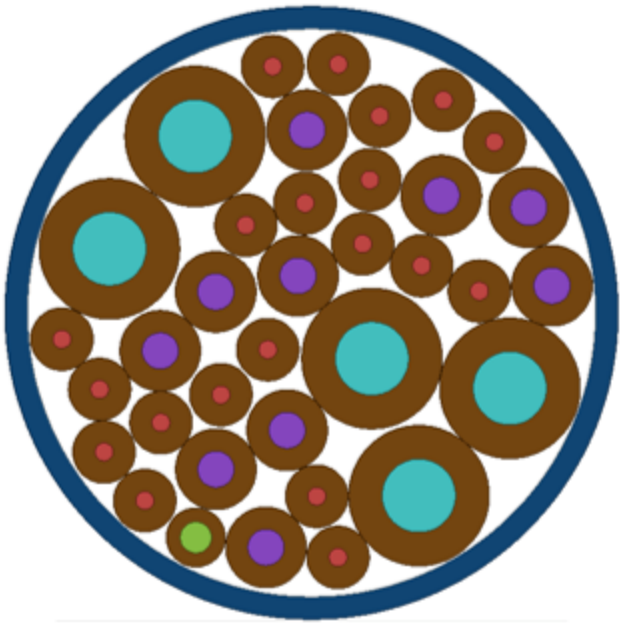
A Beta feature in HFSS lets you can add and edit new cables and harness and solve them within HFSS.



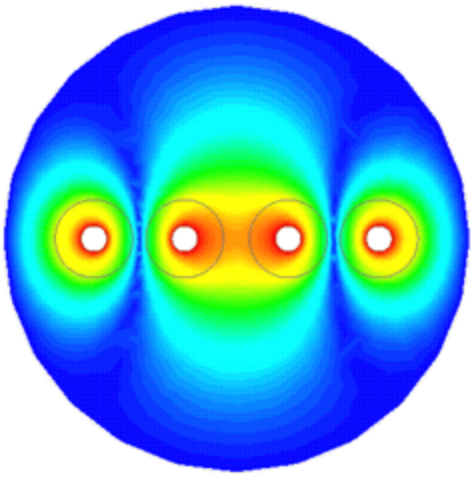
This menu provides access to editors for new and existing cables, editing cable sources, export to and import from libraries, cable harness support, as well as W-element export. The feature lets you create cables and cable bundles, and drag the cables into the bundle. The feature supports straight wire and twisted pair cable. Bundle jacket types include insulation and braided

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

shield. The software automatically generates the cross sections based on a list of conductors and their diameters. This means you have no need to draw complicated geometries. The cable editor supports hierarchy, which enables infinite nesting of bundles within bundles.

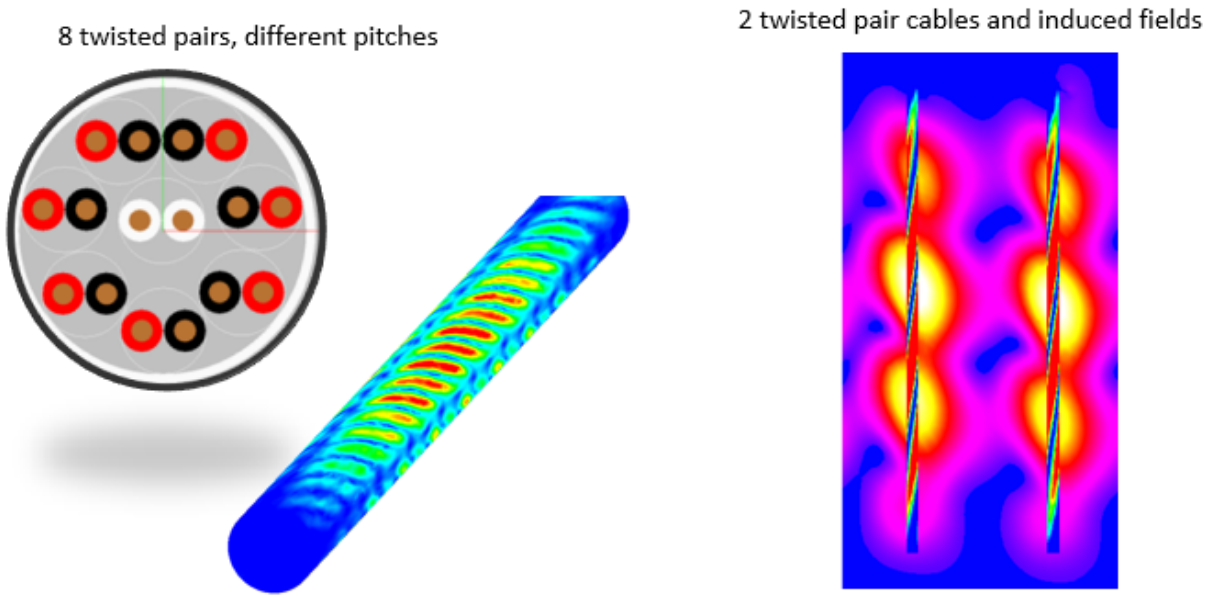


The fields on the cable's perimeter are radiation sources into the surrounding 3D domain. Fields depend on position, terminal excitations, twisting rates. The software automatically maps the 2D solutions (weighted by position-dependent voltage and currents) to the 3D cable surfaces.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Cable modeling feature employs a hybrid 2D approach: It generates “samples” of twisted pair cross sections at many locations along the cable. It does 2D solution for RLGC parameters and fields for each cross section. These calculations can be done in parallel. It then uses a semi-analytic transmission line model to compute voltages and currents at any position along the cable length.

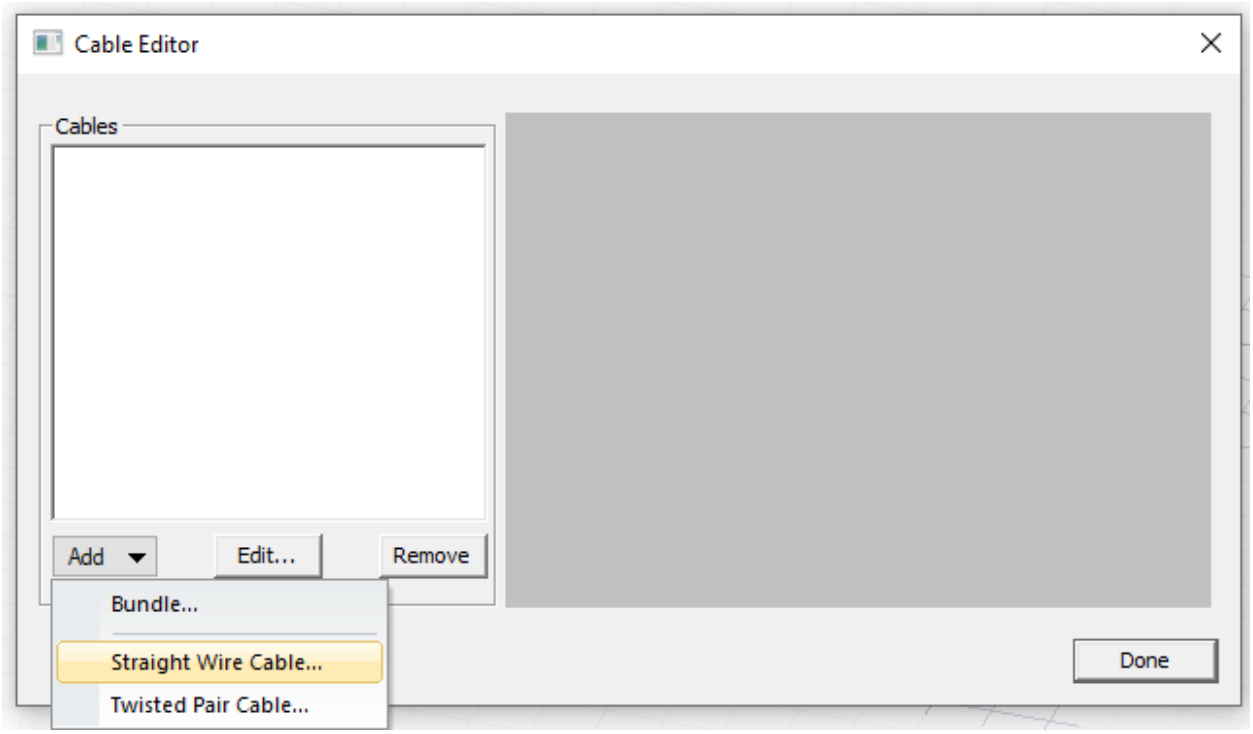


You can combine the cable model with additional 3D geometry (for example, a metallic enclosure or the chassis of a vehicle) to see how the fields radiated from the cable interact with surrounding structures. You can also export W-element models of the cable for circuit simulations in Nexxim or HSPICE.

HFSS Cable Modeling Editor Window

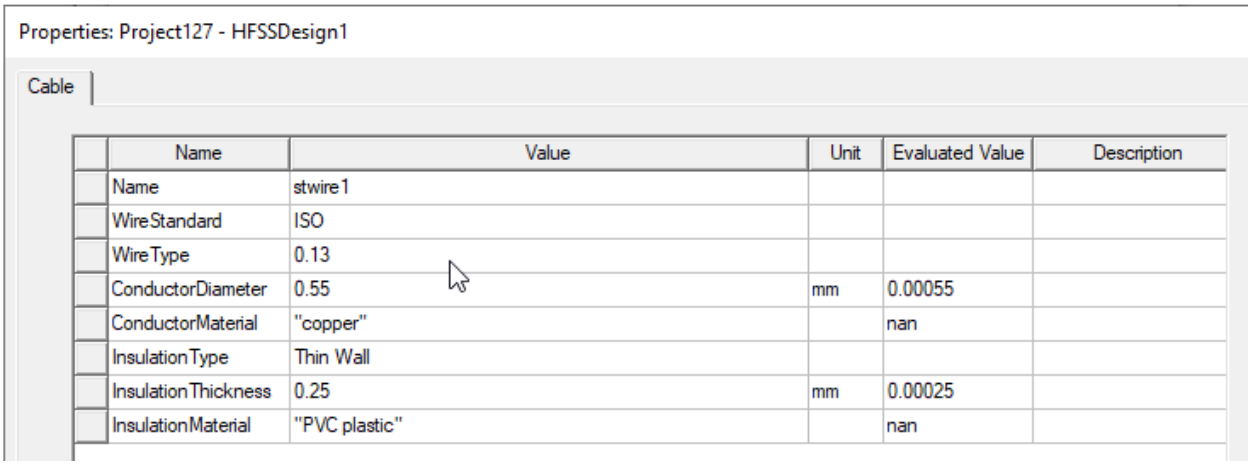
Execute the **Cables** > **Edit Cables** command to open the editor window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Beginning with a Straight Wire Cable

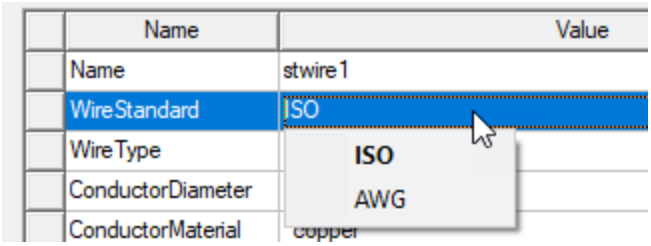
Though you can build libraries of cables, pairs, bundles and so forth, to begin, first create a Straight Wire Cable..., by selecting from the drop down menu. This opens an editable Property window.



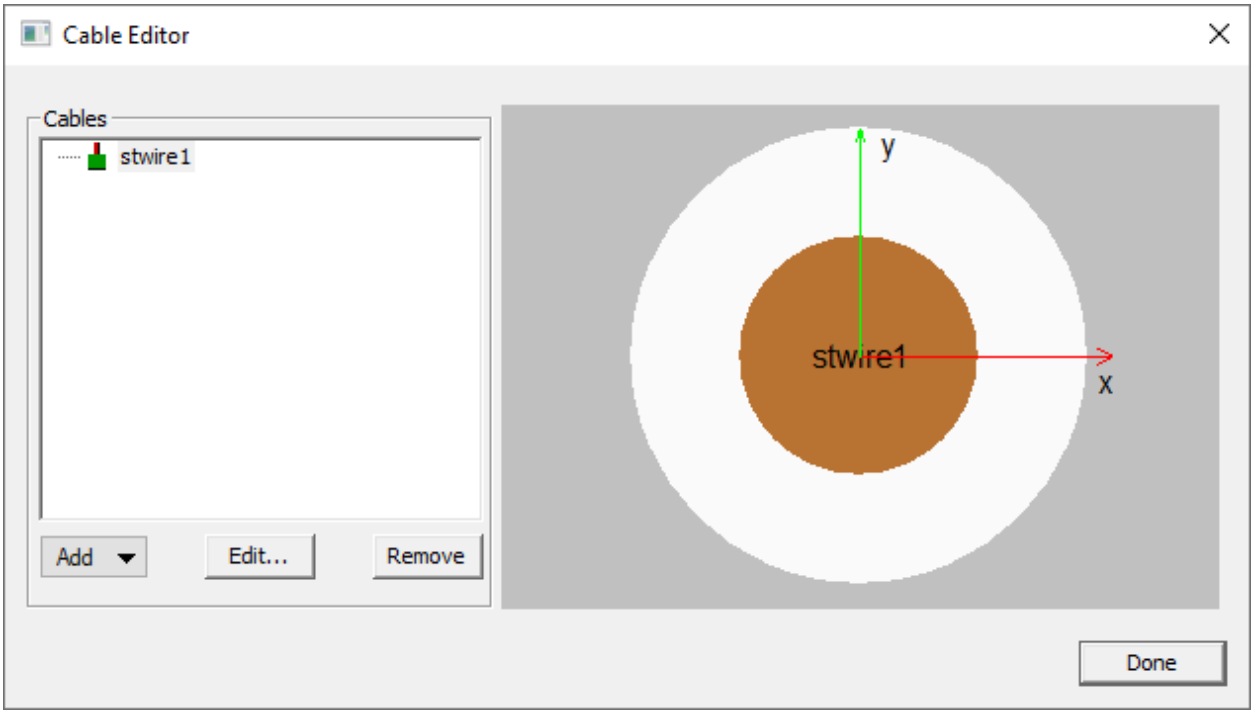
Name	Value	Unit	Evaluated Value	Description
Name	stwire1			
WireStandard	ISO			
WireType	0.13			
ConductorDiameter	0.55	mm	0.00055	
ConductorMaterial	"copper"		nan	
InsulationType	Thin Wall			
InsulationThickness	0.25	mm	0.00025	
InsulationMaterial	"PVC plastic"		nan	

You can edit the properties by typing values or by selecting from drop-down menus for Wire Standard, Wire Type, Insulation Type, or for Conductor or Insulation Material.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



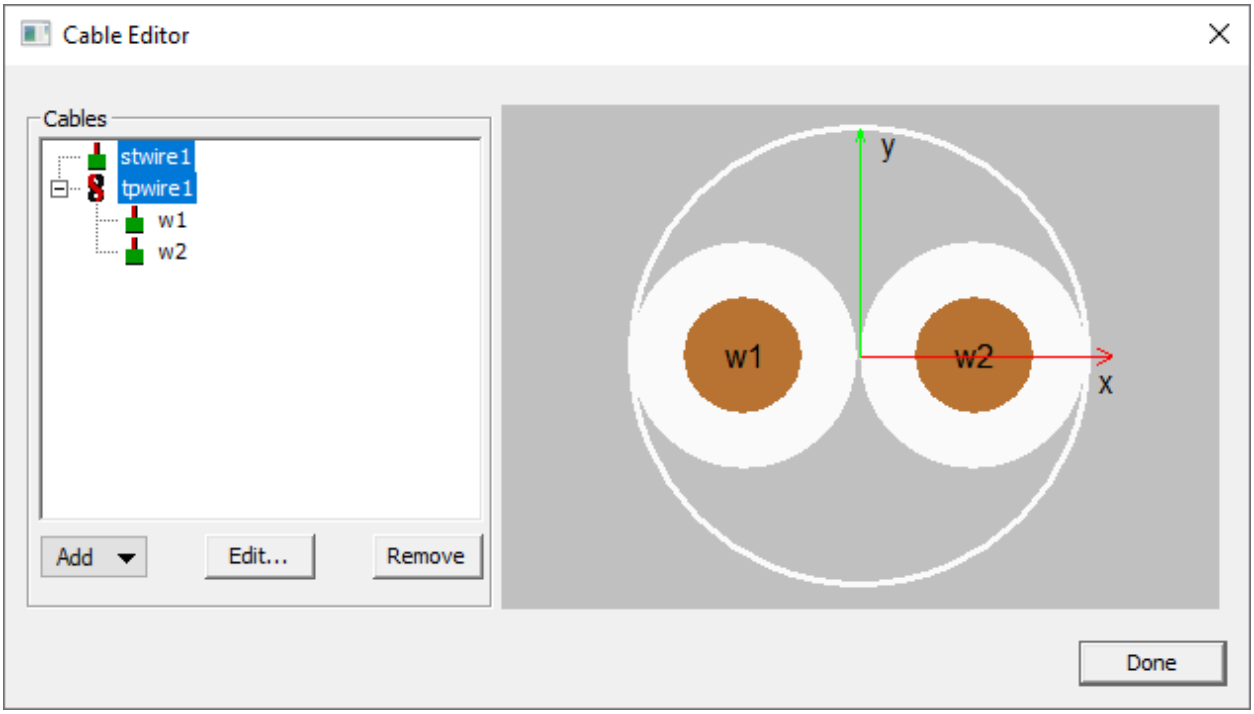
When you have specified the desired name and parameters, **OK** the Properties dialog box. The **Cable Editor** shows the current wire.



Adding Twisted Pair Cable

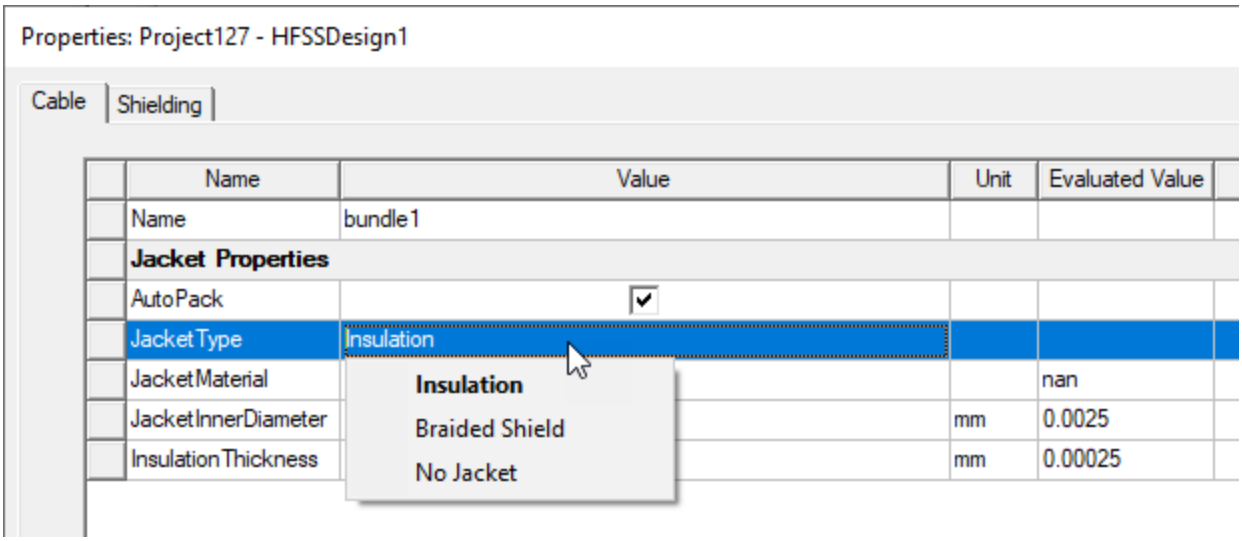
If you then select **Add>Twisted Pair Cable...**, the **Cable Editor** shows the result.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Adding Bundles

If you select **Add> Bundle...**, a Property dialog with tabs for **Cable** and **Shielding** opens. The cable editor supports hierarchy, which enables infinite nesting of bundles within bundles.

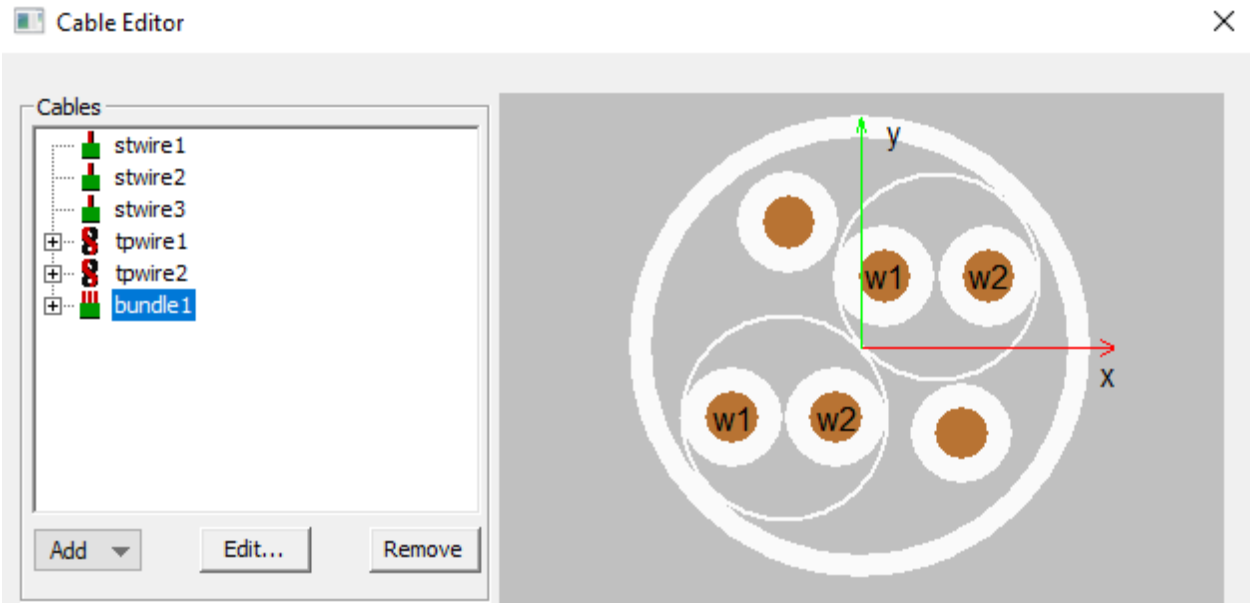


In the **Cable** tab, you can specify name, whether to AutoPack, select Jacket type from a drop down.

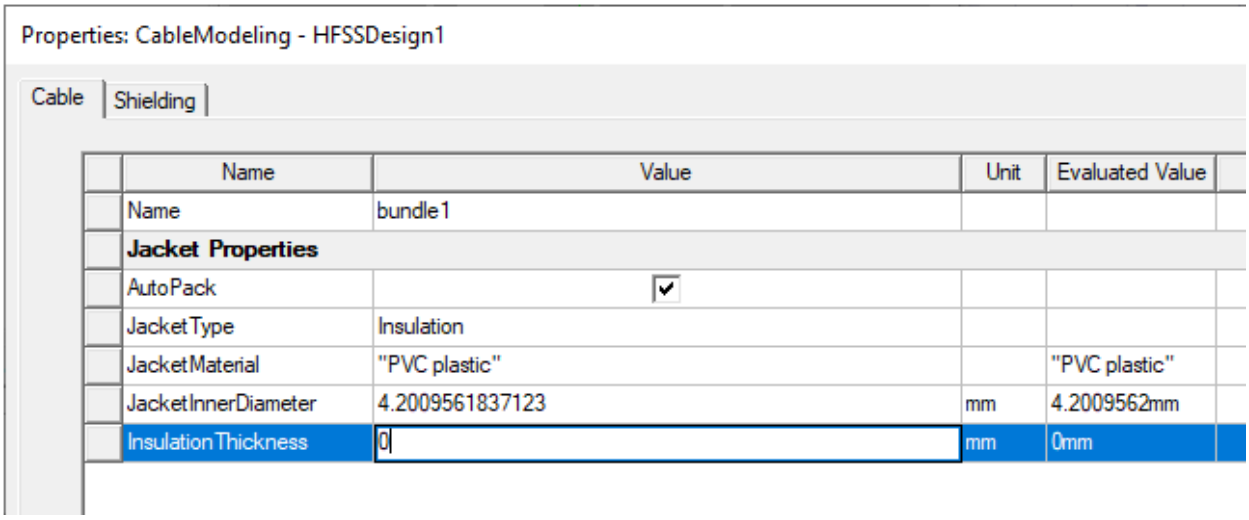
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Jacket type can be Insulation, Braided Shield, or No Jacket.

If you select Insulation, the Cable Editor image the bundle will resemble the following:

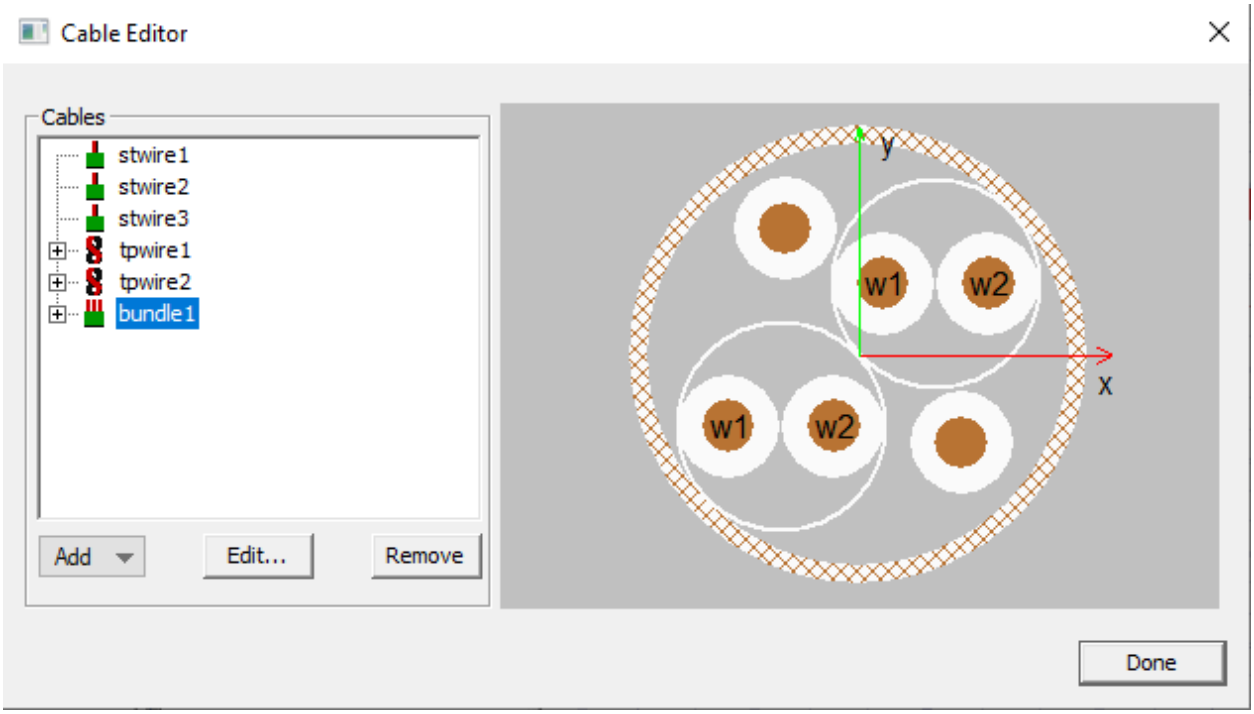


You then specify Properties for Jacket Material, inner diameter, and insulation thickness:



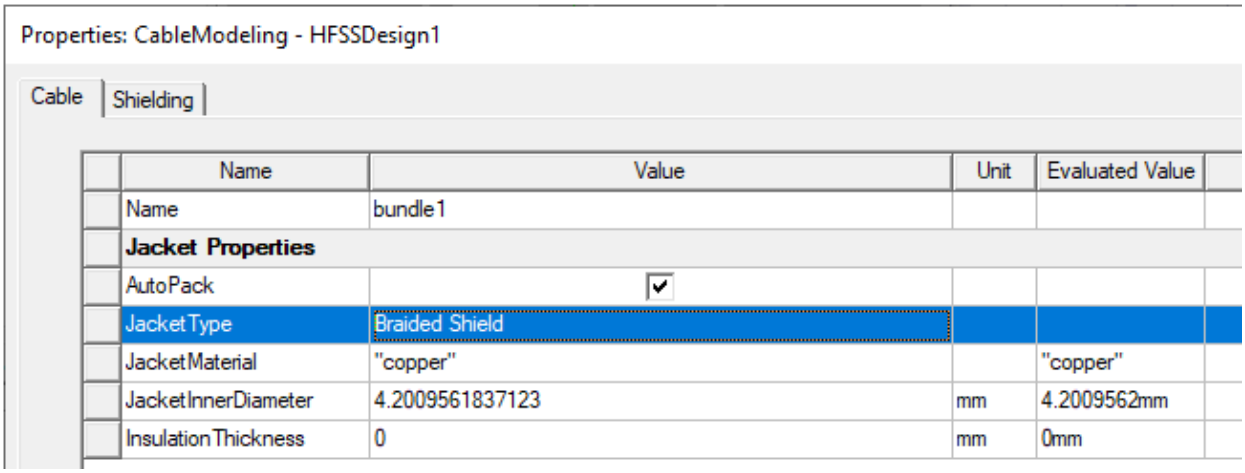
If you select Braided Shield, the **Cable Editor** image of the bundle will resemble the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



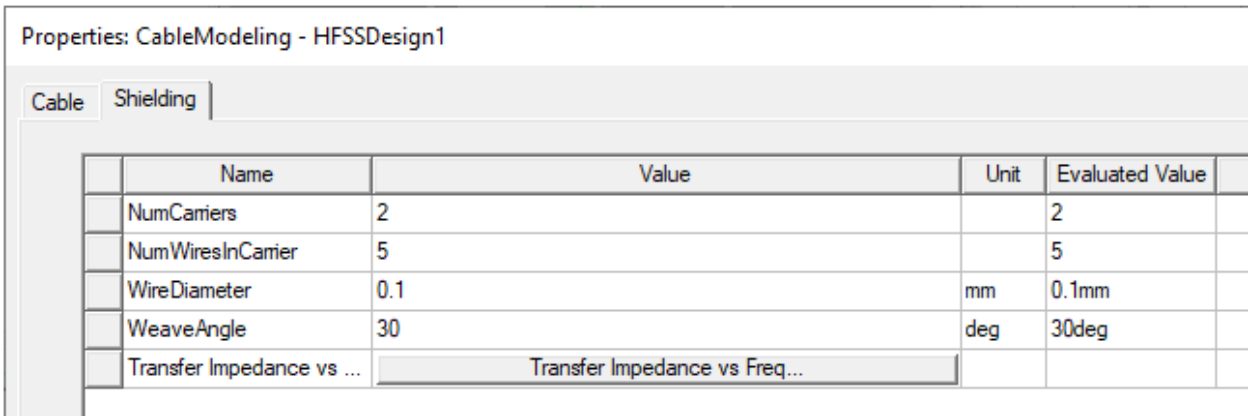
For details see *Technical Notes*: Braided Shield Models for Cable Modeling.

For Braided Shield, specify Jacket Material. You cannot edit the inner diameter and insulation thickness.

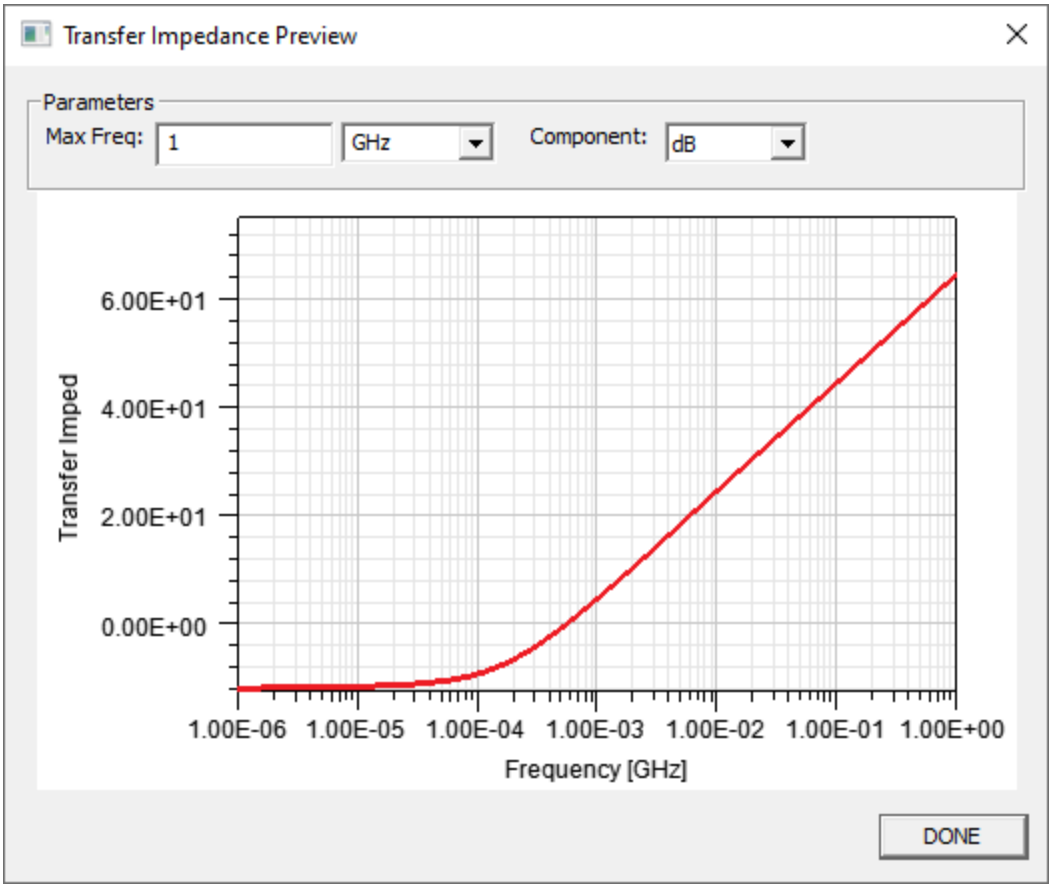


For Braided Shield, the **Shielding** tab has a set of parameters:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



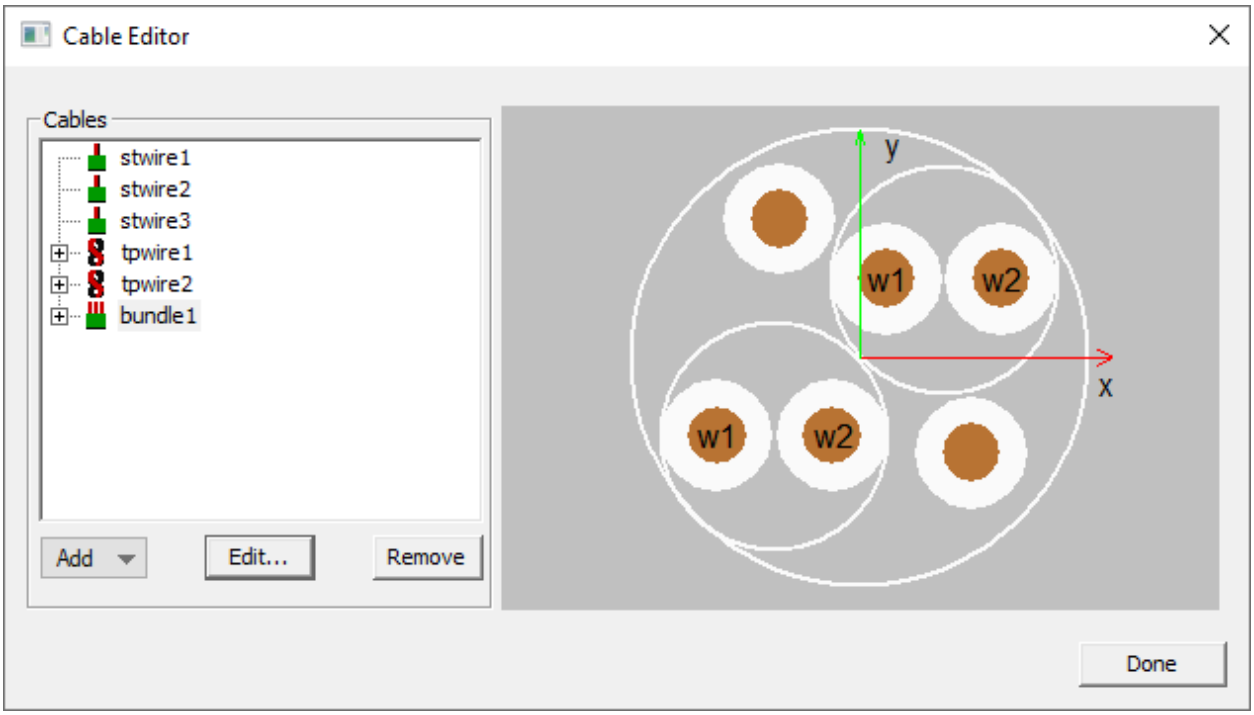
The Number Carriers, NumWiresInCarrier, Wire Diameter and Weave Angle have editable fields. Click Transfer Impedance vs Freq to see the **Transfer Impedance Preview**:



You can Edit the Max Freq value and units, and select Component as dB, Mag, IMag, and Real from the drop down.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

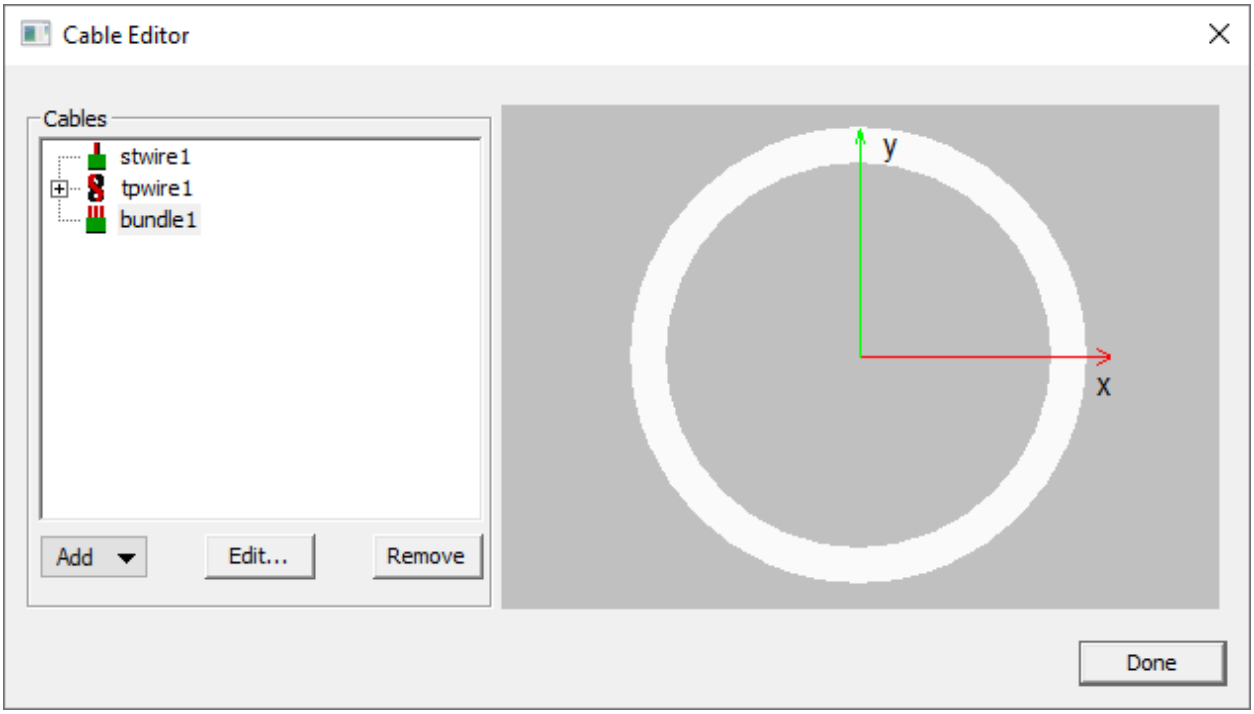
If you select No Jacket, the **Cable Editor** image if the bundle will resemble the following:



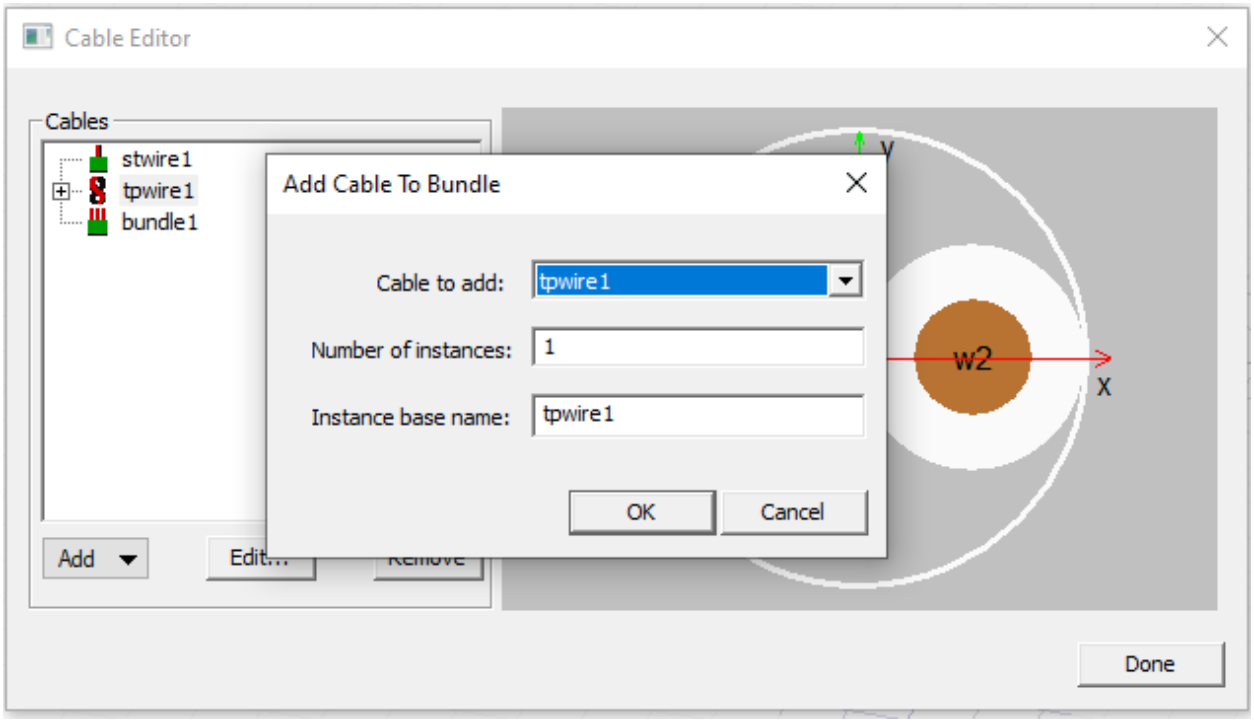
For No Jacket, the other properties are not editable.

When you click **OK** for parameters, the dialog box, the currently defined Bundles are listed and the current bundle is shown.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

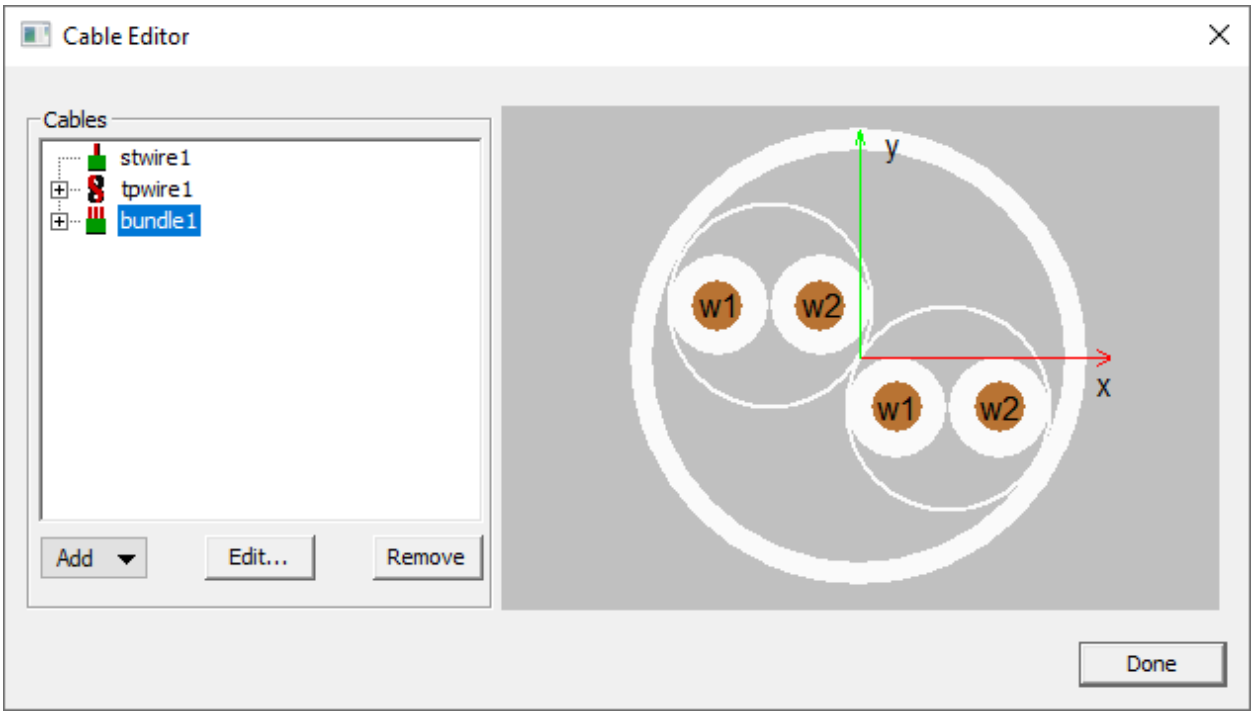


When you drag a wire from the list to the bundle image or to the bundle icon in the Cable tree, an **Add Cable to Bundle** dialog opens.



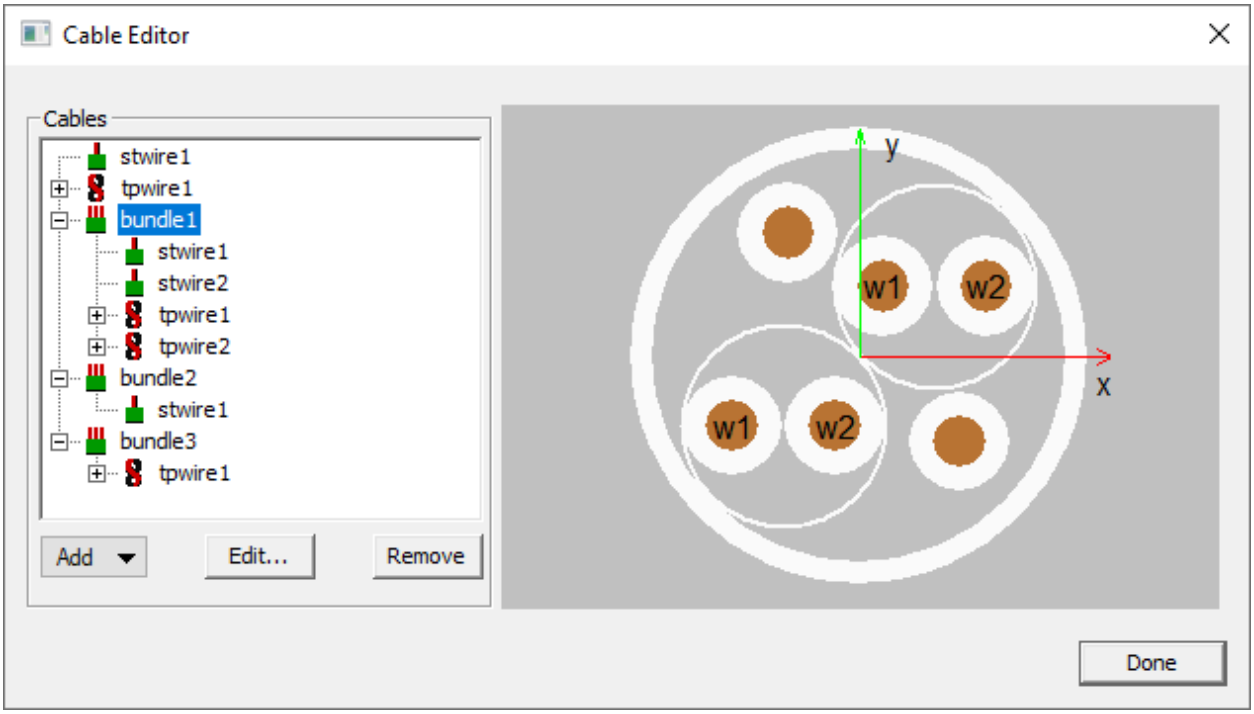
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Here you can select the Cable to add, the number of instances, and specify an Instance base name. When you have specified the values for the added cable, and **OK** the dialog box, the **Cable Editor** shows the bundle as currently defined, the following example with instances set to 2.



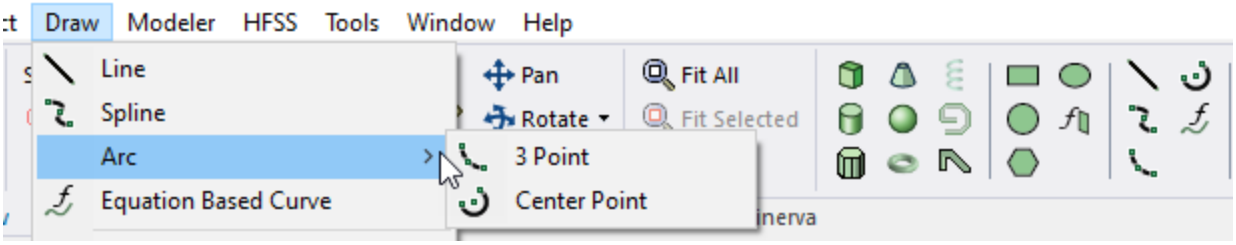
You can define as many bundles as you need, of whatever composition you require.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Define Route as Polyline

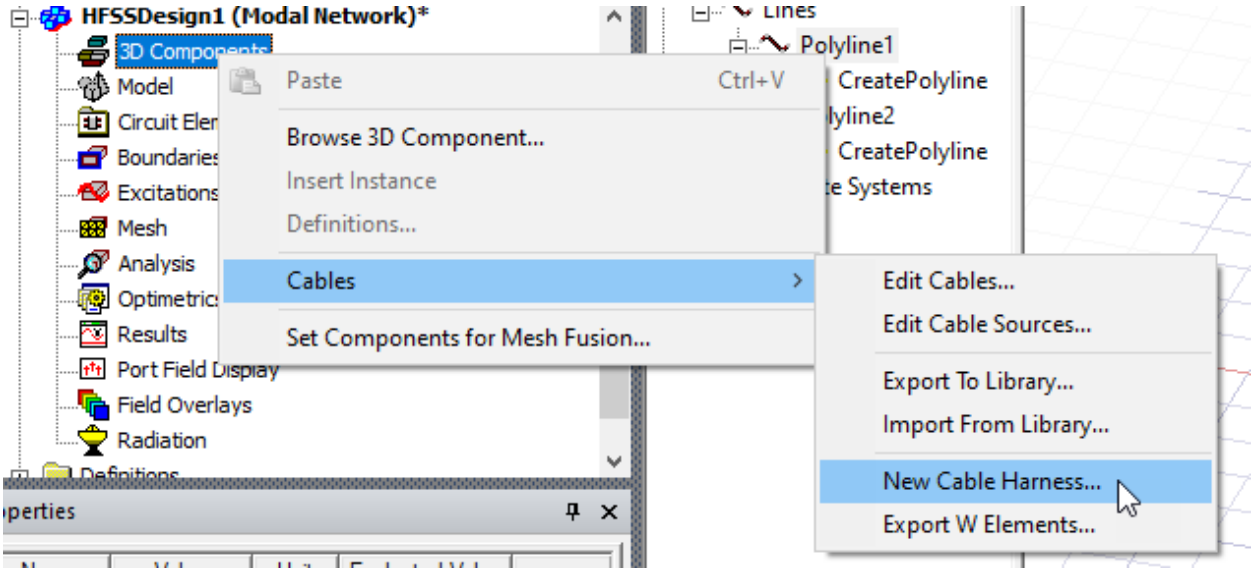
When you have defined bundles appropriate for your design, the next step is to define a route as a polyline for the cable position in HFSS. The Modeler offers several types of polylines.



Add New Cable Harness

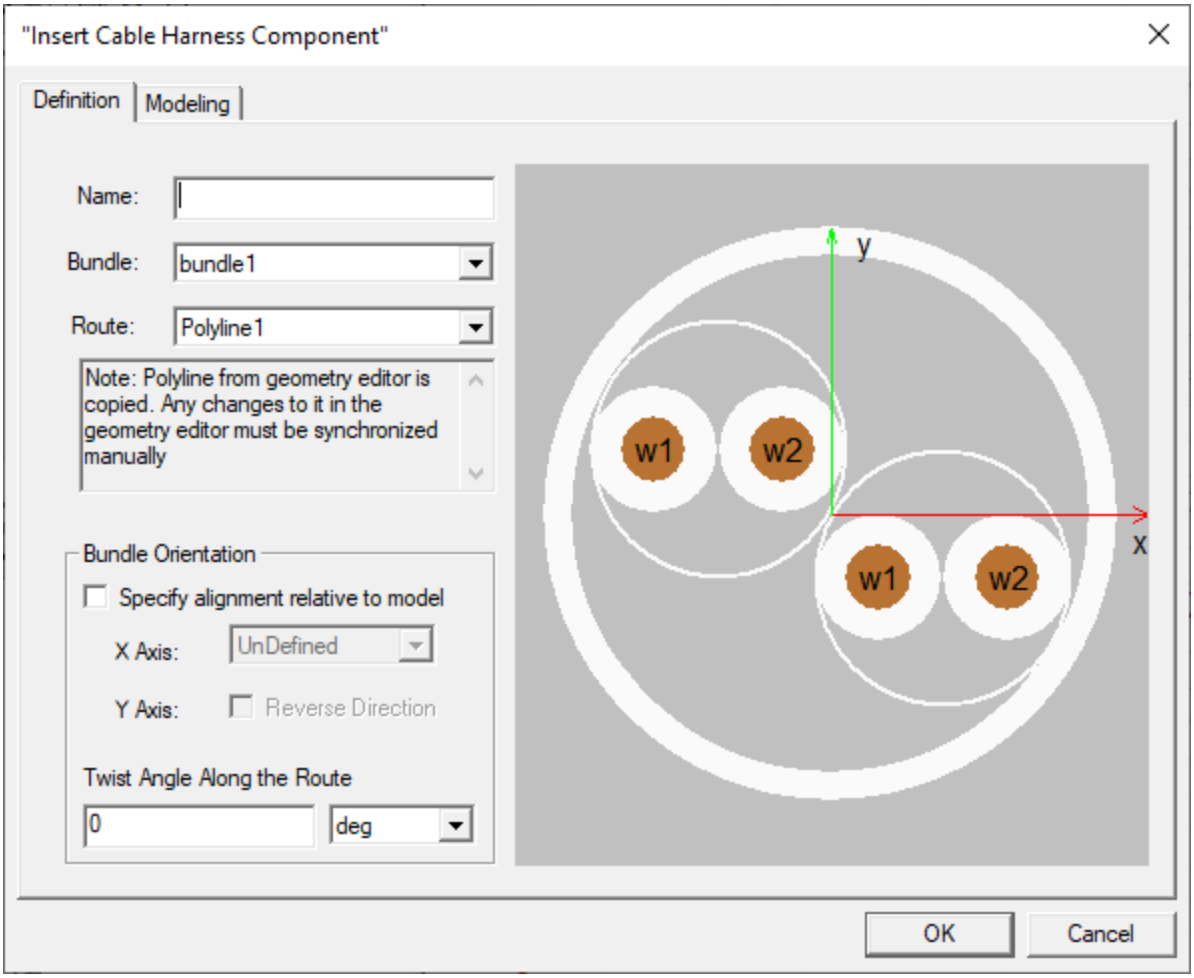
Once you have defined the polyline in the Modeler, go to the **Cable Editor**, and select **Add>New Cable Harness...**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



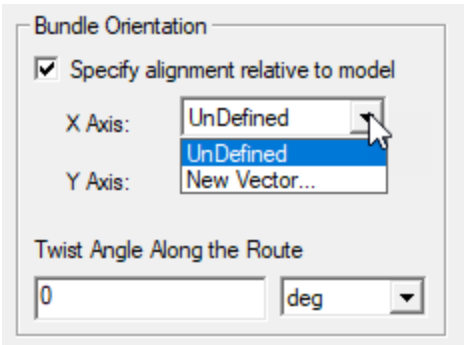
This opens the **Insert Cable Harness Component** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



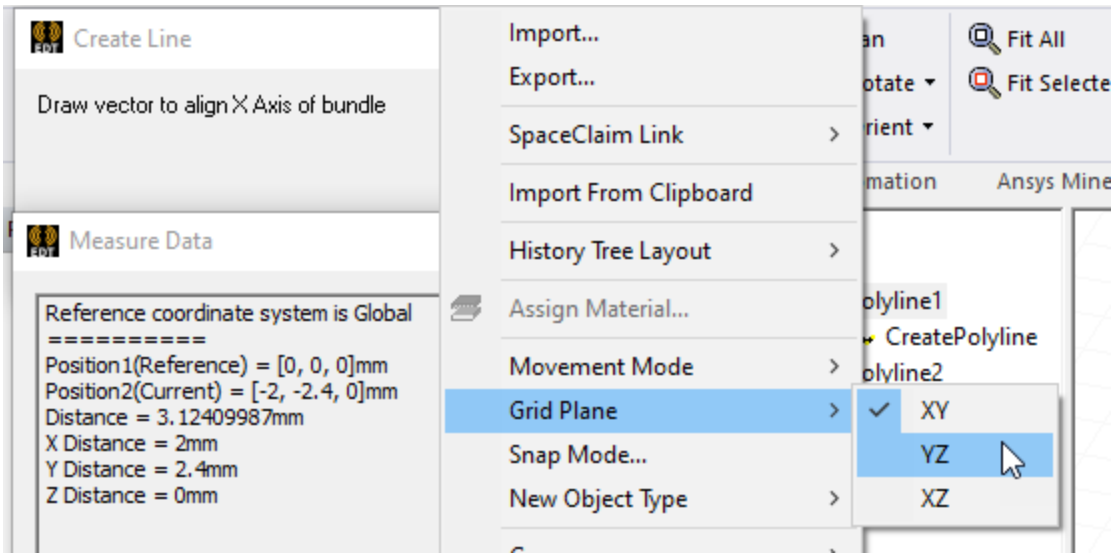
You can use drop-down menus select from bundles you have defined and from polyline routes available in the Modeler.

Under Bundle Orientation, you can specify alignment relative to the model. Check Specify alignment relative to model to specify X axis via a New Vector.



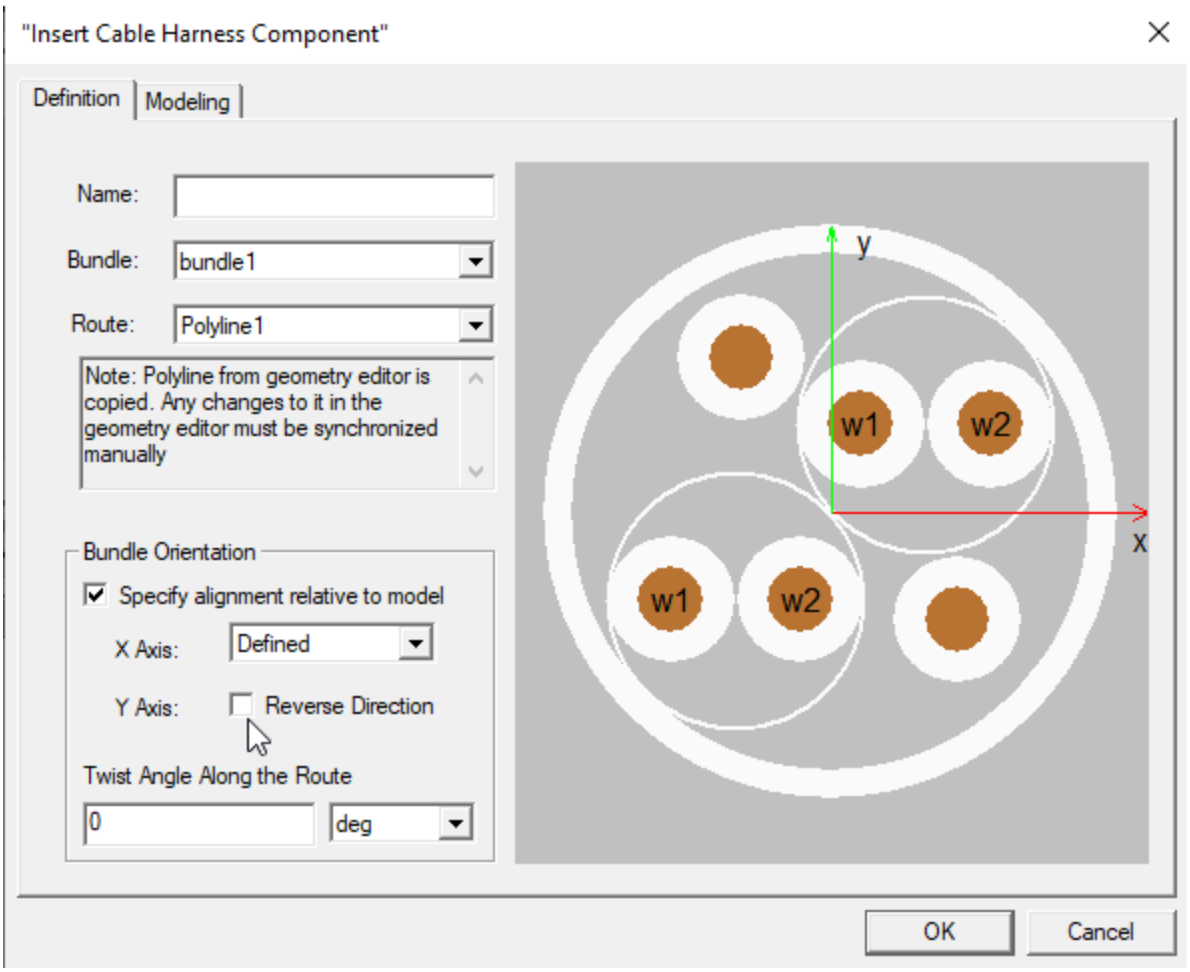
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you select New Vector, remember that you can specify the Grid Plane you use in the Modeler window.



A checkbox lets you Reverse the direction for the Orientation for the Y Axis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

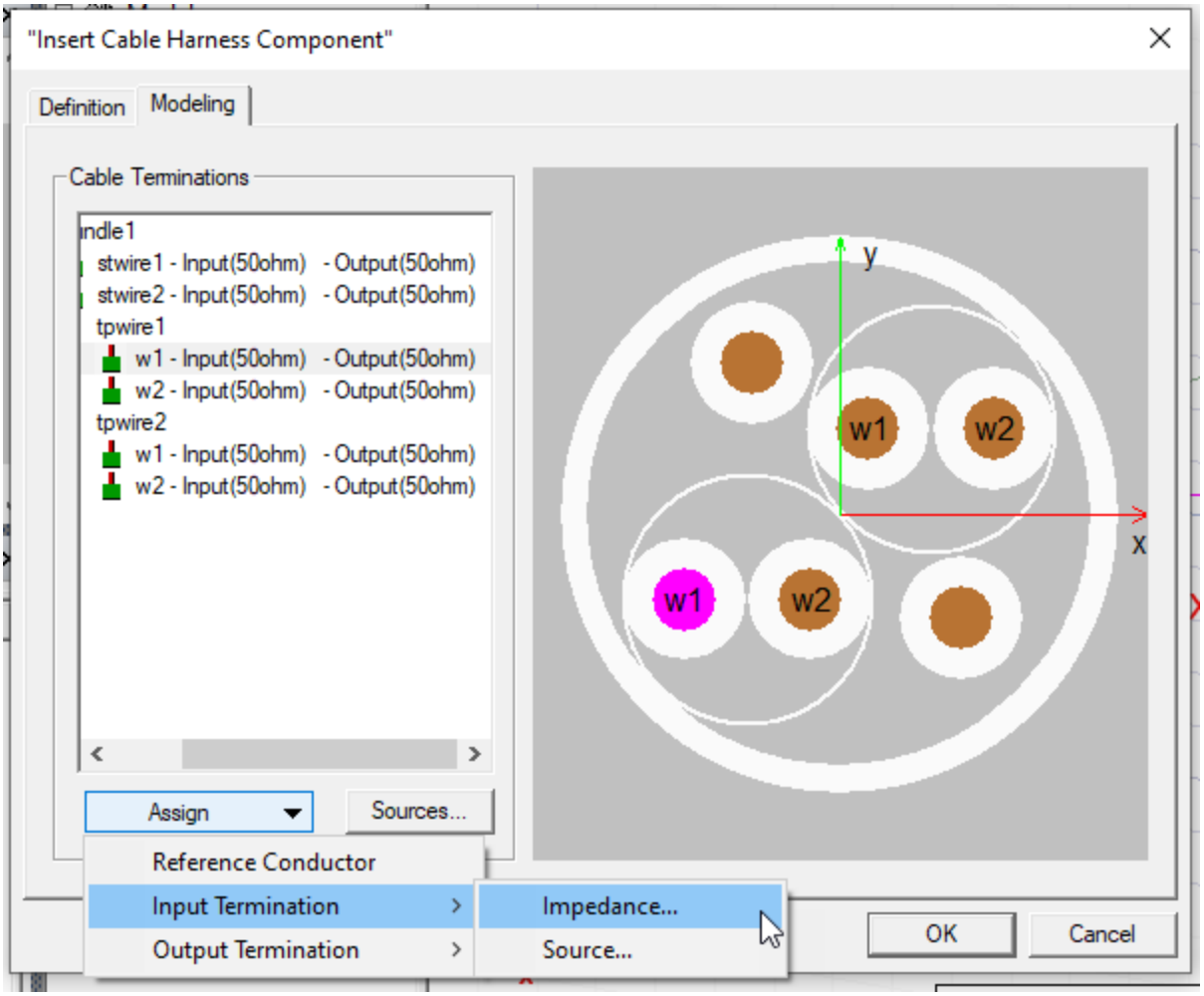


You can also specify Twist Angle Along the Route, selecting units of degrees, degree minutes, degree seconds, or radians from a drop down.

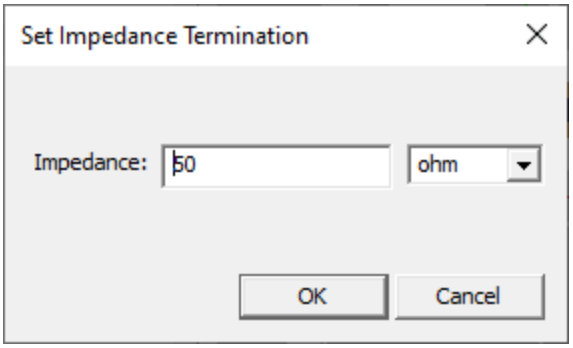
Before selecting the **Modeling** tab, you must provide a Name.

To assign termination impedance, on the **Modeling** tab, select a conductor wire, either from the list of Cable Terminations, or the clicking on the graphic, and then click **Assign>Input Termination> Impedance...**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

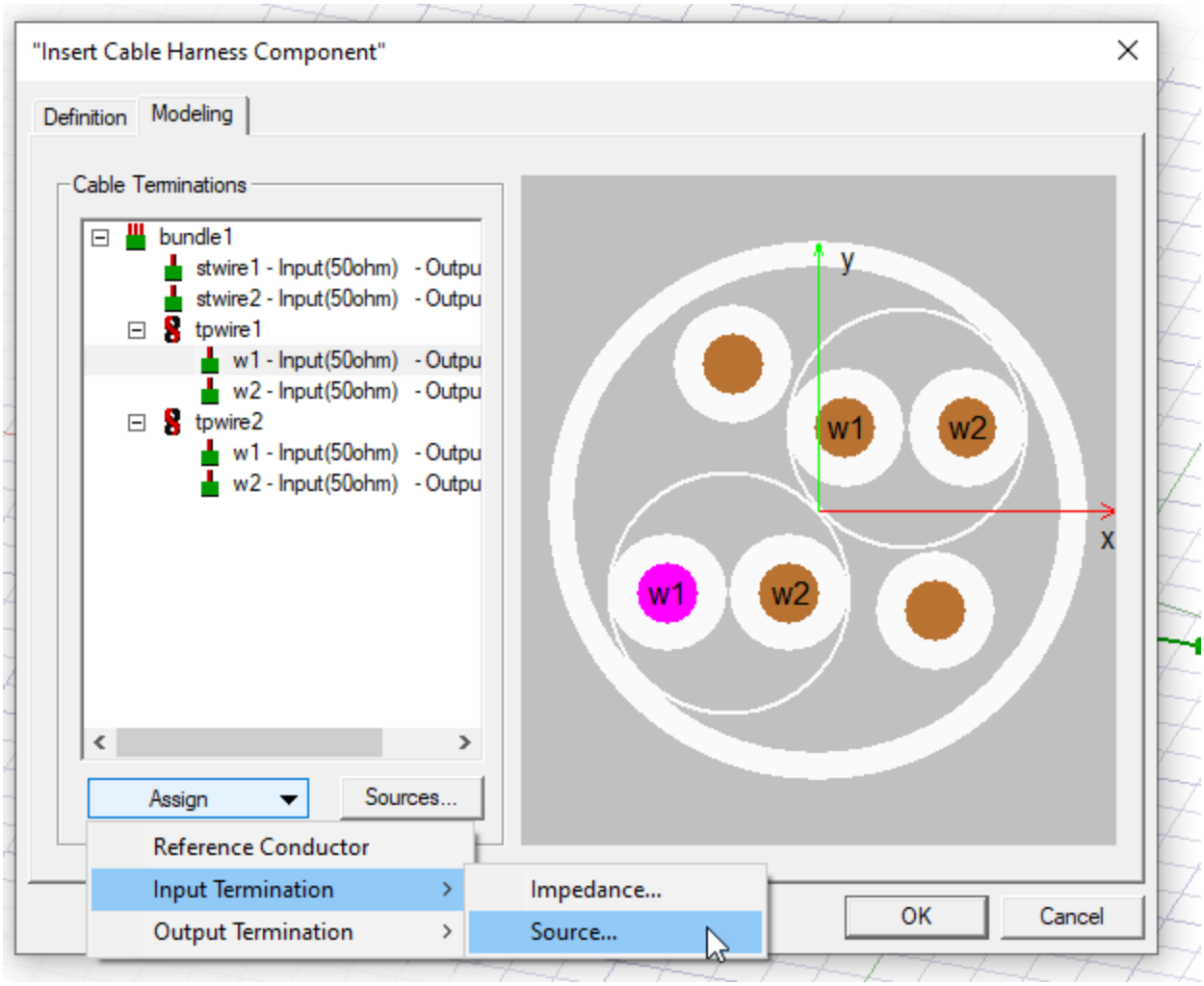


This opens a dialog for you to **Set Impedance Termination**.



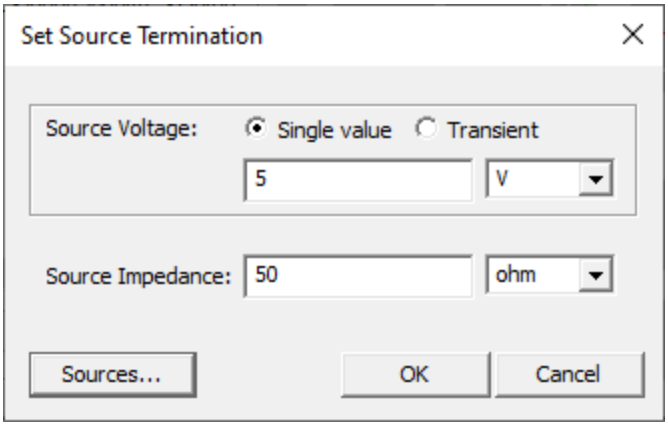
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To assign termination source, on the **Modeling** tab, select a conductor wire, either from the list of Cable Terminations, or the clicking on the graphic, and then click **Assign>Input Termination>Source...**.

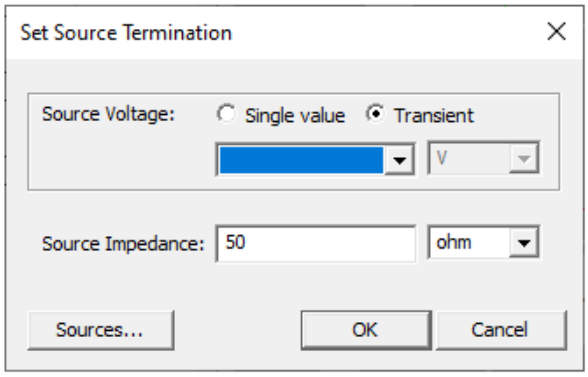


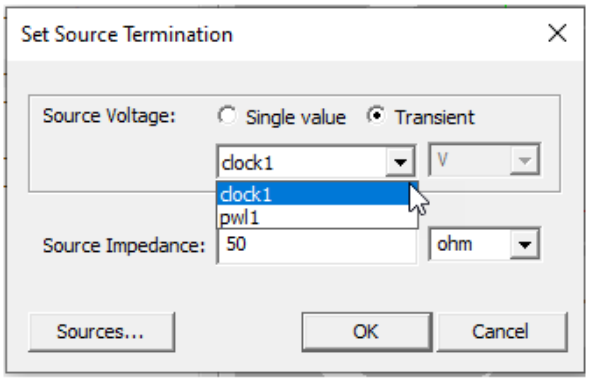
This opens a **Set Source Termination** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



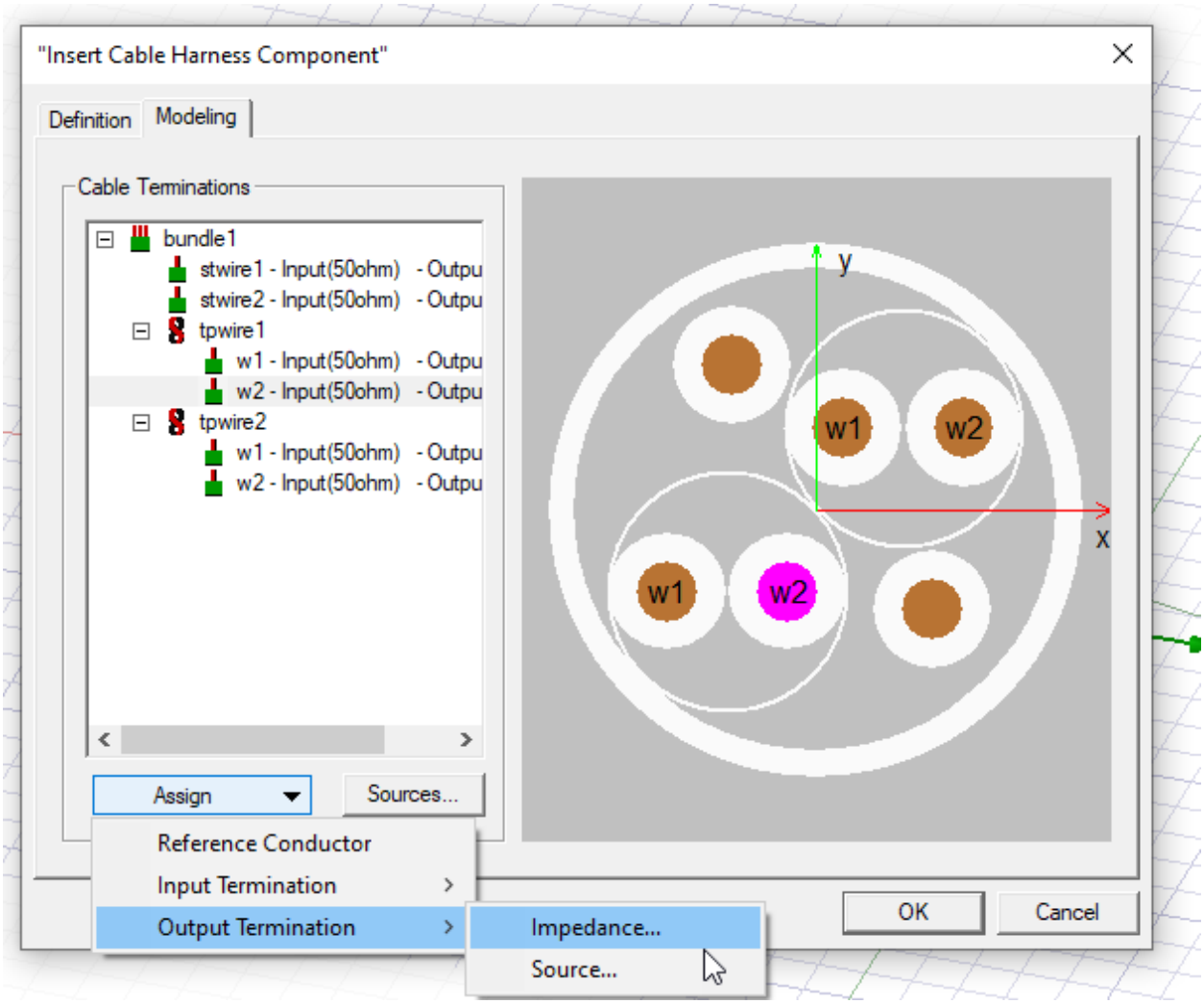
For Single Value, you set source value and units, and source impedance. Selecting Transient changes the fields. If your have not previously defined transient sources, the transient menu is blank, and you can select the **Sources...** button to open the **Cable Time Domain Sources** dialog (described below) to define them. If your have previously defined sources, you can select them from the drop-down menu, and specify impedance.



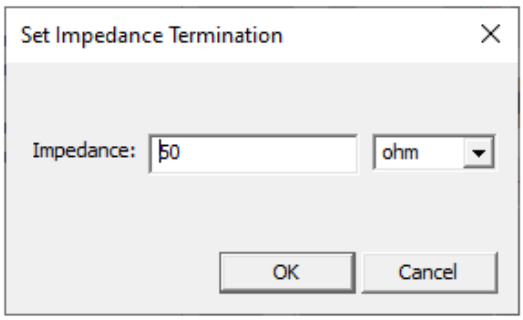


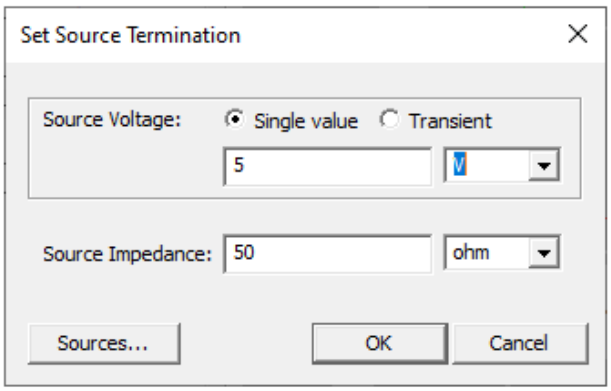
The Output Termination selections work the same way.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



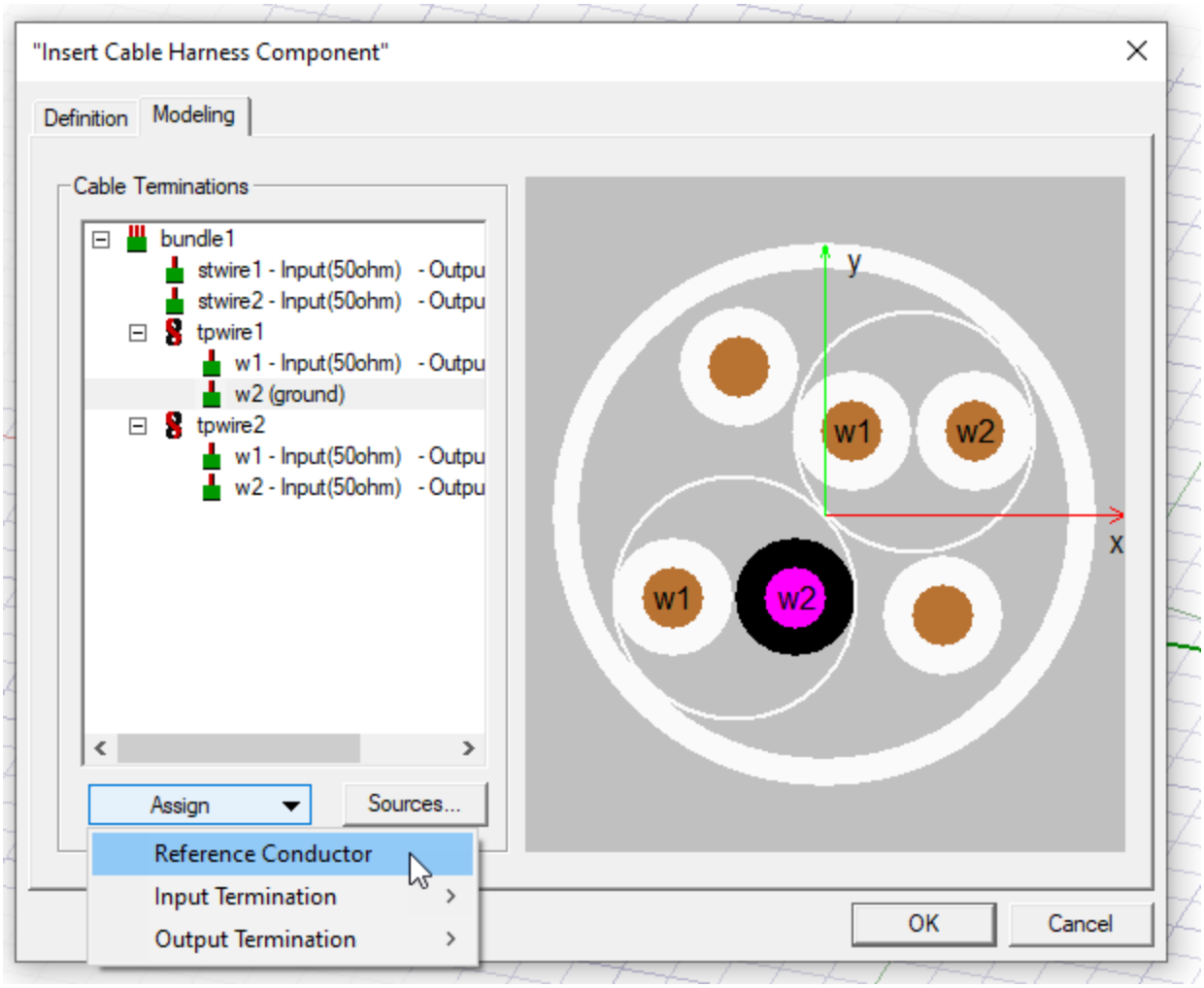
The dialogs for **Set Impedance Termination** and **Set Source Termination** work the same way.





Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

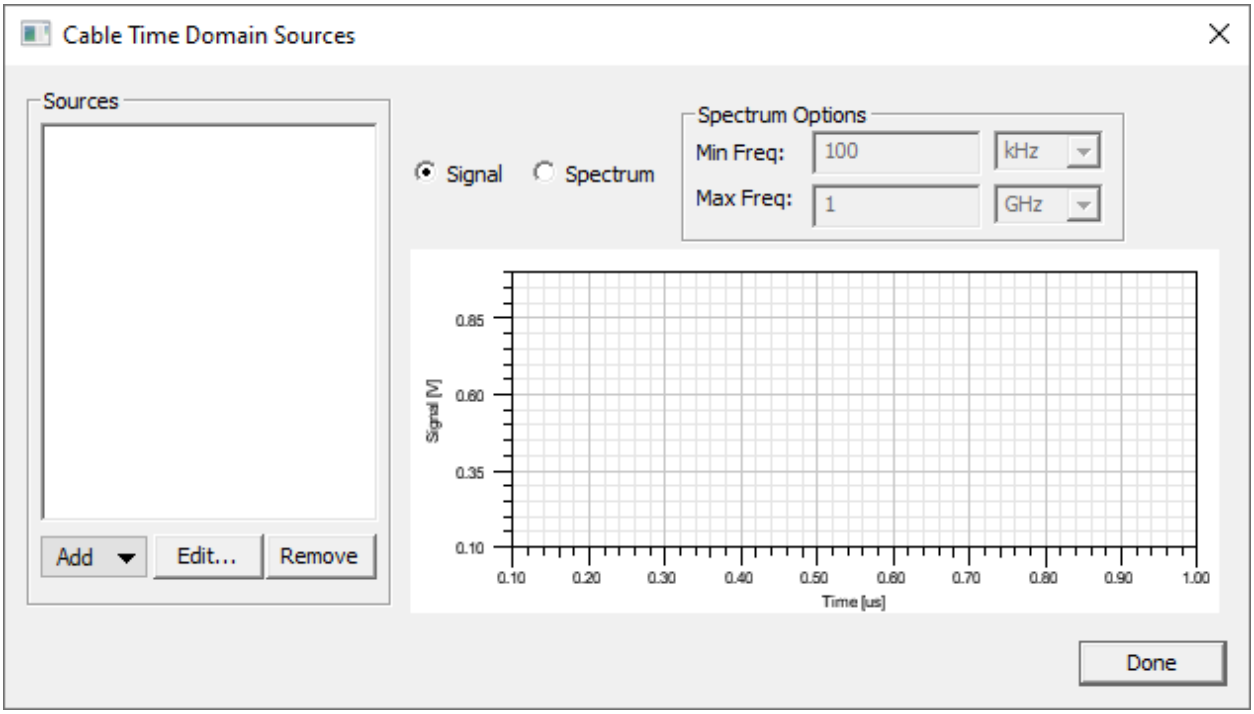
Selecting **Assign>Reference Conductor** designates a wire as ground in the list, and visually in the image. Multiple ground references are supported.



Defining and Editing Transient Sources

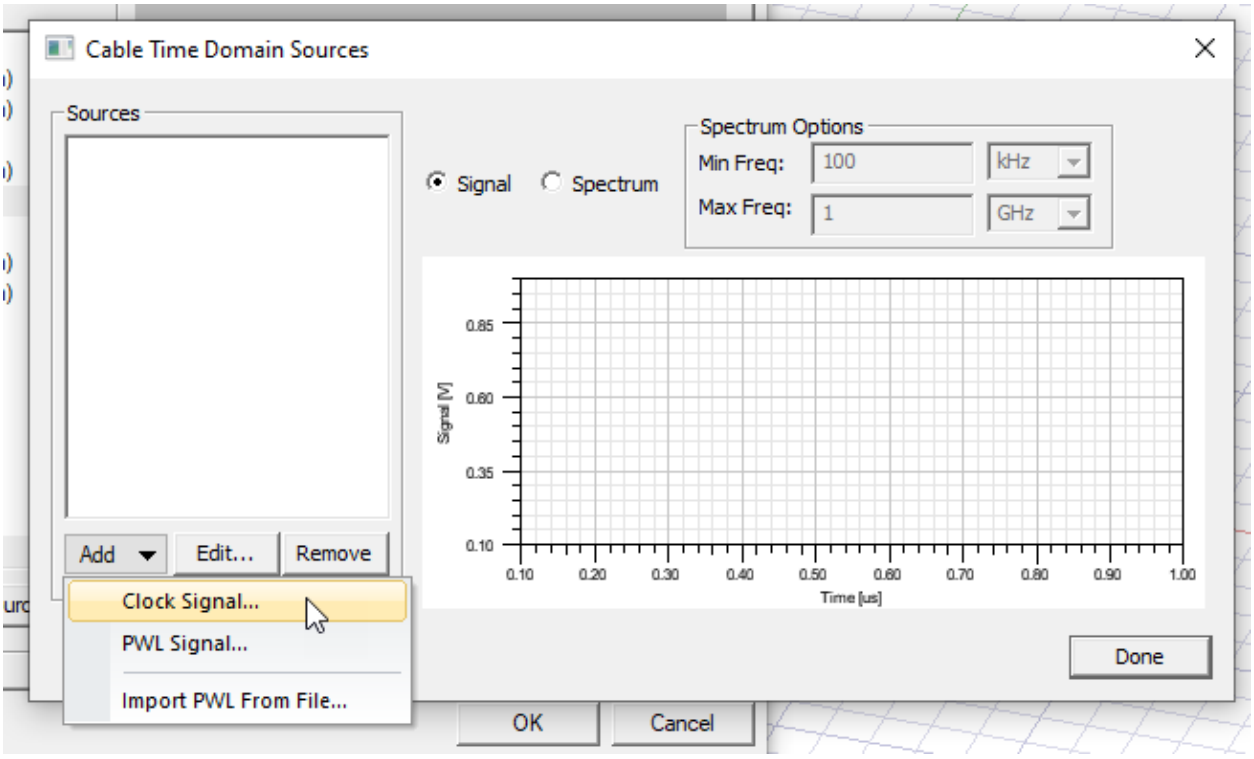
To define transient sources, or to edit previously defined sources, click the **Sources...** button, either on the **Modeling** tab **Sources...** button, or the **Sources...** button on a **Set Source Termination** dialog box. This opens the **Cable Time Domain Sources** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



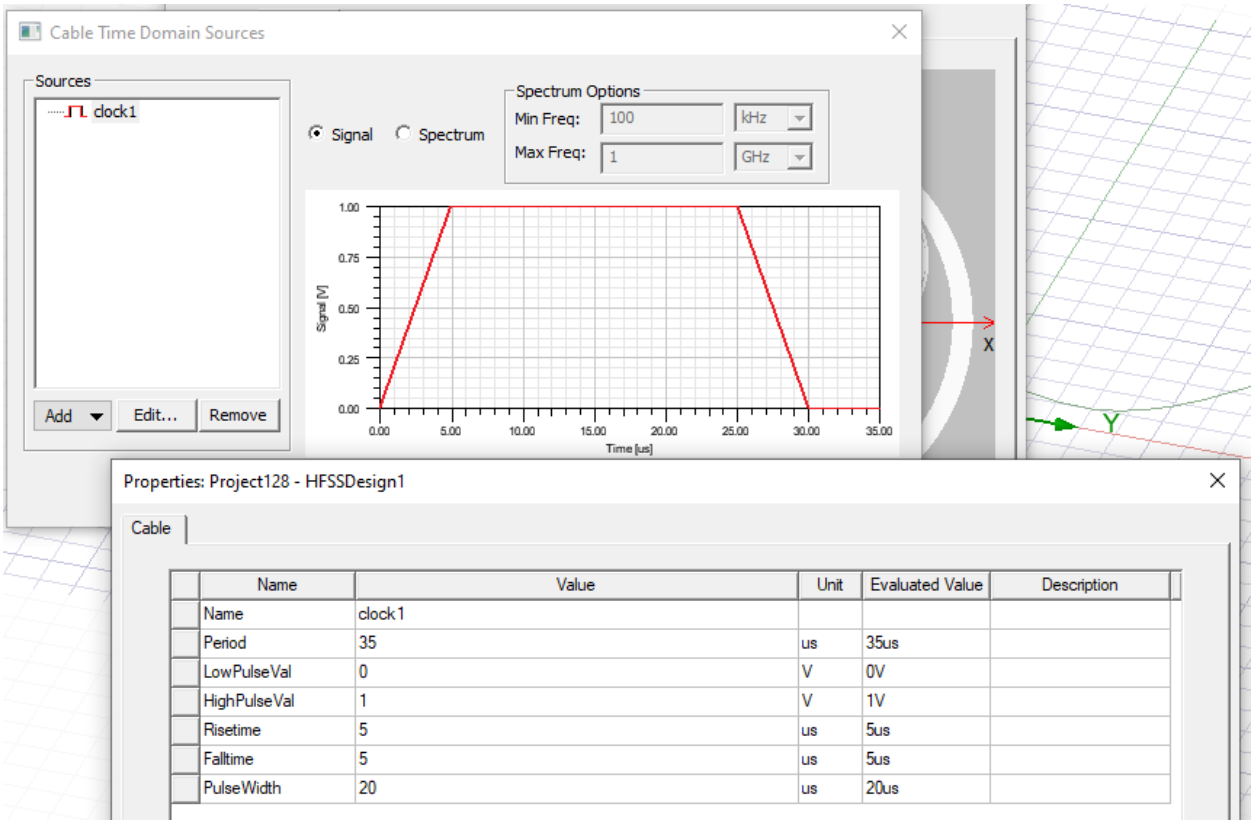
Select the from the **Add** button drop-down to define a **Clock Signal**, a **PWL Signal**, or to **Import PWL From File**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



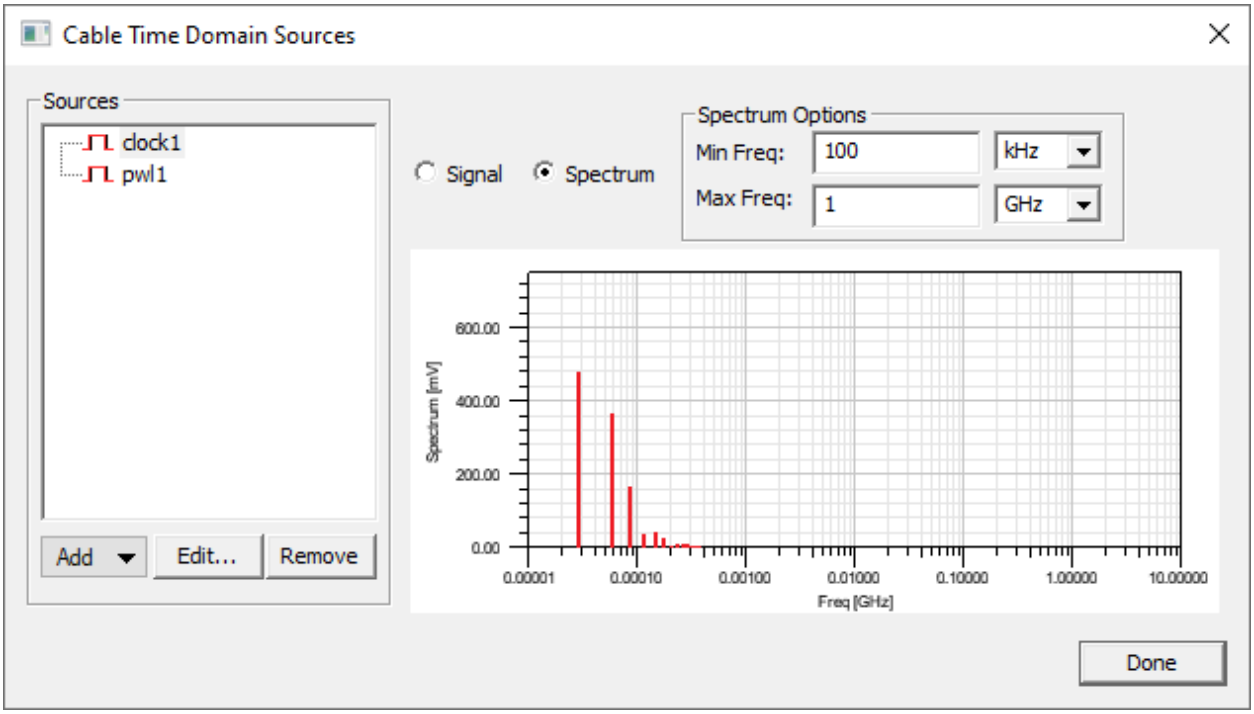
Selecting **Clock Signal...** opens a **Properties** dialog for the clock signal, and shows the signal graphed as Signal, or Spectrum, based on the parameters you specify.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

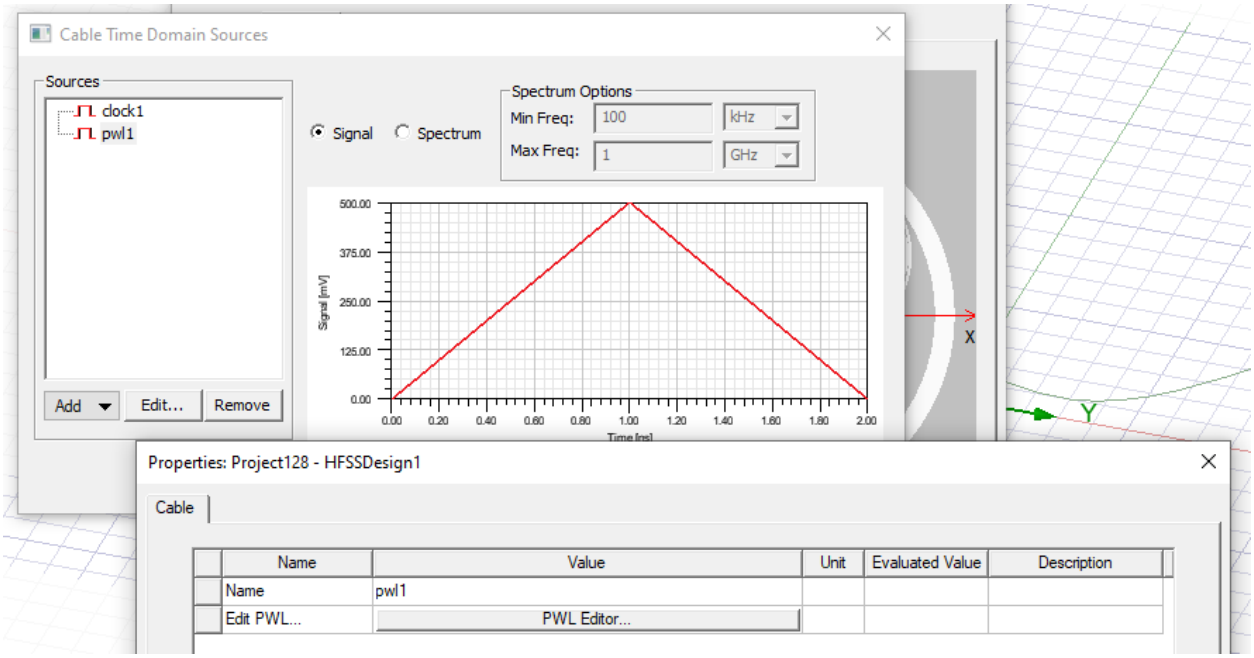


You can provide Name, Period, LowPulseVal, HighPulseVal, Risetime, Falltime, and PulseWith values and units. The units can be variables. When you select OK in the Properties, the clock is displayed and is available to Cable Modeling for selection and editing.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

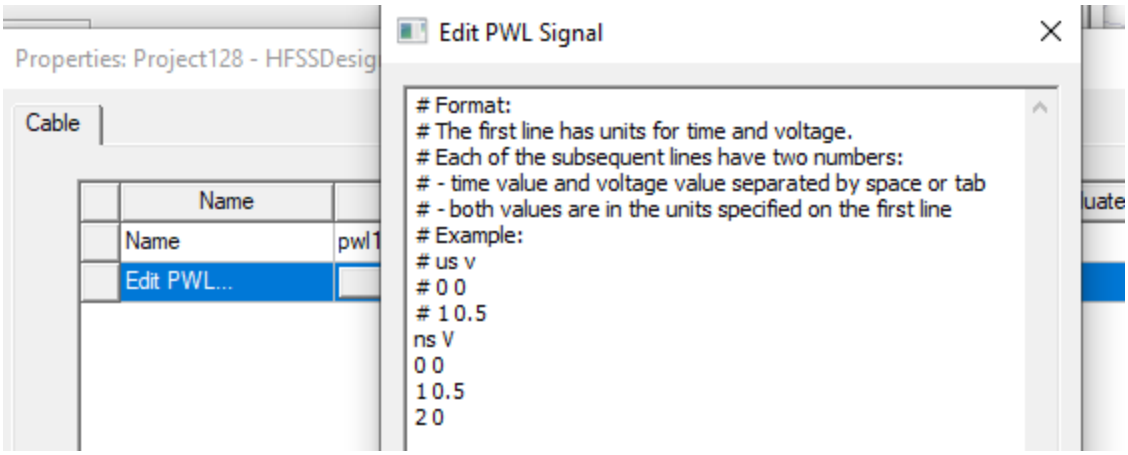


Selecting **Add>PWL Signal** opens a **Properties** dialog with a name field and a **PWL editor...** button.



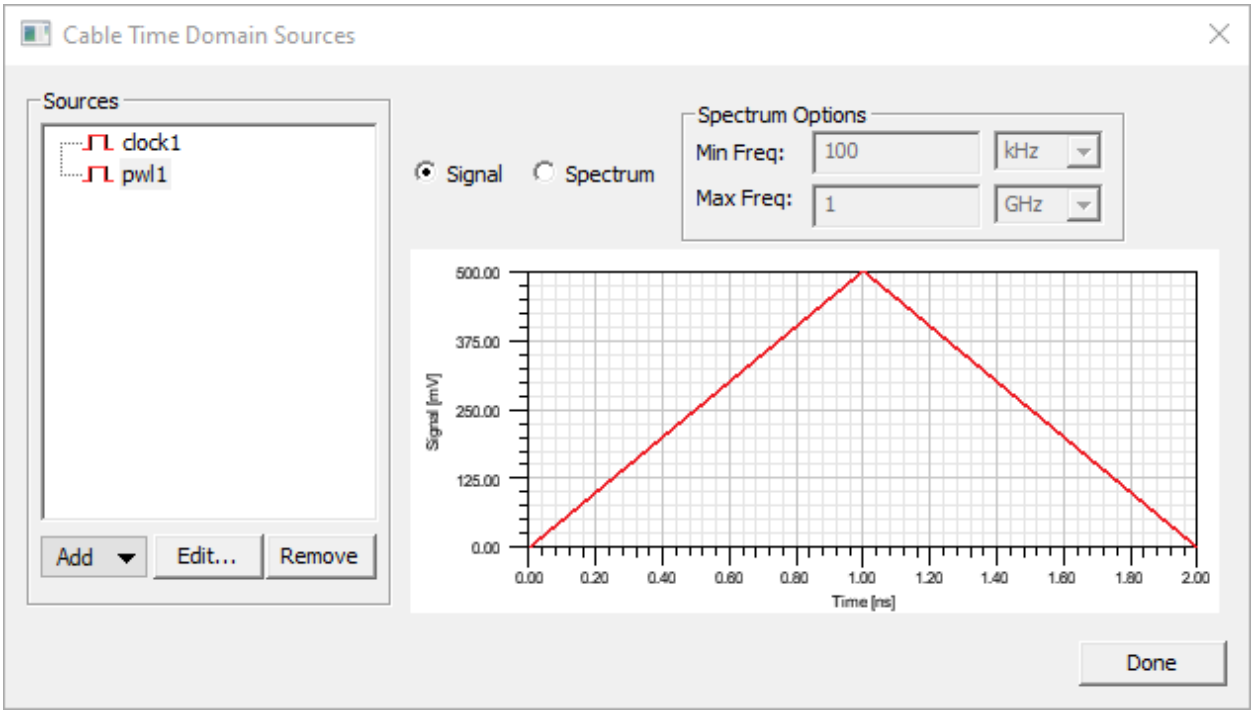
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click the **PWL Editor...** button to view a text editor that lets you specify PWL parameters as units for time and voltage. Subsequent lines have time and voltage values separated by space or tab.



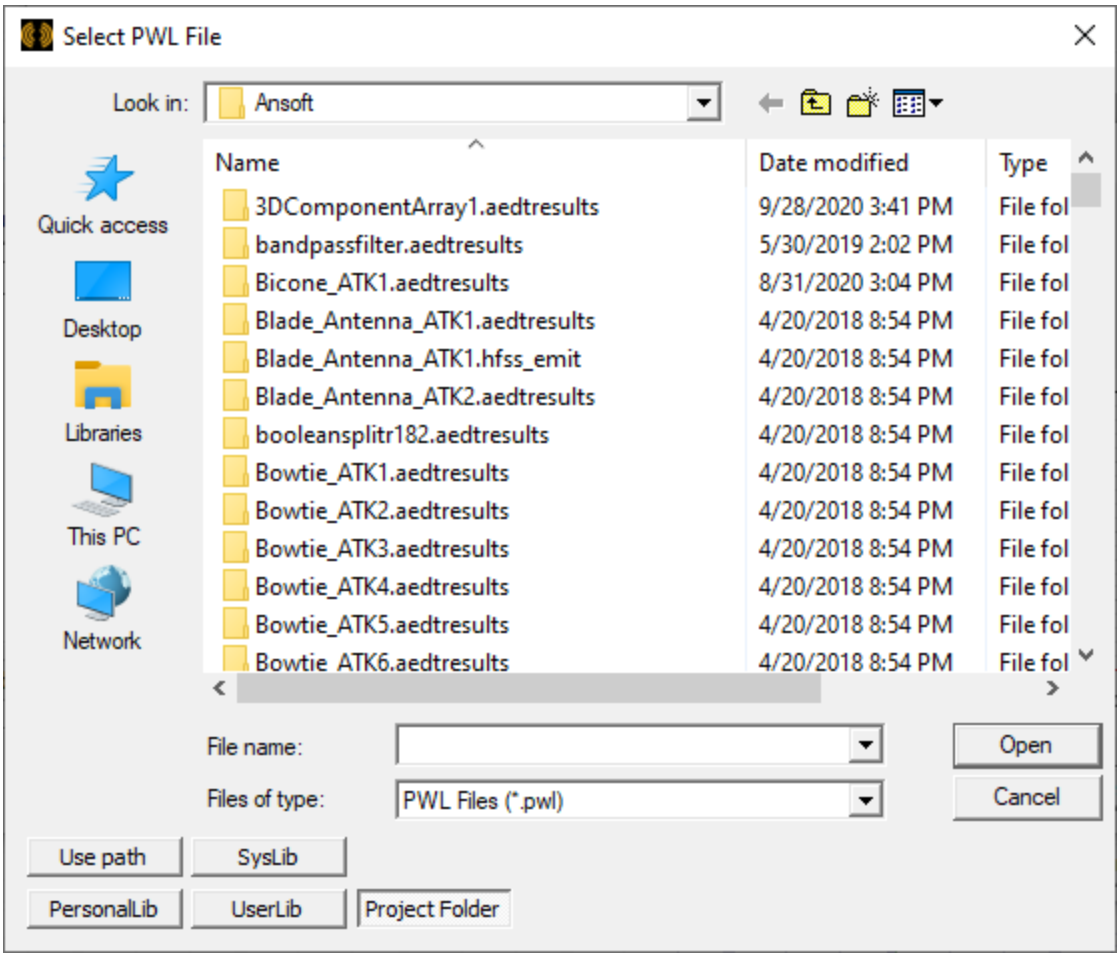
Click OK to accept the parameters from the Edit PWL Signal dialog box, and OK to accept the PWL definition for the Properties dialog.

The PWL signal is displayed in the Cable Time Domain Sources dialog and is available for selection and editing.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you select **Add> Import PWL From File...** a browser opens that lets you navigate to open a plain text file with a .pwl suffix that defines the PWL signal using the same format as the editor.

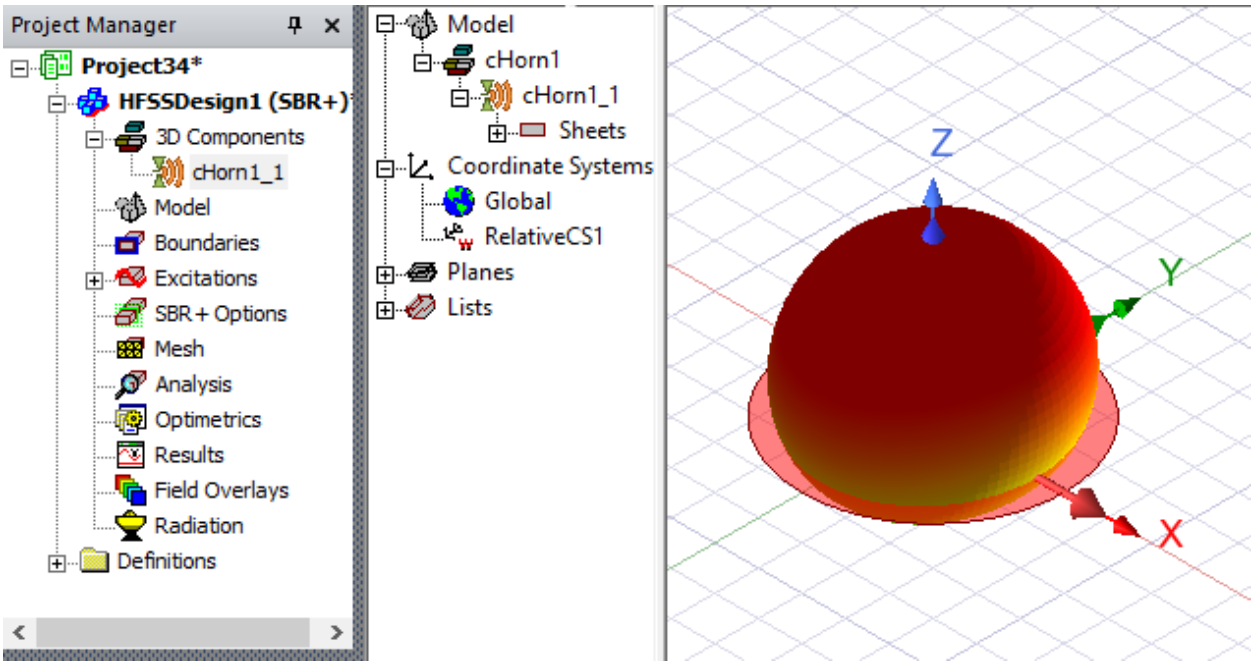


Parametric Antennas for SBR+ Solution Type

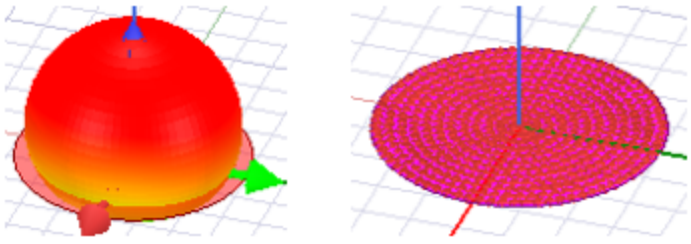
For SBR+ solution type, you can access a range of parametric antennas. Parametric antennas are antenna sources that either have no physical structure definition or a highly idealized physical structure. No model parts are constructed or used in the simulation. They create far-field pattern or current distribution sources that can provide S-parameter outputs and composite far-field radiation patterns. The SBR+ solver will automatically interpolate the source fields from the file-based antenna in order to represent the antenna at all frequencies of the user defined frequency sweep in solution setup. The file-based antenna requires at least two frequencies, and the min/max frequencies of the user defined sweep should fall within the frequency band of the file-based antenna.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parametric antennas are very quick to define and place, copy and duplicate, and are thus intended for systems studies where the specific antenna structure is either unknown or a simple, topologically representative antenna type is sufficient. The editable parameters depend on the antenna type. In the model window, the antennas display as a scaled far field pattern at the antenna origin. The model is listed under the **Project** tree under **3D Components** and automatically includes a port to support S-parameter output. You can change port post-processing setting from the **Properties** window. Selecting the antenna causes the **Properties** window to display with tabs for **General** antenna **Properties**, and **Visualization** settings.



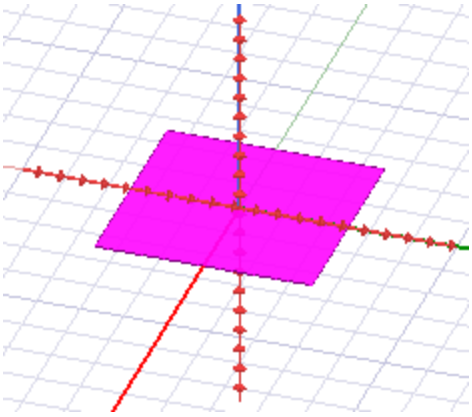
Several of the parametric antennas let you choose Antenna Representation as Far Field Source and/or as Current Source.



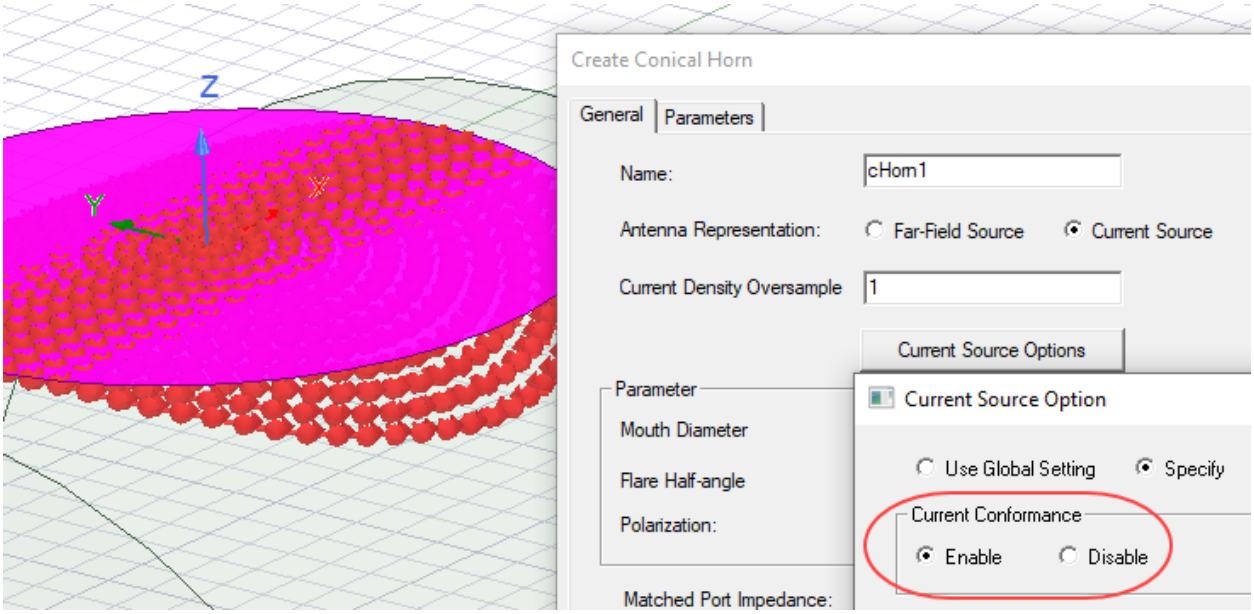
Far Field Current Source

Other parametric antennas include only Current Source representation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



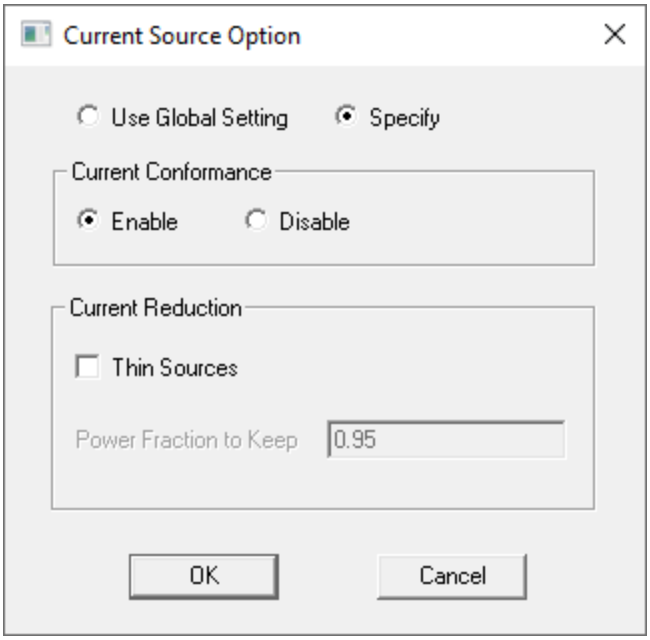
Current sources are visualized in the model window as a set of short lines with arrows to indicate the current direction. When placed near a geometric surface, these current sources can be configured to conform to the nearby surface. The current source conformance can be configured globally, to be enabled/disabled for all antennas in the HFSS design, or enabled/disabled for each parametric antenna or array.



For a technical discussion, see Current Conformance and Array Conformance in SBR+.

The Current Source Option window also includes Current Reduction option.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Enabling Thin Sources lets you specify a Power fraction. It should be greater than 0.1 and less than 1. Solutions are invalidated if you change current source conformance, use Thin Sources, Power Fraction, or change the Global option.

The outputs from SBR+ simulations with parametric antennas include:

- S-parameters.
- Composite far-field radiation patterns.
- Incident field reporter outputs provide the free space radiation patterns of the antenna.

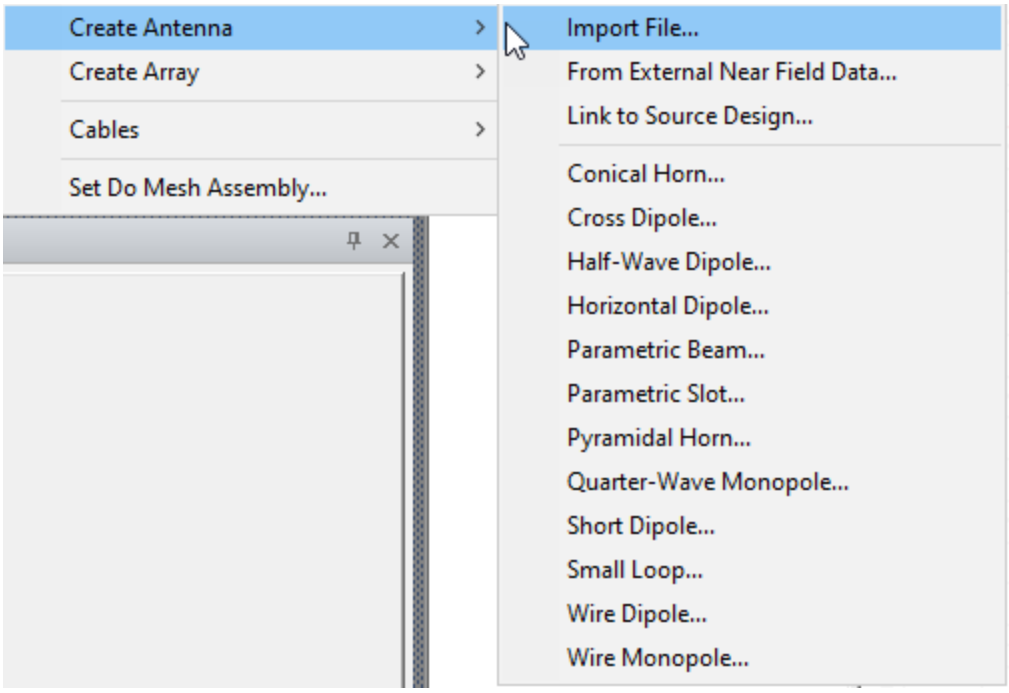
Prerequisites

- The Solution Type must be SBR+.
- If you use a File-based parametric antenna, it must be defined with at least two frequencies whose min/max meet or exceed the extents of the simulation frequency sweep.

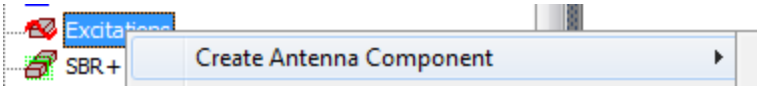
Adding Parametric Antennas

1. Right-click the 3D Component folder in the Project tree to see the **Create Antenna** menu:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

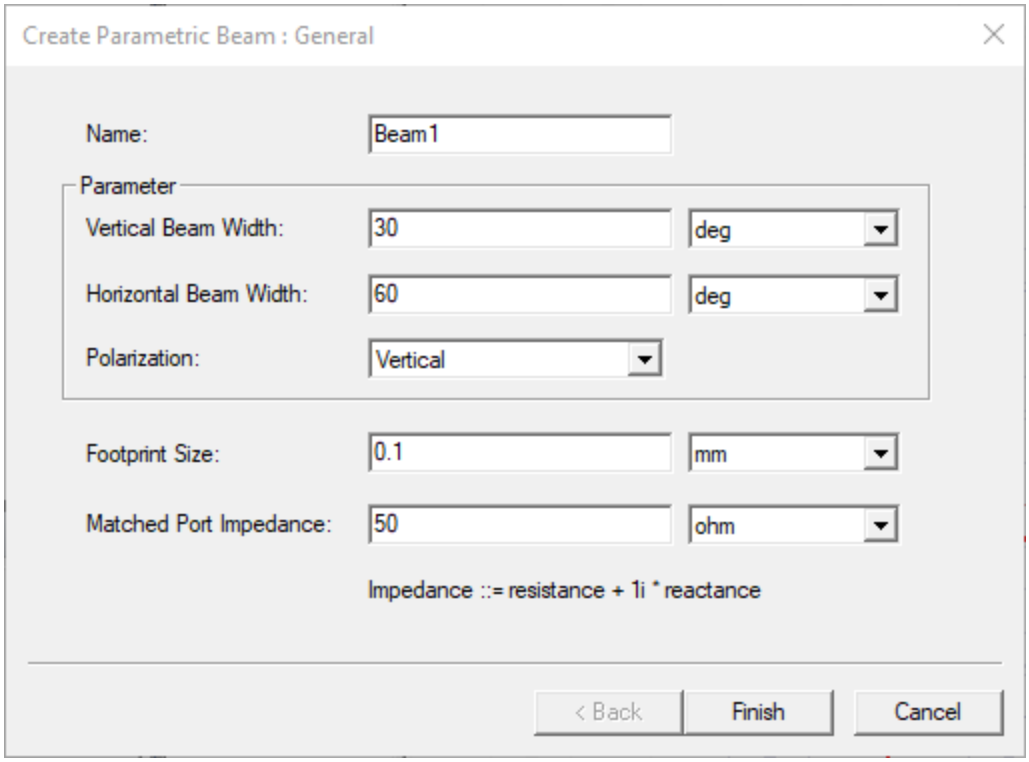


You can also view the parametric antenna menu by clicking **HFSS** > **Excitations** > **Create Antenna Component** or by right-clicking on the **Excitations** icon and selecting the submenu under **Create Antenna Component**.

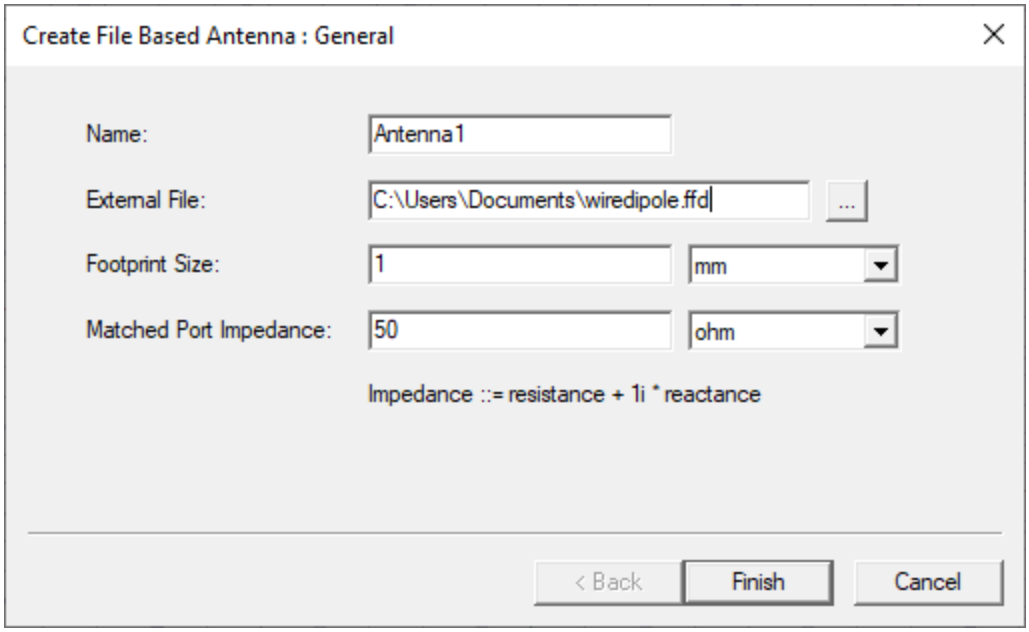


Selecting any of the predefined antennas causes a dialog to appear, which includes the name, and other parameters. For example, the dialog below shows the **Create Parametric Beam** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

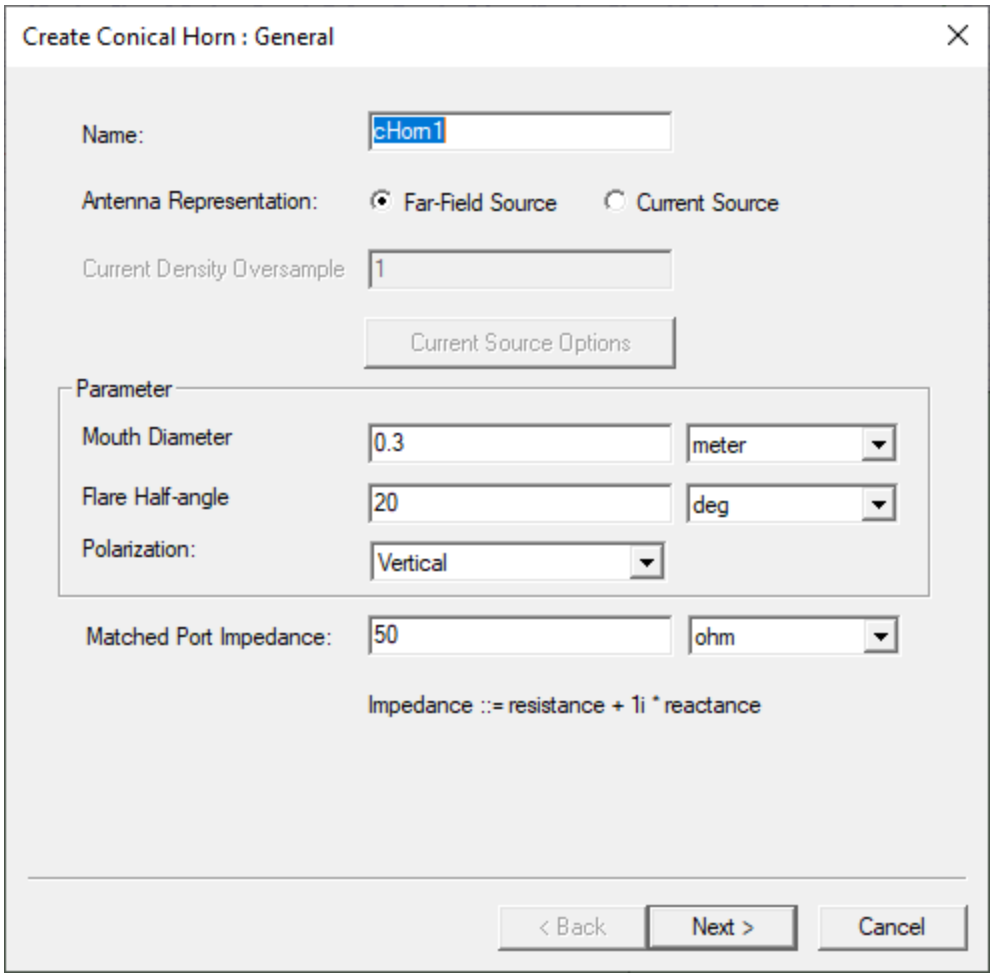


For the **By File** antenna, the Parameters include a path for a Far Field data file (*.ffd format).

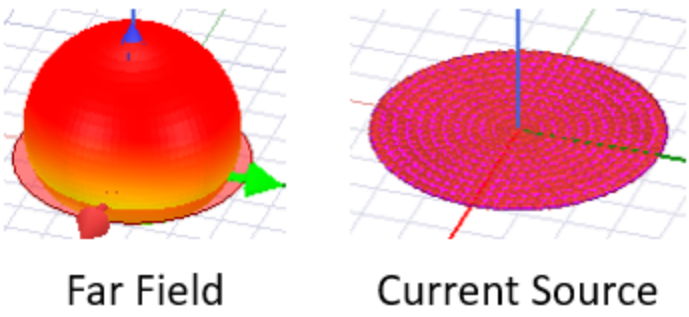


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Conical Horn dialog includes the option for representation as Far Field and/or Current Source.

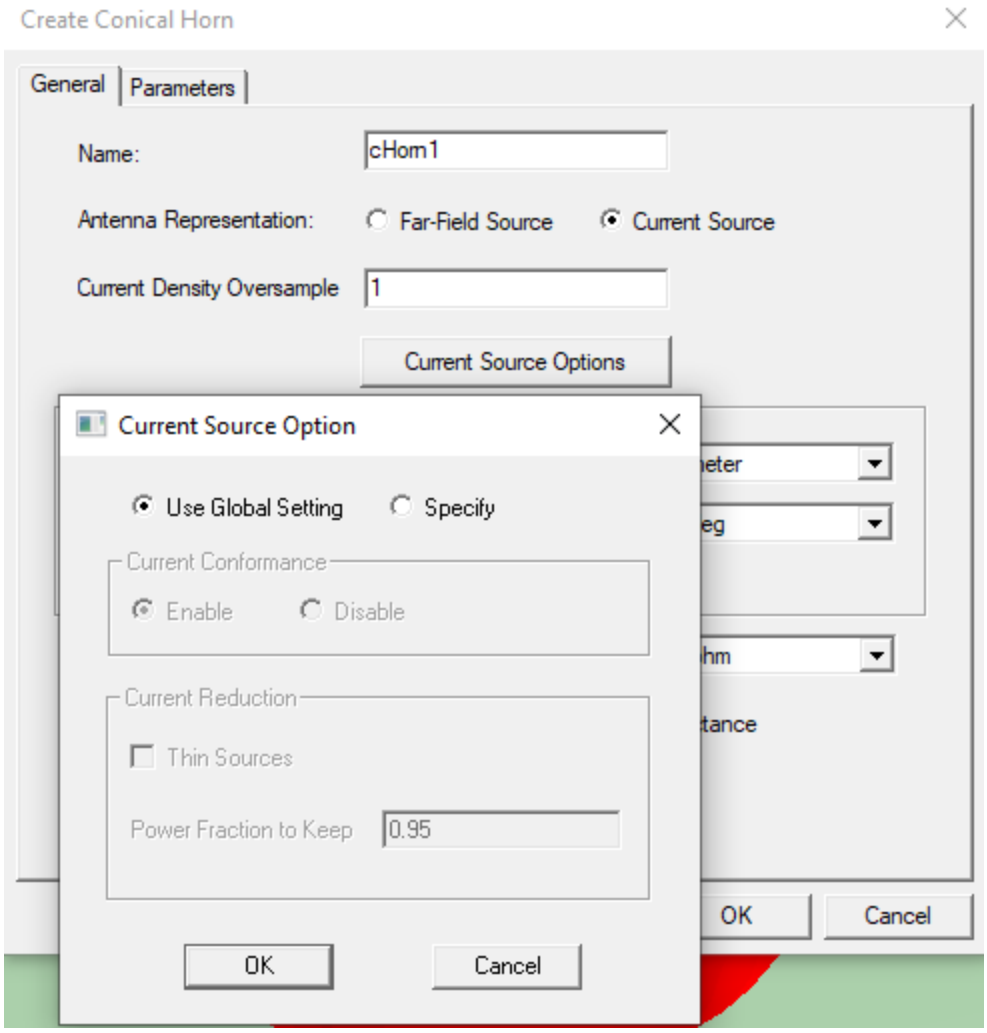


Depending on your selection, the visualization may appear as follows.



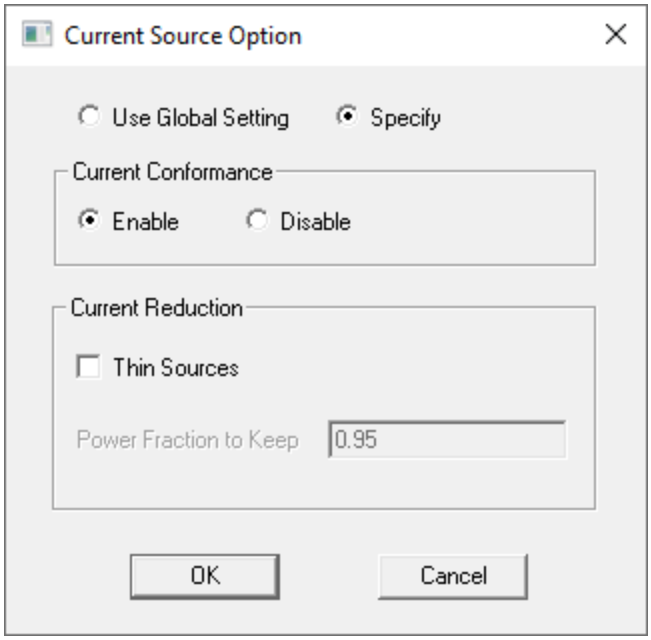
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the antenna offers current source, and you select that option, the **Current Source Options** button is enabled on the **Create** dialog box. Select it to open the **Current Source Option**. dialog.

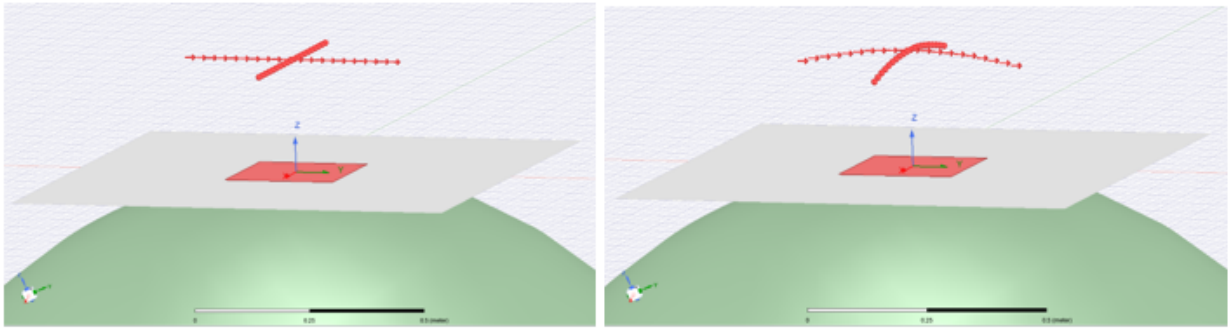


For the global current source options, see Set Current Source Option. To override the global setting, select Specify. This enables Current Conformance and Current Reduction settings for this antenna.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Current conformance controls whether antennas placed near a geometric surface conform to that surface. The figure on the left shows current conformance disabled. The figure on the right shows current conformance enabled.



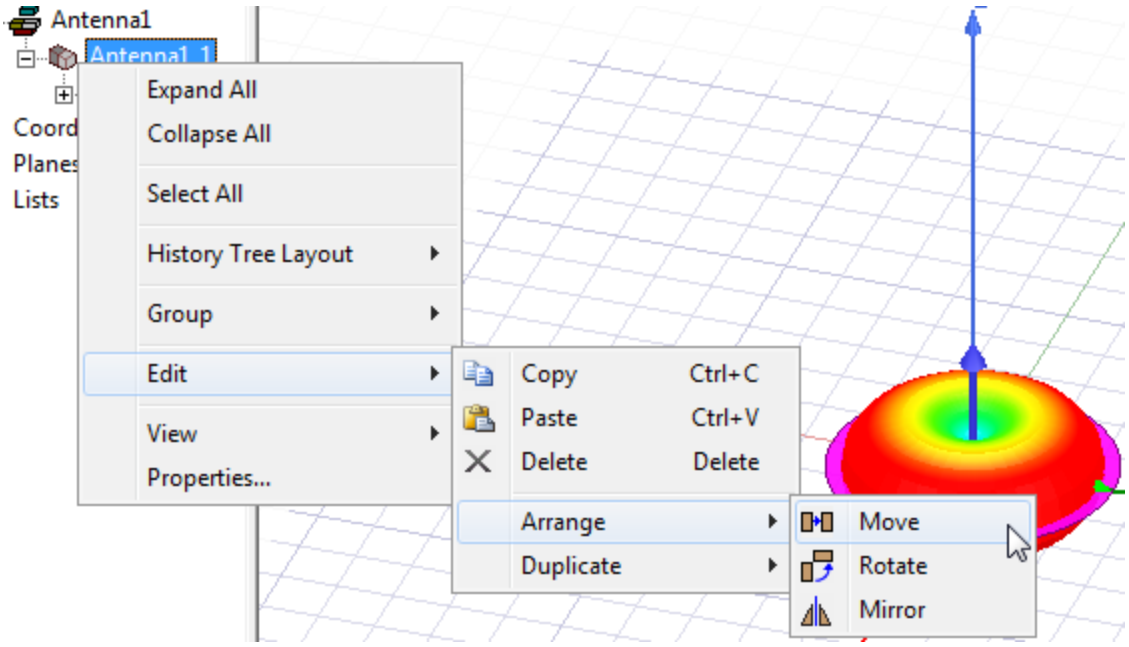
Enabling Thin Sources lets you specify a Power fraction. It should be greater than 0.1 and less than 1. Solutions are invalidated if you change current source conformance, use Thin Sources, Power Fraction, or change the Global option.

2. Specify the parameters as desired. The parametric antenna footprint is a visual to aid in placing and orienting the antenna, but it has no impact on simulation results.

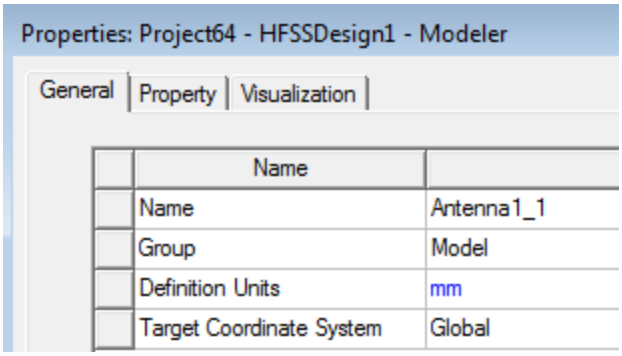
The antenna appears in the Model window.

3. If you want to edit the antenna placement, or orientation, select the model in the History tree, and right-click and select **Edit** > **Arrange**, and select from **Move**, **Rotate**, or **Mirror**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also change or edit the target coordinate system in the instance Property window.

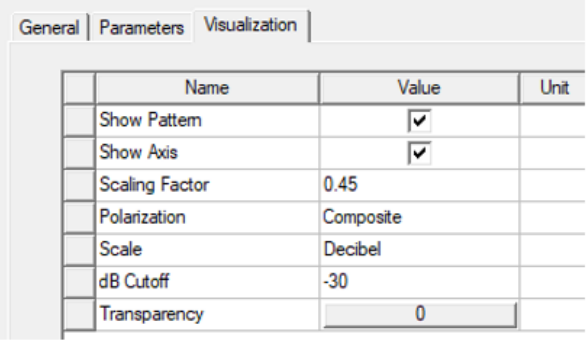


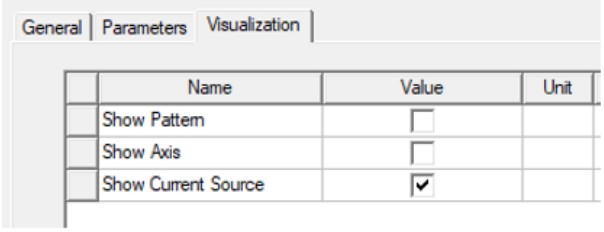
Notice that other edit commands for Copy, Paste, and Duplicate are also available.

4. If you want to edit the Visualization parameters, select the Antenna model icon under the **Project** tree or **History** tree to view the **Properties**, and select the **Visualization** tab. The visualization parameters available can be different for different antenna models. Visualization parameters have no influence on simulation results. They only determine

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

how the parametric antenna is displayed in the model viewer.

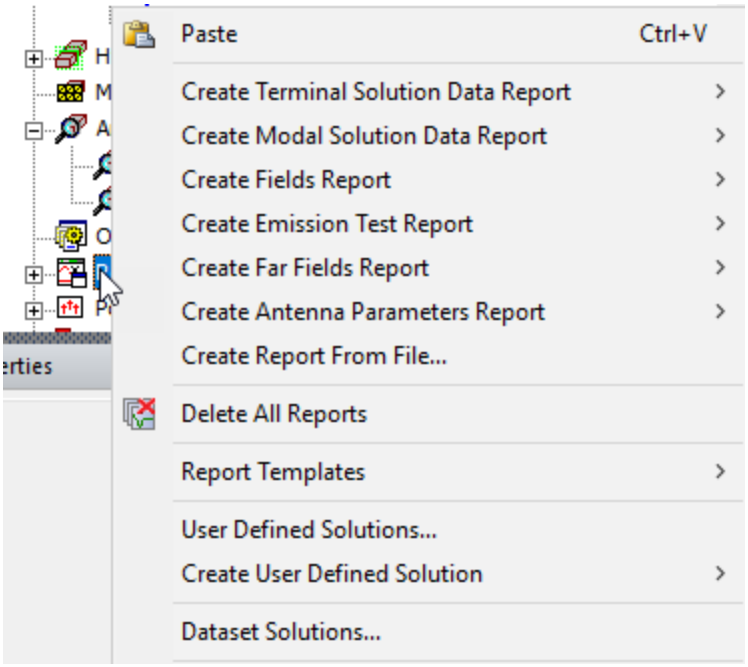




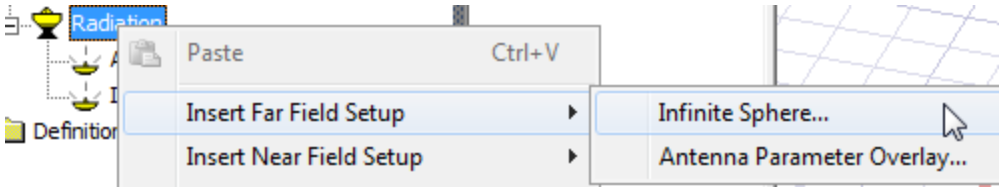
- Choose whether to show the pattern and/or axis. Parametric antennas in a current-sources representation also provide the option to view them according to their far-field Pattern, even though they always launch rays from a distribution of currents sources during SBR+ simulation.
- If the antenna parameters include Show Current Source, you can enable or disable visualization here.
- For Polarization, click on the cell for a drop-down menu selection for Composite, Horizontal, Vertical, LHCP, or RHCP. The Polarization rendering setting only controls which polarization component of the parametric antenna's radiation pattern is currently displayed. It has no impact on the polarization characteristics of the parametric antenna definition and, thus, no impact on simulation results. Some antennas, such as the Parametric Beam, also provide a control for its polarization behavior in the **Property** tab, which does influence simulation results.
- Scale can be Decibel (the default), Linear, or Power.
- dB Cutoff appears when you select Decibel as the Scale. This sets the dB level at the pattern display origin relative to its peak level.
- For Transparency, click on the cell for a dialog with a slider and text field.
- For Scaling Factor, click on the cell to edit the value. The Scaling Factor controls how large the antenna's radiation pattern is displayed relative to its Footprint Size.

5. After you run an SBR+ simulation with parametric antennas, you can create a variety of reports, including S Parameters, Far Fields, and Antenna Parameters.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You must create a Far Field Setup to access the Far Fields and Antenna Parameters reports.



Parametric Antenna Descriptions

When you specify an SBR+ Solution type, you have access to the following sources and parametric antennas:

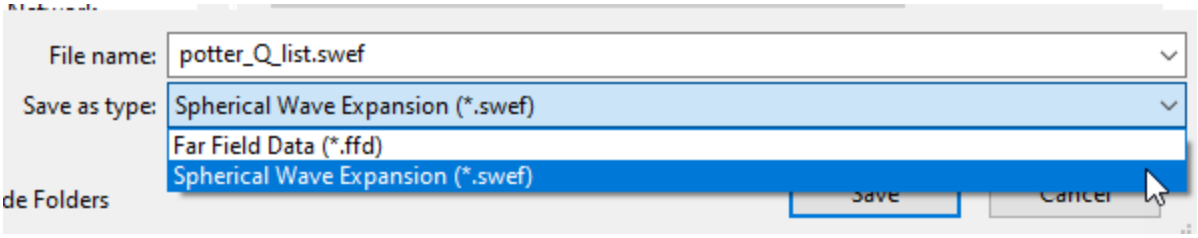
- "By File Antenna for SBR+" on the facing page
- "From External Near Field Data" on page 6-331
- "Link to Source Design" on page 6-337
- "Conical Horn" on page 6-338 (far field or current source option)
- "Cross Dipole" on page 6-340 (current source)
- "Half-Wave Dipole" on page 6-342
- "Horizontal Dipole" on page 6-343 (current source)
- "Parametric Beam" on page 6-345
- "Parametric Slot" on page 6-345 (far field or current source option)
- "Pyramidal Horn" on page 6-347 (far field or current source option)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- "Quarter-Wave Monopole" on page 6-349
- "Short Dipole" on page 6-350
- "Small Loop" on page 6-350
- "Wire Dipole" on page 6-351 (far field or current source option)
- "Wire Monopole" on page 6-352 (far field or current source option)

By File Antenna for SBR+

You define the **By File...** antenna by importing an external file containing far field data in .ffd, or for Spherical Wave Expansion Import, you can import .swef format files.



Pattern samples are entered over a two-dimensional angular sampling domain in spherical coordinates, θ and φ per frequency. The initial or final angle can be arbitrary (that is, no restriction on θ being from 0 to 180 degrees, and φ from 0 to 360 degrees). Nevertheless, the sampling of the θ and φ domain must be such that if adding more points to the domain, 0- and 180-degree samples are exactly captured.

For example, suppose one domain is defined as 1 90 90, that is, from 1 to 90 degrees with 1-degree step. This domain is compatible with the above restriction, because by adding points to the domain every 1 degree the domain can be extended to 0 180 181.

Consider another domain defined as 1 89 45, that is, from 1 to 89 degrees with 45-degree step. The domain is not compatible with the above restriction, because 0 and 180 degrees cannot be exactly sampled by continuing the domain. In other words, the external far-field can be specified over a sector which does not include θ = 0 or 180 degrees and φ = 0 or 180 degrees. However, if the defined sector is extended to cover the entire sphere using the specified sampling, the entire sphere should contain samples at θ = 0 and 180 degrees, φ = 0 and 180 degrees.

The SBR+ solver will automatically interpolate the source fields from the file-based antenna in order to represent the antenna at all frequencies of the user defined frequency sweep in solution setup. The file-based antenna requires at least two frequencies, and the min/max frequencies of the user defined sweep should fall within the frequency band of the file-based antenna.

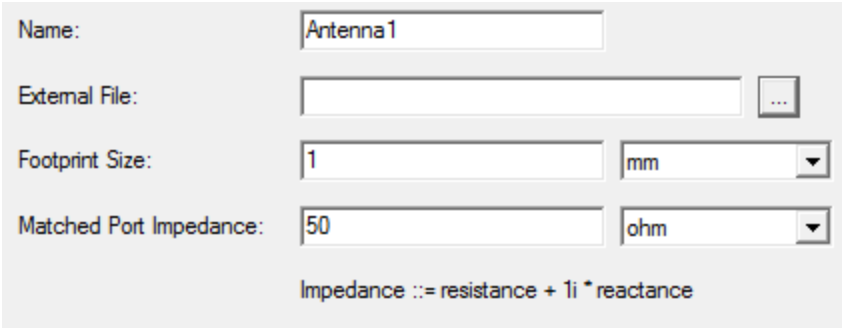
Pattern levels specified in the file are automatically renormalized so the antenna radiates at the power level specified in the **Edit Post Process Sources** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

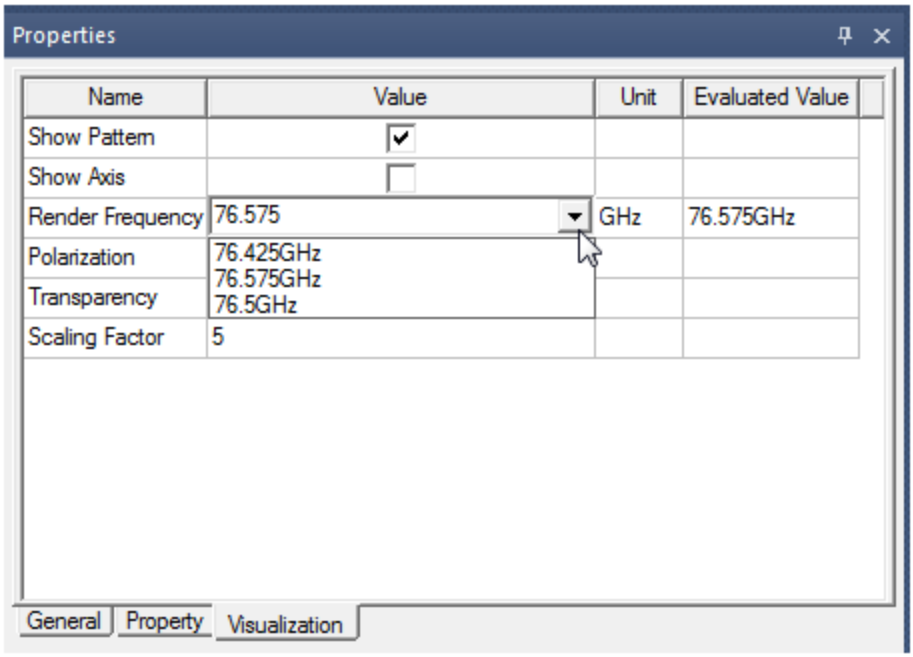
For detailed information on the swef format, and the Parameters area that appears in the import dialog, see Spherical Expansion Wave Source (*.swef).

For detailed information on the ffd format, see External Data File for Far Field Wave.

For editing properties, placement, and visualization, file based antennas resemble parametric antennas.



For the By File antenna, the **Visualization** tab for **Properties** also includes a drop down for **Render Frequency** which lists the available frequencies defined in the Far Field data file (*.ffd).

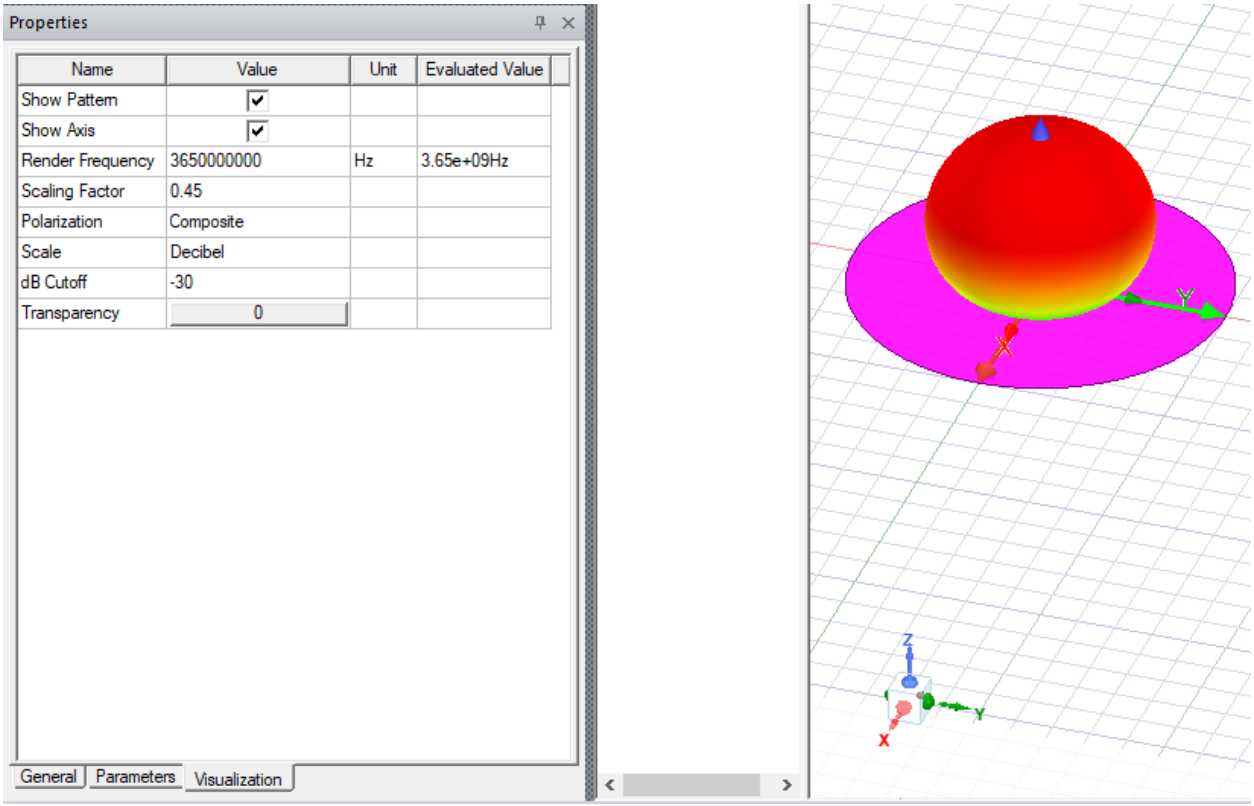


If you enable the **Show Pattern**, or load a design with the option enabled, if the .ffd file is not found, an error message reports this. Validation checks to see if the Render frequency specified is provided in the .ffd file. After you create a File Based Antenna, you have the option to reload

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the *.ffd file. If you archive a project with a File Based antenna, the process automatically includes the *.ffd file and restores it to a relative location for portability.

If you import of Spherical Wave Expansion File import for .swef files, visualization for the antenna is also available. Visualization parameters include Render Frequency, Scaling Factor, Polarization, Scale, dB Cutoff, and Transparency. You can edit fields or select from drop down menus. You can use an SWE antenna to produce S-matrix and far field results.



From External Near Field Data

Near field characteristic of the source antenna, which is saved in .and and .nfd format, can be used to create external near field antenna and act as a virtual port in SBR+ target design. Compared to near field antenna link, this "file-based" near field approach does not need any information of the source antenna design except for the near field on specific sample points, and is therefore convenient to use and easily maintainable and scalable.

For defining simple-primitive near field source, HFSS uses two kinds of external data file. First, the actual location/field data is stored in .nfd format. Second, the Ansys Near field Descriptor file (.and file) stores the general geometry information and paths of .nfd data. Adding a external near field link to HFSS design is therefore a two-steps workflow. During excitation setup, an external near field design (that is, a design with a box representing mesh points) is created based on

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

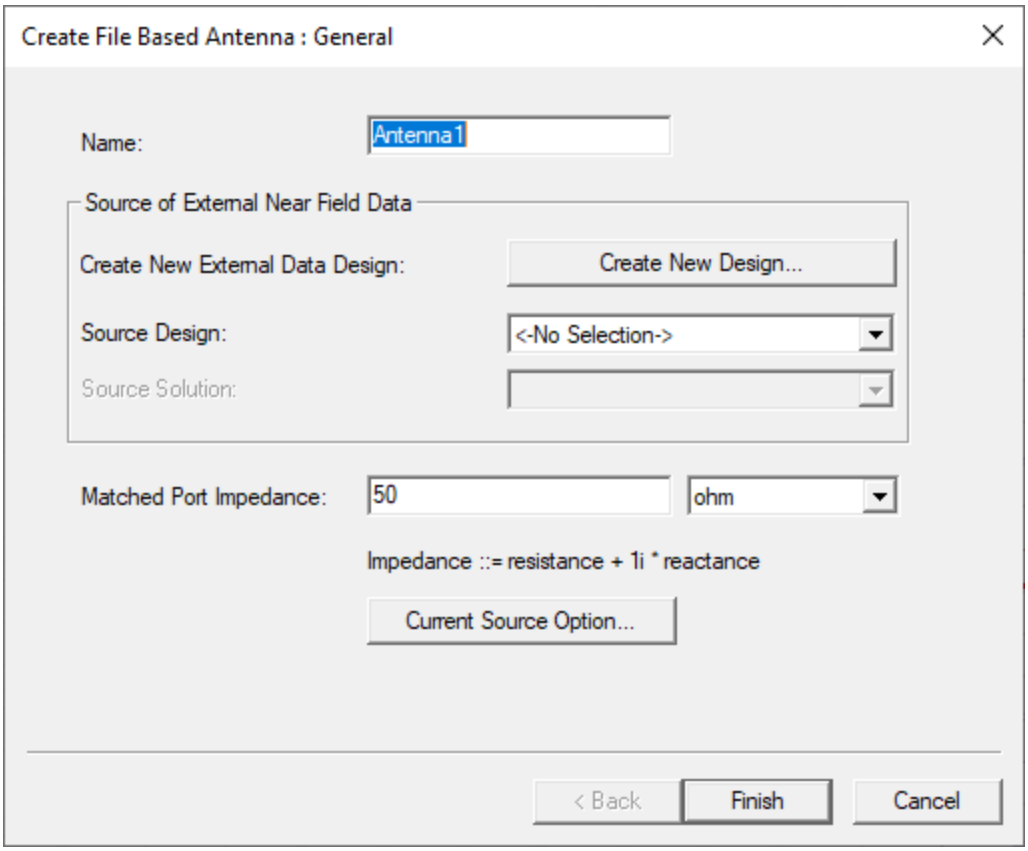
information in a .and file and linked to target design. During simulation, external near field design acquires mesh points and field from .nfd file based on specific frequency the target design asks for.

For HFSS SBR+ design you can also create external near field design based on an .and file or select from existing ones. A virtual port is created for each external near field antenna, and its corresponding footprint is created similar to an HFSS linked antenna.

To create a new Near Field design, based on an .and file:

Open an empty SBR+ design, open external near field antenna dialog by selecting **3D Component** > **Create Antenna** > **From External Near Field Data...**

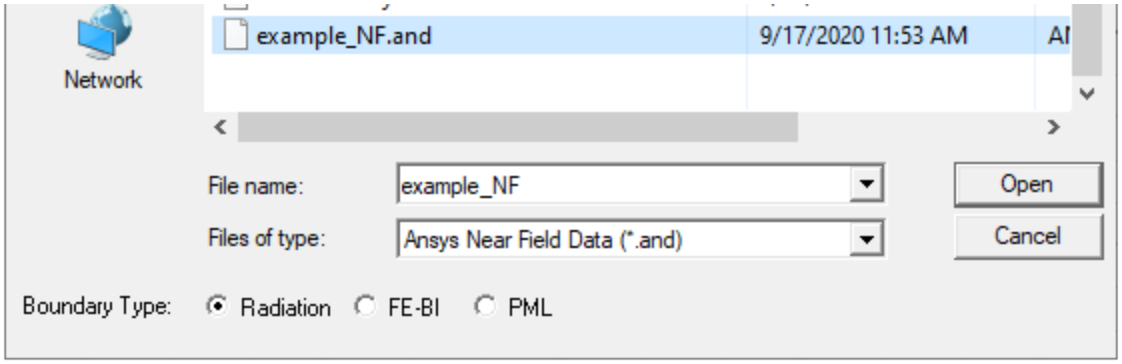
This opens the following **Create File Based Antenna** dialog box:



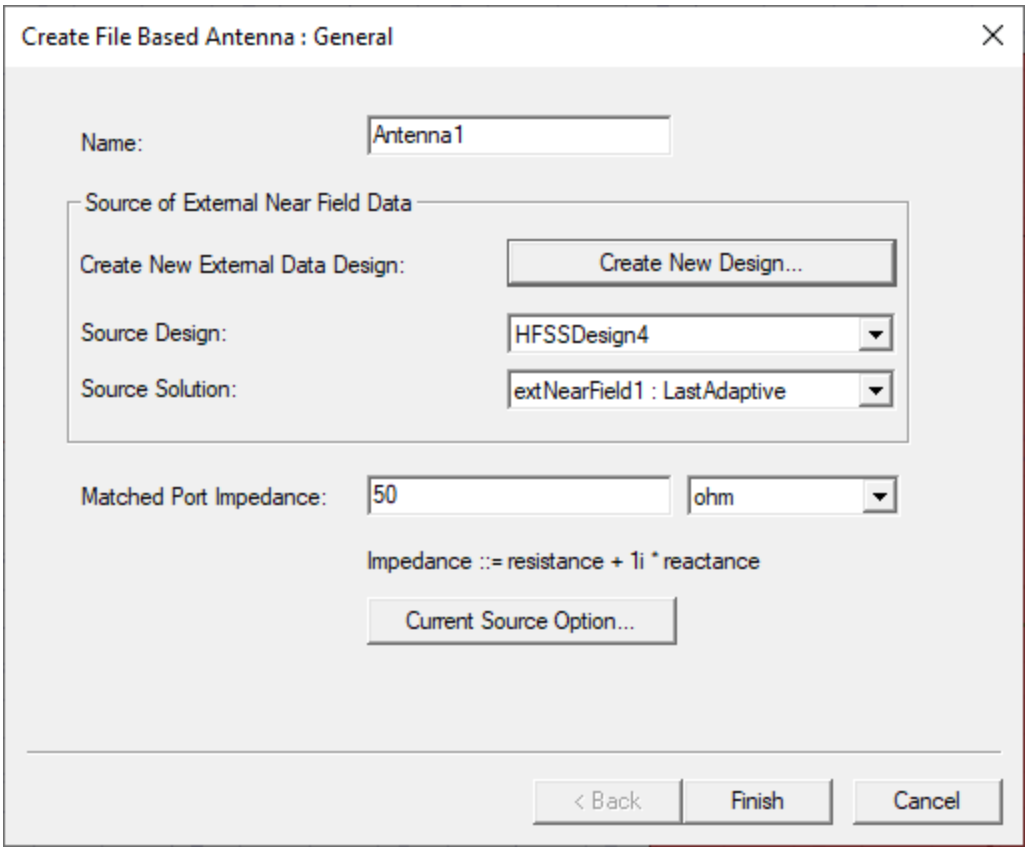
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initially, because there is no external near field design, the “Source Design” combo box shows “No Selection” and “Source Solution” combo box is grayed out.

To create a new external near field design, select the **Create New Design...** button. A file selection dialog box will pop out and boundary type is allowed for selection.

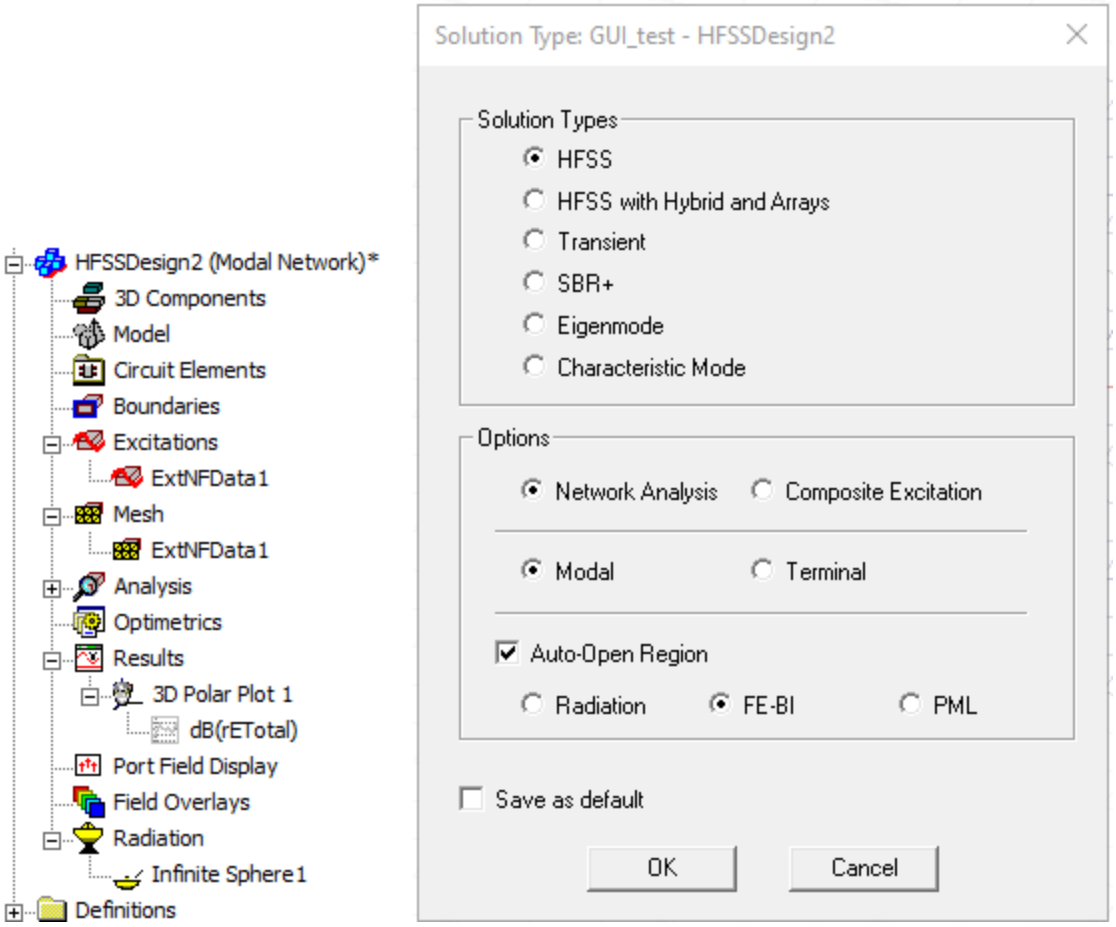


When you select an .and file, make the Boundary Type selection, and click open, the Create File Based Antenna dialog is populated with information about the Source Design and Source Solution.



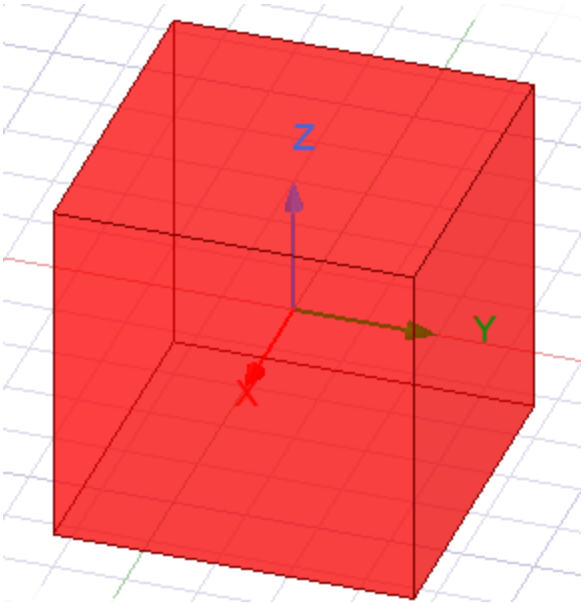
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Additionally, a new external near field design (“HFSSDesign2”) is created in the Project tree. This newly created design automatically creates several setups like mesh, solution, report and radiation. It also creates a native geometry as defined in .and file.

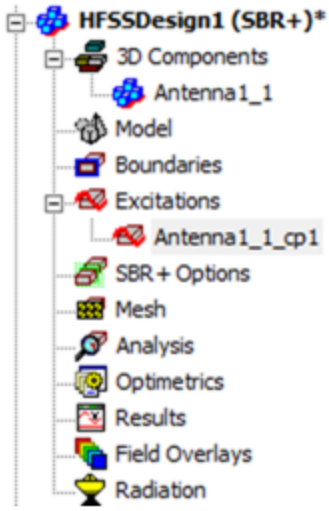


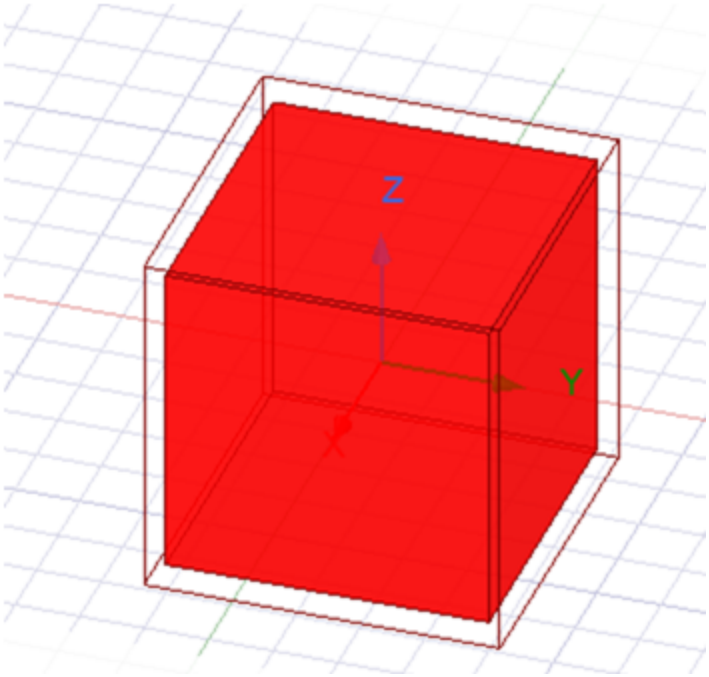
The modeler window shows a native geometry as defined in the .and file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Click **Finish** in the **Create File Based Antenna** dialog and a new external near field antenna is created, together with a virtual port and antenna footprint (including outer bounding box and inner near field geometry).

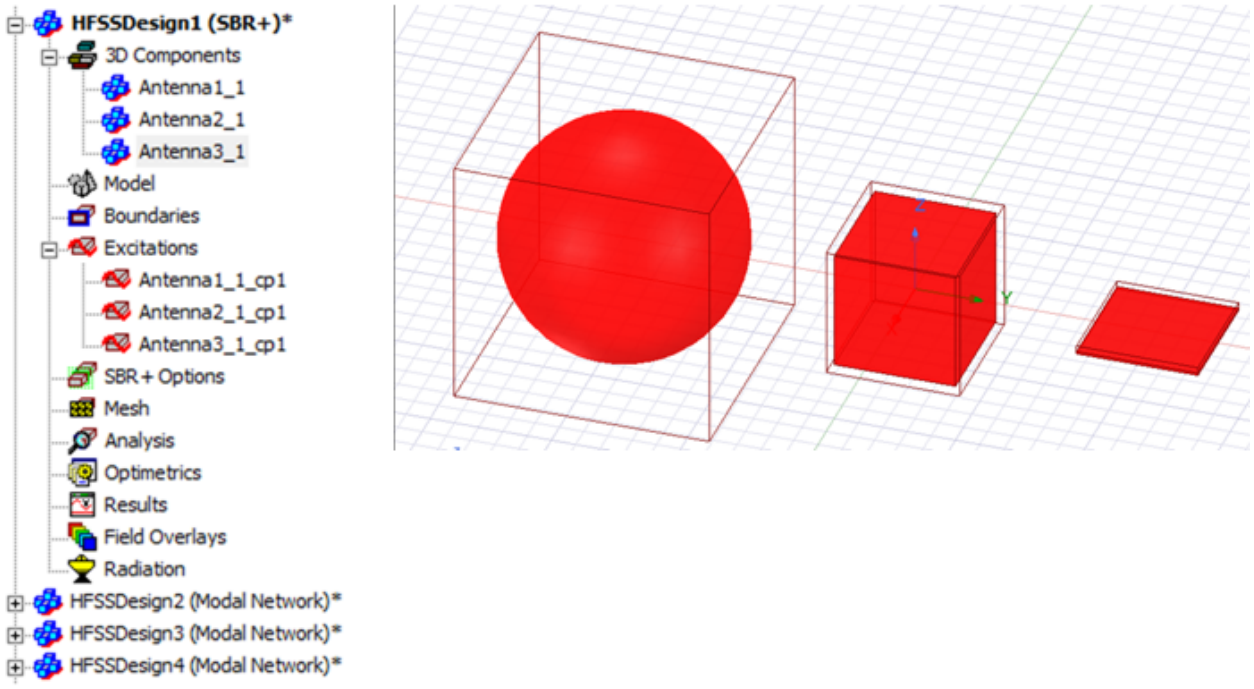




Editing an existing external near field antenna

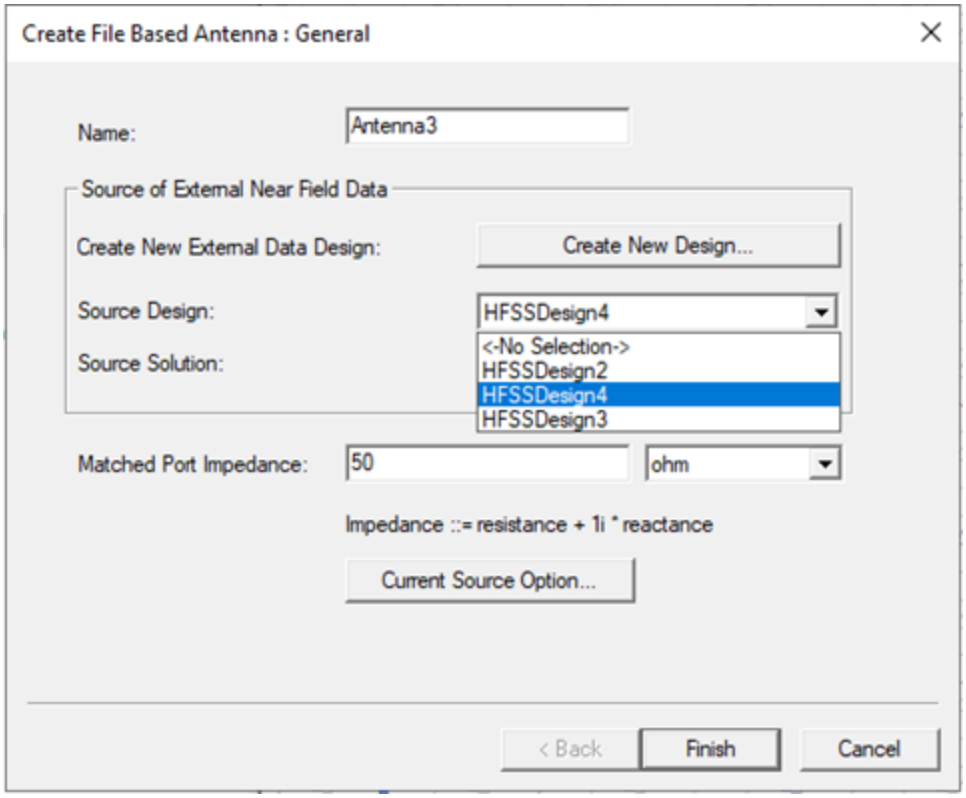
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Let say we already create several external near field antennas and their corresponding external designs using similar approaches.

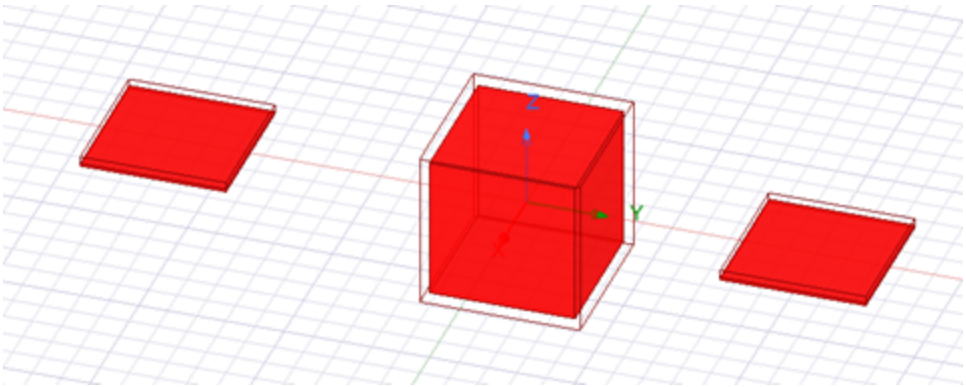


When we edit one of the antennas (in this case Antenna3), the dialog show its corresponding source design name and solution name.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If we change the source design and click **Finish**, its footprint will be updated.



Link to Source Design

HFSS lets you create a one-way link from a Driven Modal or Driven Terminal design to an SBR+ design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

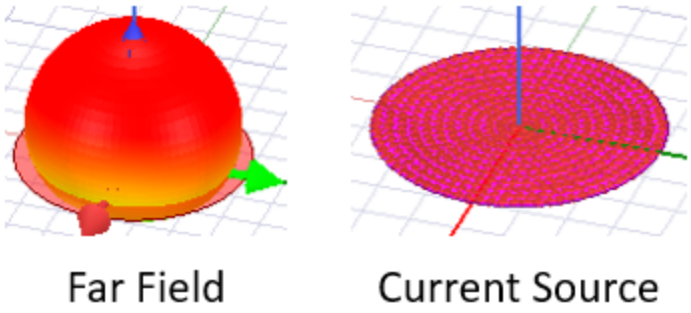
Conical Horn

This frequency-dependent antenna approximates a flared conical horn with user-defined dimensions and polarization. It is based on the approximated field distribution of a circular cross-section conical horn driven in the TE_{11} waveguide mode. The model accounts for phase taper at the edges of the flat aperture (horn mouth) due to the spherical phase front introduced by the walls flaring from the circular waveguide feed at the throat of the horn.

The aperture (horn mouth) sits in the XY plane of the antenna local coordinate system, with boresight along the +Z direction. The antenna is characterized by the following settings: Mouth Diameter, Flare Half-Angle, and Polarization.

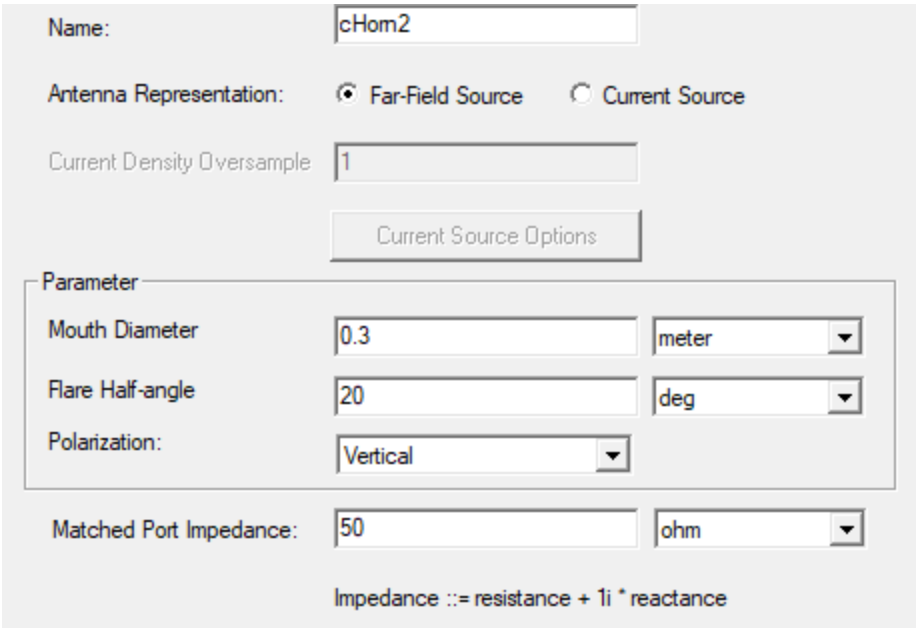
The conical horn will not operate below the TE_{11} cut-off frequency of a circular waveguide, given by $f_{cutoff}> = (x'_{11}\ c)/(\ \pi\ d_{aper})$, where $x'11 \sim= 1.8412$ is the first zero of the derivative of the first-order Bessel function of the first kind, c is the speed of light, and d_{aper} is the mouth (aperture) diameter. Attempts to simulate with a conical horn below this frequency will result in an error. Technically speaking, the cut-off frequency should be based on the smaller diameter of the waveguide, not the diameter of the horn mouth, and it should therefore be higher. However, since the waveguide diameter plays no other role in the conical horn model, it is left out of the specification parameters in the interest of simplicity, and we instead tie the cut-off frequency to the mouth diameter in this SBR+ parametric antenna model.

The dialog also lets you choose Antenna Representation as Far Field Source or as Current Source.

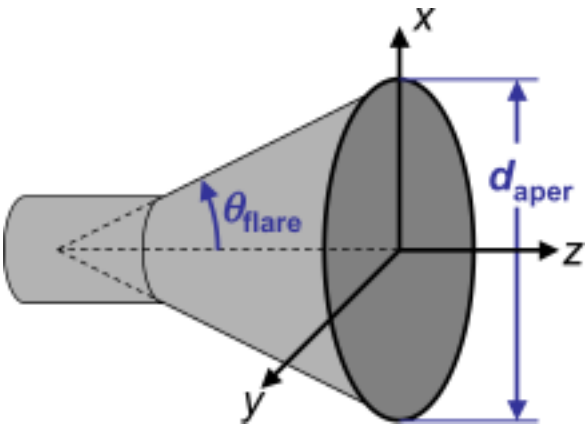


You specify parameters for mouth, flare half-angle, polarization, and matched port impedance.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For conical horn antennas, the following figure explains the meaning of the dimensions.

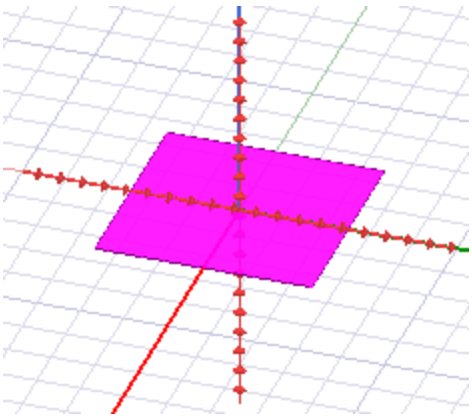


If you select Current Source, the field for Current Density Over sample is enabled and the current source options button is enabled. For Current Source, you also need to specify Current Density Oversample, where the default value of 1.0 corresponds to 4 current-source samples per wavelength. Oversample values less than 1 are usually too course for an accurate result, while those greater than 2 usually do little to improve accuracy while harming run time. The default of 1.0 is the recommended setting for most cases, and we strongly discourage setting Current Density Oversample less than 1.0.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

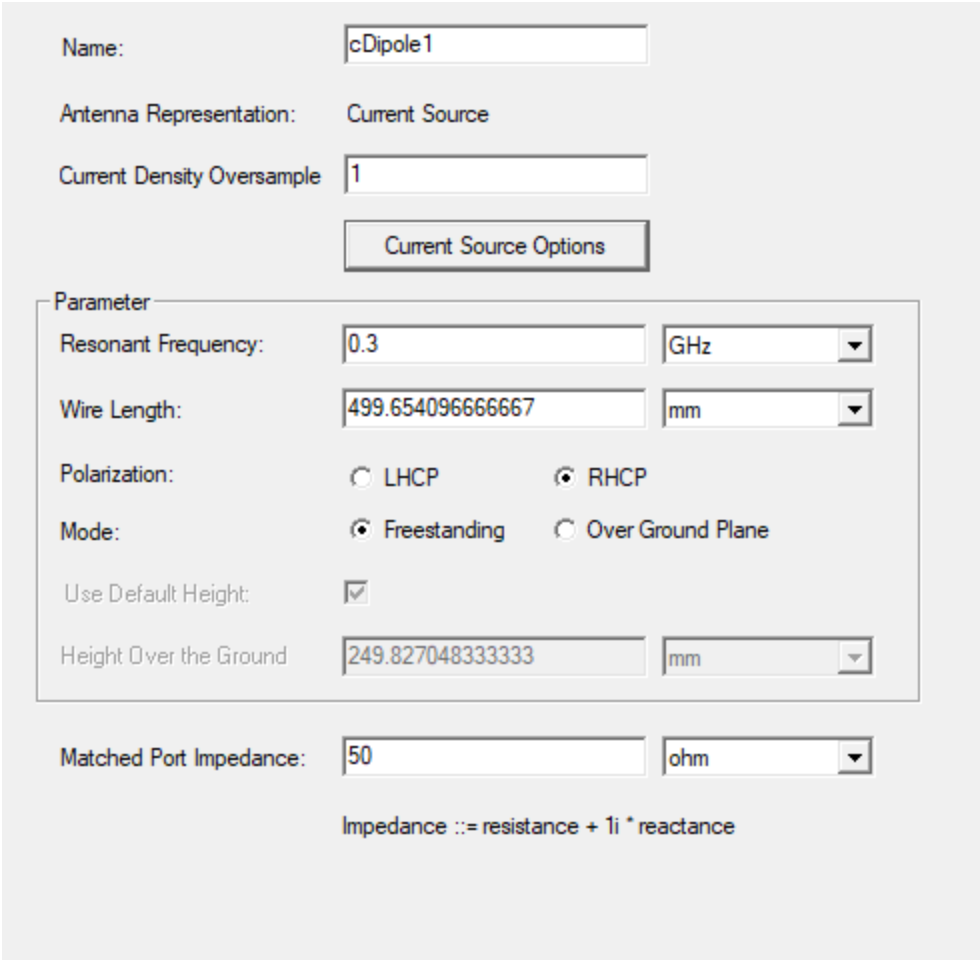
Cross Dipole

This antenna always uses a current source representation. Parameters are resonant frequency, wire length, polarization, and mode as free standing, or over ground plane. In the its Freestanding mode, the electric current sources represent two perpendicular arms of a dipole, one each along the Y and Z axes, driven with a 90 degree phase shift relative to each other to achieve the selected circular Polarization along the +X axis (it will be the opposite polarization along the -X axis). In its Over Ground Plane mode, the antenna arms are rotated to be parallel to the XY plane and at a default height along the Z axis of a quarter-wavelength at the specified Resonant Frequency. This allows the antenna to be placed directly on a metal surface of the SBR+ platform and have its reflected field constructively interfere with the direct incident field from the dipole arms when operating at the Resonant Frequency and in the +Z direction. You can switch off Use Default Height to manually set the height of the current source above the implied ground plane of this antenna if that better fits the design of the cross-dipole antenna to be simulated. You also specify matched port impedance.



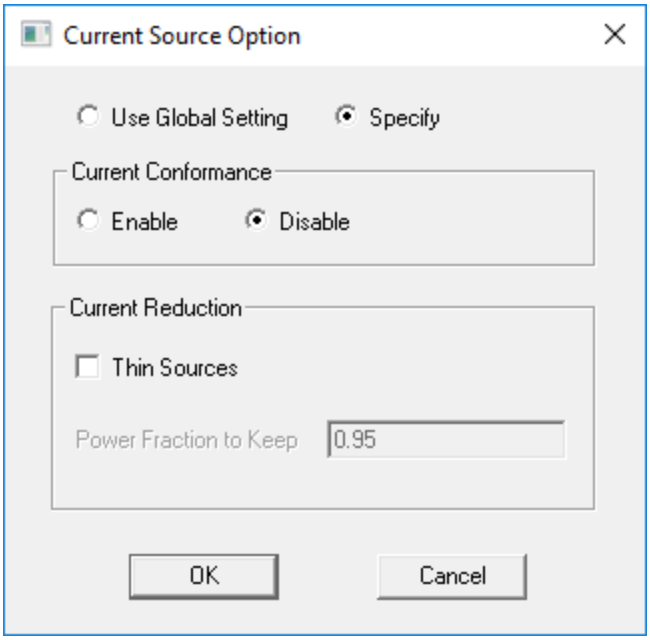
Current Source Options opens a dialog with Use Global Setting (the default) or Specify, which enables the fields for Current Conformance, and Current Reduction settings to allow you to Thin Sources, and to specify a Power Fraction to Keep.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



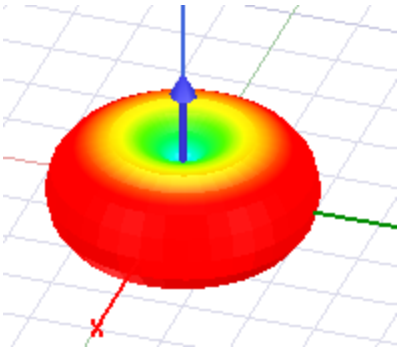
Current Source Options opens a dialog box with Use Global Setting (the default) or Specify, which enables the fields for Current Conformance, and Current Reduction settings to allow you to Thin Sources, and to specify a Power Fraction to Keep.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

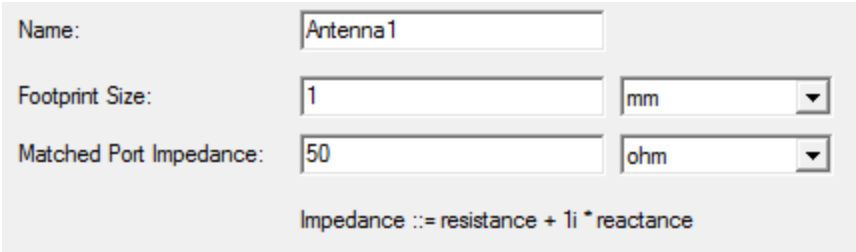


Half-Wave Dipole

This antenna produces a far-field pattern of a center-fed thin dipole with a total length of a half-wavelength, with a peak Directivity of 2.15 dBi. The Half-Wave Dipole has no current-sources representation since it needs to be resonant at all simulation frequencies. For this reason, it is not recommended to locate it within two wavelengths of any SBR+ scattering geometry. If that is needed, one should instead consider to use the Wire Dipole or Horizontal Dipole in their current-sources representation.

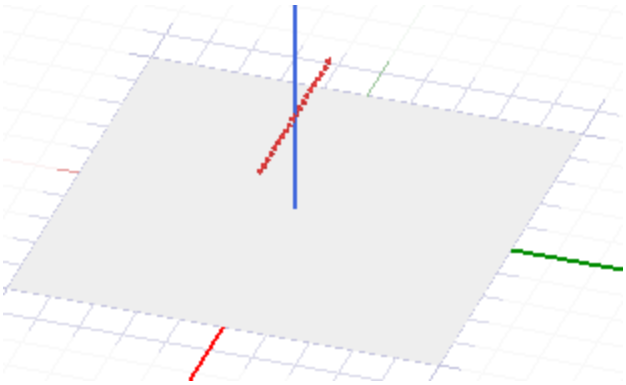


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



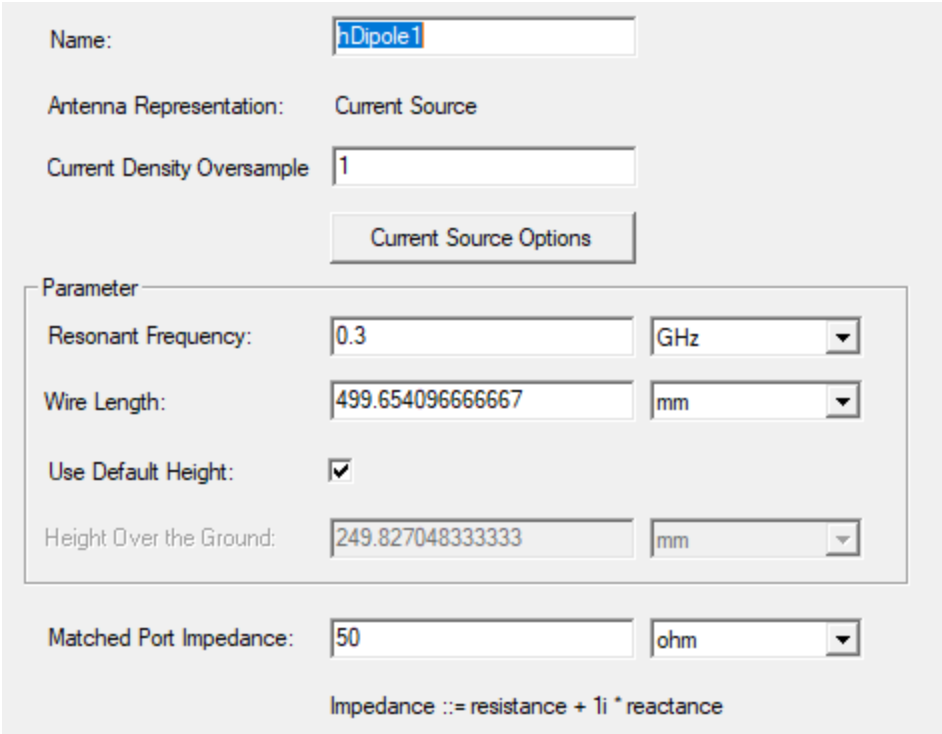
Horizontal Dipole

This antenna always uses a current-sources representation. It is intended for direct placement on a metal surface of the SBR+ platform given its built-in default height of a quarter-wavelength.

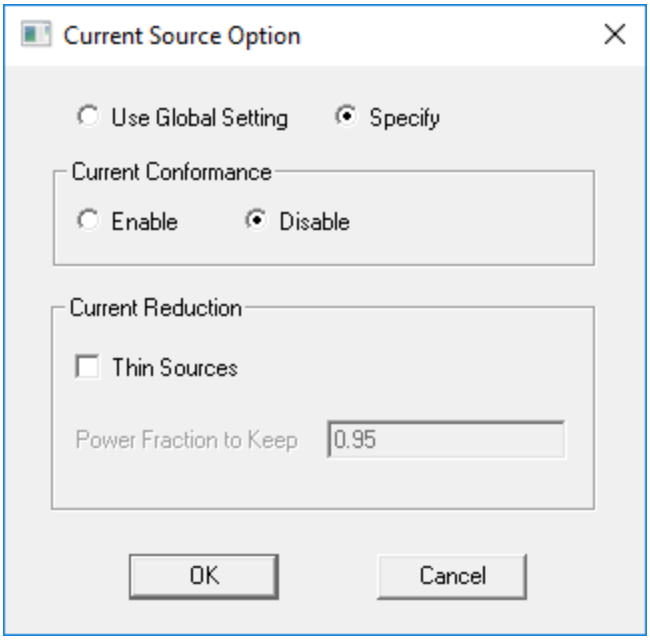


Other parameters include current source options, resonant frequency, wire length, matched port impedance, and whether to use the default height or to override it and manually specify the height of the sources above the implied grounding surface. The default height is a quarter-wavelength at the specified Resonant Frequency. Assuming direct placement on a metal surface of the SBR+ platform, the intended context of this antenna, the reflected field from the platform will constructively re-enforce the direct field from the current sources in the +Z direction of the antenna.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



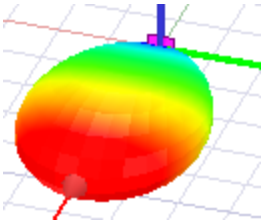
Current Source Options opens a dialog with Use Global Setting (the default) or Specify, which enables the fields for Current Conformance, and Current Reduction settings to allow you to Thin Sources, and to specify a Power Fraction to Keep.

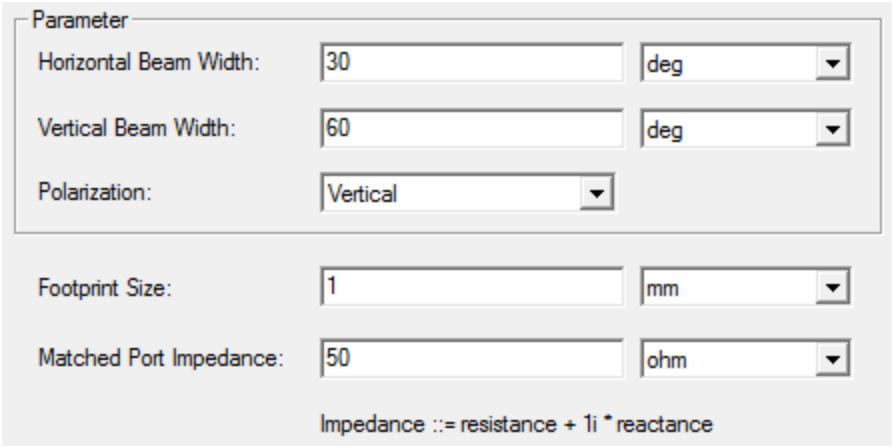


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parametric Beam

The Parametric Beam antenna for SBR+ produces an idealized pencil beam that can substitute as a simplification for a variety of aperture based antennas, including horns and broadside arrays. The main beam is pointed in the X direction of the antenna's local coordinate system and has the mathematical form, $\sin^{Nv}\theta \cos^{Nh}\phi$, where *Nv* and *Nh* are internally determined according to the Vertical and Horizontal Beam Width settings, respectively. The antenna produces an idealized pencil beam ($\cos^{n}\theta$) pattern) with user-defined beam widths and polarization. This antenna produces no side lobes or backlobes and only radiates in the forward half-space. The Parametric Beam antenna has no current-sources representation.





Parametric Slot

This antenna is based on an idealized sinusoidal (magnetic) current distribution for the slot along the X-axis of the coordinate system. The Far-Field pattern will vary according to the slot's electrical length at the visualized frequency. When operated at the user-defined Resonant Frequency, the slot length is 0.5 wavelengths.

Note that changing the Resonant Frequency will update the Slot Length and vice versa.

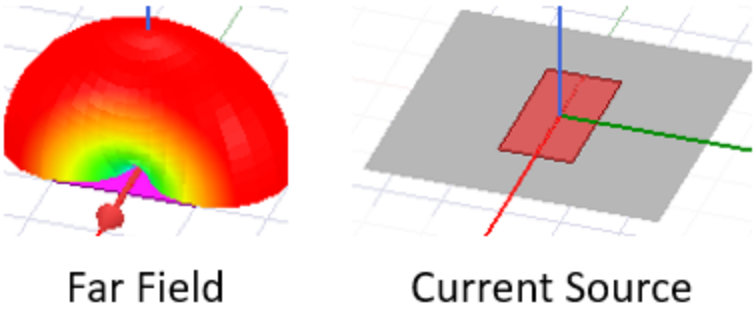
The Parametric Slot is a ground-plane antenna whose intended placement is on a metal surface of the SBR+ scattering geometry or platform. In this context, use of the current-sources

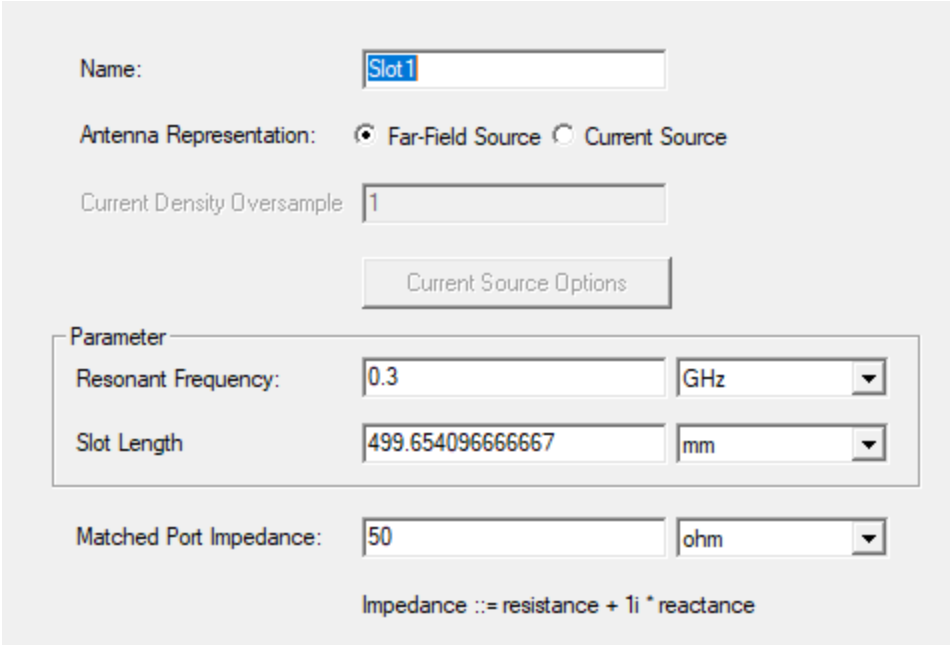
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

representation is strongly recommended. Although notionally located in the z = 0 plane of the antenna local coordinate system, the magnetic sources of the slot will instead be placed at a small offset height of λ/40 measured at the shortest wavelength of the simulation. This allows the parametric slot to be directly placed on a metal surface of the SBR+ platform without the need to manually offset it so that the sources don't inadvertently reside under the placement surface.

One can also select a far-field representation. However, given its assumed ground-plane context, the parametric slot radiates no far fields in its lower half-space, so all direct interaction with the placement platform is ignored during SBR+ simulation with this representation of the slot except for those surfaces of the platform or scattering environment that reside about the z = 0 plane of the slot antenna.

. The dialog also lets you choose Antenna Representation as Far Field Source or as Current Source.





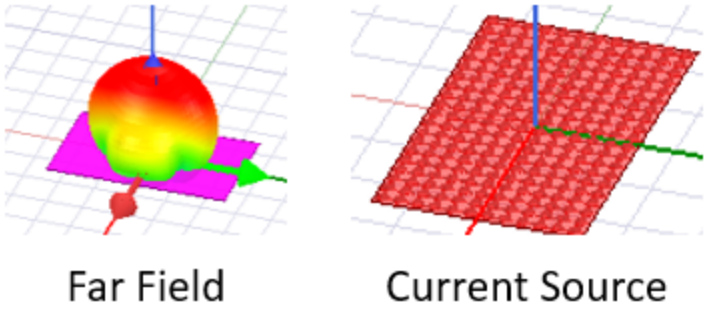
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pyramidal Horn

This frequency-dependent antenna approximates a flared rectangular horn with user-defined dimensions. It is based on the approximated field distribution of a rectangular cross-section pyramidal horn driven in the TE_{10} waveguide mode. The model accounts for phase taper at the edges of the flat aperture (horn mouth) due to the elliptical phase front introduced by walls flaring from the rectangular waveguide feed at the throat of the horn.

The aperture (horn mouth) sits in the XY plane of the antenna local coordinate system, with boresight along the +Z direction. The antenna is characterized by the following geometric dimensions: Mouth Width, Mouth Height, Waveguide Width, Width Flare Half-Angle, and Height Flare Half-Angle. From these parameters, the waveguide height is implied. If the implied waveguide height is negative, an error report is generated with suggestions for adjusting the input dimensions to resolve the conflict.

The Pyramidal Horn will not operate below the TE_{10} cut-off frequency of the rectangular waveguide, given by $f_{cutoff} = c/2w$, where w is the waveguide width and c is the speed of light. Attempts to do so will results in an error during simulation.



The dialog also lets you choose Antenna Representation as Far Field Source or as Current Source.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name: pHorn1

Antenna Representation: Far-Field Source Current Source

Current Density Oversample: 1

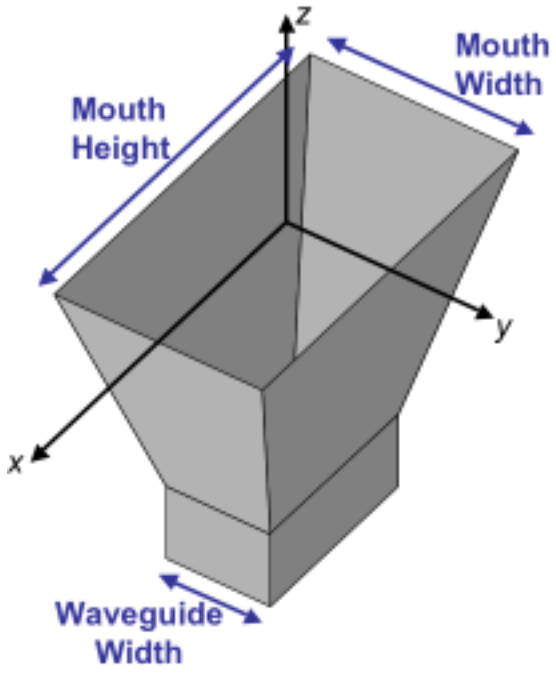
Current Source Options

Parameter

Mouth Width	0.3	meter
Mouth Height	0.5	meter
Waveguide Width	0.15	meter
Width Flare Half-angle	20	deg
height Flare Half-angle	35	deg

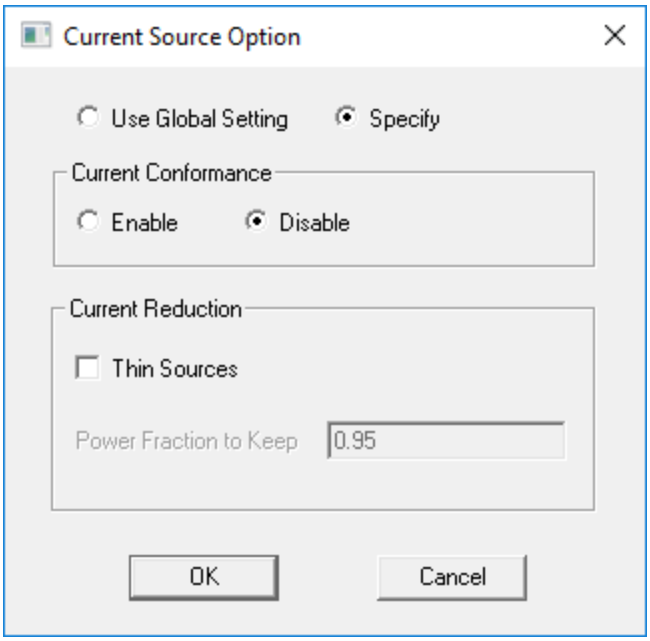
Matched Port Impedance: 50 ohm

Impedance ::= resistance + 1i * reactance



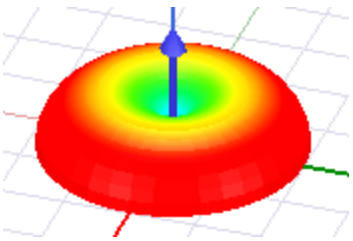
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Current Source Options opens a dialog with Use Global Setting (the default) or Specify, which enables the fields for Current Conformance, and Current Reduction settings to allow you to Thin Sources, and to specify a Power Fraction to Keep.

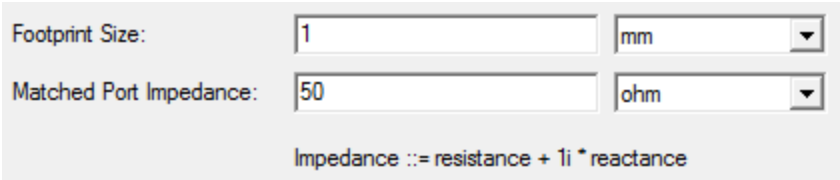


Quarter-Wave Monopole

This antenna produces a far-field radiation pattern defined over a half-space (top half of a torus), consistent with an ideal quarter-wave monopole antenna over an infinite ground plane. It has a Directivity of 5.16 dBi in the upper half-space; no radiation gain in the lower half-space. This antenna only illuminates objects above the monopole and will not capture direct interactions with its placement surface on the SBR+ geometry. To model the interaction of a monopole with its supporting SBR+ platform, users should instead use the Wire Monpole in its current-sources representation.

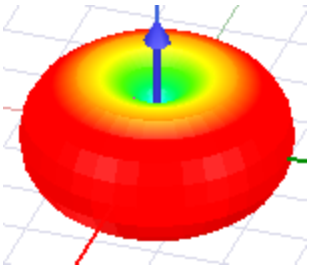


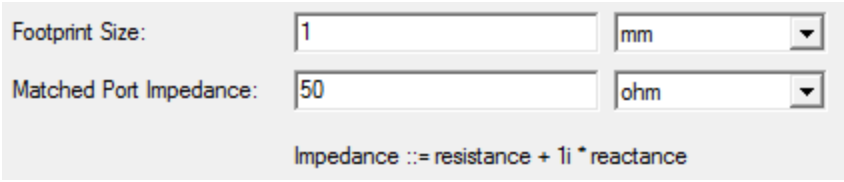
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Short Dipole

This antenna produces a far-field pattern of an ideal Hertzian(electrically short) electric dipole with a peak Directivity of 1.76 dBi. The Short Dipole is suitable as an idealized, yet still physical electromagnetic point source that stands off two or more wavelengths from any scattering geometry. It only has a far-field representation with no higher-order near-field terms. If the antenna is to be placed within one or two wavelengths of the SBR+ scattering geometry, one should consider to instead use the Wire Dipole or Horizontal Dipole in their current-sources representation since these have a valid near-field description.

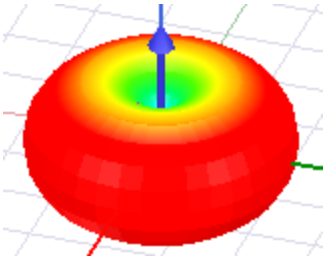


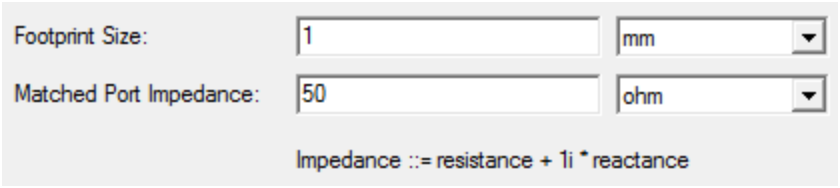


Small Loop

This antenna produces a far-field pattern consistent of an ideal Hertzian loop (electrically short) magnetic dipole with a peak Directivity of 1.76 dBi. Because it only supports a far-field representation, it should only be used when it can be placed two or more wavelengths away from any SBR+ scattering geometry.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

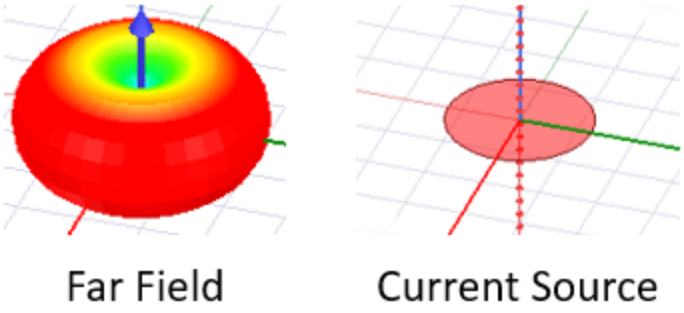




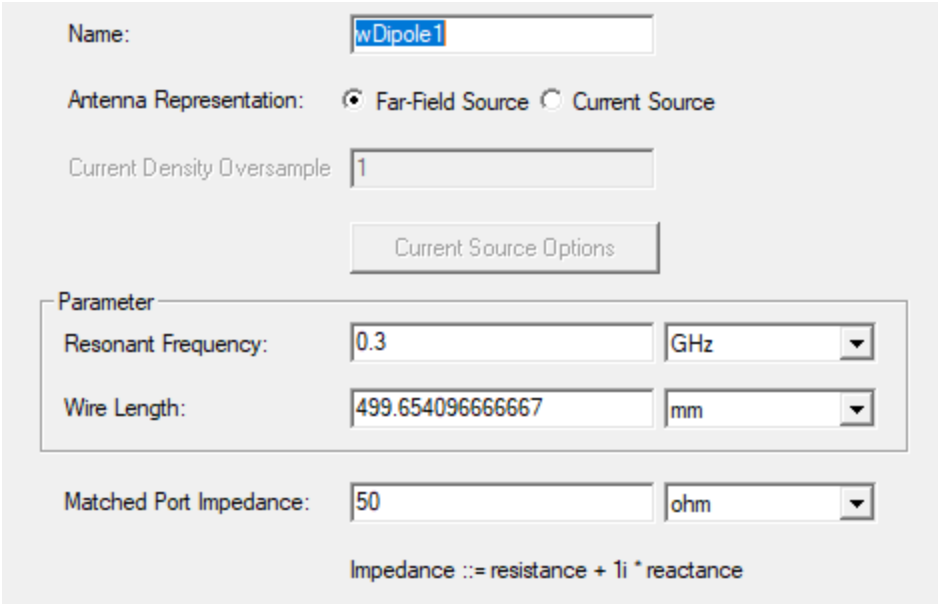
Wire Dipole

This antenna is based on an idealized sinusoidal current distribution of a wire dipole along the Z-axis of the antenna coordinate system. The Far-Field pattern will vary according to the dipole's electrical length at the visualized frequency. When operated at the user-defined Resonant Frequency, the wire is a half-wavelength long. Note that changing the Resonant Frequency will update the Wire Length and vice versa.

The Wire Dipole offers both a far-field and current-sources representation. The former is more computationally efficient to use in an SBR+ simulation, but the current-source representation should be used if the antenna is to be placed within 2 wavelengths of the SBR+ scattering geometry or its Wire Length is a significant fraction of its displacement from the scattering geometry.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



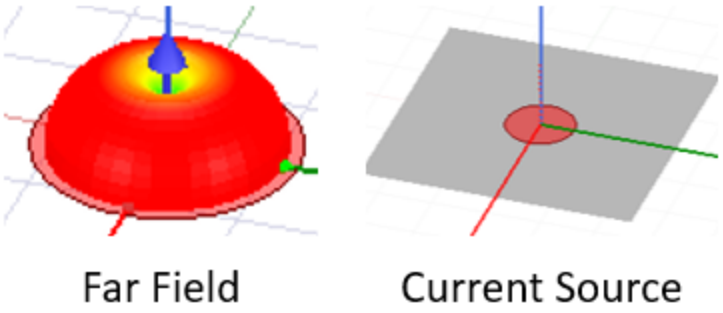
Wire Monopole

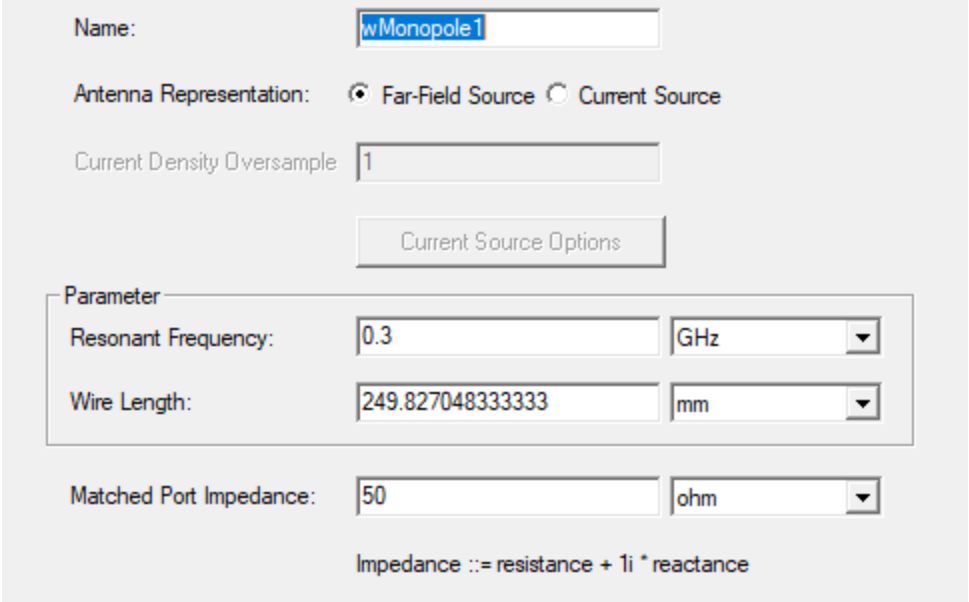
This antenna is based on an idealized sinusoidal current distribution for the monopole along the Z-axis of the antenna coordinate system. The Far-Field pattern will vary according to the monopole's electrical length at the visualized frequency. When operated at the user-defined Resonant Frequency, the monopole is a quarter-wavelength long. Note that changing the Resonant Frequency will update the Wire Length and vice versa.

The Wire Monopole is a ground-plane antenna whose intended placement is on a metal surface of the SBR+ scattering geometry or platform. In this context, use of the current-sources representation is strongly recommended.

One can also select a far-field representation. However, given its assumed ground-plane context, the Wire Monopole radiates no far fields in its lower half-space, so all direct interaction with the placement platform is ignored during SBR+ simulation with this representation of the monopole except for those surfaces of the platform or scattering environment that reside above the z = 0 plane of the monopole antenna.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Modifying Objects

You can quickly modify the position, dimensions, and other characteristics of objects created in the 3D Modeler window.

What do you want to do?

Object and History Editing	**Modify Object Appearance**
Modify Object Dimensions	**Modify Object Location or Orientation**
Surface or Edge Operations	**Boolean Operations on Objects**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Object and History Editing

- Copy and paste objects.
- Delete objects
- Delete Last Operation
- Cutting Objects
- Duplicate objects
- View and Edit Commands on History Tree Objects
- Purge History
- Generate History

Modify Object Appearance

- Assign color to an object.
- Assign transparency to an object.
- Also see Modifying the Model View

Modify Object Dimensions

- Assigning a Cross Section and Dimension to a Polyline
- Scale the size of objects
- Connect objects
- Move faces or edges
- Convert polyline segments
- Rounding the edge of an object (Fillet)
- Flattening the edge of an object (Chamfer)

Modify Object Location or Orientation

- Move objects
- Rotate objects
- Change the Orientation of an object
- Mirror objects about a plane.
- Offset an object (move every face of an object).
- Also see Modifying the Model View

Surface or Edge Operations for Objects

- Sweep objects.
- Cover lines.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Cover faces.
- Uncover faces.
- Detach faces.
- Detach edges.
- Create a new object by taking a cross-section of a 3D object.
- Wrap Command
- Project Sheet
- Thicken Sheet

Boolean Operations on Objects

- Unite objects.
- Subtract objects.
- Create objects from intersections.
- Create an object from a face.
- Create an object from an edge.
- Split objects.
- Separate objects.
- Imprint Projection Commands
- Imprinting an Object

Assigning Color to Objects

1. Select the object to which you want to assign a color.

   > **Note:**
   >
   > If the *Properties* window is not visible on the desktop, click **View> Properties Window** or use **Edit> Properties**.

2. In the *Properties* dialog box or docked *Properties* window, click the **Attribute** tab.
3. Click the color sample in the **Value** column of the **Color** row.

   The *Color* palette appears.

4. Select a *Basic* or *Custom* color from the **Color** palette.

   Alternatively, you can specify a color by typing numbers between 0 and 255 to specify the following values:

   - **Hue, Sat** (saturation), and **Lum** (luminance), or
   - **Red, Green,** and **Blue**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also specify a color by clicking within the color spectrum box and using the luminance slider to the right of this box.

5. Optionally, click **Add to Custom Colors** to save the settings as a custom color, which makes in easier to specify the same color for other objects.

> **Note:**
>
> If you wish to overwrite a previously defined or default custom color, choose the custom color swatch to overwrite *before* specifying the new color parameters. To clear your customizations and restore default choices, click **Reset Custom Colors to Defaults**.

6. Click **OK**.

   The color is assigned to the selected object.

Setting the Default Color of Objects

The default color you specify for objects is applicable to when the objects are rendered as solids (shaded) or rendered as wireframes (that is, outlines only). The outline color is the same as the object color you specify according to the following procedure:

1. Using the menu bar, click **Tools**> **Options**> **General Options**, or, from the **Desktop** ribbon tab, click **General Options**.
2. In the tree at the left side of the *Options* dialog box, select **3D Modeler**> **Display**> **Rendering**.
3. In the *Object Appearance* section, click the color sample button to the right of **Default color**.

   The **Color** palette appears.

4. Select a color from the **Color** palette, as described in the parent topic.

   Any objects you draw after this point will be assigned the default color you selected. However, if the *Material Appearance* option is selected within the *Attributes* tab of the object properties, material color and transparency assignments will override the default assignments.

   Alternatively, select **Use material appearance if available** within the *Options* dialog box to use the material color and transparency values by default (when they exist).

5. Click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning Transparency to an Object

1. Select the object to which you want to assign a transparency.

   > **Note:**
   >
   > If the *Properties* window not visible on the desktop, click **View> Properties Window** or use **Edit> Properties**.

2. In the *Properties* dialog box or docked *Properties* window, click the **Attribute** tab.
3. Click the *Value* in the **Transparent** row.

   The *Set Transparency* dialog box appears.

4. Move the slider to the right to increase the transparency of the object. Move the slider to the left to decrease the transparency of the object. Or, type a decimal number between 0 and 1 (inclusive) in the box to the right of the slider.
5. Click **OK**.

Setting the Default Transparency of Objects

1. Using the menu bar, click **Tools**> **Options**> **General Options**, or, from the **Desktop** ribbon tab, click **General Options**.
2. In the tree at the left side of the *Options* dialog box, select **3D Modeler**> **Display**> **Rendering**.
3. In the *Object Appearance* section, click and drag the **Default Transparency** slider. Move the slider to the right to increase the default object transparency. Move the slider to the left to decrease the default object transparency. Or, type a decimal number between 0 and 1 (inclusive) in the box to the right of the slider.

   Any objects you draw after this point will be assigned the default transparency you specified. However, if the *Material Appearance* option is selected within the *Attributes* tab of the object properties, material color and transparency assignments will override the default assignments.

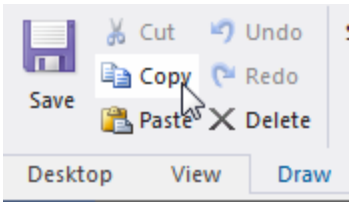
   Alternatively, select **Use material appearance if available** within the *Options* dialog box to use the material color and transparency values by default (when they exist).

Copying and Pasting Objects

To copy objects and paste them in the same design or another design, use the **Edit>Copy** and **Edit>Paste** commands. For data link purposes, where you want to include the material assignments as well as the geometry, you can use the Import from Clipboard command.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

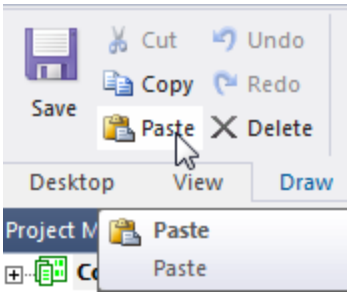
1. Select the objects you want to copy.
2. Click **Edit**>**Copy** or in the **View/Draw/Model** tabs of the ribbon, click the **Copy** icon:



The objects are copied to the Clipboard, a temporary storage area. The selected items are not deleted.

To cut an item to the clipboard and deleting the original, use the scissors icon on the toolbar.

3. Select the design into which you want to paste the objects. It can be the same design from which you copied the items.
4. Click in the **3D Modeler** window.
5. Select the working coordinate system. Objects are pasted relative to the current working coordinate system.
6. Click **Edit**>**Paste** or in the **View/Draw/Model** tabs of the ribbon, click the **Paste** icon:



The objects appear in the targeted Modeler window and in the new History tree. The pasted objects become the current active selection. If you undo the paste, the active selection goes back to the previous selection. If you redo, the pasted objects become the current active selection.

Items on the Clipboard can be pasted repeatedly. The items currently stored on the Clipboard are replaced by the next items that are cut or copied. Designs and projects containing encrypted components can be copy pasted for products using 3D Modeler

Import a Model from the Clipboard

You can import a model from the Clipboard in order to use the geometry from a different design. To use geometry with datalink, the geometry ID must be preserved. The **Import from Clipboard** command preserves geometry IDs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To import a model from the Clipboard, the model for the current design must be empty. If the design is not empty, existing geometry is deleted.

The geometry model is imported from the Clipboard with the ID preserved.

1. Select the objects you want to copy from the source design. For selecting all objects, you can use **Edit**> **Select All** or **Ctrl+A**.
2. Click **Edit**> **Copy**.

   The objects are copied to the Clipboard, a temporary storage area. The selected items are not deleted from the source location.

3. Select the design into which you want to paste the objects. It can be in the same project as the design from which you copied the items or in a different project.
4. Click in the **3D Modeler** window.
5. Click **Modeler**> **Import From Clipboard**.

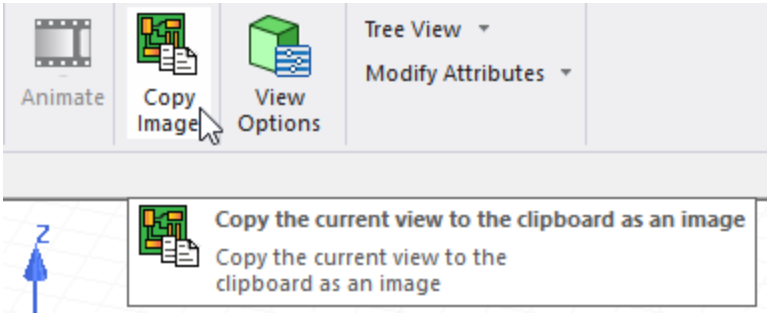
   The geometry is pasted from the Clipboard with the IDs preserved. Additionally, the source design's coordinate systems, materials, appearances, and other attributes are preserved and duplicated in the target design.

Copy Image

You can import images of the **3D Modeler** window or of Reports into any other application. The image has to be copied to the clipboard, so that it can be imported into the other application.

To copy an image of the Modeler window and paste it into another application:

1. Make the Modeler window from which you wish to capture an image active.
2. Copy an image to the clipboard using one of the following three methods:
   - From the menu bar, click **Edit> Copy Image**.
   - Right-click the Modeler window and select **Copy Image** from the shortcut menu.
   - On the **View** ribbon tab, click **Copy Image**:



   A raster image of the contents of the *Modeler* window is copied to the Clipboard.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Open the application into which you want to paste the image of your model and follow that application's image pasting procedure.

To copy an image of a Report to paste into another application:

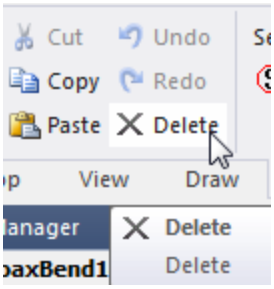
1. Make the report the active window and use the **Copy Image** command from the **Edit** menu, short-cut menu, or **View** ribbon tab.

   An image of the report is copied to the Clipboard.

2. Open the application into which you want to paste an image of the report and follow that application's image pasting procedure.

Deleting Objects

1. Select the objects to delete.
2. Delete the selected objects using *one* of the following three methods:
   - From the menu bar, click **Edit**> **Delete**.
   - From the **View, Draw,** or **Model** ribbon tab, click the **Delete** icon:

   - Press the **Delete** key.

   The objects are deleted.

> **Note:**
>
> Boundaries, excitations, hybrid regions, and mesh operations that were associated only with deleted objects are also deleted along with the objects. However, associated design variables remain, and they can be assigned to other geometry.

Deleting Start Points and Endpoints

For individual segments of polylines, you can delete the start point and/or the end point, with the following conditions:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For the *first* segment, you can only delete the *Start Point*, which effectively removes the entire segment.
- For the *last* segment, you can only delete the *End Point*, which effectively removes the entire segment.
- For all remaining (internal) segments, you can delete either the *Start Point* or the End Point. The new point becomes the opposite point of the adjacent segment, and the total number of segments is reduced by one.

To delete *Start Points* or *End Points* of polyline segments:

1. In the History Tree, locate the polyline that contains the segment you want to delete and expand the branch to expose the *CreateLine, CreateArc, CreateAngularArc,* or *CreateSpline* entries. These entries represent the individual polyline segments.
2. In the History Tree, right-click the polyline segment you want to edit and choose either:
   - **Delete Start Point**, which removes the preceding segment, or
   - **Delete End Point**, which removes the subsequent segment.

   Alternatively, you can select the segment and choose **Edit**> **Delete Start Point** or **Edit**> **Delete End Point** from the menu bar.

Delete Last Operation

You can delete the last geometry operation performed on an object. For certain situations, this functionality can offer an advantage when compared with the *Undo* command. For example, if you performed several valid operations (such as assigning boundaries, excitations, or mesh operations) after operating on an object's geometry, you could revert the last geometry change without undoing the subsequent assignments.

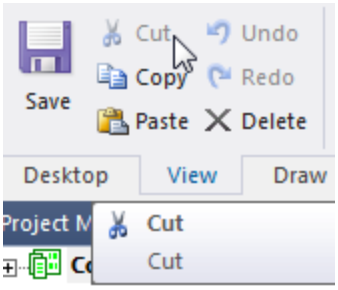
1. Select the object.
2. From the menu bar, click **Modeler**> **Delete Last Operation**.

   This command undoes the last geometry operation performed on the selected object, including removing that operation from the History Tree and updating the context for the *Undo* and *Redo* commands.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cutting Objects

1. Select the objects to cut.
2. From the menu bar, click **Edit**> **Cut** or, on any ribbon tab, click the **Cut** icon:



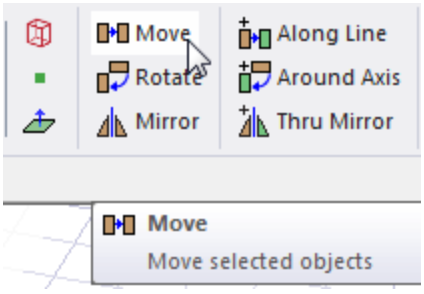
The objects are copied to the Clipboard and removed from the design. They can subsequently be pasted elsewhere, such as into a different design.

Moving Objects

The process for moving objects differs depending on the Drawing Data Entry Mode for the Drawing Options, whether Point mode (F3) or Dialog mode (F4).

For Point mode (click F3 to change to Point Mode):

1. Select the objects to move.
2. Click **Edit**> **Arrange**> **Move** or, on the **Draw** ribbon tab, click the **Move** icon:



3. Select an arbitrary anchor point in one of the following ways:
   - Click the point. (The *Measure Data* window shows the coordinates of the points you click.
   - Enter the point's coordinates in the **X, Y,** and **Z** boxes.
4. Select a target point in one of the following ways:
   - Click the point.
   - Type the coordinates of a point relative to the anchor point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

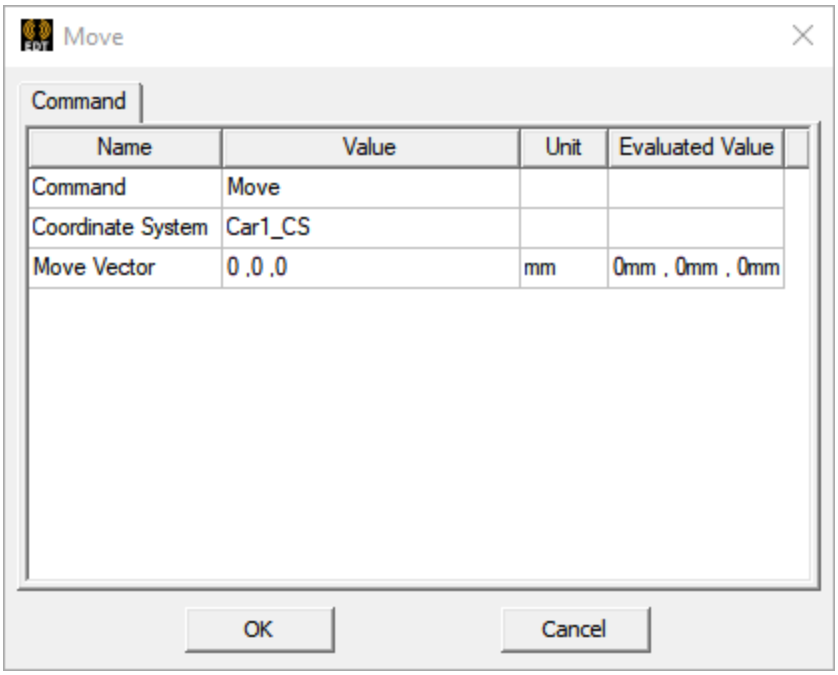
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All selected objects move the distance determined by the offset between the anchor point and the target point.

For Dialog mode (click F4 to change to Dialog mode):

1. Select the objects to move.
2. From the menu bar, click **Edit**> **Arrange**> **Move** or, on the **Drawing** ribbon tab, click the **Move** icon.

   A *Move* command dialog box appears.



3. Enter the coordinates of the location to which the object should move and click **OK**.

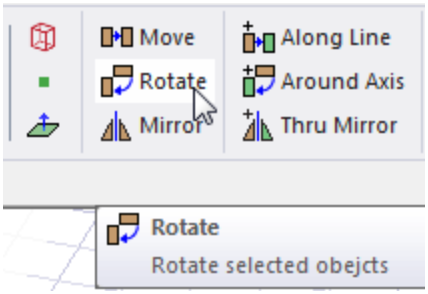
Rotating Objects

Rotate objects about the x-, y-, or z-axis using the **Edit> Arrange> Rotate** command.

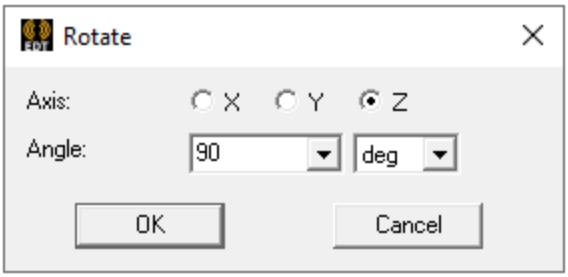
To rotate objects about an axis:

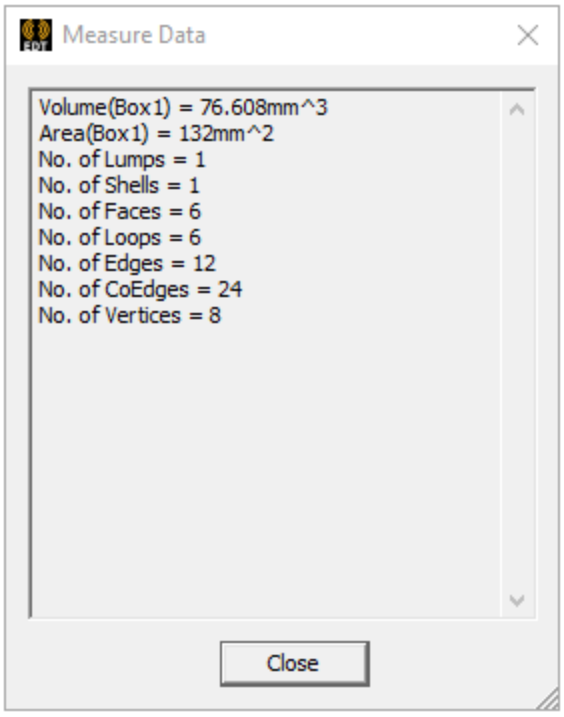
1. Select the objects to rotate.
2. From the menu bar, click **Edit**> **Arrange**> **Rotate** or, on the **Draw** ribbon tab, click the Rotate icon:

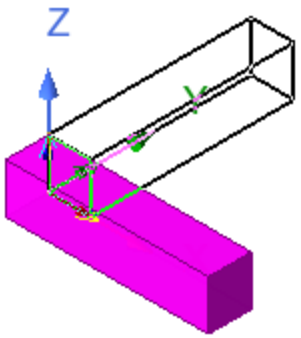
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Rotate** dialog box appears along with the **Measure Data** window. A visualization of the proposed change appears in the Modeler window.



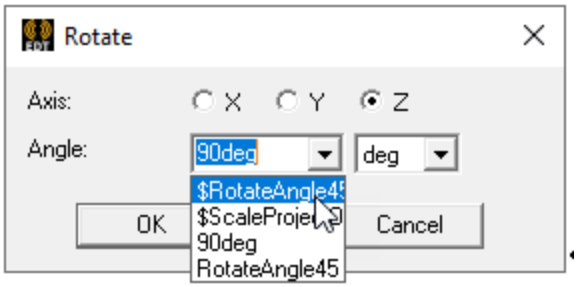




3. Select the axis about which to rotate the objects: **X, Y,** or **Z**.
4. Type the angle to rotate the objects in the **Angle** box. If you have previously specified values, or have created Project or Design Variables, this are listed in the drop-down menus.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A positive angle causes the object to be rotated in the counter-clockwise direction. A negative angle causes the object to be rotated in the clockwise direction.



5. Click **OK**.

   The selected objects are rotated about the axis.

To *rotate* and *copy* objects, use the **Edit> Duplicate> Around Axis** command.

Changing the Orientation of an Object

Each object has an **Orientation** property that specifies whether the coordinate system it uses is Global or a user-defined orientation relative to the Global coordinate system.

This property is useful in dealing with anisotropic materials. The properties of anisotropic materials are specified relative to each object's orientation. Changing the orientation of an object provides a way for objects made of the same material to have the material axes orientated differently.

To change an object's orientation:

1. Define the coordinate systems you want to have available.
2. Open the **Properties** dialog box for the object, or select the object and use the docked **Properties** window.
3. Click on the **Orientation** property and select the coordinate system from the drop-down menu. If no orientations other than **Global** have been defined, none appear on the list.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **OK** to close the dialog box and apply the changes.

> **Note:**
>
> The object will translate and rotate to make the geometry conform to the new coordinate system. If your intention was to change the material orientation, but the object had already been drawn in the correct physical orientation, you will have to redefine it's properties. The coordinates and dimensions used to define the object geometry must be based on the same coordinate system that is needed for proper material orientation. To avoid this problem, it is best to choose the correct coordinate system for material orientation before drawing the object.

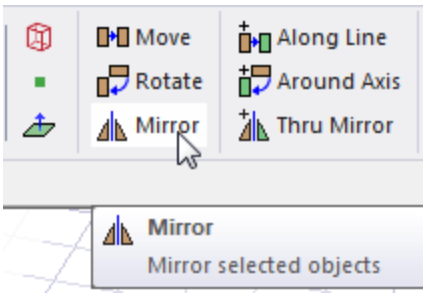
Mirroring Objects

Mirror an object about a plane using the **Edit** > **Arrange** > **Mirror** command. The plane is selected by defining a point on the plane and a normal point. This command allows you to move an object and change its orientation.

> **Note:**
>
> The distance between the point on the mirror plane and the point along the normal does not matter — only the vector direction matters.

To mirror an object about a plane:

1. Select the object or objects that you want to mirror. You can select multiple objects.
2. From the menu bar, click **Edit** > **Arrange** > **Mirror** or, on the **Draw** ribbon tab, click the **Mirror** icon:

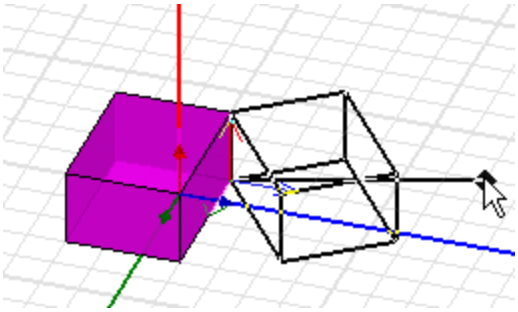


3. Select a point on the plane around which you want to mirror the object.

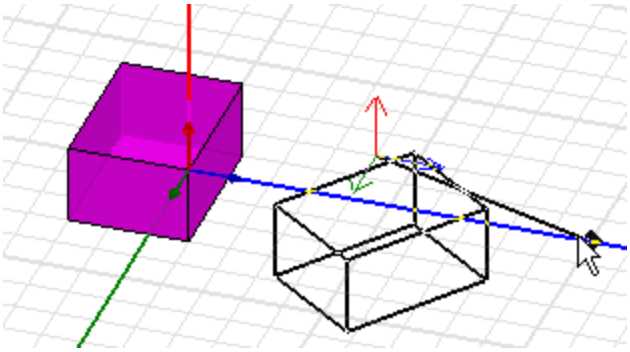
   You can do this by clicking a point, or typing coordinates in the X, Y, and Z boxes in the status bar.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you select a point on the object, the mirroring is relative to that point. In the following example, the first point clicked after selecting **Edit** > **Arrange** > ***Mirror*** was on the right-rear bottom corner of the selected object. So the axis of rotation as you move the cursor is that corner. As you move the cursor, it drags a diamond-shape on a vector extending from the initial point. The distance along the vector does not matter. Moving the mouse rotates an outline of the object to new orientations. Clicking the mouse moves the object to location indicated by the outline.



In this second example, the initial point is at a distance from the original object, designated by the triad from which the handle for rotation extends to the dragging cursor.



4. Select a second point in one of the following ways:
   - Click the point.
   - Type the coordinates of a point relative to the first point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point. As you type in the values, the outline moves to the coordinates. Press **Enter** to complete the command.

   The selected object is moved to the plane you specified and oriented relative to the normal point you specify.

To *mirror* and *copy* objects about a plane, use the **Edit** > **Duplicate** > **Mirror** command.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Offsetting Objects

Move every face of a 3D object in a direction normal to its surface using the **Edit** > **Arrange** > **Offset** command. The faces are moved a specified distance normal to their original planes. This command enables you to move every face of a solid object without having to individually select and move each face. Use the **Surfaces** > **Move Faces** > **Along Normal** command if you want to move just one or more faces of an object.

To offset every face of an object:

1. Select the object you want to offset.
2. From the menu bar, click **Edit** > **Arrange** > **Offset**.

   The *Offset* dialog box appears.

3. Type the distance you want to move the object faces from their origins, and then select a unit from the pull-down menu.
4. Click **OK**.

   The selected object's faces are moved the distance you specified.

Duplicating Objects

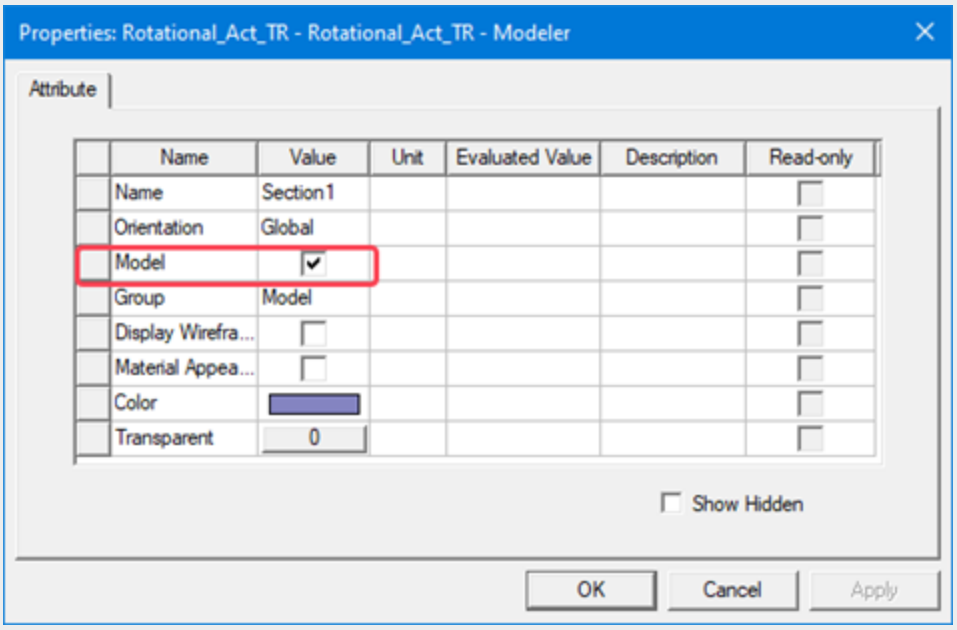
You can duplicate objects within a design using the **Edit** > **Duplicate** commands. Duplicates are dependent upon the parameters of their *parent* object at the time they were created, that is, they share the parent object's history at the time of creation. The command hierarchy in the history tree will show the duplication command, illustrating which commands affect all duplicates (those performed before the duplication) and which commands would not affect the duplicates (those performed after the duplication). For example, if you modify the radius of a parent object's hole, the change is applied to the holes of the object's duplicates because they share the radius specification history, but if you move the faces of the parent object, its duplicates are not affected because this operation took place after the duplicates were created.

Operations performed on duplicates are independent. For example, if you duplicate a cylinder twice, creating a row of three, and then split the second cylinder, the first and third cylinders are not affected by the split.

When creating duplicates, the parent object is duplicated along a line or around an axis the number of times you specify. You can also create a single duplicate that mirrors the parent object about a plane.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: When you duplicate an object, the Model\Non-Model status of the new object is not copied from the parent, but is determined by the **Modeler** > **New Object Type** menu selection. You can change the Model\Non-Model status of an existing object from its Properties window:



Choose from the following commands:

Edit > Duplicate > Along Line	Duplicates the parent object along a straight line. The child object can be designated as attached to the parent object, but if so, no ports or boundary conditions are duplicated.
Edit > Duplicate > Around Axis	Duplicates the parent object around an axis. The child object can be designated as attached to the parent object, but if so, no ports or boundary conditions are duplicated.
Edit > Duplicate > Mirror	Duplicates a mirror image of the parent object about a plane.

To copy objects to another design, use the **Edit** > **Copy** and **Edit** > **Paste** commands.

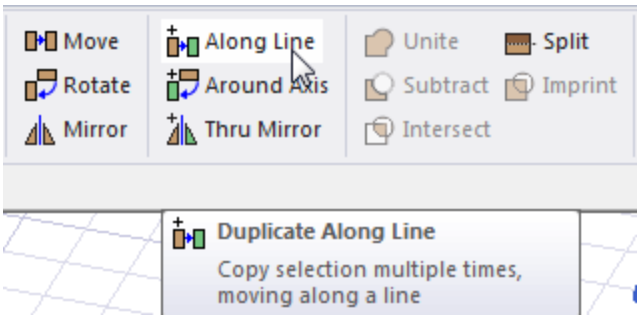
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> There is currently no method for dissolving the parent/duplicate relationship once a duplicate has been created.

Duplicating Objects Along a Line

To duplicate an object along a straight line, use the **Edit** > **Duplicate** > **Along Line** command. The line along which the object is duplicated can be vertical, horizontal, or lie at an angle.

1. Select the object you want to duplicate.
2. From the menu bar, click **Edit** > **Duplicate** > **Along Line** or, on the **Draw** ribbon tab, click the **Duplicate Along Line** icon:



3. Specify the vector along which the object will be duplicated. The *Measure Data* window appears to help you choose points:
   a. Select an arbitrary anchor point in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the in the **X, Y,** and **Z** boxes.

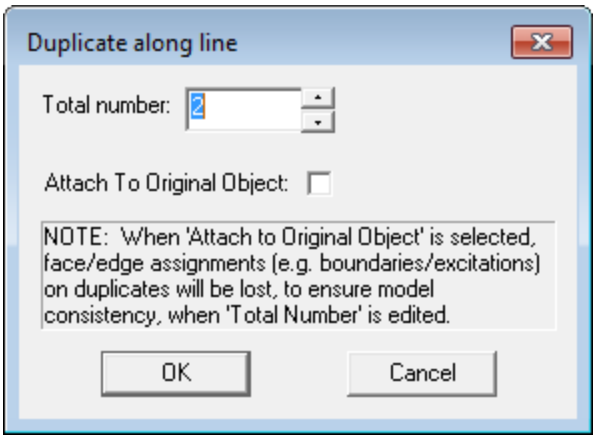
      Any point in the drawing region can be selected; however, selecting an anchor point on the object's edge or within the object makes it easier to select the duplication line.

   b. Select a second point in one of the following ways:
      - Click the point.
      - Type the coordinates of a point relative to the anchor point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

      This point defines the direction and distance from the anchor point to duplicate the object.

      The *Duplicate Along Line* dialog box appears:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



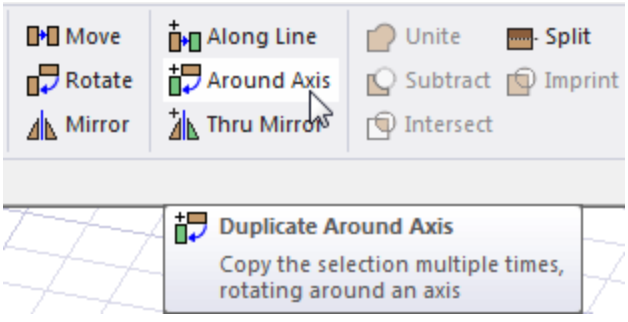
4. Type the total number of objects, including the original, in the **Total Number** box.
5. Optionally, check the **Attach to Original Object** check box. If this is checked, original object and its duplicates will be combined into one part. No ports or boundary conditions are duplicated for duplicates. If this option is used, it is recommended to run validation to ensure that the original and duplicates are not intersecting or touching each other.
6. If this option is used, it is recommended to run validation to ensure that the original and duplicates are not intersecting or touching each other.
7. Click **OK**.

   The duplicates are placed along the vector you specified.

Duplicating Objects Around an Axis

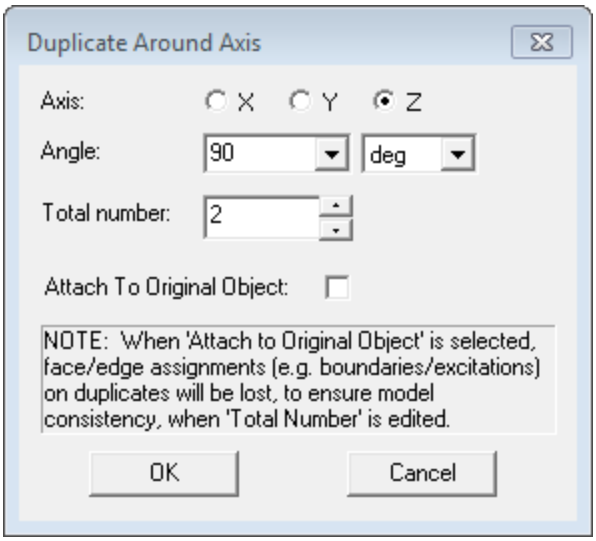
To duplicate an object around the x-, y-, or z-axis, use the **Edit** > **Duplicate** > **Around Axis** command.

1. Select the object you want to duplicate.
2. From the menu bar, click **Edit** > **Duplicate** > **Around Axis** or, on the **Draw** ribbon tab, click the **Duplicate Around Axis** icon:



The *Duplicate Around Axis* dialog box appears:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Select the axis around which you want to duplicate the object: **X, Y,** or **Z**.
4. Type the angle between duplicates in the **Angle** box.

   A positive angle causes the object to be pasted in the counter-clockwise direction.

   A negative angle causes the object to be pasted in the clockwise direction.
5. Type the total number of objects, including the original, in the **Total Number** box.
6. Optionally, check the **Attach to Original Object** check box. If this is checked, original object and its duplicates will be combined into one part. No ports or boundary conditions are duplicated for duplicates. If this option is used, it is recommended to run validation to ensure that the original and duplicates are not intersecting or touching each other.
7. Click **OK**.

   The object is duplicated around the axis at the angle you specified.

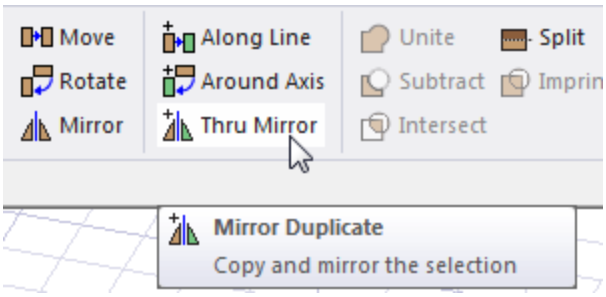
Duplicating and Mirroring Objects

To duplicate and mirror an object about a plane, use the **Edit** > **Duplicate** > **Mirror** command. The plane is selected by defining a point on the plane and a normal point. This command allows you to duplicate an object and specify the duplicate's position.

This command is similar to **Edit > Arrange > Mirror**, except that this command duplicates an object, rather than moves it.

1. Select the object you want to mirror.
2. From the menu bar, click **Edit** > **Duplicate** > **Mirror** or, on the **Draw** ribbon tab, click the **Thru Mirror** (Mirror Duplicate) icon:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Select a point on the plane on which you want to mirror the object.

   A line drawn from this point to the mirror plane will be perpendicular to the plane. The distance between the point on mirror plane and point along the normal does not matter; only the vector direction matters

4. Select a normal point on the plane in one of the following ways:
   - Click the point.
   - Type the coordinates of a point relative to the first point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

   A duplicate of the object appears on the plane you specified, oriented according to the normal point you specified.

Scaling Objects

Scale an object's dimensions in one or more directions using the **Edit** > **Scale** command.

The scale of an object is determined by the distance of each of its vertices from the origin of the model coordinate system. When an object is scaled, the distance of each vertex from the origin is multiplied by the scaling factor, causing the object to be resized and/or moved.

For example, if you specify a scaling factor of 2 in the X direction, each vertex in the model will be moved so that the distance to its origin is doubled. Note that a vertex located at the origin will not move. You can alter an object's proportions by scaling it in one direction.

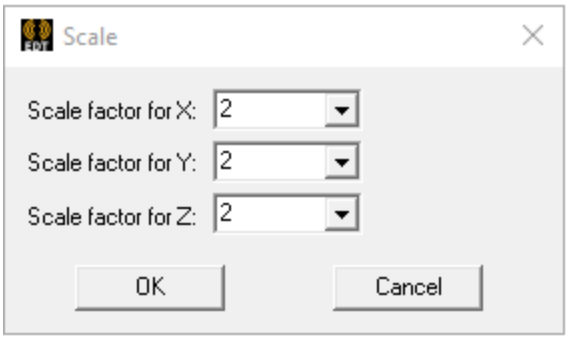
To scale an object's dimensions in one or more directions:

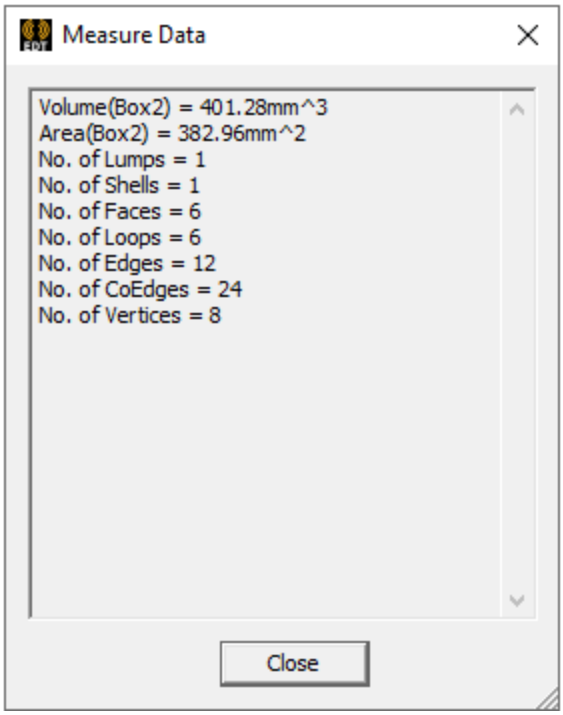
1. If necessary, set a different working coordinate system to achieve the desired scaling.
2. Select the object to scale.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. From the menu bar, click **Edit** > **Scale**.

   The **Scale** dialog box appears with the **Measure Data** window.

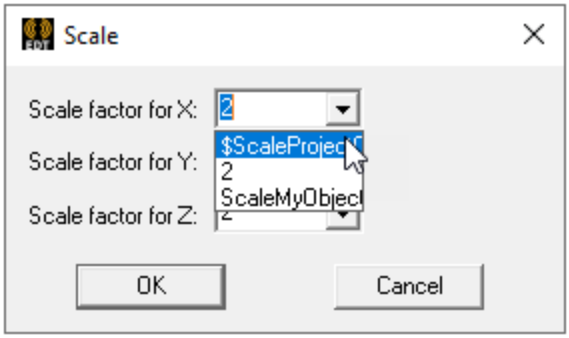




4. Type the **Scale factor** for each axis. If you have created Project or Design Variables, or have previously used a relevant value, these may be listed for selection in the drop-down

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

menu.



5. Click **OK**.

   The object is scaled about the working coordinate system's origin.

Sweeping Objects

You can sweep a 2D object around an axis, along a vector, or along a path to create a 3D solid object. Objects that can be swept include circles, arcs, rectangles, trapezoids, polylines, or any 2D object created in the **3D Modeler** window. The 2D object need not be orthogonal to the sweep path.

You can also:

- Thicken sheets to make a 3D object.
- Sweep open 1D objects, such as polylines, which results in open 2D sheet objects.
- Sweep one or more faces of a 3D object to create a new 3D object. (See Sweep Faces Along Normal).

Sweeping Around an Axis

Sweep a 1D or 2D object around the X, Y, or Z axis using the **Draw** > **Sweep** > **Around Axis** command. Sweeping circles around an axis is a convenient way to create an open coil loop.

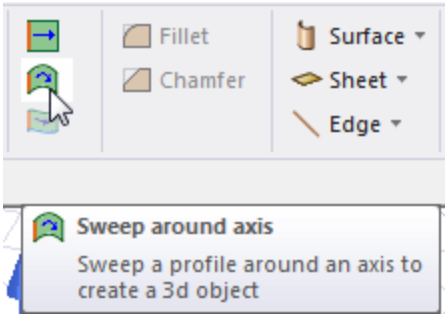
Before using this command, keep the following guidelines in mind:

- The object and the axis you are sweeping around must lie in the same plane. For example, if you are sweeping an object around the Z-axis, the object must lie in a plane that includes the Z-axis, such as XZ or YZ.
- The normal of the object's plane faces must be perpendicular to the axis around which you are sweeping.
- The object may not cross the axis around which it is being swept.

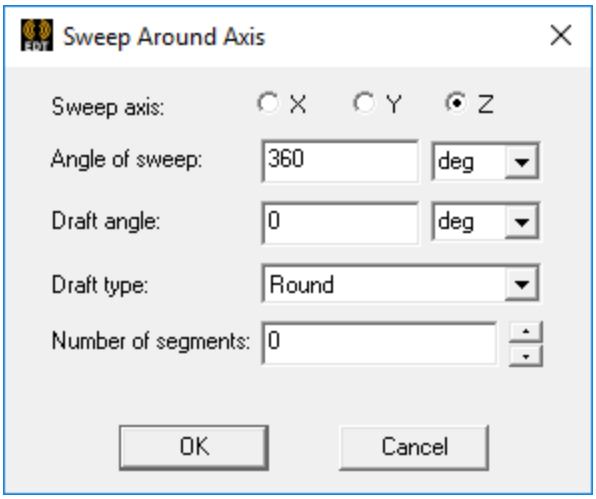
To sweep an object around an axis:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the object you want to sweep.
2. From the menu bar, click **Draw** > **Sweep** > **Around Axis** or, in the **Draw** ribbon tab, click **Sweep around axis**:



The *Sweep Around Axis* dialog box appears.



3. Select the **Sweep axis** (that is, the axis you want to sweep the object around: **X, Y,** or **Z**).
4. Type an angle into the **Angle of sweep** text box and select the desired unit of measure.

   This is the angle about the specified sweep axis through which you want the object to be twisted or rotated as it is swept. The value must be between **-360** and **360** degrees.

5. Type an angle into the **Draft angle** text box and select the desired unit of measure.

   This is the angle to which you want a 2D object's profile, or shape, expanded (positive angle) or contracted (negative angle) as it is swept. Among other purposes, drafts make it easy to remove molded or cast parts from their molds.

   For 1D objects, the draft angle progressively alters the sweep radius but does not alter the length of the object as it is swept. For example, a 10 mm straight line swept around a parallel axis with a nonzero draft angle would produce a spiral surface with a fixed width of 10 mm. When the sweep angle is *positive*, a positive draft angle produces a spiral surface with a *decreasing* radius (that is, the distance from the generated surface to the sweep

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

axis decrease along the sweep angle. A negative angle produces a spiral surface with an *increasing* radius along the sweep angle. When the sweep angle is *negative*, the spiral behavior is the opposite of that just described.

6. Select one of the following **Draft type** options from the drop-down menu. The draft type instructs the modeler whether to maintain the original object shape and how to transition between adjacent faces. This setting affects the results when sweeping 2D objects into 3D solids or when sweeping lines and curves into 3D sheets.
   - **Extended**
   - **Round**
   - **Natural**

   For a complete description of the behavior of each of these three options, see the Draft Types page.

7. Type the number of segments in the **Number of segments** text box and click **OK**.

   > **Note:**
   >
   > The default number of segments is zero, which creates a true path. A positive value results in a segmented sweep, while a negative value results in an error.
   >
   > If the sweep angle is 360 degrees, the number of segments is equal to the value specified. If the sweep angle is less than 360 degrees, half segments appear at the ends.
   >
   > Projects and scripts from previous software versions are treated as if the number of segments were zero.

   The object is swept around the axis. The new object has the properties of the original object. The *Properties* dialog box appears, enabling you to modify the object's properties.

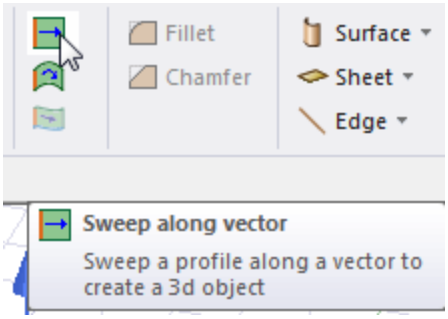
8. Click **OK**.

Sweeping Along a Vector

Sweep a 1D or 2D object along a vector using the **Draw** > **Sweep** > **Along Vector** command.

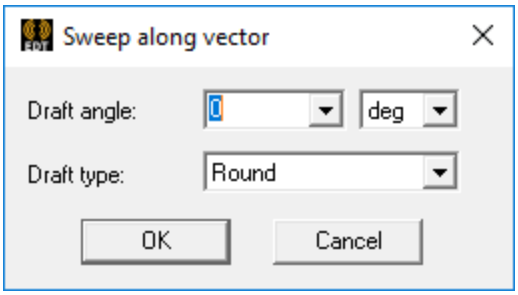
1. Select the object you want to sweep.
2. Click **Draw** > **Sweep** > **Along Vector** or, in the **Draw** ribbon tab, click **Sweep along vector**:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Draw the vector you want to sweep the object along:
   a. Select the start point by clicking the point or typing its coordinates in the **X, Y,** and **Z** boxes.
   b. Select the endpoint in one of the following ways:
      - Click the point.
      - Type the coordinates of a point relative to the start point in the **dX, dY,** and **dZ** boxes, where **d** is the relative distance from the previously selected point.

   The *Sweep along vector* dialog box appears.



4. Type an angle into the **Draft angle** text box and select the desired unit of measure.

   This is the angle to which you want a 2D object's profile, or shape, expanded (positive angle) or contracted (negative angle) as it is swept. Among other purposes, drafts make it easy to remove molded or cast parts from their molds.

   For 1D objects, the draft angle alters the sweep direction but does not alter the length of the object as it is swept. For example, a 10 mm straight line swept with a 5-degree draft angle does not produce a trapezoidal 2D object. It produces a rectangle with a constant 10 mm width but tilted 5 degrees relative to the specified sweep vector.

5. Select one of the following **Draft type** options from the drop-down menu. The draft type instructs the modeler whether to maintain the original object shape and how to transition between adjacent faces. This setting affects the results when sweeping 2D objects into 3D solids or when sweeping lines and curves into 3D sheets.
   - **Extended**
   - **Round**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Natural**

For a complete description of the behavior of each of these three options, see the Draft Types page.

6. Click **OK**.

   The object is swept along the vector. The new object has the name and color of the original profile. The *Properties* dialog box appears, enabling you to modify the object's properties.

Sweeping Along a Path

Sweep a 1D or 2D object along a path that is defined by an open or closed polyline using the **Draw** > **Sweep** > **Along Path** command.

Best Practices

Adhere to the following best practices for good sweep outcomes using the Parasolid geometry kernel:

- When sweeping along a path that is not closed (i.e., the start and end point are not the same), ensure that the start point of the path is on the profile being swept.
- When sweeping a profile around a closed path, ensure that the path intersects the profile.
- Parasolid sweep operator preserves the angle between the profile and path. Ensure that the profile is perpendicular to the path if it is supposed to be perpendicular.
- If the sweep profile is rotationally symmetric (e.g., circle, rectangle, square), position the profile so that the path intersects the profile at the center of the profile. If the profile does not have a center, allow the path to go through or start at the vertex of a profile.

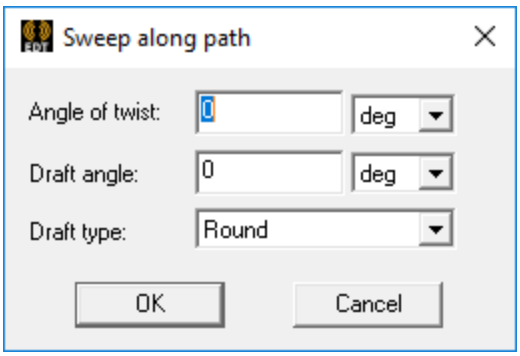
> **Note:**
>
> If the path curve is tangential to the sweep profile, then the sweep will fail.

To sweep an object along a path:

1. Create the polyline to be used as a path.
2. Select the object you want to sweep and then select the new polyline.
3. Click **Draw** > **Sweep** > **Along Path** or, from the **Draw** ribbon tab, click the **Sweep along path** icon:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Sweep along path** dialog box appears.



4. Type an angle in the **Angle of twist** text box and choose the desired unit of measure.

   This is the angle you want the profile to rotate about the sweep path vector as it is swept through the complete path.

5. Type an angle into the **Draft angle** text box and select the desired unit of measure.

   This is the angle to which you want the object's profile, or shape, expanded (positive angle) or contracted (negative angle) as it is swept. Among other purposes, drafts make it easy to remove molded or cast parts from their molds.

   For 1D objects, the draft angle alters the sweep direction but does not alter the length of the object as it is swept. For example, a 10 mm straight line swept with a 5-degree draft angle does not produce a trapezoidal 2D object. It produces a rectangle with a constant 10 mm width but tilted 5 degrees relative to the specified sweep path.

6. Select one of the following **Draft type** options from the drop-down menu. The draft type instructs the modeler whether to maintain the original object shape and how to transition between adjacent faces. This setting affects the results when sweeping 2D objects into 3D solids or when sweeping lines and curves into 3D sheets.
   - **Extended**
   - **Round**
   - **Natural**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For a complete description of the behavior of each of these three options, see: Draft Types.

7. Click **OK**.

   The object is swept along the path. The polyline object used as the path is deleted. The new object has the properties of the original object. The **Properties** dialog box appears, enabling you to modify the object's properties.

Sweeping Faces Along Normal

To create a new object by sweeping select 3D object's face a specified distance in a direction normal to its original plane, use the **Modeler** > **Surface** > **Sweep Faces Along Normal** command. Note that the adjoining faces will not be sheared or bent.

This command is useful for extruding faces, resizing holes, and removing rounded corners.

To sweep selected object faces in a normal direction:

1. Right-click in the Modeler window and choose **Select Faces** from the shortcut menu.

   Alternatively, with the Modeler window active, press **F** to switch to the *Face* selection mode.

2. Select the faces of the object you want to sweep.
3. From the menu bar, click **Modeler** > **Surface** > **Sweep Faces Along Normal**.

   The *Sweep Faces Along Normal* dialog box appears.

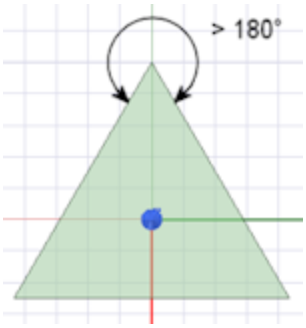
4. Type the distance you want to sweep the object face from its origin.
5. Click **OK**.

   The face is swept the distance you specified to create a new 3D object.

Draft Type Options

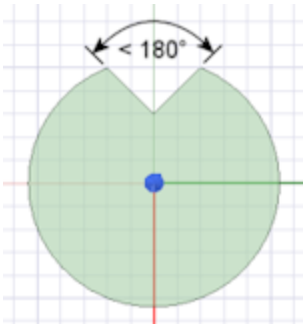
In many cases, the **Draft type** setting may have little to no effect on the resultant geometry of a swept object unless a *positive, non-zero draft angle* is specified. Consider a *sharp corner* feature. For the purpose of this discussion let's define a sharp corner as any protruding feature where the *external* angle between the adjacent faces is greater than 180-degrees:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When this type of feature is expanding, the corner can gradually become rounded. However, if the same corner is contracting, the radius is already zero at the start of the sweep and cannot contract or decrease further. Therefore, no rounding occurs in the latter case.

Now, consider a *groove* in the perimeter of a 2D object to be swept. Let's define a groove as any indented feature, where the *external* angle between the adjacent faces is less than 180-degrees:



A groove will grow in size as the object is swept with a *negative* draft angle specified. In this case, progressive rounding of the corners is possible.

You may need to experiment with the three draft type options to determine what the effect will be for nontrivial object shapes. The behavior of each option can differ according to the shape of the 2D object, the draft angle, and whether it is swept along a vector, a path, or about an axis.

The following three **Draft type** options are available:

- **Natural:** This draft type generally maintains the shape of the original object as it is swept along the specified direction. That is, all sharp corners remain sharp, producing sharp edges in the sweep direction. There are exceptions to this general behavior. Rounding of certain corners may occur if required to accommodate sides with complex shapes (for example, a complex side might be split into a combination of planar, cylindrical, helical, or conic-section faces).
- **Extended:** This option maintains the shape of the original object as it is swept along the specified direction. Sharp corners of the original object remain sharp corners along the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

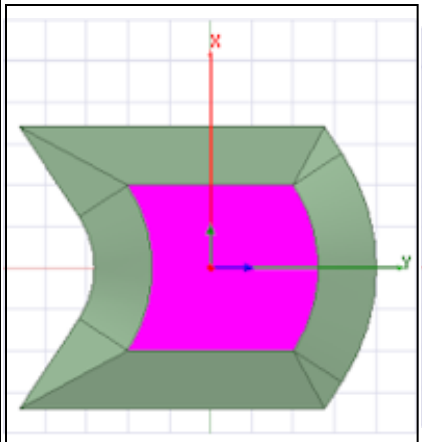
entire swept direction. However, the side faces may be split into three separate tangential faces, as needed and as described in the preceding bullet.

- **Round:** This option rounds all sharp corners of the original object, when possible, as it is swept along the specified direction.

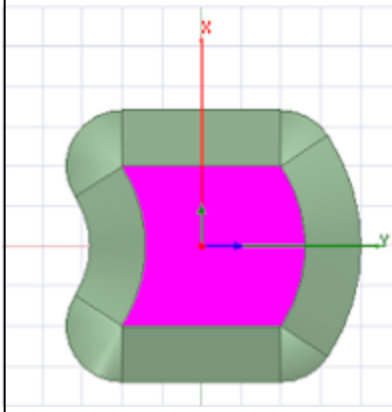
Examples

The images below compare the results of sweeping an object using each of the available **Draft type** options. The first six images are of an object swept with a positive draft angle defined. The final three images are of a different object swept with a negative draft angle defined. For all examples, the original 2D object is highlighted in magenta.

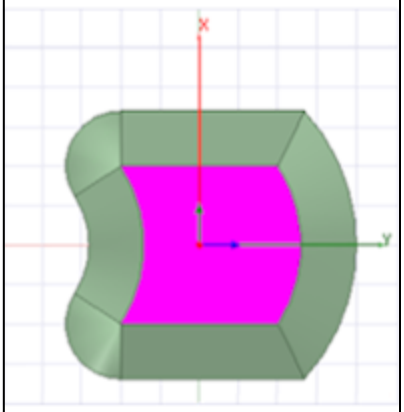
Sweep Along Vector (Positive Draft Angle):



1a: Extended

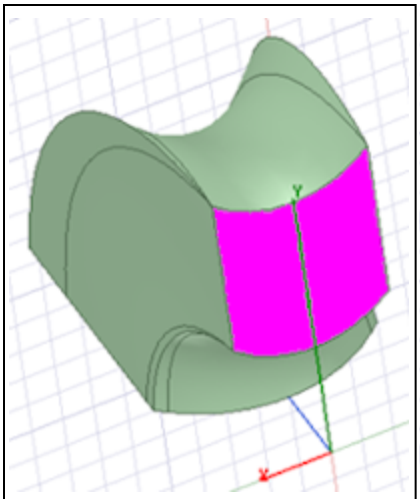


1b: Round

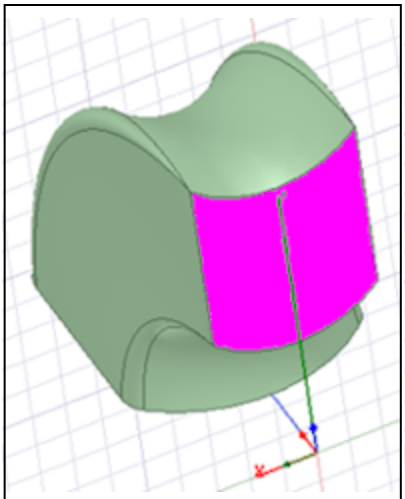


1c: Natural

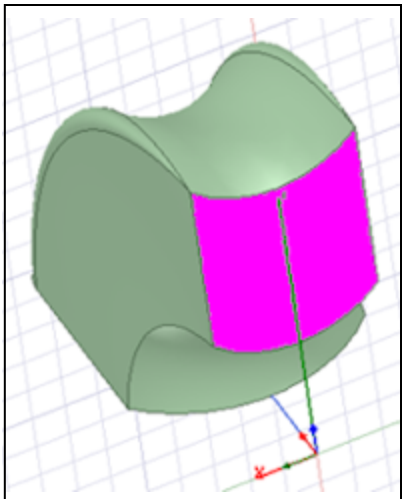
2. **Sweep About Axis** (Positive Draft Angle):



2a: Extended



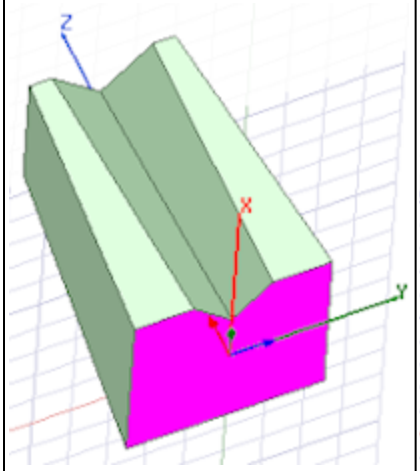
2b: Round



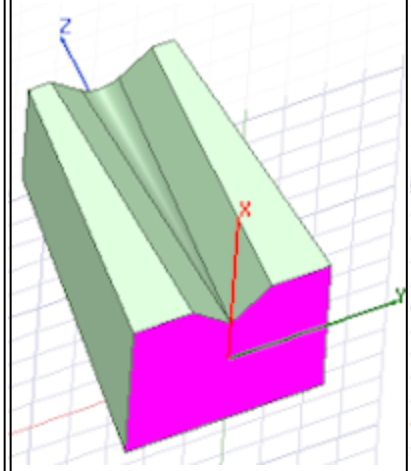
2c: Natural

3. **Sweep Along Vector** (Negative Draft Angle):

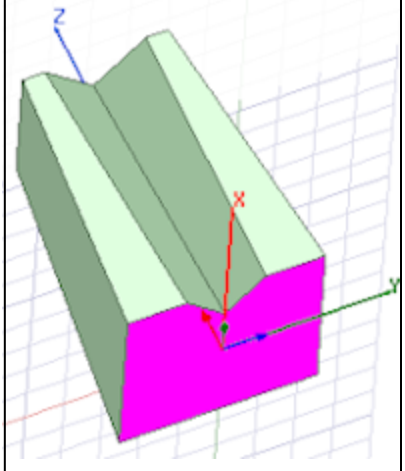
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3a: Extended



3b: Round



3c: Natural

Observations:

- **Extended Option:** Notice how two of the four sides of example **1a** have been divided into three faces each (the sides generated by sweeping curved edges). However, the other two sides (those generated from straight edges) have not been divided.

  In example **2a**, all four sides have been divided into three faces each. Even the straight sides generate a complex surface due to the combination of a positive draft angle and a circular sweep path.

  In example **3a** (negative draft angle), no face is split and no edge is rounded.

- **Round Option:** In both examples **1b** and **2b**, all corners of the original 2D object have been progressively rounded (with the radius increasing from zero at the plane of the original object to a maximum radius at the end of the sweep).

  In example **3b** (negative draft angle), only the corner of the groove has been rounded. The indented feature is the only one increasing is size along the sweep direction.

- **Natural Option:** In both examples **1c** and **2c**, the corners adjacent to the convex curved face remain sharp. However, the corners adjacent to the concave curved face are progressively rounded in both cases (with the radius increasing from zero at the plane of the original object to a maximum radius at the end of the sweep).

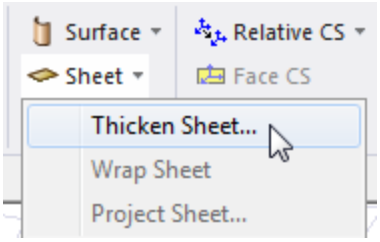
  In example **3c** (negative draft angle), no face is split, and no edge is rounded.

Thicken Sheet

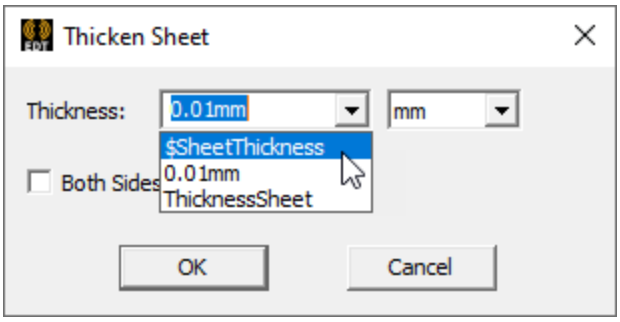
To thicken one or more sheet objects to make 3D objects:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the sheet or sheets you want to thicken into 3D objects.
2. From the menu bar, click **Modeler** > **Surface** > **Thicken Sheet** or, on the **Draw** ribbon tab, choose **Sheet** > **Thicken sheet**:



The *Thicken Sheet* dialog box appears:



3. Specify the **Thickness**. If you have set Project or Design variables with relevant values, or have previously assigned values, this are listed in the dropdown.
4. If necessary, specify the units by selecting from the drop-down menu.
5. If you want to thicken symmetrically (half of the thickness on each side of the sheet or sheets), select the **Both Sides** check box.

   The resulting solid is the requested thickness, but the original sheet location corresponds to the middle of the resulting solid's thickness.

6. Click **OK**.

   The dialog closes and the sheets are changed into 3D objects of the desired thickness.

Wrap Sheet Command

You can use **Modeler** > **Surface** > **Wrap Sheet** command to wrap a sheet object around a suitable 3D object. The sheet object does not have to be in contact with the 3D object. It should have smaller dimensions than the 3D object.

When the sheet to be wrapped and the target body do not touch, the wrap sheet command chooses among several different methods of bringing them into contact.

- If an edge from the sheet body and the target body are close enough and at a small angle (10 degrees), the edges are brought into alignment before the wrap.

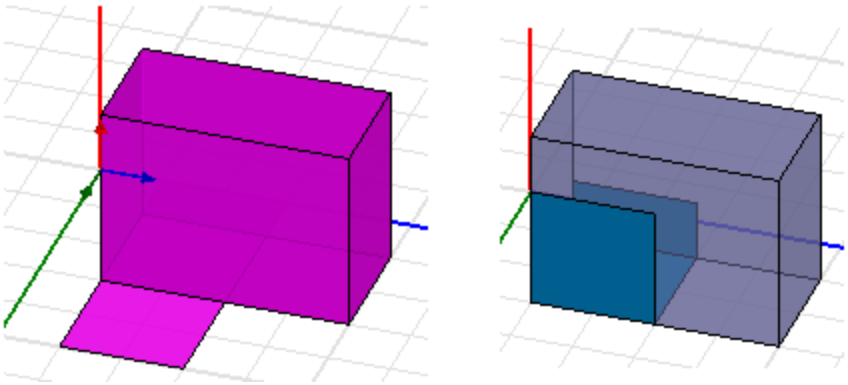
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If a vertex from the sheet body and an edge on the target body are close enough, the vertex is moved to the target edge.
- If the previous criterion are not matched, a vertex from the sheet body is moved to the closest face of the target body.

If the sheet to be wrapped and the target face are both planar and have the same or opposite normal, the closest vertex from the sheet body is moved to the target face. No moving to match or align edges is performed.

If the sheet body has multiple lumps, a single rectangle is fitted around all of the lumps. The rectangle is moved to the target surface using the above procedure.

If the sheet object does not overlap the corners of the 3D object, the wrap is straightforward, as shown in the following figure:



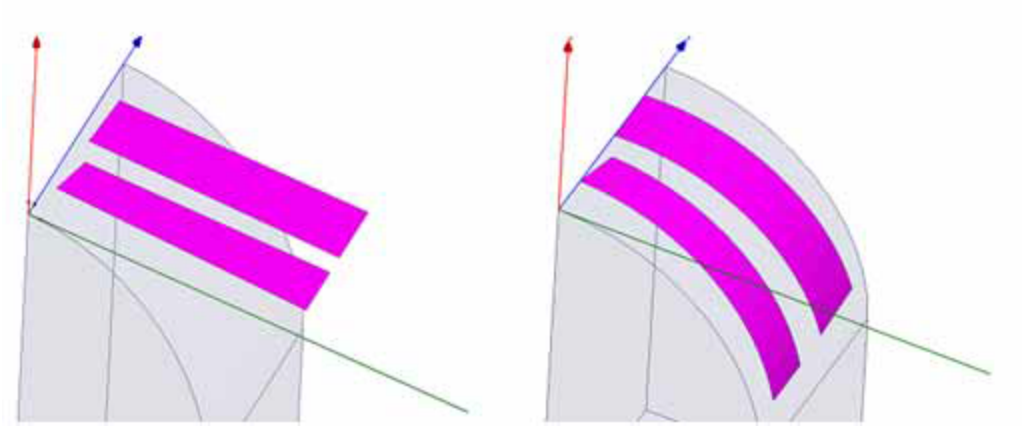
If you delete the 3D object, the wrapped sheet retains the form it took when wrapped.

A sheet object that overlaps corners may not wrap in straightforward fashion, depending on both the angle(s) involved, and the sheet object. While it is possible, it is not recommended.

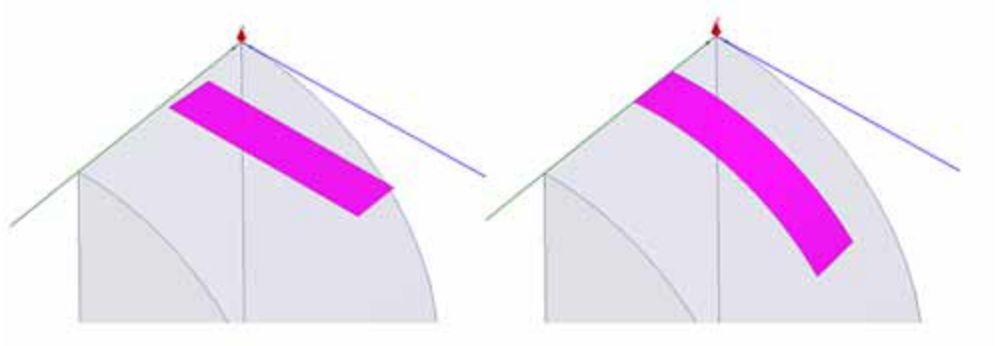
Examples:

The following figure illustrates wrapping on a target surface with the snap to an edge. The surface will snap to an edge if the distance is close relative to the length of the edge. If the angle between the edge is less than 10 degrees, the edge to be wrapped will be aligned to the target body. In the example on the left, the lower sheet has an edge angle more than 10 degrees. The vertex will be snapped to the edge. The top sheet in the picture has an edge angle of less than 10 degrees. The vertex is snapped to the target edge and the sheet edge is aligned to the target. To avoid snapping to an edge or edge alignment, the wrap sheet can be put into contact with the target surface. In that case the sheet vertex will not be moved to the target edge unless it is within modeling tolerances.

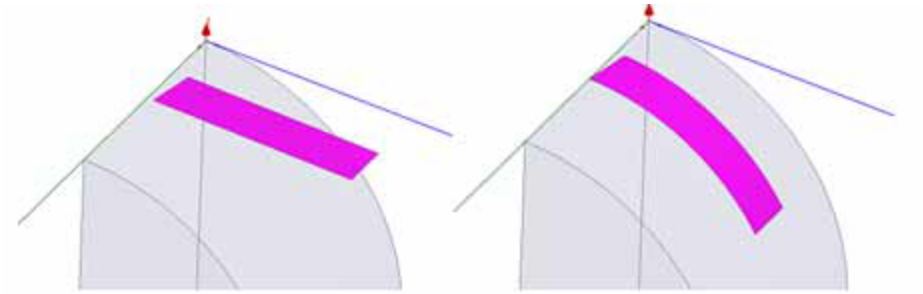
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following figure shows the situation when you move the edge to the edge before wrapping.

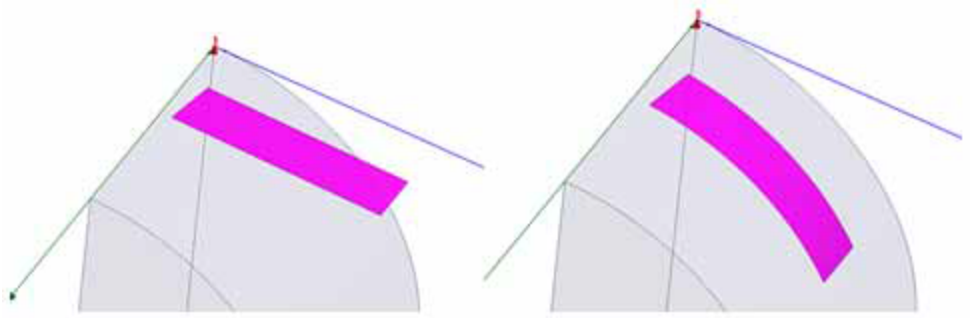


The following figure shows the situation when you move a vertex to an edge before wrapping.

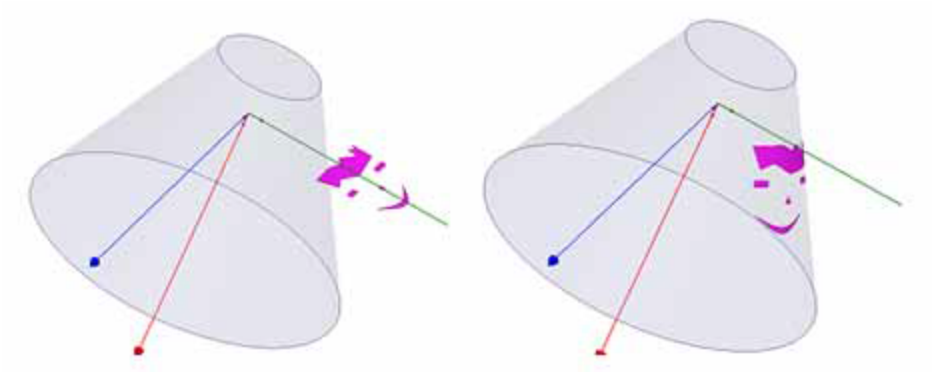


The following figure shows the situation where you move a vertex to a face before wrapping.

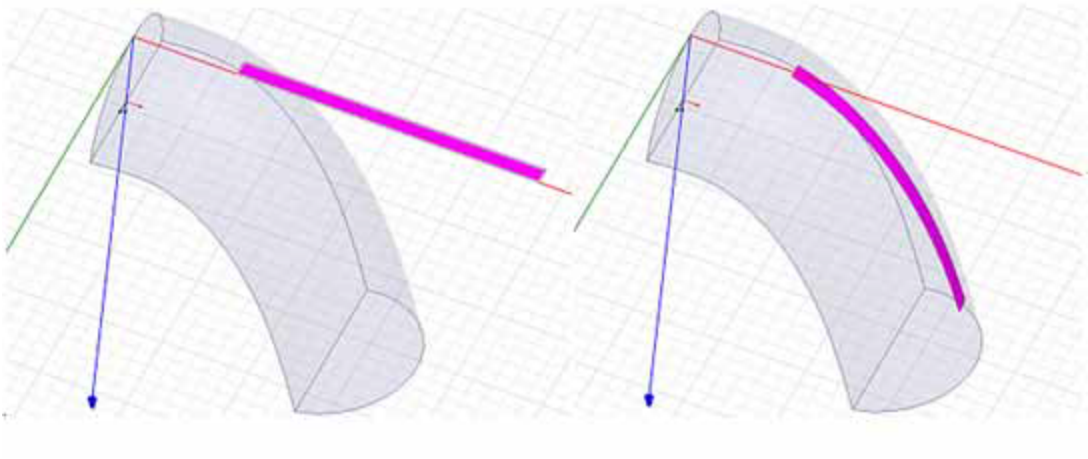
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following figure shows the situation when you move a multilump sheet to touch the target body. A rectangle that encloses all lumps is used to position the sheet on the target.

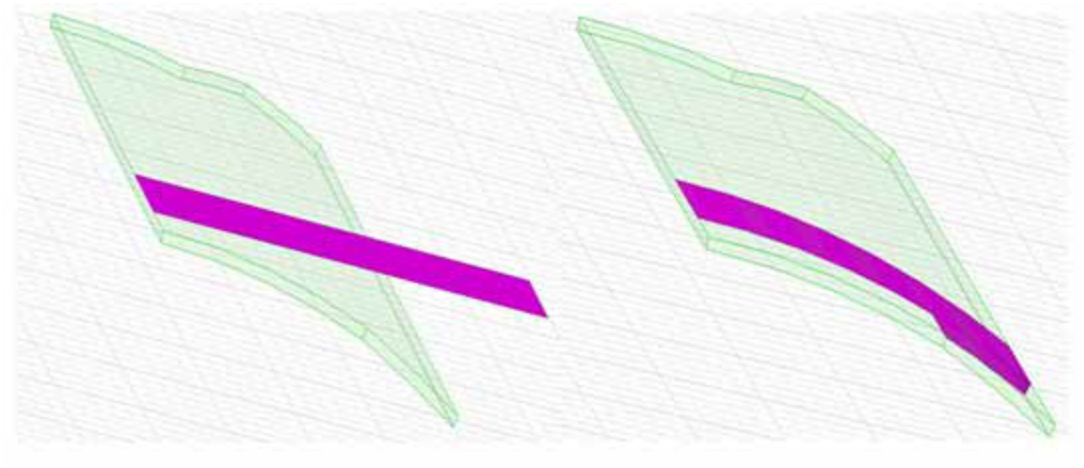


The next figure illustrates wrapping to a non-developable surface. The area of the sheet changed from 0.7000 mm^2 before wrapping to 0.6986 mm^2 after wrapping.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following figure illustrates wrapping on a target surface which has tolerant edges. The tolerant edge is at the beginning of the wrap. The tolerant edge is not evident visually.



Limitations/Error Conditions:

- Wrapping on non-developable surfaces creates distortion in lengths.
- Wrapping is not allowed across curved boundaries between faces.
- Working with tolerant edges is successful only if the gaps can be tightened sufficiently for the operation.
- Wrapping on the pole of a surface (such as sphere or cone) is not supported. A warning is given:

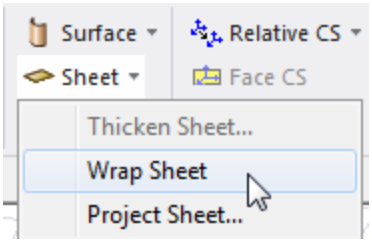
  WrapSheet: Wrapping sheet contacted pole of target surface. Recommend moving sheet and splitting out pole from target surface.

- The choice of method for how the sheet body is moved to touch the target face is not directly selectable by the user. If the sheet to be wrapped and target body touch, then the sheet body is only rotated at point of contact.

To wrap a sheet object:

1. Create a sheet close to an appropriate 3D object.
2. Select both objects.
3. From the menu bar, click **Modeler** > **Surface** > **Wrap Sheet** or, on the **Draw** ribbon tab, select **Sheet** > **Rap Sheet**:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The sheet object wraps around the 3D object. You can select the wrapped sheet object and the 3D object separately, and assign properties separately.

If the object cannot wrap, the Message window contains a warning and description.

You can wrap multiple sheets on the same 3D object.

If you delete the 3D object, the wrapped sheet retains the form it took when wrapped.

You can use the **Tools** > **Options** > **General Options**, 3D Modeler options to automatically perform a **Modeler** > **Boolean** > **Imprint** command after performing a **Wrap.**

Covering Lines

To cover a closed 1D polyline object with a face, use the **Modeler** > **Surface** > **Cover Lines** command. The polyline object becomes a 2D sheet object.

To convert a closed polyline object to a sheet object:

1. Select the closed polyline object you want to cover.
2. From the menu bar, click **Modeler** > **Surface** > **Cover Lines**.

   The object is now covered. It is now a 2D sheet object that can be swept to form a 3D solid object.

> **Note:**
>
> If you want the modeler to automatically cover all closed polyline objects you draw, including circles, ellipses, rectangles, and regular polygons, select the **Automatically cover closed polylines** option in the *3D Modeler > Drawing* section of the *General Options*. A closed polyline object can also be created by using boolean unite operations on two or more polylines.

Covering Faces

To cover object faces, the faces must be united into a 3D sheet object. To cover the face of a 2D or 3D object, use the **Modeler** > **Surface** > **Cover Faces** command.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Covering the face of an open 2D sheet object that had previously been uncovered results in a 3D solid object. For example, for a box, when you select and uncover a face, the solid box becomes a sheet with five faces. When you then select that sheet body box and use the **Cover Faces** command, the box becomes a solid again with six faces.

To cover the faces of objects:

1. Select the faces of the objects you want to cover.
2. From the menu bar, click **Modeler** > **Surface** > **Cover Faces**.

   The object faces are now covered.

Uncovering Faces

Uncover a surface of a 3D object using the **Modeler** > **Surface** > **Uncover Faces** command. Uncovering the surface of a 3D solid object results in an open 2D sheet object.

To uncover the face of a 3D object:

1. Switch to the *Face* selection mode: Click **Edit** > **Selection Mode** > **Faces**.
2. Select a face of the object you want to uncover.

   From the menu bar, click **Modeler** > **Surface** > **Uncover Faces**. The selected face is uncovered, leaving an open face on the object.

> **Note:**
>
> You can uncover one face of a 3D object at a time. If you select multiple faces, only the first face will be uncovered.

Detaching Faces

The **Modeler** > **Surface** > **Detach Faces** command enables you to remove the face of a 3D object, resulting in two separate objects.

To detach the face of an object:

1. Switch to *Face* selection mode: Click **Edit** > **Selection Mode** > **Faces**.
2. Select the face of the object you want to detach. You can select multiple faces to detach.

   Click **Modeler** > **Surface** > **Detach Faces**. The selected face is now detached, resulting in two 2D sheet objects.

Detaching Edges

The **Modeler** > **Edge** > **Detach Edges** command enables you to remove an edge of a wire object, resulting in two separate wire objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To detach an edge of an object:

1. Switch to *Edge* selection mode: Click **Edit** > **Selection Mode** > **Edges**.
2. Select the edge of the object you want to detach. You can select multiple edges to detach.
3. Click **Modeler** > **Edge** > **Detach Edges**.

   The selected edge is now detached, resulting in multiple wire objects.

> **Note:**
>
> Only edges from wire bodies can be used in a detach edge operation.

Creating a Cross-Section

You can take a cross section of a 3D object to create a new 2D object. This is done using the **Modeler** > **Surface** > **Section** command.

Use this command to create cross sections of 3D objects on the XY, YZ, or XZ plane. The cross sections are created as 2D closed polyline objects.

To create a cross section of an object:

1. Make sure the working coordinate system you want to use for the cross section plane is set.
2. Select the object from which you want to create a cross section.
3. From the menu bar, click **Modeler** > **Surface** > **Section**.

   The *Section* dialog box appears.

4. Select the **Section Plane** you will use to divide the object: **XY, YZ,** or **XZ**.
5. Click **OK**.

   A closed polyline object is created at the intersection of the 3D object and the specified section plane. The original 3D object is unmodified.

Connecting Objects

Use the **Modeler** > **Surface** > **Connect** command to perform the following operations:

- Connect two or more 1D polyline objects. Ansys Electronics Desktop will modify the first polyline you select to be a 2D sheet object that connects to the second and any subsequently selected polylines. The second and subsequent polylines selected are deleted.
- Connect two or more 2D sheet objects. Ansys Electronics Desktop will modify the first 2D object you select to be a 3D solid object that connects to the second and any subsequently selected objects. The second and subsequent objects selected are deleted.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

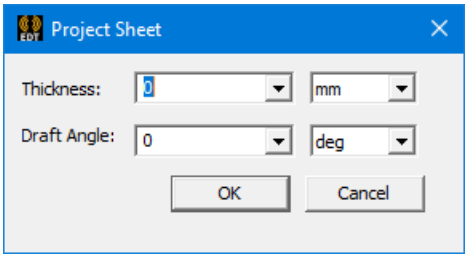
To connect objects:

1. Select the objects you want to connect. Select 3D objects as objects, not as faces.
2. Click **Modeler** > **Surface** > **Connect**.

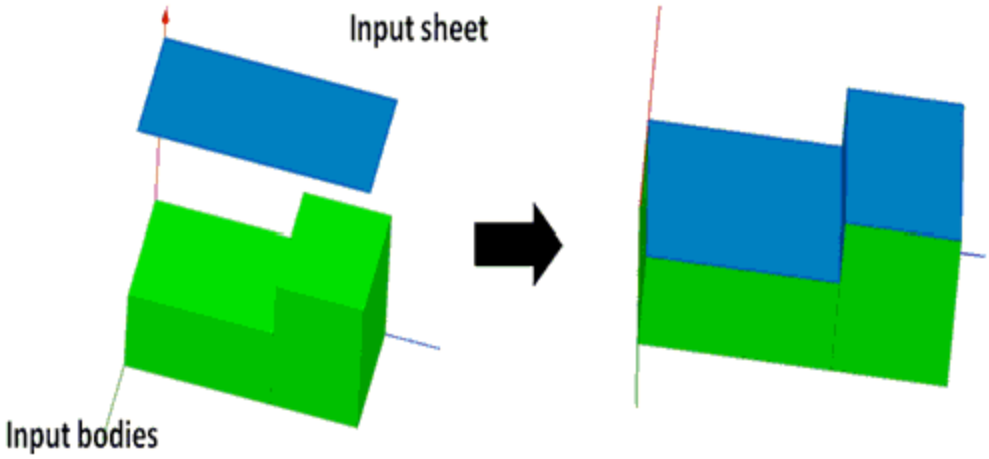
   A new object is created that connects the objects you selected. The first object you selected was modified to create the new object and all subsequently selected objects were deleted.

Project Sheet Object

Use the **Modeler** > **Surface** > **Project Sheet** command to apply a selected planar sheet object to a suitable 3D object. This permits easy modeling of thin conformal deposits. (The **Wrap Sheet** and **Imprint Projection** commands are not suitable for this use.) For modeling conformal deposits when using **Project Sheet**, you can specify a thickness and a draft angle.

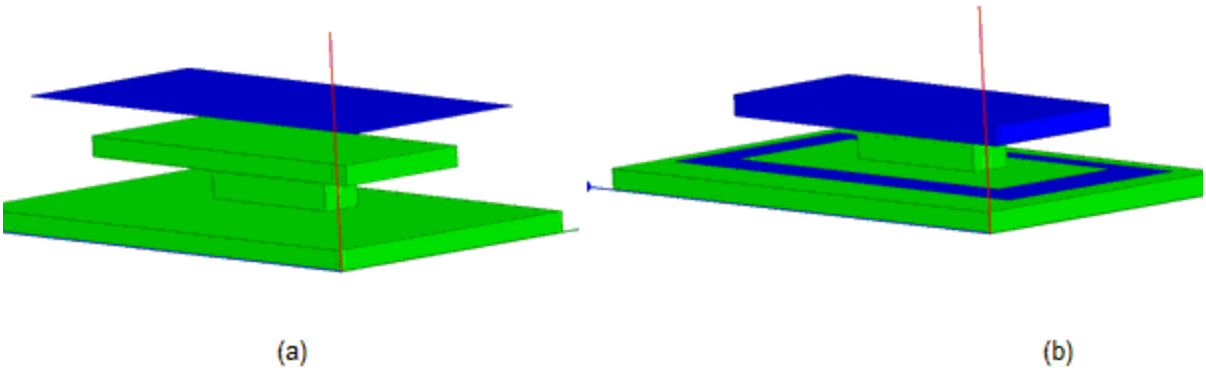


To use the **Project Sheet** command you must select one planar sheet and at least one solid body. If you select multiple solids, the sheet is project on all bodies as if they have been united. After the commend executes, a new sheet that lies on the surface of the selected bodies is created. The solid bodies are not changed.

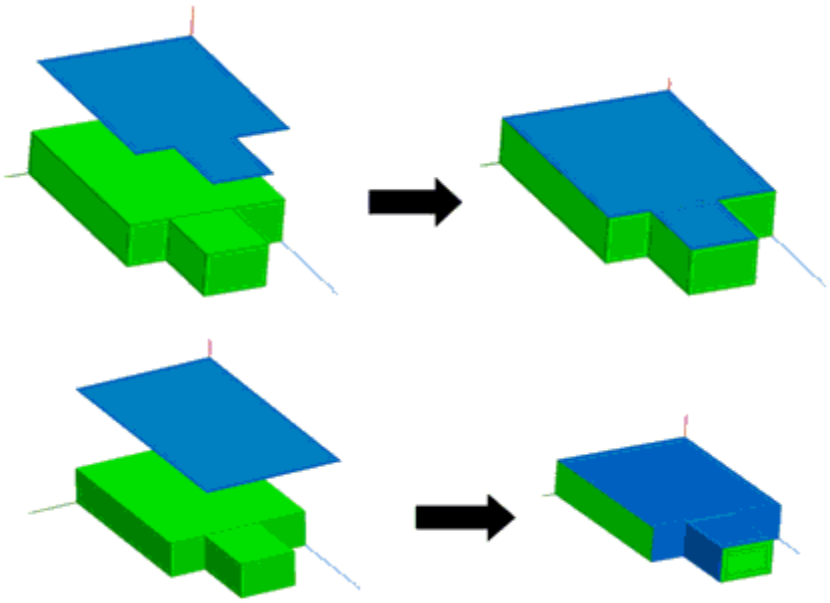


Faces that are hidden from the sheet being projected are not covered, due to the projected sheet.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



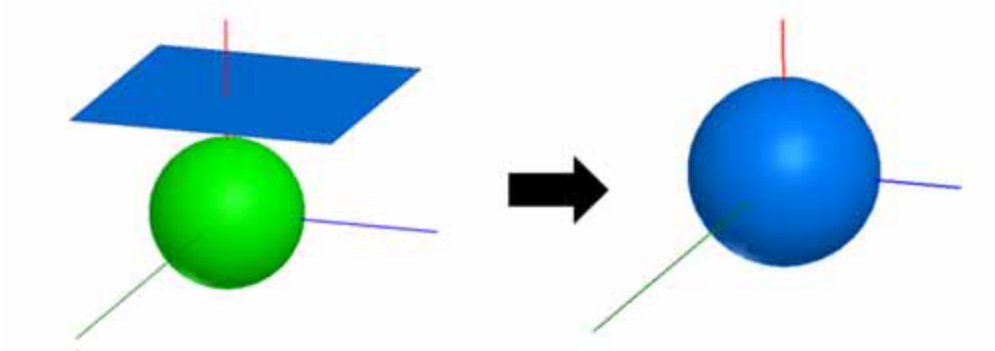
Only vertical faces (or faces parallel to the projection direction) that lie completely within the projection of the sheet will be covered by the projected sheet. Vertical faces that lie on the boundary of the projection are not covered.



If the sheet cross-section exactly matches the body cross-section, no vertical faces are covered. If the projected sheet exceeds the cross-section of solids, faces that lie with in the sheet projection are covered.

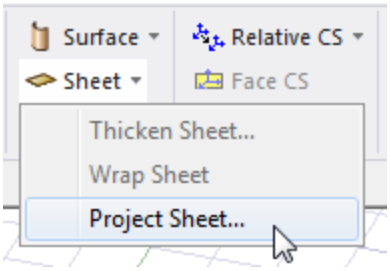
If a surface like a cylinder or sphere lies completely within the sheet projection, the entire surface will be covered, though there may be a warning that “Output may be incorrect because a self-obstructing face was found.”

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To Project a Sheet:

1. Select a planar sheet object and one or more appropriate solid objects.
2. From the menu bar, click **Modeler** > **Surface** > **Project Sheet**, or on the **Draw** ribbon tab, click **Sheet** > **Project Sheet**:

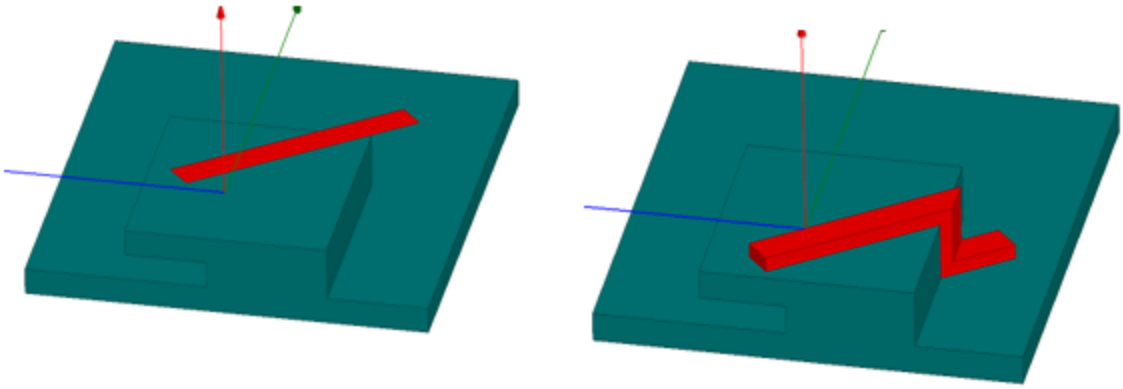


   For modeling conformal deposits, you can also specify a thickness and optional draft angle.

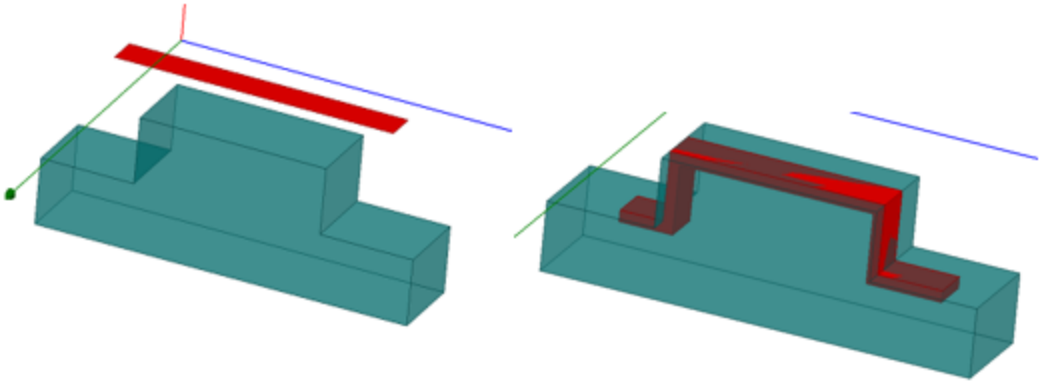
After you perform **Project Sheet**, the History Tree shows the **Project Sheet** command and the create command for the imprinted object. If you select the **Project Sheet** command in the History Tree, you can suppress the command via the docked *Properties* window. If you select the Create <object> icon for the object, you can edit the properties of that object. The changes applied to the object carry over to the imprinting.

Specifying a Thickness value has different effects depending on whether the value is positive or negative. The following figure shows the effect of a positive thickness, with the Projected sheet taking its thickness outward from the target object.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

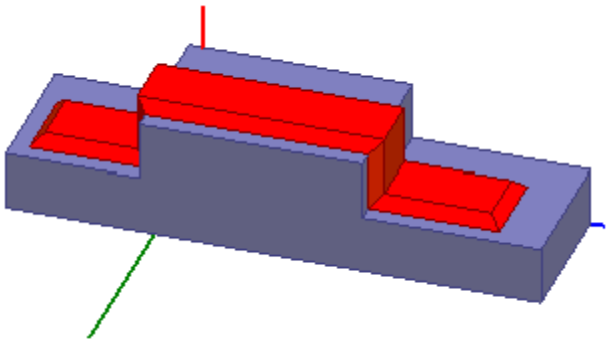


The following figure shows the effect of a negative value for thickness.



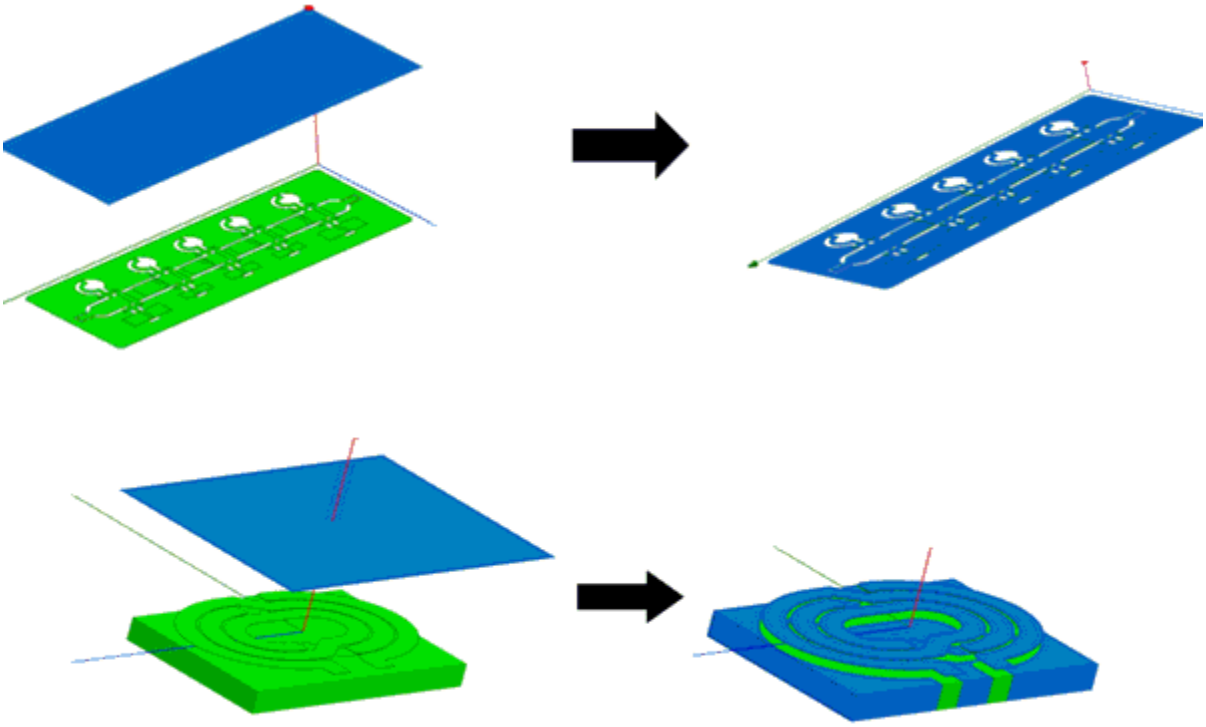
When a negative value is given the thickness will be limited by the target solid body. That is, the thickness can not be beyond the original solid body which was chosen. If you use a negative thickness it is also possible that not all overhangs will be eliminated.

The following figure shows the effect of a 45 degree draft angle.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Other examples of Project Sheet on complex models are show in the following figure.



Moving Faces or Edges

You can move the faces of a 3D object in a normal direction using the **Modeler> Surface> Move Faces** commands. Moving object faces enables you to resize, reshape, or relocate an object.

Moving Faces Along the Normal

To move a 3D object's face a specified distance in a direction normal to its original plane, use the **Modeler> Surface> Move Faces> Along Normal** command. The faces that adjoin the original face are extended or shortened along their own planes to meet the new face. Note that the adjoining faces will not be sheared or bent.

This command is useful for extruding faces, resizing holes, and removing rounded corners, as shown below.

To move an object face in a normal direction:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right-click in the Modeler window and click **Select Faces** on the shortcut menu.
2. Select the face of the object you want to move.

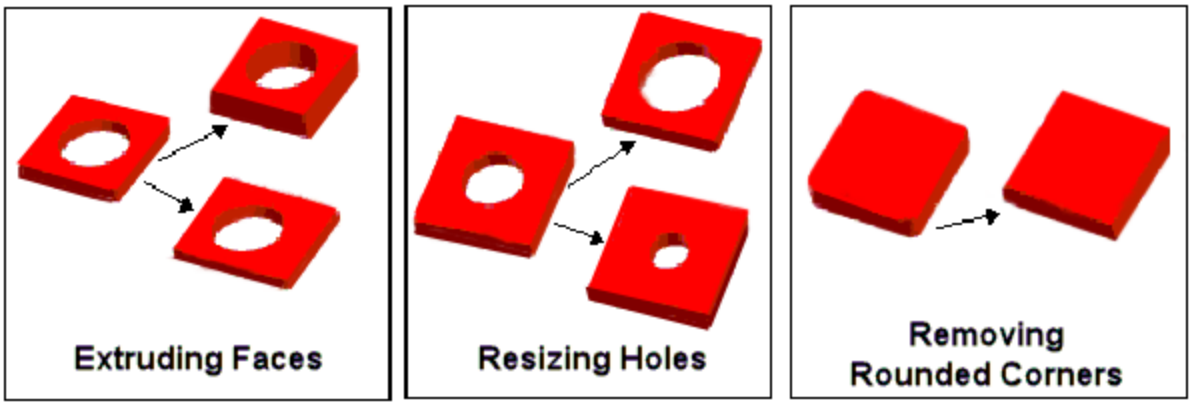
   If you have created a suitable face list, right-click the list and click **Select Assignment** from the shortcut menu. You can operate on faces in the list.

3. Click **Modeler> Surface> Move Faces**> **Along Normal**.

   The *Move Faces Along Normal* dialog box appears.

4. Type the distance you want to move the object face.
5. Click **OK**.

   The face will be moved the distance you specified.



To move every face of an object normal to its surface, use the **Edit> Arrange> Offset** command.

Moving Faces Along a Vector

To move the faces of a 3D object a specified distance along a vector use the **Modeler> Surface> Move Faces> Along Vector** command. Each selected face is moved along the vector, normal to its original plane. The faces that adjoin the original face are extended or shortened along their own planes to meet the new face. Note that the adjoining faces will not be sheared or bent.

This command is useful for relocating holes in an object, as shown below.

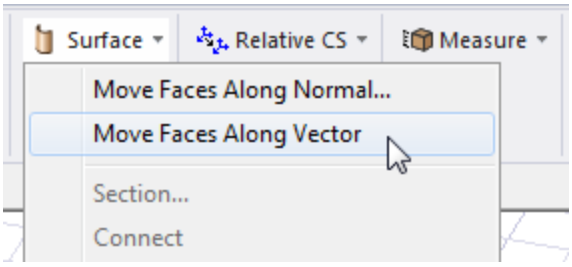
To move an object face along a vector:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Select Faces** on the shortcut menu.
2. Select the face of the object you want to move.

   If you have created a suitable face list, right-click the list and click **Select Assignment** from the shortcut menu. You can operate on faces in the list.

3. From the menu bar, click **Modeler> Surface> Move Faces> Along Vector**, or, on the **Draw** ribbon tab, click **Surface> Move Faces Along Vector**:



4. Specify the vector along which the face will be moved:
   a. Select an arbitrary anchor point in one of the following ways:
      - Click the point.
      - Type the point's coordinates in the **X, Y,** and **Z** boxes.

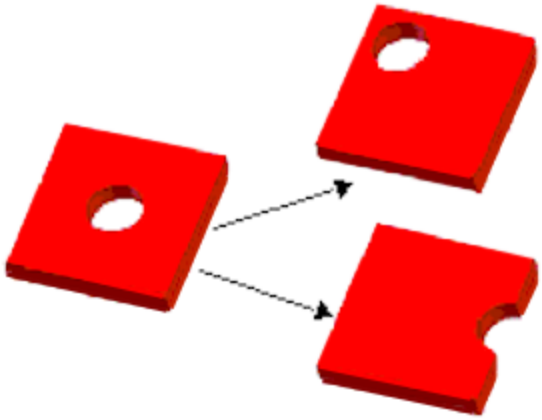
      Any point in the drawing region can be selected; however, selecting an anchor point on the object's edge or within the object makes it easier to select the vector.

   b. Select a second point in one of the following ways:
      - Click the point.
      - Type the coordinates of a point relative to the anchor point in the **dX, dY,** and **dZ** boxes, where **d** is the distance from the previously selected point.

      This point defines the direction and distance from the anchor point to move the face.

      The face is moved along the vector you specified.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Relocating Holes

To move every face of an object normal to its surface, use the **Edit>Arrange>Offset** command.

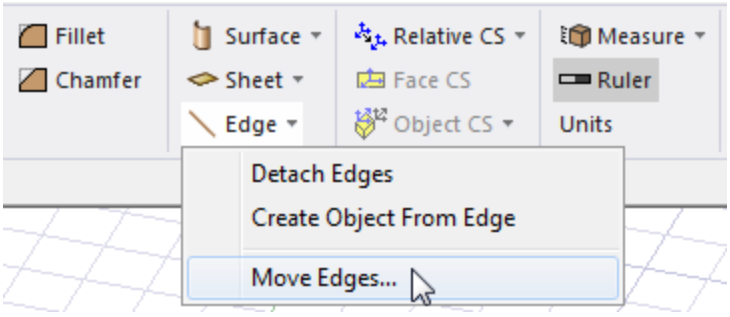
Moving Edges Along Normal

To move a 2D object's edge a specified distance in a direction normal to its original plane, use the **Modeler> Edge> Move Edge** command. The edge is extended or shortened along its own plane. Note that the adjoining faces will not be sheared or bent. The edge can be on a rectangle, an ellipse, a circle, a regular polygon, or an equation based surface.

This command is useful for extending or shrinking faces and resizing holes.

To move an object edge in a normal direction:

1. Right-click in the Modeler window and choose **Select Edge** on the shortcut menu.
2. Select the edge of the object you want to move.
3. From the menu bar, click **Modeler> Edge> Move Edge** or, on the **Draw** ribbon tab, click **Edge> Move Edges**:



The *Move Faces Along Normal* dialog box appears.

4. Type the distance you want to move the selected edge.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

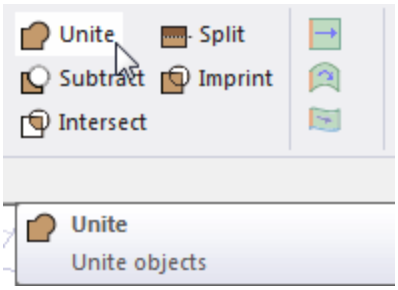
5. Click **OK**.

   The edge of the object is moved based on the value you specified.

Uniting Objects

To join two or more objects into one object, use the **Modeler> Boolean> Unite** command. The new object has the name, color, boundary, and material assignment of the first object selected. The objects are united at the point of intersection.

1. Select the objects you want to join.
2. From the menu bar, click **Modeler> Boolean> Unite** or, on the **Draw** ribbon tab, click the **Unite** icon:



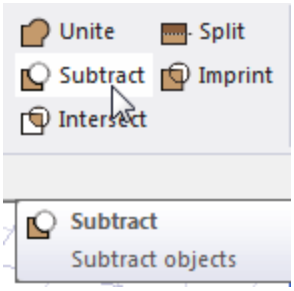
   The objects are united.

> **Note:**
>
> By default, the objects being joined to the first object selected are *not* preserved for later use. If you want to keep a copy of the objects being joined to the first object selected, do one of the following:
>
> - Copy the objects, and then paste them back into the design after uniting them.
> - Select **Clone tool objects before uniting** in the *3D Modeler> Operation* section of the *General Options*. This option instructs the modeler to always keep a copy of the original objects being joined.

Subtracting Objects

1. Select the object from which you want to subtract other objects.
2. Hold down the **Ctrl** key and select the objects you want to subtract.
3. From the menu bar, click **Modeler**> **Boolean**> **Subtract** or, on the **Draw** ribbon tab, click the **Subtract** icon:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The *Subtract* dialog box appears.

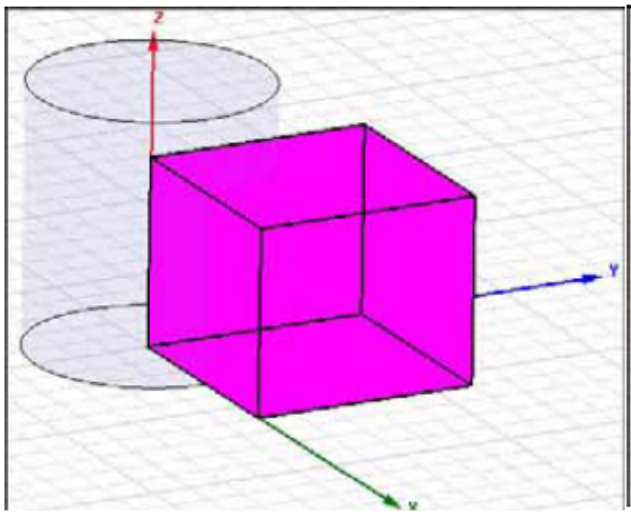
Objects listed in the *Tool Parts* list will be subtracted from the object or objects listed in the *Blank Parts* list.

4. Optionally, select an object name in either list and use the left and right arrow buttons to move the object name to the opposite list.

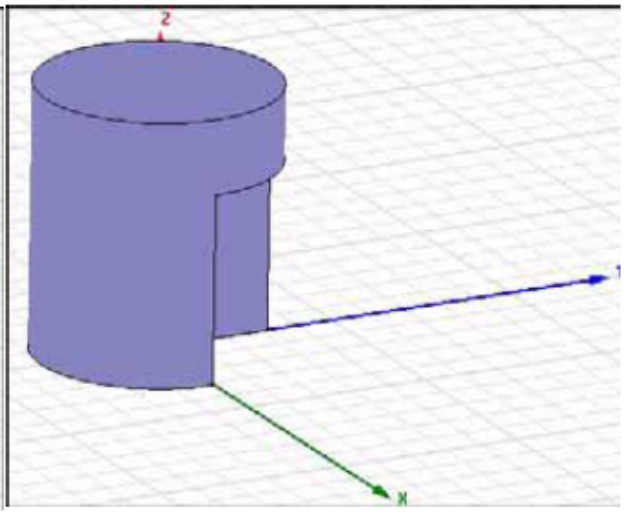
   Though you can type the name of objects into either list, you cannot access the *Subtract* command unless you first select two or more model objects. Therefore, the lists will initially be populated with the selected objects.

5. Optionally, select **Clone tool objects before subtracting**. This option instructs Ansys Electronics Desktop to keep a copy of the original objects being subtracted.
6. Click **OK**.

   The revised *Blank Parts* retain their original name, color, and material.



An intersecting box and cylinder



A box subtracted from a cylinder (the cylinder was selected first)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

By default, the *Tool Parts* (that is, the objects being subtracted from the *Blank Parts*) are *not* preserved for later use. If you want to keep a copy of the objects being subtracted from the blank parts, do one of the following:

- In the *Subtract* dialog box, select **Clone tool objects before subtracting**, as stated in step 5 above. This action is a one-time override of the defdefault behavior.
- In the *3D Modeler*> *Operation* section of the *General Options*, select **Clone tool objects before subtracting**. This option instructs Ansys Electronics Desktop to always (that is, by default) keep a copy of the original objects being subtracted.

Creating Objects from Intersections

To create a new object from the intersection of two or more objects, use the **Modeler** > **Boolean** > **Intersect** command.

To create an object from an intersection:

1. Select the objects from which you want to take the intersection.

   Warning:

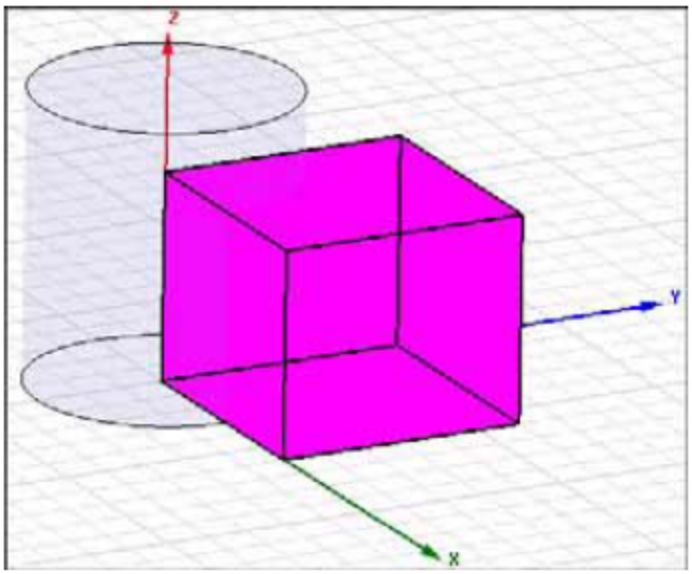
   If the objects you selected do not overlap, the result is a null object, and both objects vanish. When three or more objects have been selected, an intersection is created only if a common volume exists in which all the selected objects overlap. Otherwise, a null object is created and all original objects vanish.

   For example, assume that Part 1 overlaps Part 2, and Part 2 overlaps Part 3, but all three parts do not overlap in a common volume. This situation results in no intersection being created, and all three parts vanishing.

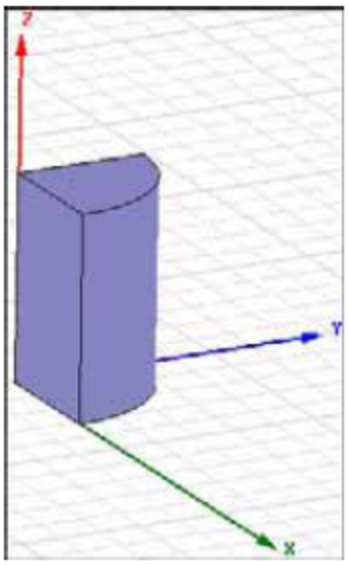
2. From the menu bar, click **Modeler** > **Boolean** > **Intersect**. (You can also access this command from the **Draw** ribbon tab.)

   The original objects vanish, leaving only the new object that was formed from their intersection.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



An intersecting box and cylinder.



Object formed from the intersection of the box and cylinder.

Note:

By default, the original intersecting objects are *not* preserved for later use. If you want to keep a copy of the objects that intersect the first object selected, do one of the following:

- Copy the objects, and then paste them back into the design after creating the new object from the intersection.
- Select **Clone tool objects before intersecting** in the *3D Modeler> Operation* section of the *General Options*. This option instructs the modeler to always keep a copy of the original tool parts (the second and subsequent objects selected) that intersect the blank part (the first object selected). The blank part is not preserved. To preserve the blank part, do as instructed in the preceding bullet.

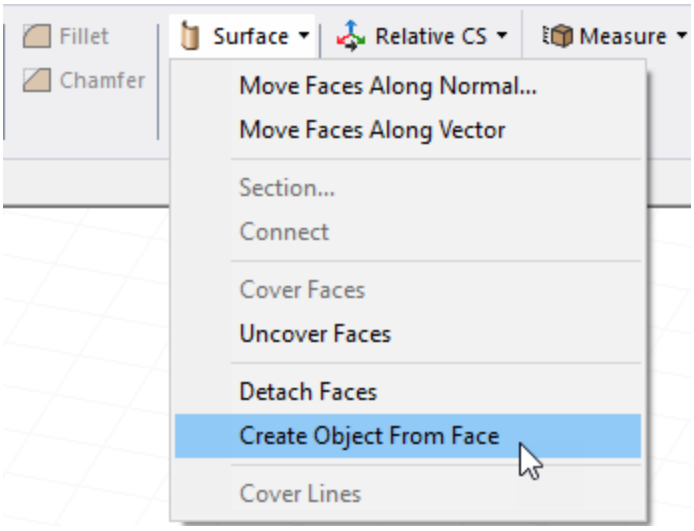
Creating an Object from a Face

The **Modeler> Surface> Create Object from Face** command copies a selected face, resulting in a new 2D sheet object.

To create a new object from a face:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right-click in the Modeler window and choose **Select Faces** from the shortcut menu.
2. Select the object face you want to copy. If you select multiple faces, each becomes a new object.
3. From the menu bar, click **Modeler**>**Surface>Create Object From Face** or, on the **Draw** ribbon tab, click **Surface**> **Create Object From Face**:



The face is copied, resulting in a new 2D sheet object in the same location.

> **Tip:**
>
> This command is useful for assigning a boundary to the partial intersection of two faces. To do this, first select the faces and create objects from them using the procedure shown above. Next, make sure the **Clone tool objects before intersecting** option is cleared in the *3D Modeler> Operation* section of the *General Options*. Then, select the two sheet objects and use the **Modeler> Boolean> Intersect** command to create a sheet object that includes only the intersection of the two faces. Finally, assign the boundary to the new sheet object.

Creating an Object from an Edge

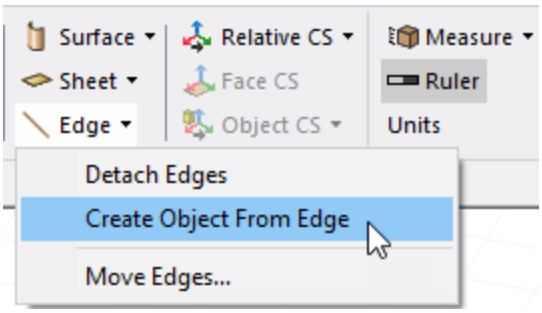
The **Modeler> Edge> Create Object From Edge** command copies a selected edge, resulting in a new line object.

To create a new object from an edge:

1. Right-click in the modeler window and choose **Select Edges** from the shortcut menu.
2. Select the object edge you want to copy. If you select multiple edges, each edge becomes a new line object.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. From the menu bar, click **Modeler**> **Edge**> **Create Object From Edge** or, on the **Draw** ribbon tab, click **Edge**> **Create Object From Face**:

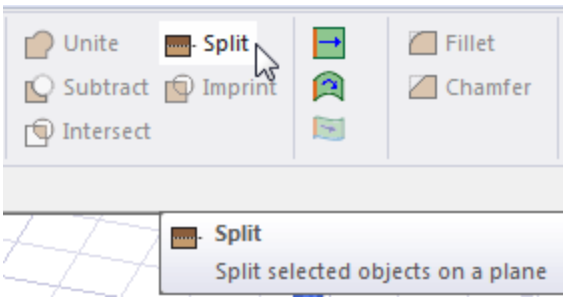


The edge is copied. The resulting object appears in the History Tree as a line object.

Splitting Objects

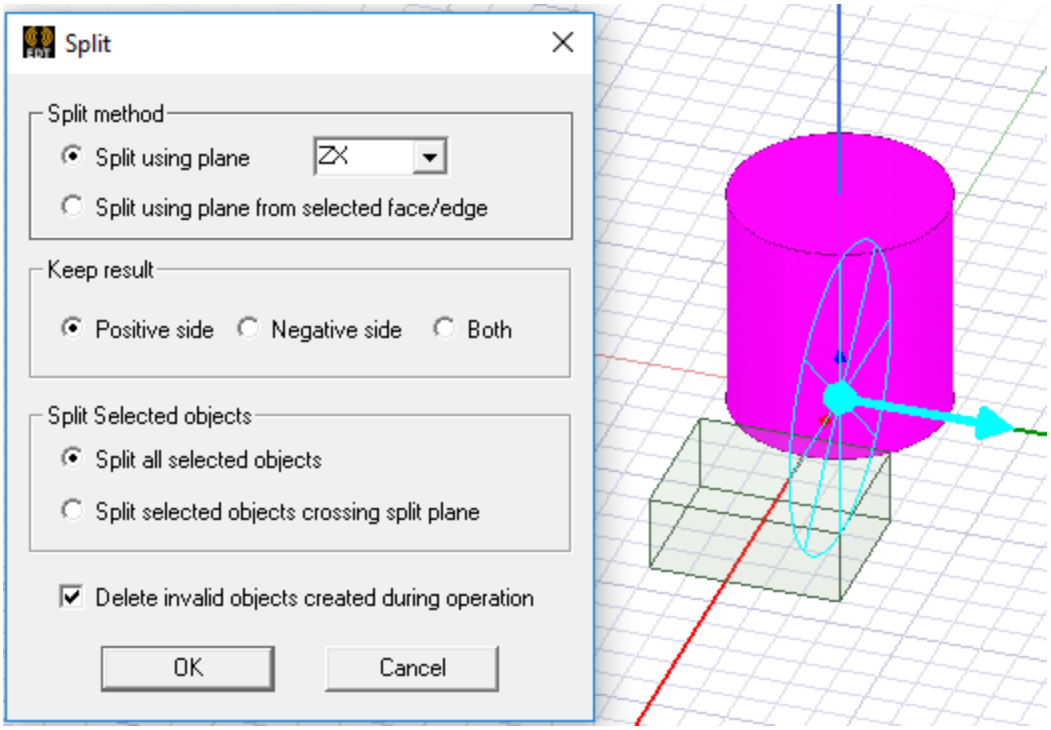
To split an object or several objects using one of the global planes (XY, YZ, or XZ) or a plane based on a selected face or arc, use the **Modeler> Boolean> Split** command.

1. Select the object you want to split. You can select more than one object.
2. From the menu bar, click **Modeler> Boolean>** **Split** or, on the **Draw** ribbon tab, click the **Split** icon:



The *Split* dialog box appears. Also, a cyan colored wheel shows the current split plane and points toward the positive side of the plane.

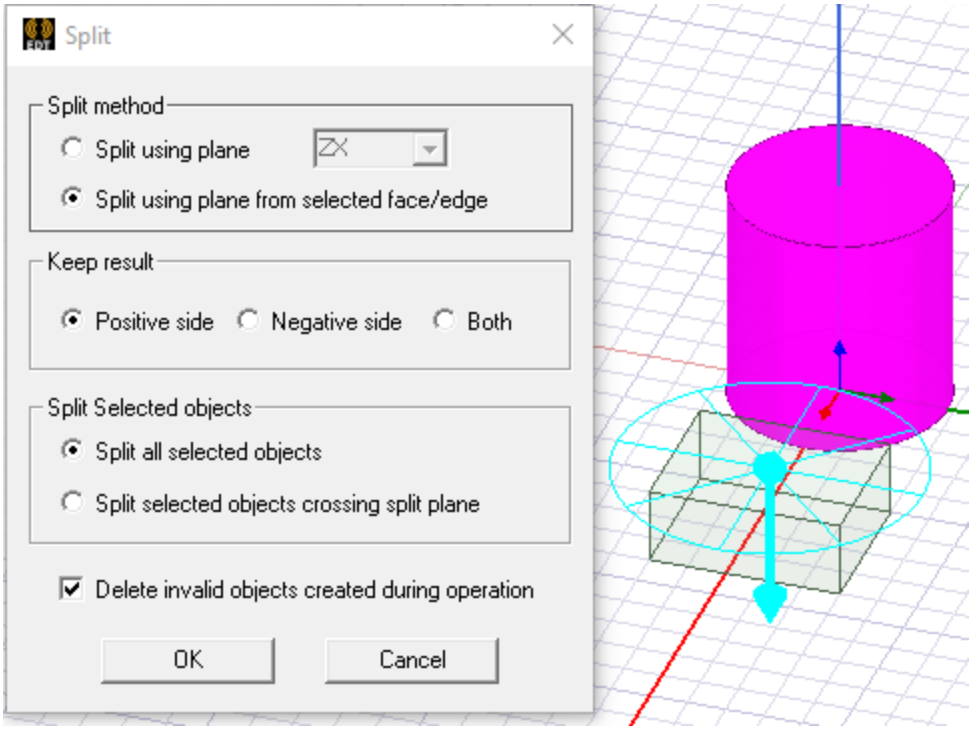
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Choose the desired *Split method* and specify the splitting plane:
   - **Split using plane:** Select the desired splitting plane (**XY, YZ,** or **XZ**) from the drop-down menu, or
   - **Split using plane from selected face/edge:** Select a single face or edge (specifically, a planar face or arc) to define the splitting plane.

     The following example shows the top face of the box being used to define the splitting plane for the cylinder:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

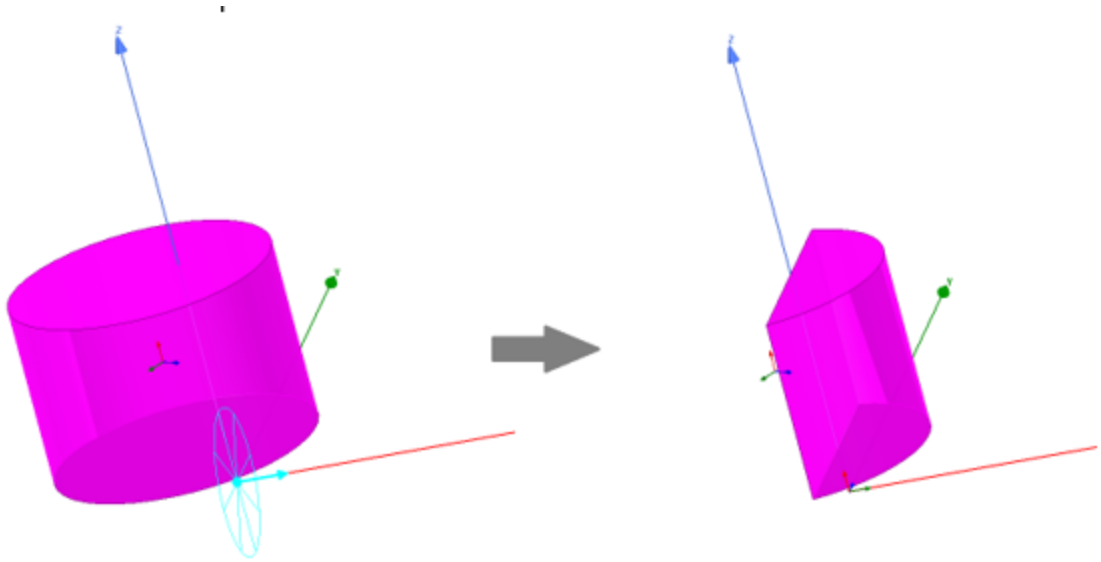


4. Select one of the following *Keep result* options to specify which object fragments you want to keep:
   - **Positive side:** Keep objects on the positive side of the splitting plane.
   - **Negative side:** Keep objects on the negative side of the splitting plane.
   - **Both:** Keep objects on both sides of the splitting plane.

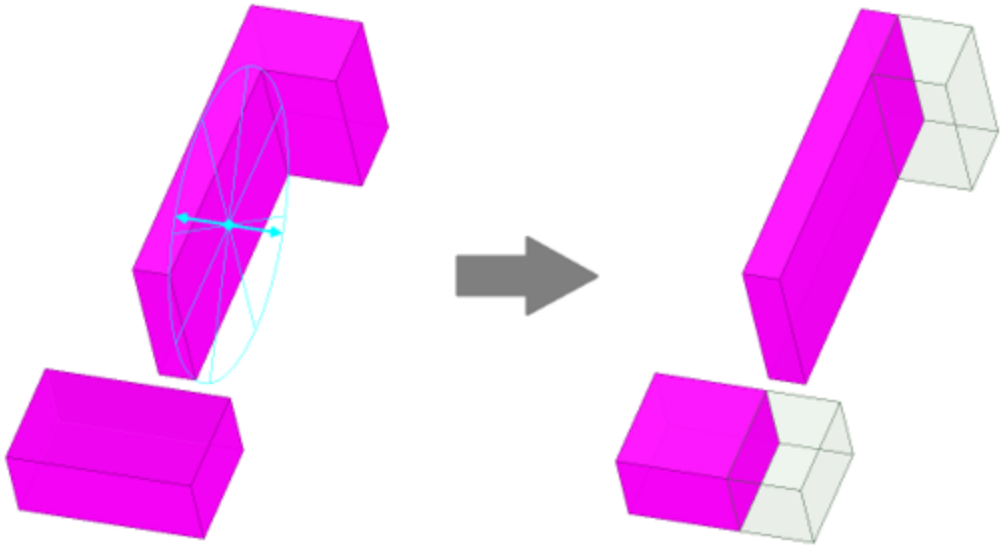
   The split plane is shown using a cyan colored wheel with spokes, as seen in the images above and below left. A cyan arrow indicates which side of the split plane is the positive side.

   In the following example, the *YZ* plane (green and blue axes) is the splitting plane, and the result on the *positive side* was kept. The picture on the left is before the split. The picture on the right is the result of the completed *Split* command:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The next example shows the second *Split method* (*Split using plane from selected face/edge*). You can only select one entity, a planar face or an arc (selected as an edge). The face or edge used to define the splitting plane need not belong to one of objects selected to be split. You can select any face or arc edge in the model.. You can pan, zoom, or rotate the model viewpoint as needed to make the selection easier. Also, the *Keep result* for *Both* sides option was selected. (Notice the arrow is in both directions for the split plane graphics.) The picture on the left is before the split. The picture on the right is the result of the completed *Split* command:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Select one of the following *Split Selected objects* options:
   - **Split all selected objects:** Select this option if you do not want to preserve objects that are not crossing the split plane but are part of the selection.
   - **Split all selected objects crossing split plane:** This option allows you to identify selected objects that do not cross the split plane and ignore them for the operation. For a multiple selection, only those objects that cross the split plane are split; others are kept intact. By design, splits in existing designs from previous software versions are not changed.
6. Optionally, select the **Delete invalid objects created during operation**. If you choose to *Split all selected objects*, the operation could create invalid objects (if split plane does not cut any of selected objects and any are on the side not being kept). This option deletes any such invalid objects that might be created.
7. Once the proper selections have been made, click **OK**.

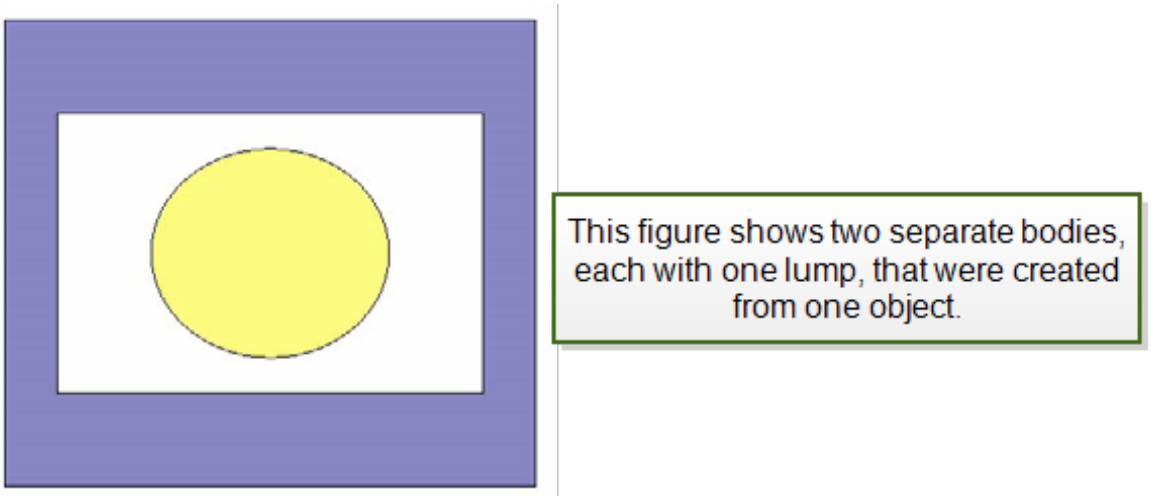
   The objects are divided and retained as specified.

Separating Bodies

To separate an object with multiple lumps (geometric bodies) into individual single-body objects:

1. Select the object you want to separate.
2. From the menu bar, click **Modeler**> **Boolean**> **Separate Bodies**.

   The bodies are separated into multiple objects.



Converting Polyline Segments

A polyline is a single object that includes any combination of straight line, arc, or spline segments. You can convert a polyline segment from one type to another. The following

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

conversions are supported:

- Straight line segments to arc or spline segments.
- Arc segments to straight line or spline segments.
- Spline segments to straight line segments.

To convert polyline segments:

1. In the History Tree, locate the polyline that contains the segment you want to convert. Expand this part of the tree.
2. Click the polyline segment operation you want to change to select it.

   The segment properties appear in the *Segment* tab of the docked *Properties* window.
3. Click in the **Value** text box in the **Segment Type** row and select the desired polyline segment type from the drop-down menu.

   The polyline segment you selected is changed to the new type.

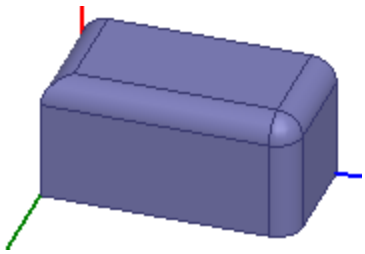
   > **Note:**
   >
   > Converting an arc or spline segment to a straight line segment results in two straight line segments; one segment is created between the start point and midpoint and one segment is created between the midpoint and endpoint.
4. By default, curved segments are generated as smooth (true) curves. If you want to approximate an arc or spline using multiple discrete straight segments, enter an integer of 2 or greater in the **Number of Segments** value. A value of zero produces a smooth curve.

   Segmented curves can be useful when sweeping polylines into surfaces. Segmented surfaces provide some regularity to the mesh and can enable you to visualize surface twisting when you sweep along a path with a twist angle specified.
5. If the changes are not what you expected, undo the change using the **Edit> Undo** command or by pressing **Ctrl+Z**.

Rounding the Edge of Objects (Fillet Command)

The fillet command rounds the object at the original edges and vertices. The edges and vertices are replaced by new rounded surfaces that are tangential to the original adjacent surfaces, providing a smooth transition.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Vertices are only going to be replaced by new rounded surfaces if all the edges connecting to the original vertex are selected; otherwise, the vertex is preserved but moved (if necessary). The edges are replaced by quarter-cylindrical surfaces, of which the radius can be customized (see the **Fillet Radius** property). Vertices are replaced by more complicated new faces. You can control the setback distance.

The fillet command is disabled if an edge is not selected.

To switch to edge selection mode:

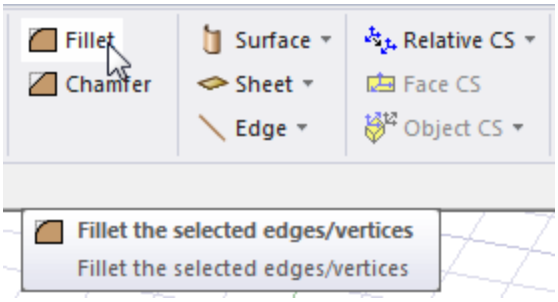
- Right-click the desktop, and choose **Select Edges** from the shortcut menu.

To round the edge of an object:

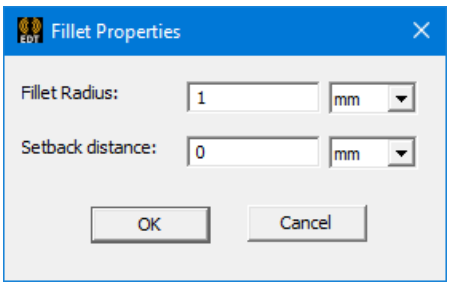
1. Select the edge you want to change.

   The edge is highlighted and the *Fillet* command is enabled.

2. From the menu bar, click **Modeler** > **Fillet** or, on the **Draw** ribbon tab, click the **Fillet** icon:



   The *Fillet Properties* dialog box is displayed:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Enter a value for the **Fillet Radius** and select units from the drop-down menu.
4. Enter a value for the **Setback distance**.

   The setback distance controls the shape of the vertex. It is the distance of the cross curve from the vertex at the end of the edge. If it is less than the fillet radius the *Setback distance* has no effect. Also, you will get an error if it is greater then the length of the edge. Note that the setback feature only works on corners where three or more edges meet and only if all edges meeting at the vertex are selected.

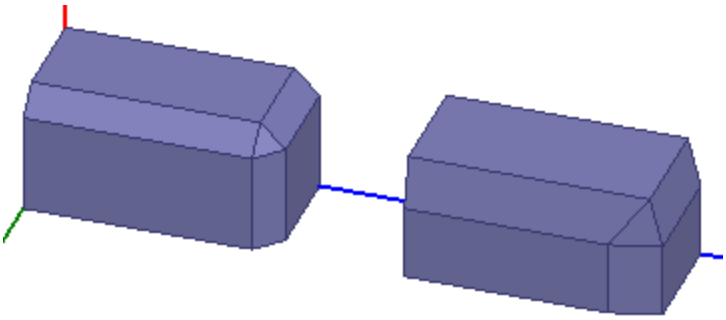
5. Click **OK** to apply the change to the edge.

   The dialog box closes and the object edges and vertices are rounded as specified.

Flattening the Edge of Objects (Chamfer Command)

The chamfer command flattens the selected edges and vertices of the object. Selected edges and vertices are replaced by new flat surfaces known as chamfers.

Chamfers can be symmetric, or asymmetric:



The angle between the original faces and the chamfer's face varies depending on the angle between the original faces, and it also depends on the left and right chamfer distances. For example, for two faces at 90° with an equal distance (symmetrical) chamfer, the edge chamfer's face will be 45° from both of the original faces.

Vertices are only going to be replaced by new flat faces if all the edges connecting to the original vertex are selected; otherwise, the vertex is preserved but moved (if necessary).

The chamfer command is disabled if an edge is not selected.

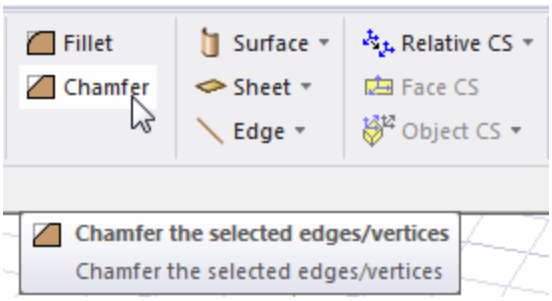
To flatten an object's edge for a symmetric chamfer:

1. Right-click the desktop, and select **Select Edges** from the shortcut menu.
2. Select the edge (or edges) you want to change.

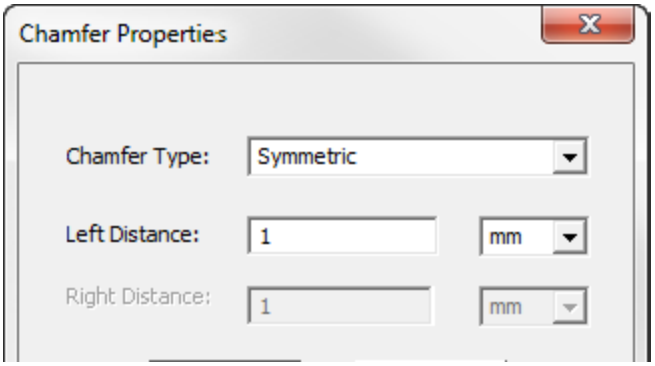
   The edge(s) is highlighted, and the **Chamfer** command is enabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. From the menu bar, click **Modeler> Chamfer** or, on the **Draw** ribbon tab, click the **Chamfer** icon:



The *Chamfer Properties* dialog box appears.



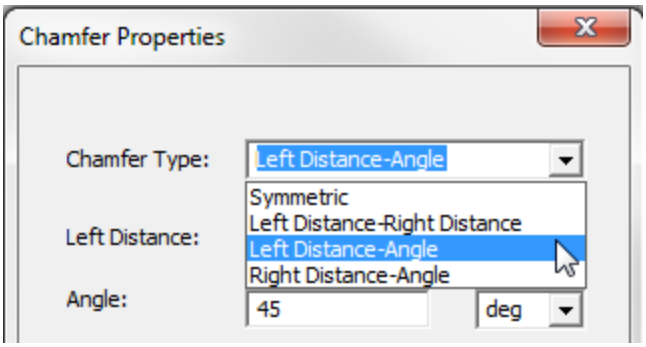
4. Type a **Left Distance** value and select the units from the drop-down menu.
5. Click **OK** to apply the change to the selected edges.

To flatten object's edge for an asymmetric chamfer:

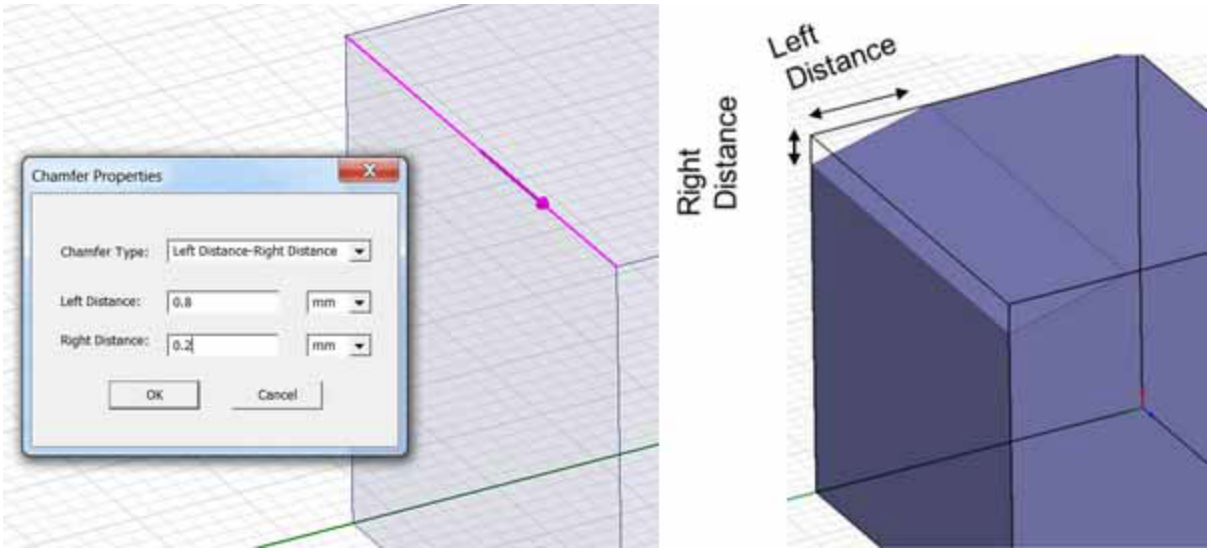
1. Select one or more edges and click **Modeler> Chamfer** to open the *Chamfer Properties* dialog box.
2. Use the **Chamfer type** drop-down menu to select the type:

   You control an asymmetric chamfer by selecting a type that defines the chamfer asymmetry as **Left Distance-Right Distance**, as **Left Distance-Angle**, or as **Right Distance-Angle**. Notice that red directional arrows on the selected edges provide the direction against which left distance and right distance are determined.

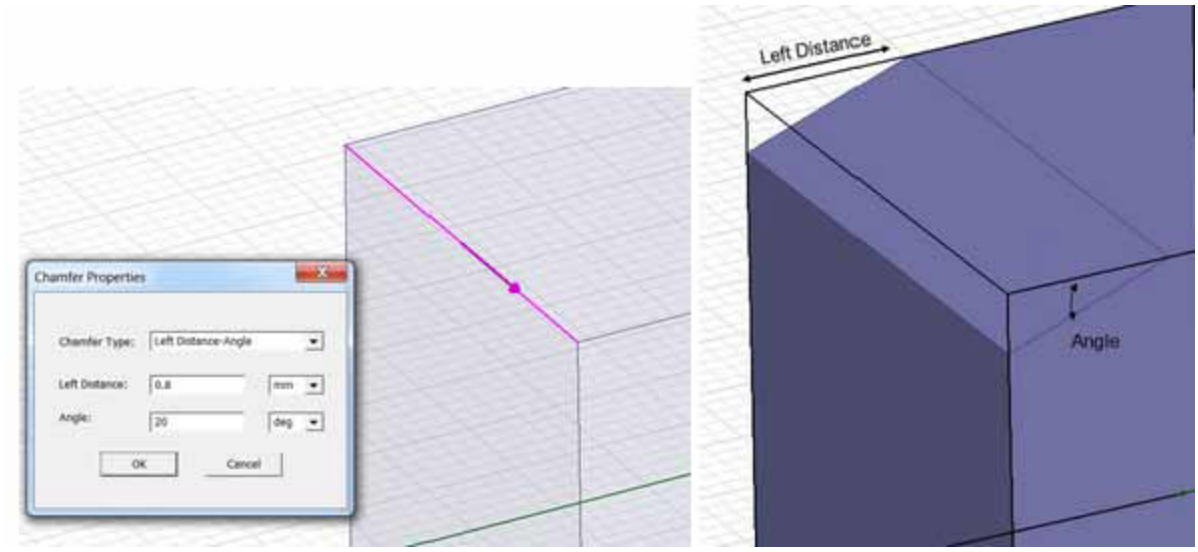
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Example 1: Left Distance-Right Distance

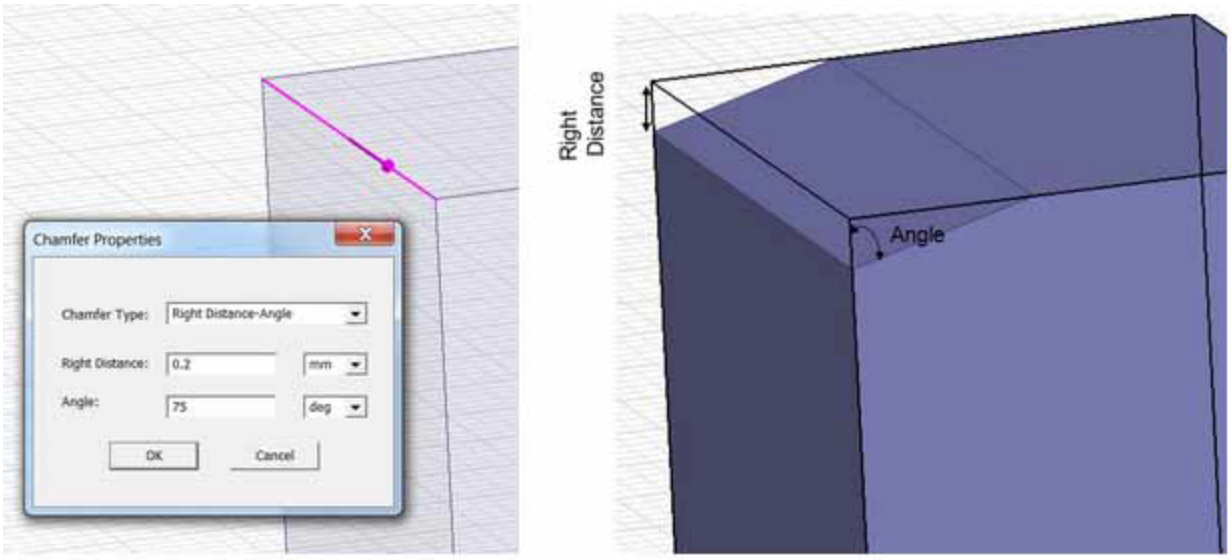


Example 2: Left Distance-Angle



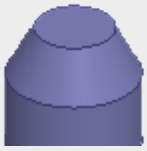
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example 3: Right Distance-Angle



Note:

Only symmetric and left distance-right distance chamfers are supported for edges formed by a curved surface.



Angle-distance chamfers are not supported in such cases and do not appear on the Chamfer Type drop-down menu.

3. Click **OK** to apply the chamfer to the selected edges.

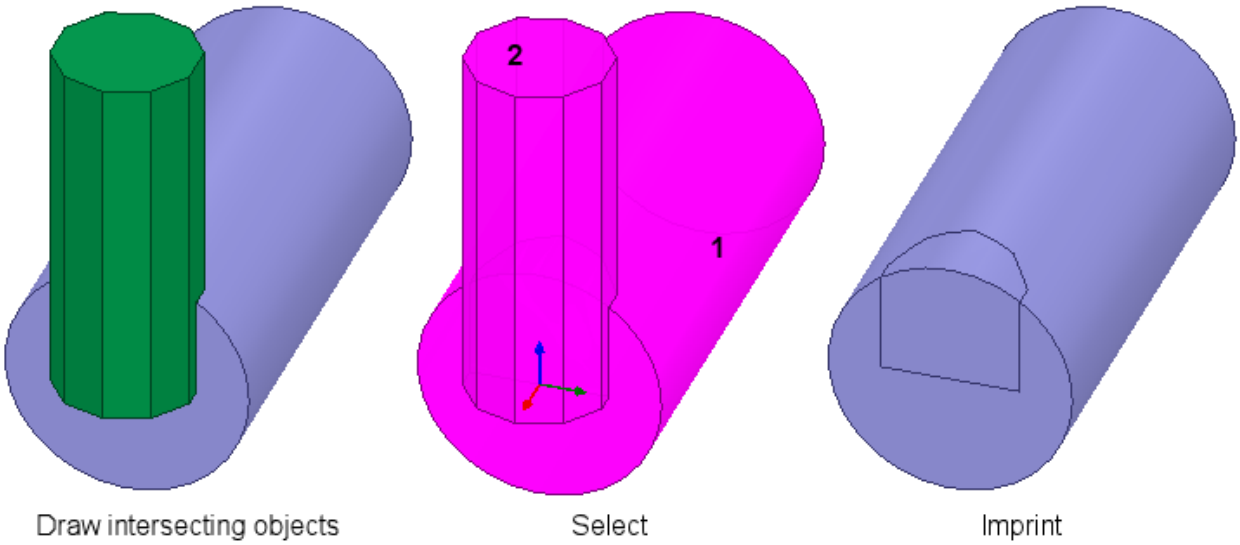
Imprinting an Object

The **Boolean** > **Imprint** command lets you imprint the geometry of one object upon another. For example, you could draw a polyhedron intersecting a cylinder, and then imprint the intersecting lines on the cylinder. This process splits the larger face into two or more faces, with the intersecting lines representing the edges of the imprinted face. The original face is cut so that it no longer extends into the imprinted area.

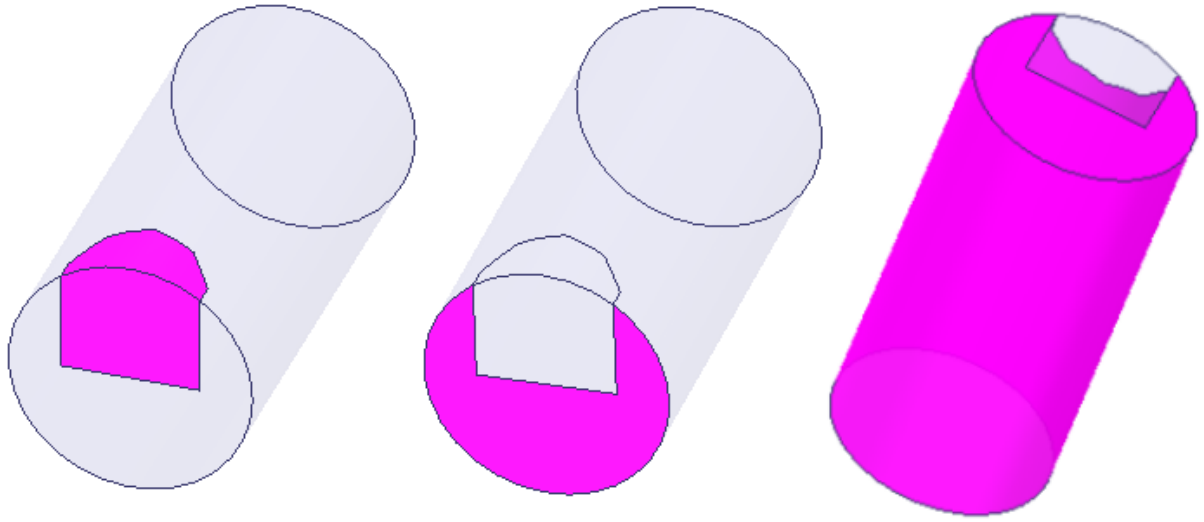
Imprinting is often an essential capability for finite element analysis. It enables a boundary or excitation to be applied to the exposed area of a model face while excluding it from areas of intersection with other parts. One example is a thermal analysis where convection is applied to

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the exposed surfaces of a heat sink. The area in contact with a semiconductor (the source or heat) is not exposed to ambient air and should not receive a convection boundary. In this case, you would imprint the semiconductor on the heat sink, producing a separate heat sink face in the contact area. This face is excluded from the selection set when applying the convection boundary.



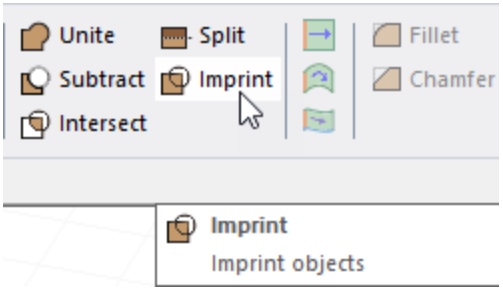
You can select the faces of the imprinted surface separately and assign properties as needed.



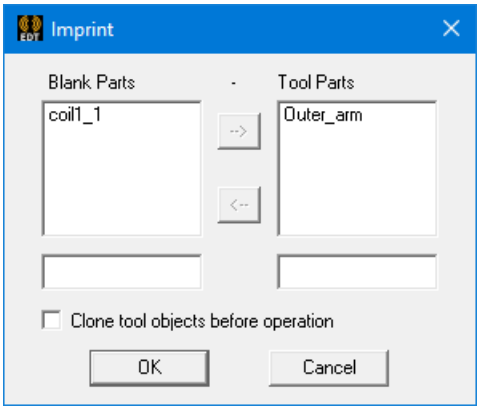
To imprint one object with another:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the intersecting objects.
2. From the menu bar, click **Modeler** > **Boolean** > **Imprint...** or, on the **Draw** ribbon tab, click the **Imprint** icon:

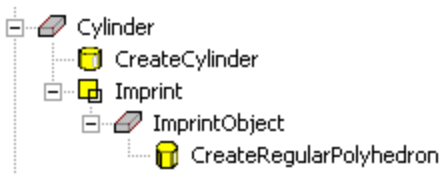


The **Imprint** dialog box displays in which you designate which objects are the *Blank Parts*, and which the *Tool Parts*. If necessary, you can select the objects in lists, and use the arrow keys to move them. If desired, you can clone the tool objects before the imprint operation.



3. Optionally, select **Clone the tool objects before operation** to retain the tool part. Otherwise, only its imprint on the blank part will remain.
4. Click **OK** to close the dialog box and perform the Boolean imprinting.

   The History Tree retains the ImprintObject command and the Create<object> command for the imprinted tool object:



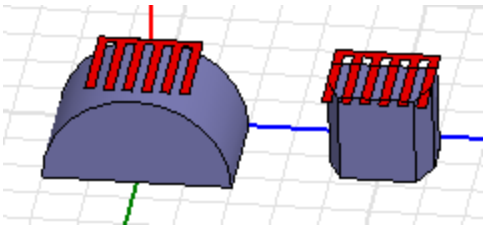
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you select the Imprint command in the History tree, you can suppress the command via the docked *Properties* window. If you select the *Create<object>* icon, you can edit the properties of that object. The changes you apply carry over to the imprinting operation, modifying the imprinted face accordingly.

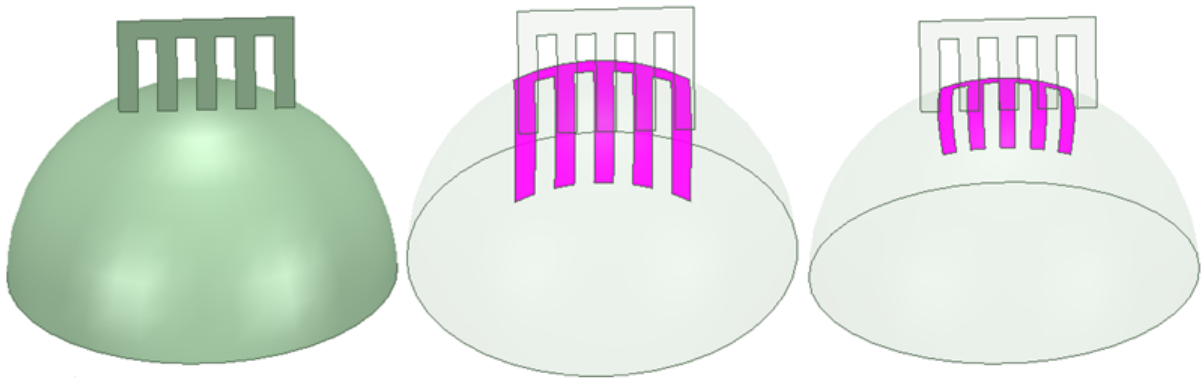
In the *3D Modeler* > *Operation* section of the *General Options*, you can also enable the **Automatically imprint wrapped sheets** option to automatically perform an *Imprint* operation command after executing the **Modeler** > **Surface** > **Wrap** command.

Imprint Projection Commands

The **Boolean** > **Imprint Projection** commands lets you project the form of a sheet object onto the face or faces of another object (either solid or sheet). The receiving object faces can be flat, curved, or faceted.



If the receiving surface is curved or inclined, and you choose to project *Along Normal* direction, the dimensions of the projection are affected. The reason is that the projection direction is normal to the curved or inclined face of the receiving object and not normal to the object being projected.



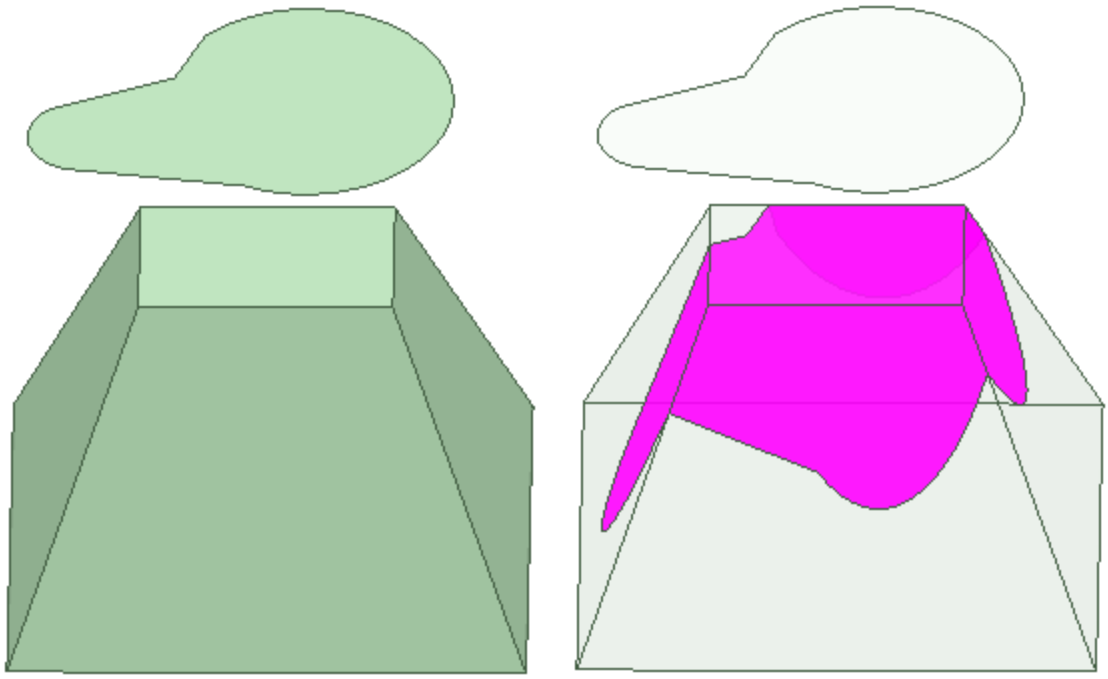
Comb & Hemisphere

Comb Projected Along Vector

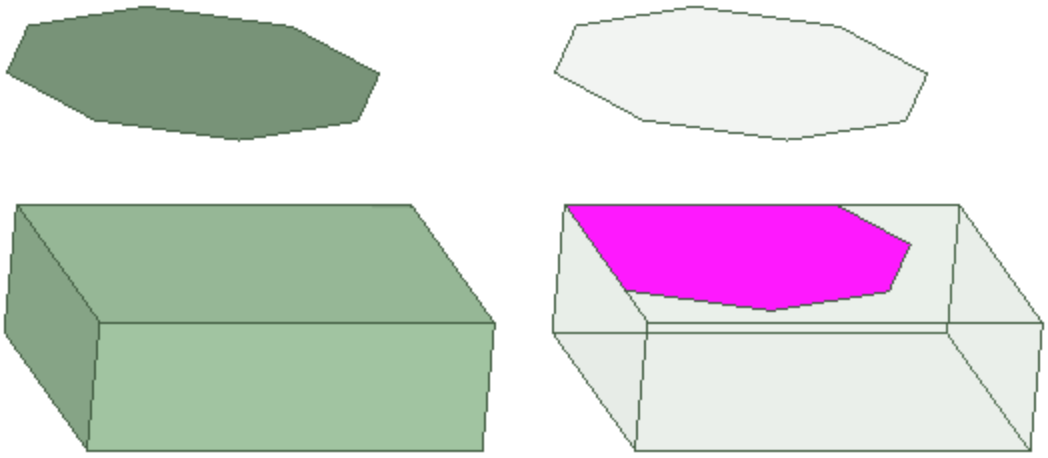
Comb Projected Along Normal (normal to hemispherical target object)

The projected shape will wrap around the target object, from the face nearest to the center of the imprinting area and extending to the adjacent target object faces:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the projection of the imprinting object extends beyond the extents of the receiving object, the imprinted shape is truncated:

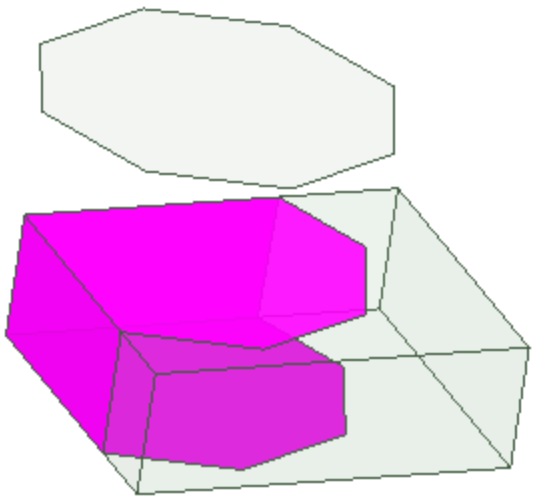


You can select the imprinted faces on the target object separately, and perform subsequent operations (such as creating sheet objects from the faces, assigning boundaries or excitations, or selecting for field overlays).

To perform an Imprint Projection:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

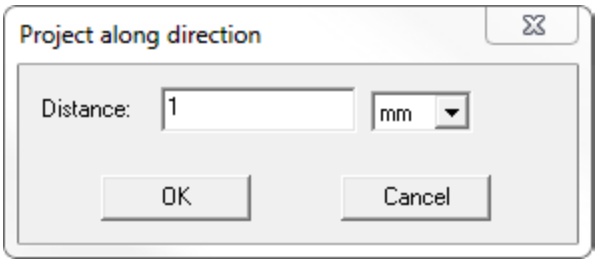
1. If you want to preserve the tool object that will be imprinted on the target object, use one of the following methods:
   - Copy the tool object to the clipboard so that you can paste it back into the model after the imprinting operation.
   - In the *3D Modeler > Operation* section of the *General Options*, select the **Clone tool objects before projecting** option.
2. Select the projecting and target *objects*.
3. From the menu bar, click **Modeler** > **Boolean** > **Imprint Projection** > **Along Normal** or **Modeler** > **Boolean** > **Imprint Projection** > **Along Direction...**
   - If you select **Along Normal**, the projection occurs along the direction normal to the target faces and the operation is completed immediately. Imprinting occurs only on target faces visible from the imprinting object's point of view.
   - If you select **Along Direction**, you must specify two points that define the imprinting vector direction (step 3 below). Once you have done this, you see a dialog box for specifying the projection **Distance**. This option allows you to limit the projection distance so that only near faces are affected. Unlike the *Along Normal* option, the *Along Direction* option can imprint onto the target object faces that are not visible from the imprinting object's point of view (that is, those on the opposite side of the target object):



4. If performing an imprint projection operation using the **Along Direction** option:
   a. Specify the imprinting vector direction. You can click in the Modeler window or use the coordinate entry text boxes in the Status Bar.

      The *Project along direction* dialog box appears:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. Specify the maximum projection **Distance** and select the length unit from the drop-down menu.
c. Click **OK**.

   The dialog box closes and the Boolean imprinting operation is performed.

After you perform the imprinting, the History Tree shows the ImprintProjection command and the create command for the imprinted object.

If you select the *Imprint Projection* command in the History Tree, you can suppress the command via the docked *Properties* window. If you select the *Create<object>* icon for the object, you can edit the properties of that object. The changes applied to the object carry over to the imprinting operation.

Purging Object History

Each object is a sequence of modeler-based operations. The history for each object is shown under its name in the History Tree. You can use the **Purge History** command to remove the history of operations while not affecting the geometry itself. This is useful when you wish to perform healing operations on the object. If there is an object for which you want to keep the history, you should make a copy of the object for that purpose before purging.

To purge the history:

1. Select the object.
2. From the menu bar, select **Modeler**> **Purge History**. The history for the model is purged, and the context for the **Undo** and **Redo** commands is updated.

Related Topics

Working with the History Tree

Generate History

Generate History to Reproduce Portions of Model

If a primitive object (cylinder, cone, box, circle, 3D polygon, etc.), polyline object (line, spline, or arc), circle, or ellipse is imported, or if the history was previously purged, you can select the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

object and use the **Generate History** command to reproduce, in the History Tree, the primitive object definition or the individual line segments used to create a polyline. *Generated History* uses the original coordinate system in which the objects were created. *Generate History* also retains boundary conditions.

To reproduce the line segments in the model history tree:

1. Select the primitive or polyline object.
2. From the menu bar, click **Modeler> Generate History**.

Selecting Items in the 3D Modeler Window

To modify or learn more about an item's properties, you must first select it. All commands you choose while an item is selected are performed on or in reference to the selected item.

What selection mode do you want to use?

- Select Submodels
- Select Objects
- Select Faces
- Select Edges
- Select Vertices
- Select Multi
- Coordinates in the drawing space
- Select By Area including optional filters
- Select by History Tree Group
- Select by Submodel
- Select by intersection error message
- Selecting the Face or Object Behind

Selection menu for:

The **Edit>Select Objects** menu and **Modeler** window shortcut menu includes selection for:

- By Name
- By Material
- By Variable
- All Model Objects
- All Non-model Objects
- All Primitive Shapes
- All Non Primitive Shapes
- All Solids

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- All Sheets
- All Lines

Selections enabled for features of selected Object:

Some **Edit>Extend Selection** menu commands are enabled when you select an appropriate object:

- All Object Faces
- All Object Edges
- All Face Edges
- Faces on Plane
- Select Connected Vertices
- Select Connected Edges
- Select Connected Faces
- Select Edge Chain
- Select Face Chain
- Select Sheet Edges

History Tree Icon Menu Selection

You can also right-click **History** tree icons to **Select All** for ungrouped:

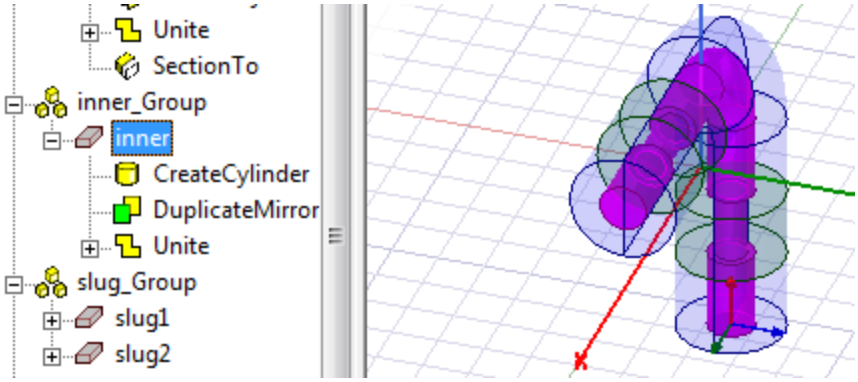
- Model
- Group
- Solids
- Selected Material, for example copper or air.
- Sheets
- Lines

> **Note:**
>
> If selected objects do not display correctly, for some graphics card, you can improve performance by changing settings in the *NVIDIA Control Panel* application. In the *Select a Task* tree, navigate to **3D Settings> Manage 3D Settings**. Then, under the **Global Settings** tab, choose **Workstation App - Dynamic Streaming** from the **Global presets** drop-down menu.
>
> For more information about graphics cards, navigate to the appropriate Windows or Linux Ansys Installation Guide and search for **OpenGL**.

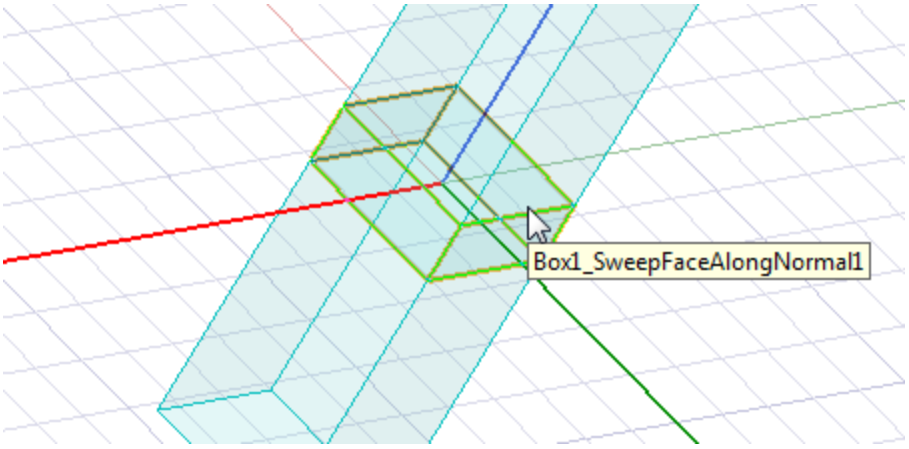
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting Objects

By default, the modeler is in *object selection mode*. Simply click an object in the view window or an object name in the history tree and it will be selected. All other objects become relatively transparent. Selected objects become the color specified under the **Display** tab of the **Modeler Options** dialog box.



When the mouse hovers over an object in the view window, that object is highlighted, which indicates that it will be selected when you click. As you hover the cursor over an entity, tooltips indicate its type/ID (object name in the case of objects, Face_id in the case of faces, and so on). This feature helps you distinguish between face-of-sheet-object pick versus sheet-object pick.

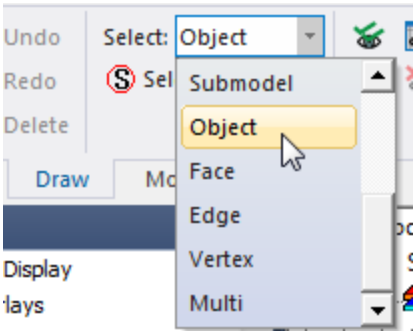


If the modeler is not currently in object selection mode, you can switch to it using one of the following methods:

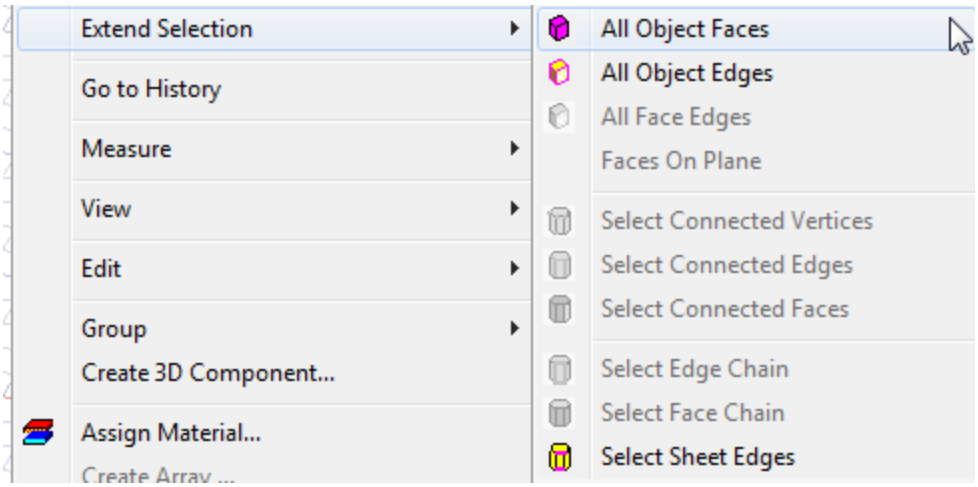
- Press the **O** shortcut key while the Modeler window is active.
- Right-click in the Modeler window and click **Selection Mode**> **Select Objects**.
- From the menu bar, click **Edit**> **Selection Mode**> **Objects**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- On the **Draw** or **Model** ribbon tabs, choose **Object** from the **Select** drop-down menu:



When you've selected an object, the menu bar's **Edit**> **Extend Selection** submenu and the **Extend Selection** submenu in the Modeler window's shortcut menu show the following options:

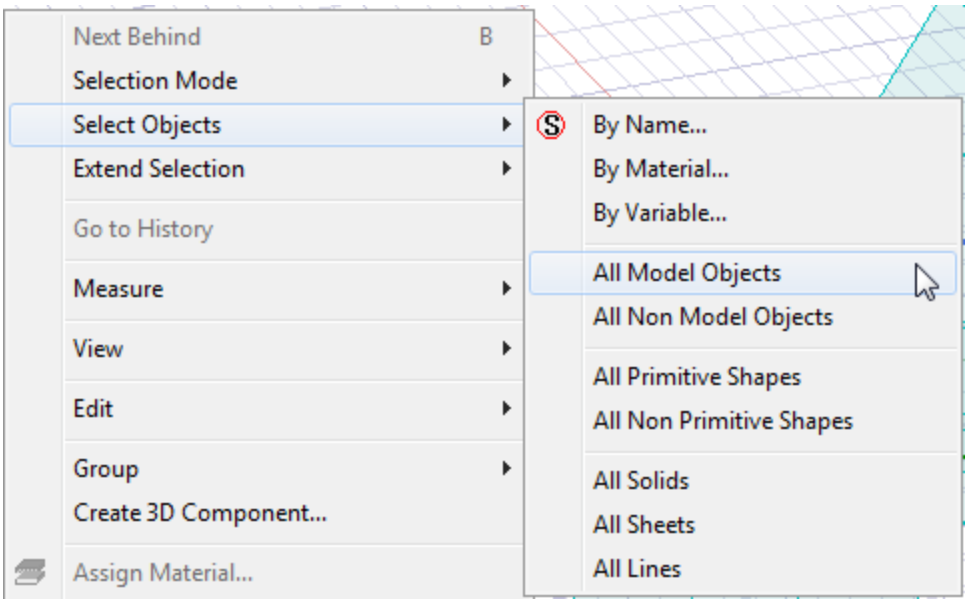


Selecting Several Objects

1. If you are selecting objects in the Modeler window make sure that the modeler is in object selection mode by pressing the **O** shortcut key. You can always select objects in the History Tree.
2. Select several objects in one of the following ways:
   - Hold down **Ctrl** and click the objects in the Modeler window that you want to select.
   - Hold down **Ctrl** and click the object names in the History Tree that you want to select.
   - In the History Tree, select a range of objects by first clicking one object to select it, and then Shift+click to extend the selection of visible items.
   - In the History Tree, under *Lists*, select **AllObjects**, which is an automatically created list of all model objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From the menu bar, click **Edit> Select All** to select all objects that were drawn in the active Modeler window, including objects that are not currently visible.
- Press **Ctrl+A** or, from the menu bar, click **Edit> Select All Visible** to select all visible objects in the active Modeler window.
- From the menu bar, click **Edit> Select Objects>** or right-click in the Model window and go to the **Select Objects>** submenu. Then, choose a command from the submenu suitable for the objects you want to select:



- Right-click a History Tree icon for Model, Group, object, category (such as Solid or Sheet), or material to perform context-sensitive selections. See Select Objects in the History Tree.

To deselect all objects, do one of the following:

- From the menu bar, click **Edit> Deselect All**.
- Press **Ctrl+Shift+A**.

Selecting Objects by Name or Material

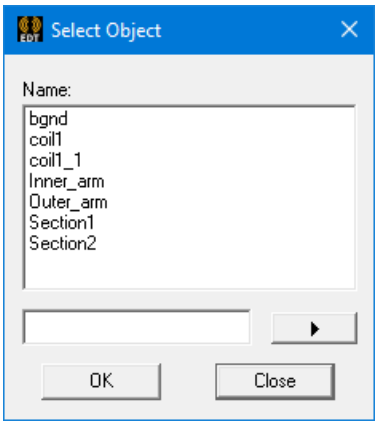
1. Make sure that the modeler is in the *Object* selection mode by pressing the **O** shortcut key or by choosing **Object** from the **Select** drop-down menu on the **Draw** or **Model** ribbon tab. Then, proceed to one of the following two procedures.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selection by Object Name

2. Access the *Select Object* dialog box using one of the following three methods:
   - From the menu bar, click **Edit** > **Select Objects** > **By Name**.
   - On the **Draw** or **Model** ribbon tab, click **Select by Name**.
   - Right-click in the *Modeler* window and choose **Select Objects** > **By Name** from the shortcut menu.

   The *Select Object* dialog box appears, populated with a list of the objects in the model, by name:



3. In the *Name* list, click the name of the object you want to select. Use the **Ctrl** key to select more than one.

   Alternatively, type the name of an object in the empty text box, click the arrow to the right of the text box, and choose **Select** or **Deselect**.

4. Click **OK**.

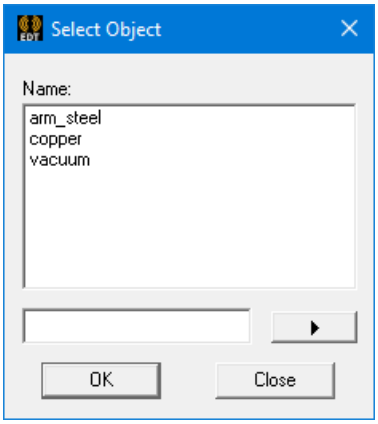
   The object or objects specified are selected.

Selection by Object Material

2. Access the *Select Object* dialog box using one of the following two methods:
   - From the menu bar, click **Edit** > **Select Objects** > **By Material**.
   - Right-click in the *Modeler* window and choose **Select Objects** > **By Material** from the shortcut menu.

   The *Select Object* dialog box appears, populated with a list of the materials comprising the model objects:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. In the *Name* list, click the material of the object or objects you want to select. Use the **Ctrl** key to select more than one material.

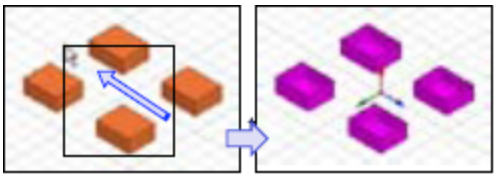
   Alternatively, type the name of a material in the empty text box, click the arrow to the right of the text box, and choose **Select** or **Deselect**.

4. Click **OK**.

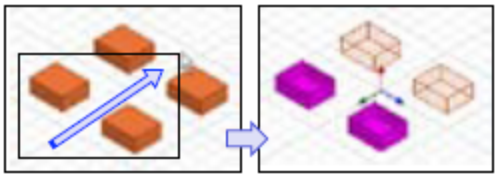
Select By Area

You can select items by area in the Modeler window by clicking and dragging a rubber band (selection box) around objects. Rubber band selection works differently depending on the direction you drag the mouse:

- **From right-to-left:** Selects all items that are wholly or partly enclosed within the rubber band.



- **From left-to-right:** Selects only the items that are wholly enclosed within the rubber band.

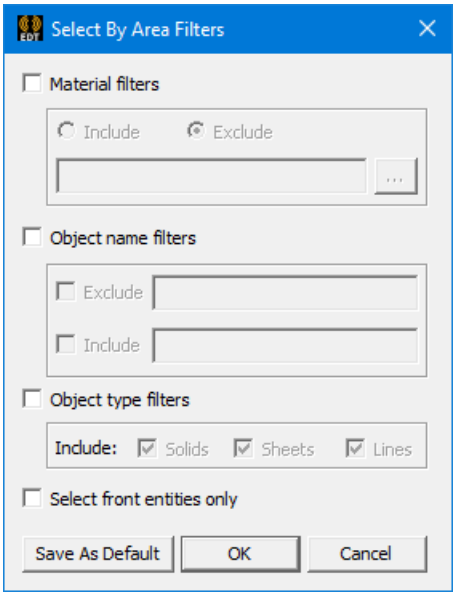


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Area selection works with Object, Face, Edge, and Vertex selection modes, but not for the Multi selection mode.

You can control which objects to include or exclude from area selections based on material, object name, or object type. To do this:

1. From the menu bar, click **Edit** > **Select By Area Filter** to display the following dialog box:



2. Check **Material filters** to enable the **Include** and **Exclude** radio buttons. Use the text box to specify filters by name, or use the ellipsis [...] button to display the *Select Definition* dialog box, from which you can browse and select materials.
3. Check **Object name filters** to enable the **Exclude** and **Include** check boxes and associated text boxes in which you can specify object names.
4. Check **Object type filters** to enable the *Include* check boxes for including **Solids, Sheets,** and/or **Lines**.
5. Check **Select front entities only** to prevent entities in the background (that is, those hidden behind other entities) from being selected.
6. If you click **Save As Default**, the settings persist for the project (until you change the settings and *Save as Default* again.
7. Click **OK** to close the dialog box.

   Now when you click and drag around an area, those objects which meet the filter criteria are highlighted in the Modeler window, and those objects are shown as selected in the History Tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Default Color and Transparency of Selected Objects

To set the color of objects when they are selected:

1. From the menu bar, click **Tools> Options> General Options** or, on the **Draw** ribbon tab, click **General Options**.

   The *Options* dialog box appears.

2. In the tree at the left side of the dialog box, select the **3D Modeler> Display> Rendering** group of settings.
3. In the *Rendering Defaults* section, choose **Select** from the **Default color** pull-down menu.
4. Click the color button to the right of *Select*.

   The *Color* palette appears.

5. Select a color and then click **OK**.

   Any objects you select after this point will temporarily become the default color you selected.

By default, the modeler shows selected objects as nearly opaque and shows non-selected objects as nearly transparent. This features helps you to easily distinguish between selected and non-selected objects and to see selected objects on the back side of the model.

To set the transparency of selected and non-selected objects:

1. From the menu bar, click **View> Options**.

   The *3D UI Options* dialog box appears. The *When there is a selection* region contains check boxes for setting the transparency for selected and non-selected objects.

2. Ensure that the check box is selected for any value you want to change.

   Each check box enables the associated value box. The default transparency for selected objects is 0.1, which makes them almost opaque. The default transparency for non-selected objects is 0.9, which makes them highly transparent.

3. Enter a new value, and click **OK**.

Setting the Default Color of Highlighted Objects

1. From the menu bar, click **Tools> Options> General Options** or, on the **Draw** ribbon tab, click **General Options**.
2. In the tree at the left side of the dialog box, select the **3D Modeler> Display> Rendering** group of settings.

   The *Options* dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the *Rendering Defaults* section, choose **Highlight** from the **Default color** pull-down menu.
4. Click the color button to the right of *Highlight*.

   The *Color* palette appears.

5. Select a color and then click **OK**.

   Outlines of all object you hover over temporarily become the default color you specified above.

Creating an Object List

Create an object list when you want to group multiple objects for convenient selection and processing. Creating an object list is a convenient way to identify and select a group of objects for a field plot or calculation. Objects in a list can still be treated as separate objects. The same object can be included in several different lists.

To create an object list:

1. If you are selecting in the Modeler window, make sure that the modeler is in object selection mode by pressing the **O** shortcut key. This is not necessary for selecting in the History Tree.
2. Select the objects you want to include in the list.
3. Click **Modeler> List> Create> Object List**.

   The object list is created with the default name Objectlist*n*. It appears in the History Tree under **Lists**. Selecting an object list displays the properties of that list in the docked *Properties* window. One of the properties is a list of objects contained in the list.

   To rename the Object list, edit its Name in the docked *Properties* window. Object lists are sorted in alphanumeric order.

The object list is treated as one volume when you plot and perform fields calculations. It will be listed in the *Geometry* window of the Fields Calculator when you select **Volume**.

There is an automatically created list called AllObjects. Select this list selects all objects in the model. If a list contains mixed types of geometry, for example, volume and sheet objects, the volume calculation only uses the geometry of the highest dimension in plots or integral, and so forth.

- Line objects in the list do not affect where the fields are saved. If all objects in the list are lines, fields will be saved on all objects.
- To post processing on line object when saving fields on object/face list, include both the line and its containing object/face in the list.

Example: To plot the E-field on a surface formed by the intersection of the XY-plane and several objects, first define a list of these objects. Then, when plotting fields, select the ***object list name***

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

from the **Geometry** window of the *Fields Calculator*. Fields will be plotted only at the intersection of the plane and the objects in the list.

Reassigning Objects to Another Object List

You can assign objects after you have created object lists. Creating an object list is a convenient way to identify and select a group of objects for a field plot or calculation. Objects in a list can still be treated as separate objects. The same object can be included in several different lists.

To reassign objects to an existing object list:

1. If you are selecting in the Modeler window, make sure that the modeler is in object selection mode by pressing the **O** shortcut key. This is not necessary for selecting in the History tree.
2. Select the objects you want to reassign.
3. Click **Modeler** > **List** > **Reassign**

   A dialog box with the existing object lists is displayed. (They appear in the History Tree under **Lists**.)

4. Select the list to which you want to assign the selected object(s) and click **OK**.

   The object is reassigned to the selected list, replacing previous list members. The *Objects* property of the List shows the objects contained in the list, with each name separated by a comma.

The object list will be treated as one volume when you are plotting and performing fields calculations. It will be listed in the *Geometry* window of the *Fields Calculator*, when you select **Volume**.

Using or Viewing List Objects

To view the objects included in an Object list:

1. In the History Tree, expand the **Lists** branch.
2. Right-click the ***list*** you want to select, and click **Select Assignment**.

   The objects that are included in that list are selected in the Modeler window, and the attributes appear in the docked *Properties* window.

To view the properties of the object list (including a list of the objects included):

1. In the History Tree, expand the **Lists** branch.
2. Under *Lists*, right-click the ***list*** you want to view and click **Properties**.

   The *Properties* dialog box appears. The objects included in the list appear as comma-separated names in the *Objects* row.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK** or **Cancel** to close the *Properties* dialog box.

To use an object from a list in another operation:

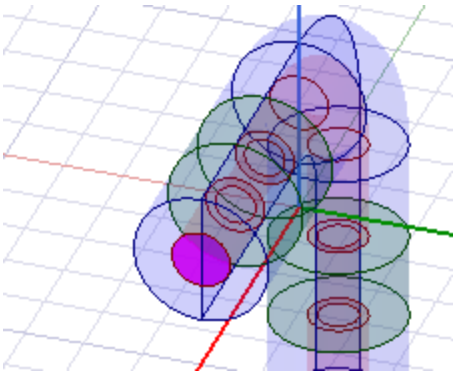
1. In the History Tree, expand the **Lists** branch.
2. Right-click the ***list*** you want to select and click **Select Assignment**.

   The objects that are included in that list are selected in the Modeler window, and the attributes appear in the docked *Properties* window.
3. Holding **Ctrl**, select any additional objects you might want to use in the operation.
4. Complete the operation.

   For example, you could select an object list and an additional object, and then execute one of the Boolean commands (such as *Unite* or *Subtract*).

Selecting Faces

If the modeler is in face selection mode, click an object face in the view window to select it. The selected face is rendered in the section color (magenta by default) and is relatively opaque by default. All other objects become relatively transparent.



When the mouse hovers over a face in the view window, that face is highlighted, which indicates that it will be selected when you click. Selected faces become the color specified under the *3D Modeler> Display> Rendering* section the *General Options*.

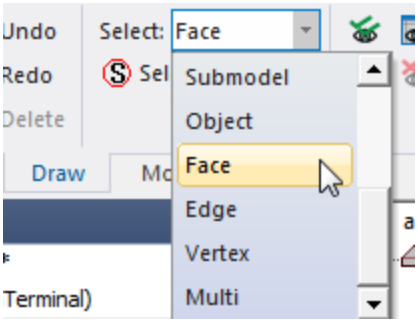
To select multiple faces, hold the **Ctrl** key as you click the faces. You also have the option to create face lists, which define a list of object faces, or you can make face selections from a Face ID list in the Select Face dialog box.

Switch to face selection mode using one of the following methods:

- From the menu bar, click **Edit> Selection Mode> Select Faces**
- Right-click in the Modeler window and choose **Selection Mode> Select Faces** from the shortcut menu.

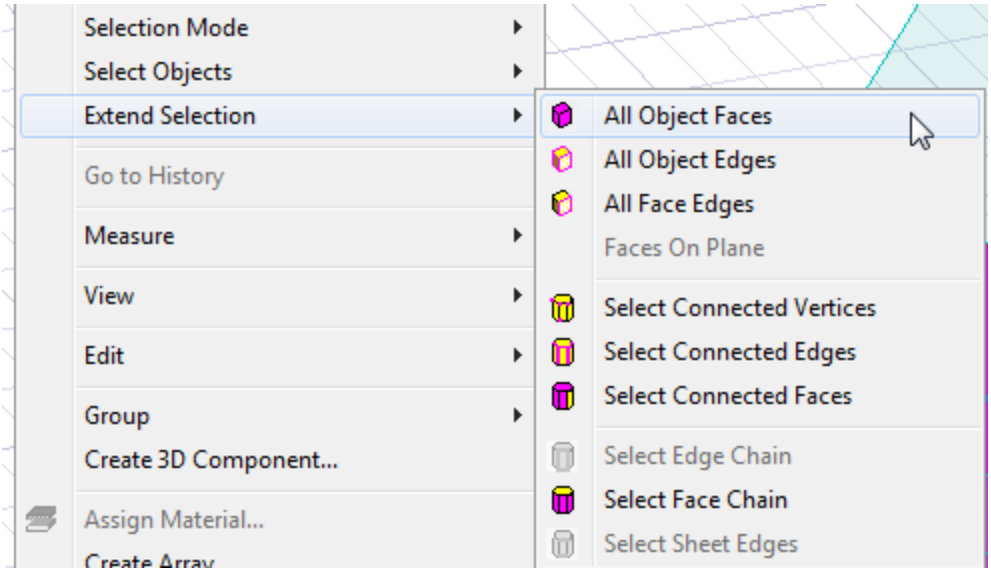
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- With the Modeler window active, press the **F** shortcut key.
- On the **Draw** ribbon tab, choose **Faces** from the **Select** drop-down menu.



You can also select faces in the Select Multi mode.

When you've selected a face, the menu bar's **Edit**> **Extend Selection** submenu and the **Extend Selection** submenu in the Modeler window's shortcut menu show the following options:



Various tools are available for face selection, as described in the *Related Topics* listed below.

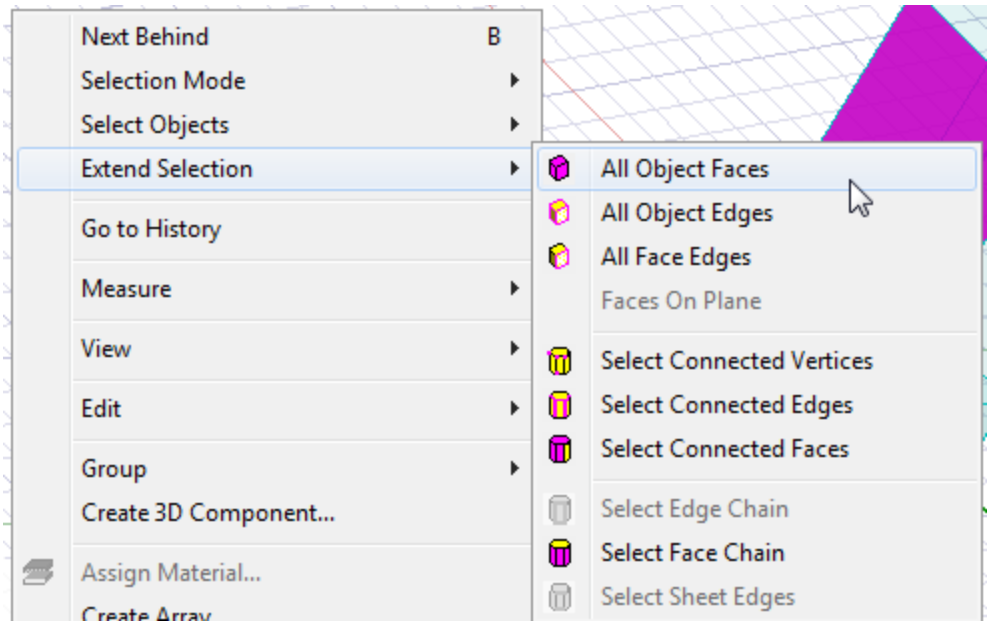
Selecting All Faces of an Object

1. With the Modeler window active, switch to the face selection mode by pressing the **F** shortcut key.
2. Click a face on the object of interest to select it.

   You can select a face on each of two or more objects if desired.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

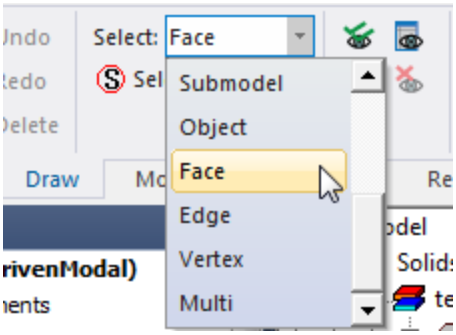
3. Use one of the following methods of selecting all faces of the objects of interest:
   - From the menu bar, click **Edit> Extend Selection> All Object Faces.**
   - Right-click in the Modeler window and choose **Extend Selection> All Object Faces** from the shortcut menu:



The selection is extended to include all faces of each object that had at least one face selected originally.

Selecting Faces by Name

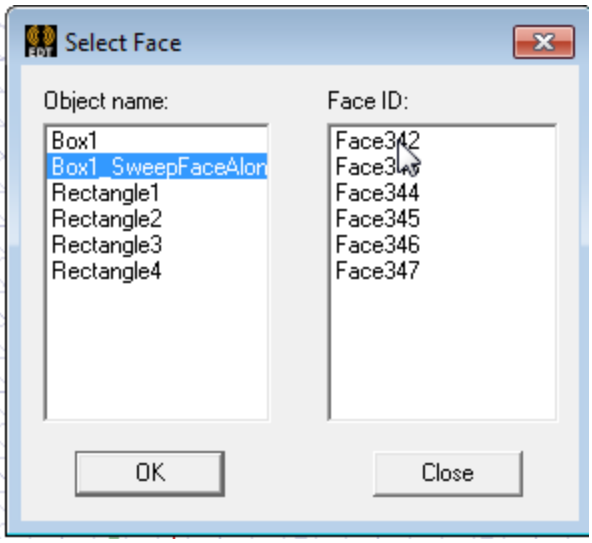
1. Make sure that the modeler is in face selection mode using one of the following methods:
   - With the Modeler window active, press the **F** shortcut key.
   - From the menu bar, click **Edit> Selection Mode> Faces**
   - On the **Draw** or **Model** ribbon tabs, select **Face** from the **Select** drop-down menu.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. From the menu bar, click **Edit> Select Objects> By Name** or, on the **Draw** or **Model** ribbon tabs, click **Select by Name**.

   The *Select Face* dialog box appears:



3. In the *Object name* box, click the name of the object with the face or faces you want to select.

   The object's faces are listed in the *Face ID* box.

4. Click the ***face*** you want to select in the *Face ID* column. Use **Ctrl+click** to select more than one face or **Shift+click** to select a range of faces.

   The selected faces are highlighted in the list and in the Modeler window.

5. Click **OK**.

Selecting Faces by Plane

To select a face that is aligned with a global plane, use one of the following two methods.

Method 1:

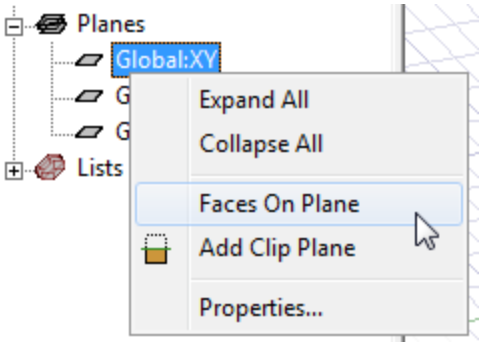
1. Make sure that the modeler is in face selection mode by pressing the **F** shortcut key.
2. In the History Tree, expand the **Planes** branch and select a plane to display (a global plane, a user-defined plane, or a plane associated with a relative coordinate system).
3. From the menu bar, click **Edit> Extend Selection> Faces on Plane** or right-click in the Modeler window and choose **Extend Selection> Faces on Plane**.

   All faces that lie on the chosen plane are selected.

Method 2:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the History Tree, expand the **Planes** branch.
2. Right-click one of the listed planes (a global plane, a user-defined plane, or a plane associated with a relative coordinate system).
3. On the shortcut menu, click **Faces on Plane**.



All faces that lie on the chosen plane are selected.

Creating a Face List

Create a face list when you when you want to group multiple faces for convenient selection and processing. Creating a face list is a convenient way to identify and select a specific set of surfaces for field plots, calculations, boundaries or excitations. The same face can be included in several different lists.

To create a face list:

1. Make sure that the modeler is in face selection mode by pressing the **F** shortcut key.
2. Select the object faces you want to include in the face list.
3. From the menu bar, click **Modeler** > **List** > **Create** > **Face List**.

   The face list is created. It is listed in the History Tree under **Lists**. The default name is Facelist*n*. The lists appear in alphanumeric order.

4. Optionally, to change the name of a face list (for example, to a name describing the listed faces as ports or boundaries), select the list in the History Tree and change the **Name** value in the docked *Properties* window.

   If necessary, the lists in the History Tree are resorted to restore the proper alphanumeric order.

The face list will be treated as one selection of surfaces when you are plotting and performing fields calculations. The face list will be listed in the *Geometry* window of the *Fields Calculator*, when you select the *Surface* option.

- Line objects in the list do not affect where the fields are saved. If all objects in the list are lines, fields will be saved on all objects.

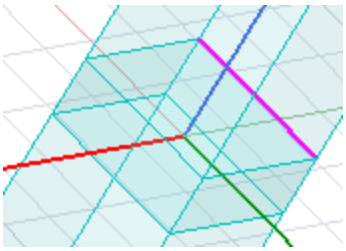
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To post processing on line object when saving fields on object/face list, include both the line and its containing object/face in the list.

If you right-click an existing face list and choose **Select Assignment** from the shortcut menu, you can make boundary assignments or execute the **Move Faces** geometry operations for all faces in the list.

Selecting Edges

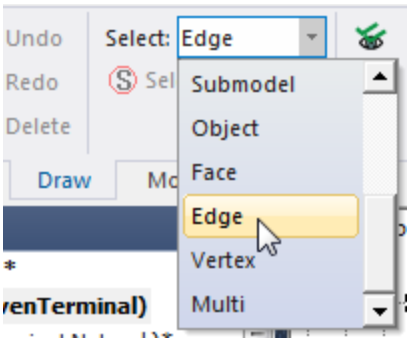
If the modeler is in edge selection mode, simply click an object's edge in the view window and it will be selected. The edge is rendered as nearly opaque using the default *Select* color. All other objects become relatively transparent.



When the mouse hovers over an edge in the view window, that edge is highlighted, which indicates that it will be selected when you click.

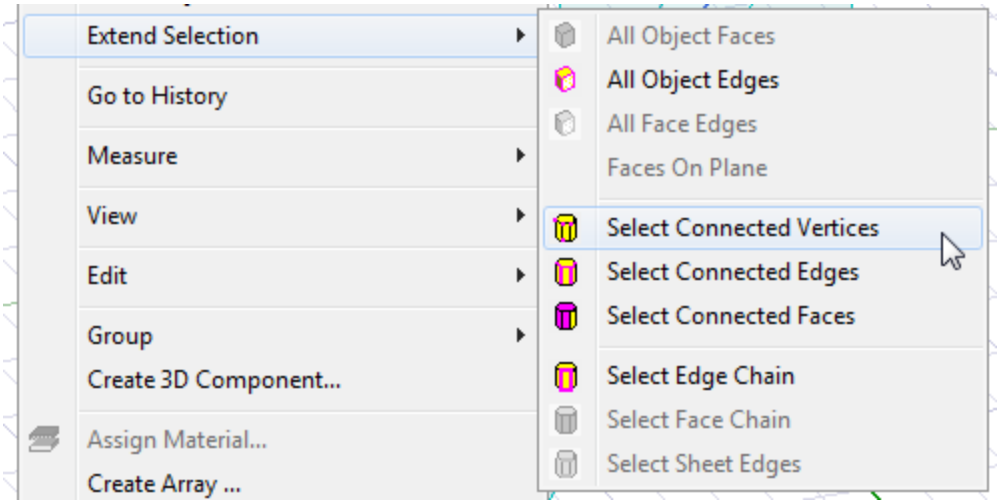
Switch to edge selection mode using one of the following methods:

- From the menu bar, click **Edit> Selection Mode> Edges**.
- Right-click in the Modeler window and click **Selection Mode> Edges**.
- With the Modeler window active, press the **E** shortcut key.
- On the **Draw** or **Model** ribbon tab, choose **Edge** from the **Select** drop-down menu.



Selecting an edge enables the following commands in the menu bar's **Edit> Extend Selection>...**submenu and the **Extend Selection>...** submenu that appears when you right-click in the Modeler window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can use these commands to modify the current selection.

- All Object Edges
- Select Connected Vertices
- Select Connected Edges
- Select Connected Faces
- Select Edge Chain

You can also select multiple edges by holding the **Ctrl** key as you click additional edges.

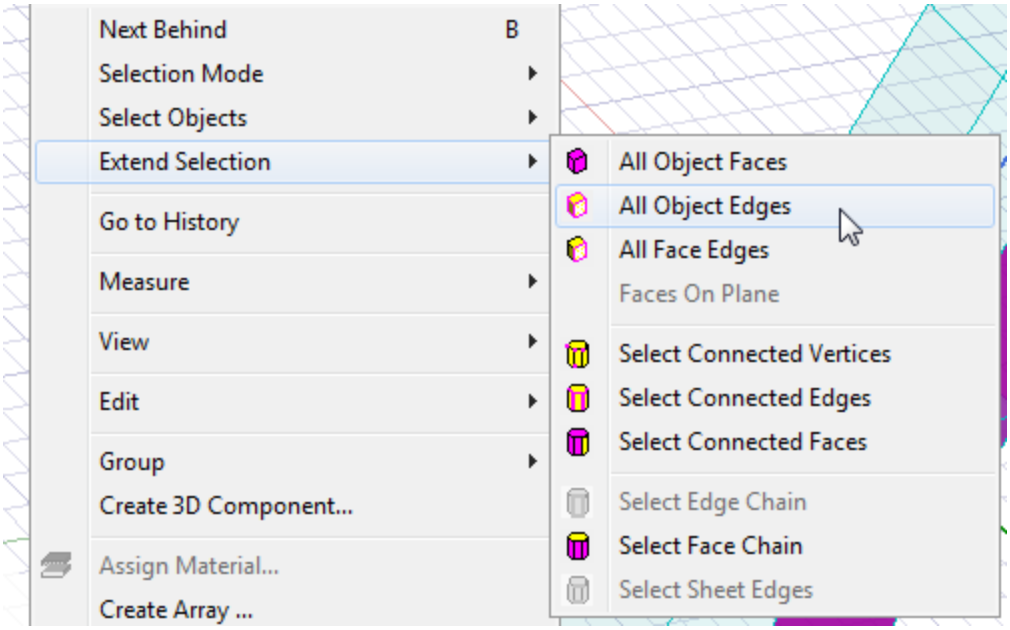
Selecting All Edges of an Object or Face

To select all edges of an object or face:

1. Select a single edge or face that belongs to the object for which you wish to select all edges.

   You may also select the object itself. The type of entity you initially choose affects which selection tools will be available in the next step:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. If you selected an **object** or an **edge** in step 1, the following *All Edge* command is available:
   - From the menu bar click **Edit> Extend Selection> All Object Edges** or right-click in the Modeler window and choose **Extend Selection> All Object Edges** from the shortcut menu.

     All edges of the object are selected.
3. If you selected a **face** in step 1, the following two *All Edge* commands are available:
   - From the menu bar click **Edit> Extend Selection> All Face Edges** or right-click in the Modeler window and choose **Extend Selection> All Face Edges** from the shortcut menu.

     All edges belonging to the face are selected.
   - From the menu bar click **Edit> Extend Selection> All Object Edges** or right-click in the Modeler window and choose **Extend Selection>All Face Edges** from the shortcut menu.

     All edges of the object to which the face belongs are selected.

If you selected multiple objects, all edges of those objects are selected.

Selecting Vertices

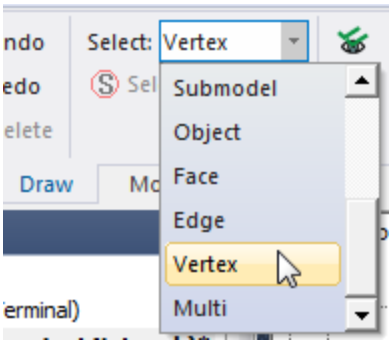
If the modeler is in vertex selection mode, simply click an object's vertex in the view window and it will be selected. To select multiple vertices, hold the **Ctrl** key as you click the vertices.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

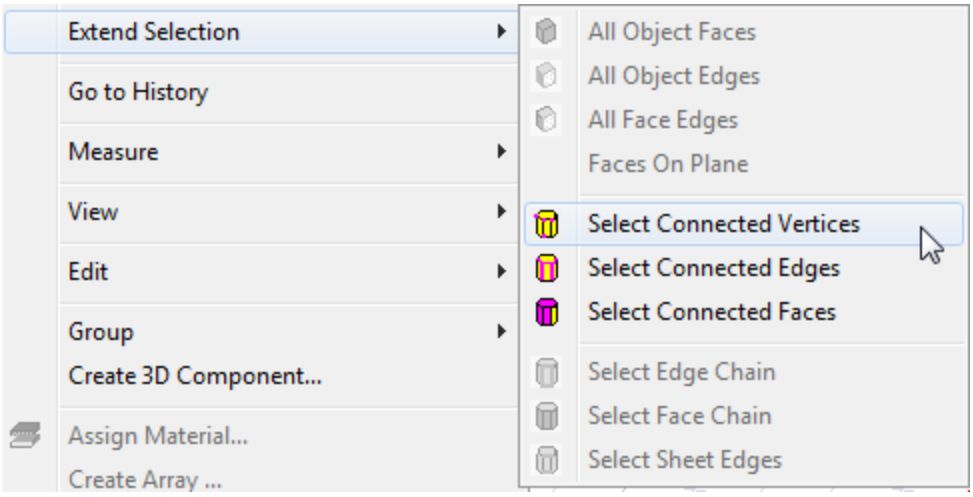
When the mouse hovers over a vertex in the view window, that vertex is highlighted, which indicates that it will be selected when you click. Selected vertices are rendered as using the default *Select* color. All other objects become relatively transparent.

Switch to vertex selection mode using one of the following methods:

- From the menu bar, click **Edit**> **Selection Mode**> **Vertices**.
- Right-click in the Modeler window and click **Selection Mode**> **Vertices**.
- With the Modeler window active, press the **V** shortcut key to begin the vertex selection mode.
- On the **Draw** or **Model** ribbon tabs, choose **Vertex** from the **Select**drop-down menu.



Selecting a vertex enables the following commands in the menu bar's **Edit> Extend Selection>...** submenu and the **Extend Selection>...** submenu that appears when you right-click in the Modeler window.



You can use these commands to modify the current selection:

- **Select Connected Vertices:** Selects the vertices of all edges that touch the initially selected vertices.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

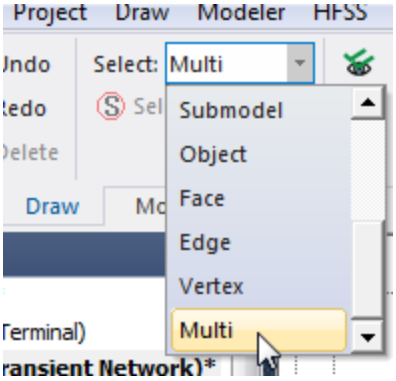
- **Select Connected Edges:** Selects all edges that touch the initially selected vertices.
- **Select Connected Faces:** selects all faces touching the initially selected vertices.

> **Note:**
>
> Multiple vertices can exist at the same coordinates, one vertex for each object that intersects at that point. There's no visual way to discern to which object a selected vertex belongs. If you click a vertex and use any of the commands listed above, only the entities that belong to the same object will be selected. To select the associated entities for all objects, start by box-selecting the vertices at the point of interest (clicking and dragging from left-to-right). This method selects all coincident vertices at the point. Then, use one of the above three commands.
>
> You can also pick a vertex and press **B** (*Next Behind*) to select a different vertex at the point of interest. Use the trial and error method to select the related vertices, edges, or faces of the desired object. For example, you may have to press *B* two or more times to select the vertex of the desired object.

Selecting Multi

The Select Multi mode permits you to select a mixture of objects, faces, vertices, or edges, depending on where you click. This mode can be very useful in conjunction with Measure Mode, for measuring the distances between different entities. Enter the *Multi* selection mode using one of the following methods:

- With the Modeler window active, press the **M** shortcut key.
- Right-click in the Modeler window and choose **Selection Mode**> **Multi** from the shortcut menu.
- From the menu bar, click **Edit**> **Selection Mode**> **Multi**.
- On the **Draw** or **Model** ribbon tabs, choose **Multi** from the **Select** drop-down menu.

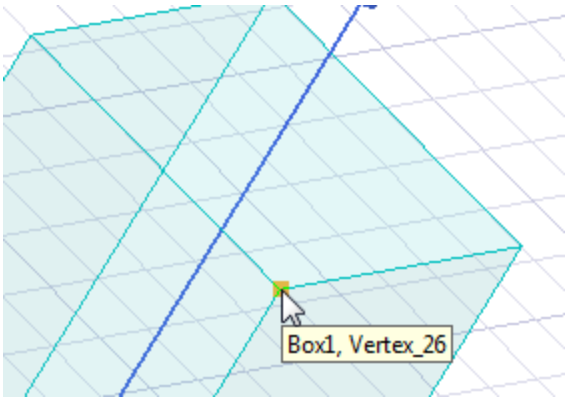


With Multi mode active:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To select a vertex, click near a vertex, within 10 pixel radius.
- To select an edge, click near an edge (and at least 10 pixels away from any vertex).
- To select an object, click little farther from any edge (between 10 and 20 pixels away).
- To select a face, click anywhere else on the interior of a face.

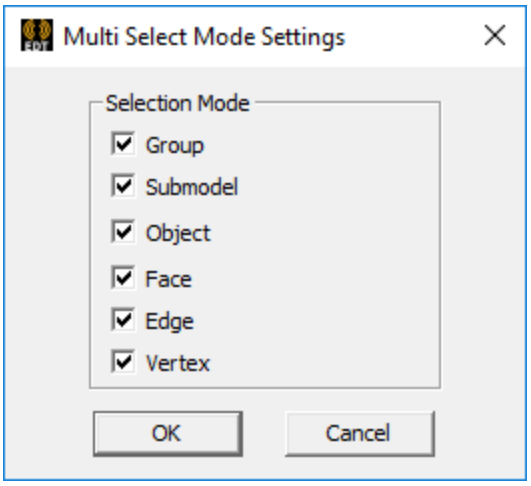
As you hover the cursor over an entity, a Tooltip indicates the type/ID of the entity (object name in the case of objects, Face_id in the case of faces, and so on). For example, this feature helps you to distinguish between the selection of the face of a sheet object versus a sheet object.



By holding down the **Ctrl** key, you can make multiple selections.

Controlling the Selection in Multi Mode

You can control the behavior of this selection mode. From the menu bar, click **Edit> Selection Mode> Multi Mode Settings**. This action displays a dialog box with check boxes for enabling the selection of specific types of entities (Group, Submodel, Object, Face, Edge, and Vertex):



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clear a box filters out the associated item, preventing that type of entity from being selected while the *Multi* selection mode is active.

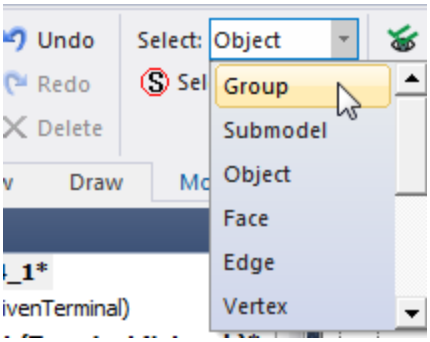
Selecting Groups and Submodels

With the *Group* selection mode, the parent group of selected object is selected. Selecting a group highlights all children under the group in the Modeler window. If you click an object that has not been assigned to a group, nothing is selected. To select multiple groups, hold the **Ctrl** key as you click the groups.

The *Group* selection mode is only available when the model contains at least one defined group.

Use one of the following methods of setting the selection mode to *Group*:

- From the menu bar, click **Edit**> **Selection Mode**> **Groups**.
- With the Modeler window active, press the **G** shortcut key.
- Right-click in the Modeler window and choose **Selection Mode**> **Groups** from the shortcut menu.
- On the **Draw** or **Model** ribbon tabs, choose **Group** from the **Select** drop-down menu. You may have to scroll up to see this option.



If your design contains submodels, you can set the selection mode to *Submodel*:

- From the menu bar, click **Edit**> **Selection Mode**> **Submodels**.
- With the Modeler window active, press the **U** shortcut key.
- Right-click in the Modeler window and choose **Selection Mode**> **Submodels** from the shortcut menu.
- On the **Draw** or **Model** ribbon tabs, choose **Submodel** from the **Select** drop-down menu. You may have to scroll up to see this option.

Clearing a Selection

To clear an object, face, edge, or vertex selection, do one of the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click in an empty area of the Modeler window.
- To clear an object selection, click a point away from the object name in the History Tree.
- From the menu bar, click **Edit> Deselect All**.
- Press **Shift+Ctrl+A**.

The items are no longer selected.

Selecting the Face, Edge, Vertex, or Object Behind

To select the face, edge, vertex, or object, behind another selected face, edge, vertex, or object, do one of the following:

- From the menu bar, click **Edit**> **Next Behind**.
- Right-click in the Modeler window and choose **Next Behind** from the shortcut menu.
- Press the **B** shortcut key.
- Press **Ctrl+B**.

> **Note:**
>
> The desired entity must be behind the point where you clicked, not just behind the initially selected face, edge, vertex, or object.

This option is useful when you are trying to select a face, edge, vertex, or object that is in the interior of a model, or when you do not want to change the model viewpoint to select an item.

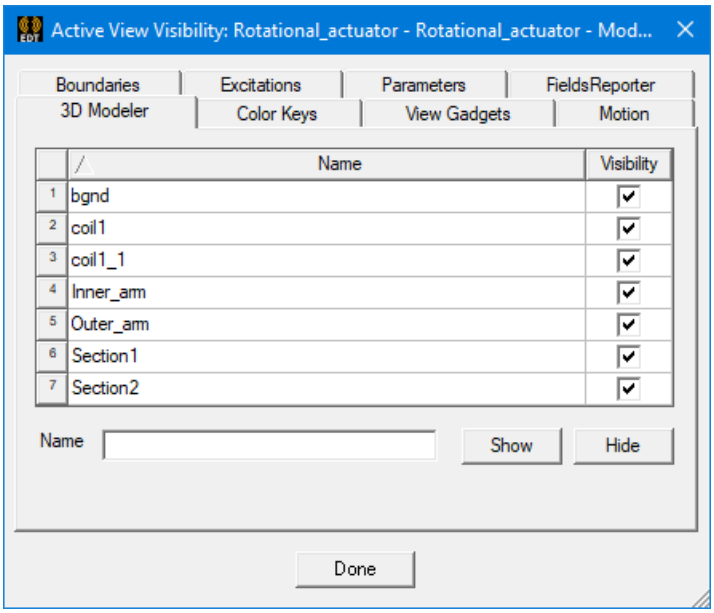
Selecting Objects and Surfaces That Lie Inside Other Objects

To select objects and faces that lie inside other objects (such as an object that lies within an air box, a conductive shield, or the background object), do one of the following:

- From the menu bar, **View** > **Visibility** > **Active View Visibility...** to access the *Active View Visibility* dialog box. Then, under the *3D Modeler* tab, use the check boxes in the **Visibility** column to control which objects are visible and which are hidden.

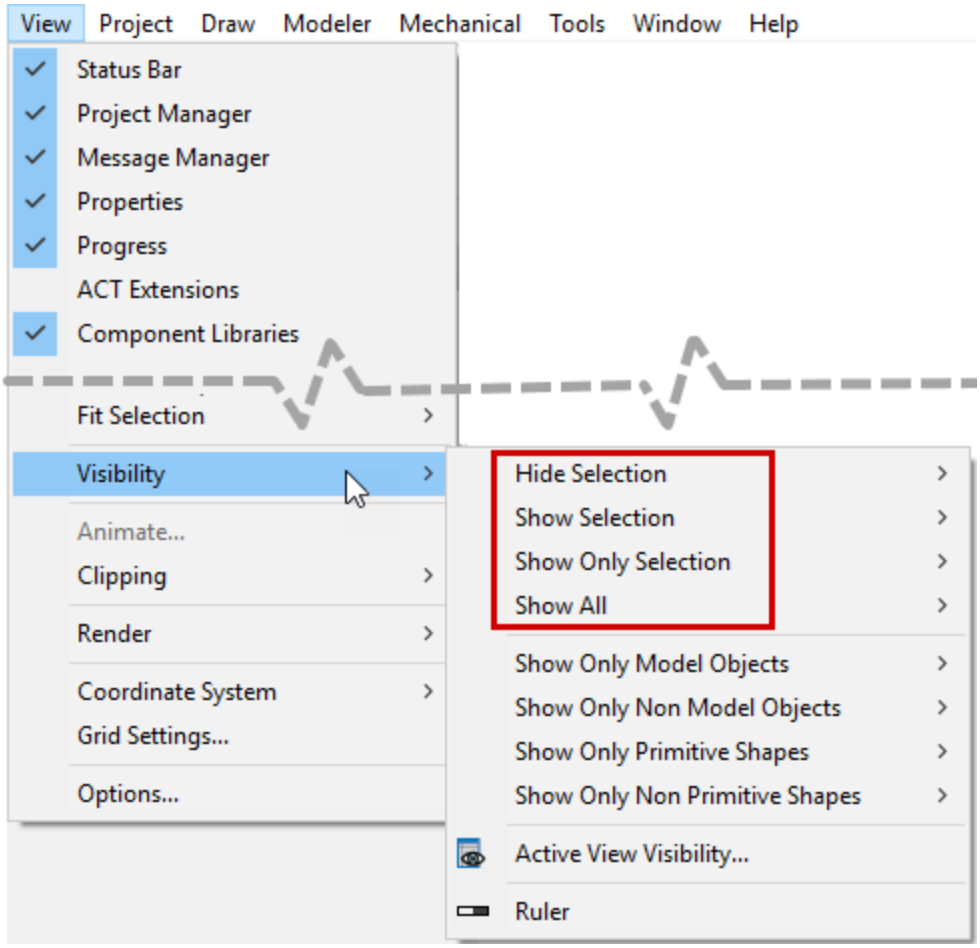
  This method, and the method that follows, is useful when you want to select objects using the mouse, since the mouse cannot select invisible objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

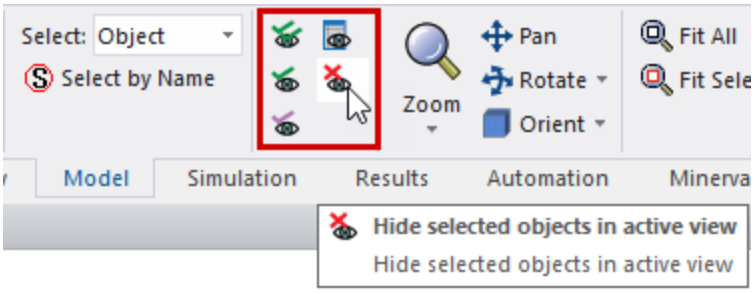


- Make the objects on the outside of the model invisible using the menu bar's other **View** > **Visibility** > commands:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



or the same commands on the **View, Draw,** or **Model** ribbon tabs:



- Use the **Select by Name** command to select Objects or Faces inside the model.
- Use the **Next Behind** command in the *Edit* menu or Modeler shortcut menu, or use the **B** shortcut key. This command selects the object that lies behind the one you initially selected. This command does nothing if no objects have previously been selected or if no object is behind the one you initially selected at the location you clicked.

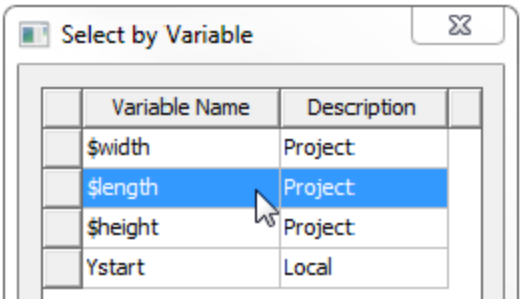
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting Objects by Variable

You can select an object based on a variable that affects it. If your design includes variables, do so as follows:

1. From the menu bar, click **Edit> Select Objects> By Variable**.

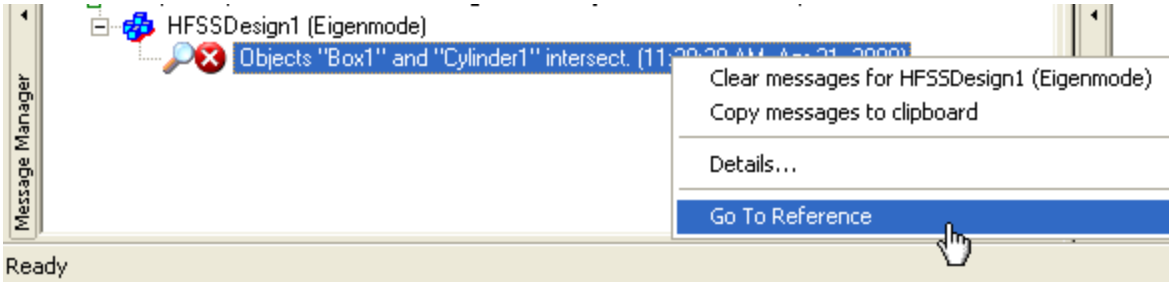
   The *Select by Variable* dialog box appears, which lists the variables in your design. Both columns are sortable by clicking the header. You can resize and move the dialog box. When you next open it, it remembers that size and location.



2. Select the variable of interest and click **OK**.

   The dialog box closes, and the object affected by the variable is highlighted in the Modeler window.

   The Message window contains a reference that you can select and use to go to the affected object.



   If you execute the command again, without clearing the current selection(s), the additional object can be highlighted. You can resize and move the dialog.

Assigning Coordinates to New Objects

When you insert and draw new 3D, 2D, or 1D objects in model, you may need to define coordinate systems, locations of points, distances between objects, and other geometry items.

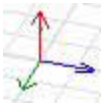
- Setting the Reference Point
- Defining Cartesian Coordinates

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Defining Spherical Coordinates
- Defining Relative Coordinates
- Defining Absolute Coordinates

Setting the Reference Point

When you draw objects, the cursor's location is always relative to a reference point. The reference point is displayed with a mini xyz-axis:



To change the reference point:

1. Select the drawing command to use.

   The Measure Data dialog box opens. As you move the cursor over the Modeler window, the top line in the *Measure Data* dialog box shows the coordinates of the current reference point.

2. Move the cursor to the desired reference point and press **Ctrl+click** or right-click and select **Set Reference Point** from the short cut menu.

   This action moves the reference point marker to the new location. The *Measure Data* dialog box updates. The coordinates boxes in the Status Bar change to accept Relative distance information. If you prefer, rather than setting the reference point with the cursor, you can press **Tab** to jump to a coordinate box in the status bar, where you can specify the coordinates numerically. Navigate among the three coordinate boxes by pressing the **Tab** key again.

Defining Cartesian Coordinates

When you draw an object, define a point using Cartesian coordinates by typing the point's x, y, and z distances from the origin in the **X, Y,** and **Z** text boxes, respectively. When defining a second point, specify its x, y, and z distances from the previously selected point in the **dX, dY,** and **dZ** text boxes, respectively.

1. Select the desired drawing command.
2. Select **Cartesian** from the drop-down menu in the Status Bar.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Type the point's x, y, and z coordinates in the **X, Y,** and **Z** text boxes.

   > **Tip:**
   >
   > Press **Tab** to move from one coordinate text box to the next. Press **Ctrl+Tab** to move to the previous coordinate text box.

   - Alternatively, click the point in the Modeler window.
4. When drawing objects other than polylines and helices, the second point you select is relative to the first point. Type the second point's x, y, and z distances from the previously selected point in the **dX, dY,** and **dZ** text boxes, respectively.

Defining Cylindrical Coordinates

To define a point using cylindrical coordinates, specify the point's radius, measured from the origin, in the **R** text box, the angle from the x-axis in the **Theta** text box, and the distance from the origin in the z direction in the **Z** text box. When defining a second point, specify its distance from the previously selected point in the in **dR, dTheta,** and **dZ** text boxes.

1. After clicking the desired drawing command, select **Cylindrical** from the pull-down menu in the Status Bar.
2. Type the point's r, theta, and z coordinates in the **R, Theta,** and **Z** boxes.

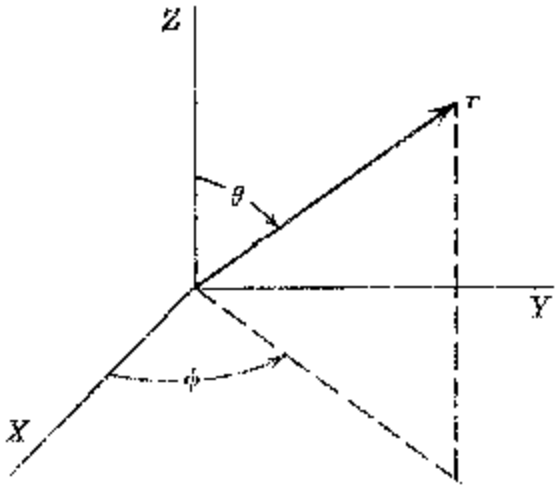
   > **Tip:**
   >
   > Press **Tab** to move from one coordinate text box to the next. Press **Ctrl+Tab** to move to the previous coordinate text box.

   - Alternatively, click the point in the Modeler window.
3. When drawing objects other than polylines and helices, the second point you select is relative to the first point. Type the second point's distance from the previously selected point in the **dR, dTheta,** and **dZ** text boxes.

Defining Spherical Coordinates

To define a point in spherical coordinates, specify the point's radius, measured from the origin, in the **Rho** text box, the angle from the positive z-axis in the **Theta** text box, and the angle from the positive x axis in the **Phi** text box. When selecting a second point, specify its distance from the previously selected point in the in **dRho, dTheta,** and **dPhi** text boxes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> Even though you are inputting spherical coordinates, all data is internally stored in Cartesian coordinates.

To define a point in spherical coordinates:

1. After clicking the desired drawing command, select **Spherical** from the drop-down menu in the status bar.
2. Type the point's r, theta, and phi coordinates in the **Rho, Theta,** and **Phi** text boxes in the Status Bar.

   > **Tip:**
   >
   > Press **Tab** to move from one coordinate text box to the next. Press **Ctrl+Tab** to move to the previous coordinate text box.

   - Alternatively, click the point in the Modeler window.
3. When drawing objects other than polylines and helices, the second point you select is relative to the first point. Type the second point's distance from the previously selected point in the **dRho, dTheta,** and **dPhi** text boxes.

Using Absolute Coordinates

When entering a point's location, you can specify it using *absolute* or *relative* coordinates. Absolute coordinates are relative to the working coordinate system's origin (0, 0, 0). This entry mode is the default setting for the first point you select after clicking a drawing command. Relative coordinates are relative to the reference point, or the previously selected point.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To enter a point's absolute coordinates:

1. Click the desired drawing command.
2. Select **Absolute** from the **Absolute/Relative** drop-down menu in the Status Bar.
3. Specify the point's coordinates in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the appropriate text boxes in the Status Bar.

> **Note:**
>
> When drawing objects other than polylines and helices, by default, the second point you select is relative to the first point; **Relative** is automatically selected in the **Absolute/Relative** drop-down menu in the Status Bar. Be sure to select **Absolute** from this drop-down menu if you want the second point to be relative the origin of the working coordinate system.

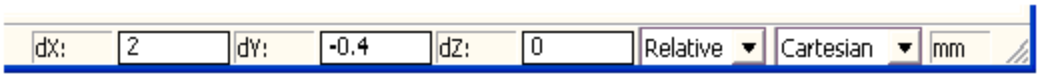
Using Relative Coordinates

When entering a point's location, you can specify it using *absolute* or *relative* coordinates. Relative coordinates are relative to the reference point or the previously selected point. Absolute coordinates are relative to the working coordinate system's origin (0, 0, 0).

To enter a point's relative coordinates:

1. Click the desired drawing command.
2. Select **Relative** from the **Absolute/Relative** drop-down menu in the Status Bar.

   When you are in *Relative* mode, the text boxes for a coordinate show an "d" before the coordinate description, to indicate "distance from" the working reference. For example:

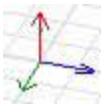


3. Specify the point's coordinates in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the appropriate text boxes in the Status Bar.

Choosing the Movement Mode

When drawing objects, the cursor's location is always relative to a reference point. The reference point is displayed with a mini xyz-axis when you click a drawing command:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To change the reference point, move the cursor to the desired point and **Ctrl+click** the point.

When creating 1D, 2D, or 3D geometry graphically (that is, by clicking points instead of by defining coordinates numerically), points that you can select are dependent on the current *Movement Mode* setting. You can move the cursor and place a point at one of the following locations:

- In the same plane as the reference point (In Plane movement mode)
- Perpendicular to the reference point (Out of Plane movement mode)
- If an object is present, you can snap to one of its points – any snapable point in 3D space (3D movement mode).
- Along the X-axis of the working coordinate system (CS)
- Along the Y-axis of the working CS
- Along the Z-axis of the working CS

Drawing commands that you choose may temporarily override the current movement mode. For example, when you draw a box or cylinder, the first two points you click (defining the base) may be in the drawing plane, but the third (defining the height) must be out of plane relative to the base. In this case, you do not have to manually change the movement mode; it is automatically overridden for the third click.

> **Note:**
>
> For all six movement modes covered in this section, the following rules apply regarding mode persistence:
>
> - If you change the movement mode *before* executing a drawing command, the new mode setting will be retained until you change it again.
> - If you change the movement mode *after* executing a drawing command, it will be treated as a temporary override. Upon completion of the drawing operation, the mode will revert to the previous setting.

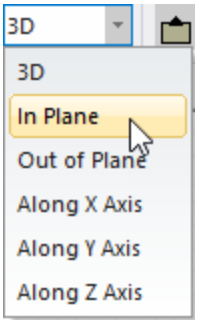
Moving the Cursor In Plane

Use the *In Plane* movement mode to specify points of objects being drawn that are *on the same plane* as the reference point:

Before or after clicking a desired drawing command, change the movement mode using one of the following methods:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

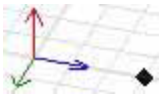
- From the menu bar, click **3D Model**> **Movement Mode**> **In Plane**.
- On the **Draw** ribbon tab, choose **In Plane** from the **Movement mode** drop-down menu:



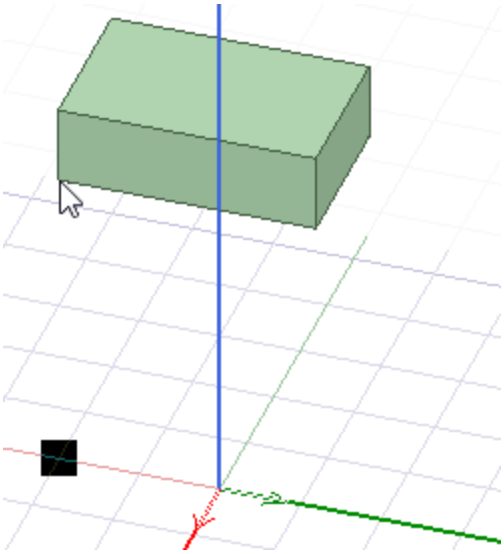
The following additional method is available only after clicking a drawing command:

- Right-click in the Modeler window and choose **Movement Mode**> **In Plane** from the shortcut menu.

The next point you select will be on the same plane as the reference point. The cursor's location is displayed with a black diamond that indicates where it will snap to the drawing plane grid.



If you click a snapping point on an object in 3D space (that is, one not on the drawing plane), the point location is projected to the drawing plane along a perpendicular vector. In the following example, a filled black square indicates the projected snapping point:



The symbol used for projected point matches the type of snapping point on the object (vertex, edge center, face center, and so on).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

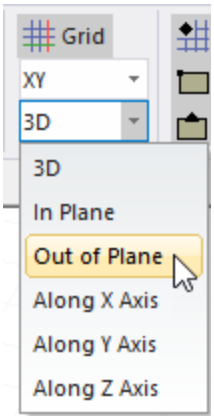
The *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

Moving the Cursor Out of Plane

Use the *Out of Plane* movement mode to specify points of objects being drawn that are along a vector *perpendicular* to the drawing plane and intersecting it at the reference point:

Before or after clicking a desired drawing command, change the movement mode using one of the following methods:

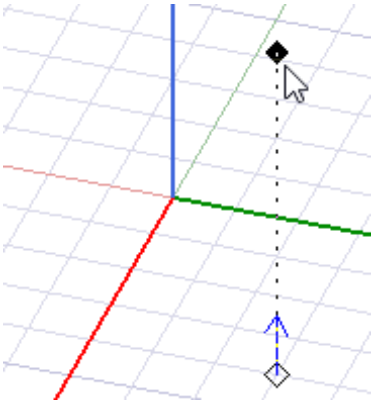
- From the menu bar, click **3D Model**> **Movement Mode**> **Out of Plane**.
- On the **Draw** ribbon tab, choose **Out of Plane** from the **Movement mode** drop-down menu:



The following additional method is available only after clicking a drawing command:

- Right-click in the Modeler window and choose **Movement Mode**> **Out of Plane** from the shortcut menu.

A dashed line perpendicular to the drawing plane is displayed, and a filled black diamond indicates the cursor's location:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

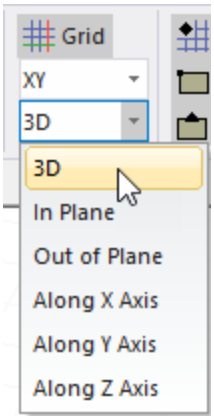
The *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

Moving the Cursor in 3D Space

To snap to drawing plane grid points or to object snapping points anywhere in 3D space, use the *3D* movement mode.

Before or after clicking a desired drawing command, change the movement mode using one of the following methods:

- From the menu bar, click **3D Model**> **Movement Mode**> **3D**.
- On the **Draw** ribbon tab, choose **3D** from the **Movement mode** drop-down menu:

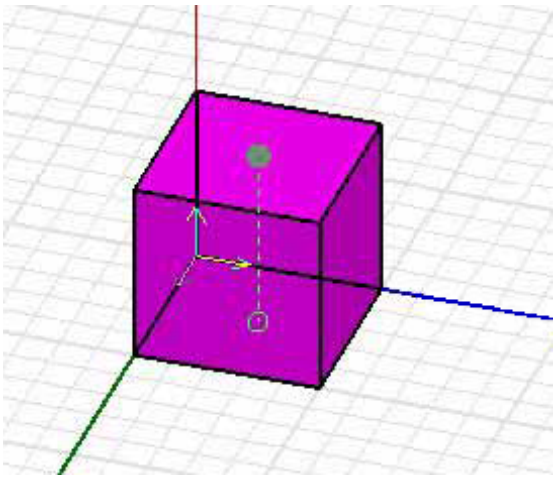


The following additional method is available only after clicking a drawing command:

- Right-click in the Modeler window and click **Movement Mode**> **3D** from the shortcut menu.

If one of an object's snapping points (vertex, edge center, face center, and so on) is within snapping range of the cursor, the geometry you are drawing will snap to that point. The symbol will indicate the type of snapping point. In the following example, the cursor will snap to the face center point of the boxes top face:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If an object is not within snapping range of the cursor, the *3D* movement mode is identical to the *In Plane* mode. That is, the geometry being drawn will snap to a grid point on the drawing plane.

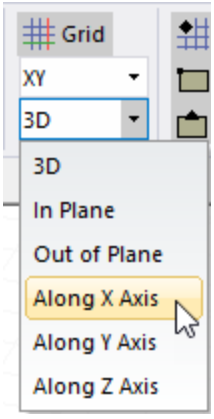
In either case, the *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

Moving the Cursor Along the X-Axis

Use this movement mode to constrain the cursor to a moving only in the +/- X direction relative to the reference point. Movement is constrained to the X direction as defined by the working coordinate system, not necessarily the global X direction. The line of motion passes through the reference point, so it may be parallel to the X-Axis and not necessarily along the actual X-axis.

Before or after clicking a desired drawing command, change the movement mode using one of the following methods:

- From the menu bar, click **3D Model**> **Movement Mode**> **Along X Axis**.
- On the **Draw** ribbon tab, choose **Along X Axis** from the **Movement mode** drop-down menu:



The following additional methods are available only after clicking a drawing command:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Press and hold the **X** key while clicking the next point.
- Right-click in the Modeler window and choose **Movement Mode**> **Along X Axis** from the shortcut menu.

The next point you select will be on the same plane as the reference point and in the positive or negative X direction relative to the reference point.

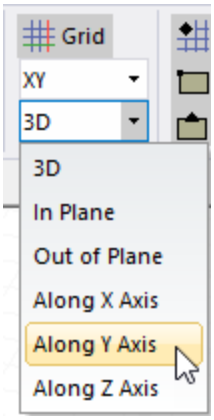
The *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

Moving the Cursor Along the Y-Axis

Use this movement mode to constrain the cursor to a moving only in the +/- Y direction relative to the reference point. Movement is constrained to the Y direction as defined by the working coordinate system, not necessarily the global Y direction. The line of motion passes through the reference point, so it may be parallel to the Y-Axis and not necessarily along the actual Y-axis.

Before or after clicking a desired drawing command, change the movement mode using one of the following methods:

- From the menu bar, click **3D Model**> **Movement Mode**> **Along Y Axis**.
- On the **Draw** ribbon tab, choose **Along Y Axis** from the **Movement mode** drop-down menu:



The following additional method is available only after clicking a drawing command:

- Right-click in the Modeler window and choose **Movement Mode**> **Along Y Axis** from the shortcut menu.
- Press and hold the **Y** key while clicking the next point.

The next point you select will be on the same plane as the reference point and in the positive or negative Y direction relative to the reference point.

The *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

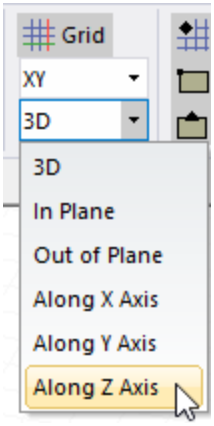
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Moving the Cursor Along the Z-Axis

Use this movement mode to constrain the cursor to a moving only in the +/- Z direction relative to the reference point. Movement is constrained to the Z direction as defined by the working coordinate system, not necessarily the global Z direction. The line of motion passes through the reference point, so it may be parallel to the Z-Axis and not necessarily along the actual Z-axis.

Before or after clicking a desired drawing command, change the movement mode using one of the following methods:

- From the menu bar, click **3D Model**> **Movement Mode**> **Along Z Axis**.
- On the **Draw** ribbon tab, choose **Along Z Axis** from the **Movement mode** drop-down menu:



The following additional methods are available only after clicking a drawing command:

- Press and hold the **Z** key while clicking the next point.
- Right-click in the Modeler window and choose **Movement Mode**> **Along Z Axis** from the shortcut menu.

The next point you select will be on the same plane as the reference point and in the positive or negative Z direction relative to the reference point.

The *Measure Data* window and the coordinate text boxes in the Status Bar provide precise cursor location information.

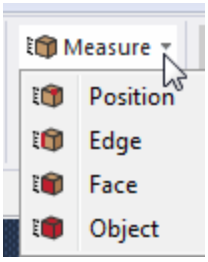
Measure Modes

The Measure modes lets you measure the position, length, area, and volume of objects. With two faces selected, with two edges selected, or with an edge and a face selected, the Measure Mode displays the angle and distance between them. The Measure Position mode dynamically measures the distance between a reference point and the cursor location. You can pause

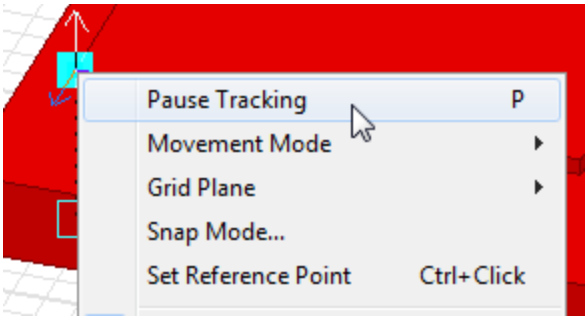
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dynamic tracking in order to copy text information from the *Measure Data* window, and then Resume dynamic updating.

1. To access a Measure mode, do one of the following:
   - From the menu bar, click **Modeler** > **Measure** > [**Position, Edge, Face,** or **Object**].
   - Right-click in the Modeler window and choose **Measure** > [**Position, Edge, Face,** or **Object**] from the short-cut menu.
   - On the **Draw** ribbon tab, choose **Position, Edge, Face,** or **Object** from the **Measure** drop-down menu.



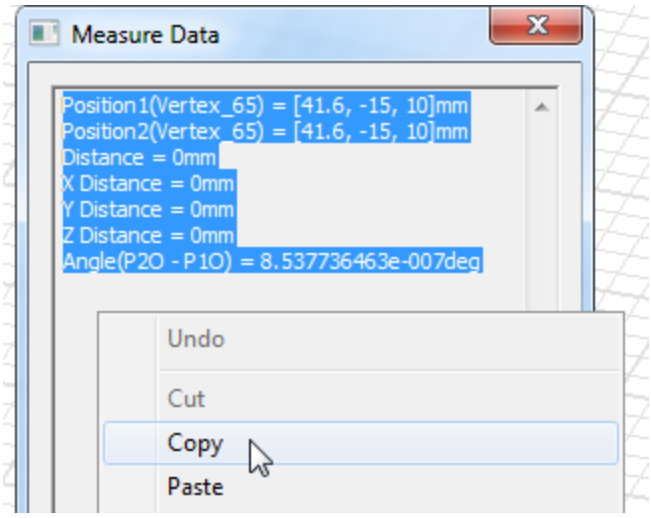
2. Select items to measure as described below under Measurement Modes and Data.
3. Optionally, to *Pause* the dynamic tracking, do one of the following:
   - Right-click in the Modeler window and select **Pause Tracking**:



   - Press **P**.

   While paused, you can move the cursor without changing the *Measure Data* window contents. You can also copy and paste the data to a text file:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To resume measurement tracking, do one of the following:

- Right-click in the Modeler window and choose **Resume Tracking**.
- Press **P** again.

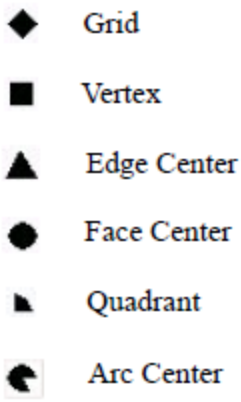
4. To exit the Measure mode, do one of the following:
   - Click **Close** on the *Measure Data* window.
   - Press **Esc**.

Measurement Modes and Data:

- **Position:** Use the following procedure to measure the position of points and the distance between them:
  1. Click a snap point anywhere on the model, which becomes *Position 1* (the measurement reference point).
  2. Hover over another snap point but do not click the mouse (or **Ctrl+click** the second point to redefine the reference point but retain the measurement data).

     The following symbols indicate the available types of snap points:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> As you move the cursor, the Measure Data window displays the reference point, current cursor location, and measurement information. Clicking on a new vertex or other snap point updates the reference to the new location and resets the measurement results to zero. To resent the reference point location without clearing the measurement data, **Ctrl+click** the point.

The *Measure Data* window gives you the following information:

- Position1 coordinates
- Position2 coordinates
- Distance between Position1 and Position2
- X Distance: X-component of the Distance.
- Y Distance: Y-component of the Distance.
- Z Distance: Z-component of the Distance.
- Angle: The angle of the line connecting Position1 and Position2

- **Edge:** The *Measure Data* window gives you the following information:
  - Edge name
  - Length
  - Reference coordinate system
  - Start Vertex Position (with Vertex name)
  - End Vertex Position (with Vertex name)
  - Additional data for circles and arcs only:
    - Arc Center Position
    - Arc Radius
    - Arc Diameter
    - Plane

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Face:** Each item listed below only appears in the *Measure Data* window when applicable to the selected face.
  - Face ID
  - Area
  - Perimeter
  - Cylinder Radius or Sphere Radius
  - Cylinder Diameter or Sphere Diameter
  - Plane (orientation of the plane – the face may be parallel to the listed plane and does not necessarily lie on it)
  - Plane Normal (normal vector for planes not aligned with a global or relative coordinate system)
  - Reference coordinate system
  - Center Position
  - No. of Loops
  - No. of Edges
  - No. of CoEdges
  - No. of Vertices
- **Object:** Each item listed below only appears in the *Measure Data* window when applicable to the selected object.
  - Volume(with *ObjectName* — zero for shell objects)
  - Area(with *ObjectName* — total surface area for solids and shells)
  - No. of Lumps
  - No. of Shells
  - No. of Faces
  - No. of Loops
  - No. of Edges (the total number of unique edges)
  - No. of CoEdges (the total number of coincident edges, where adjacent faces intersect)
  - No. of Vertices

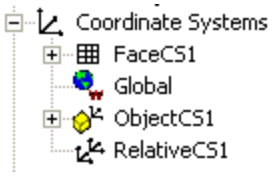
Setting Coordinate Systems

The modeler has four types of coordinate systems (CS) that enable you to easily orient new objects:

- Global CS
- Relative CS
- Face CS
- Object CS

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Every coordinate system has an x-axis that lies at a right angle to a y-axis, and a z-axis that is perpendicular to the xy plane. The origin (0,0,0) of every CS is located at the intersection of the x-, y-, and z-axes. The default Global coordinate system and any additional coordinate systems that you create for a project appear in the History Tree of the Modeler window.



- The *Global* coordinate system (CS) is the fixed, default CS for each new project. It cannot be edited or deleted.
- A *Relative CS* is user-defined. Its origin and orientation can be set relative to an existing CS. Relative CSs enable you to easily draw objects that are located relative to other objects. If you modify a relative CS, all objects drawn on that CS will be affected and change position and/or orientation accordingly. You can define a relative CS to be *Offset* and/or *Rotated* from an existing CS. This feature provides a way for objects made of the same anisotropic materials to have different orientations.

  When you set a new relative coordinate system, you specify whether to express the coordinates as Absolute or Relative Coordinates. Absolute uses the specified values in terms of the global coordinate system. Relative interprets the values as differences from the current working CS.

  You have choices for expressing the coordinates as Cartesian, Cylindrical, or Spherical. These are evaluated as Cartesian for the coordinate system properties.

- A *Face CS* is also user-defined. Its origin is specified on a planar object face. Face CSs enable you to easily draw objects that are located relative to an object's face.
- An *Object CS* is user-defined as attached to a specific object.

Switch between global, relative, object and face CSs by changing the *working CS*. Simply click the CS you want to use in the History Tree. The working CS is indicated by a red *W* that appears at the lower-left corner of the CS name in the History Tree (as shown on Global in the figure above). The docked **Properties** window lists the CS associated with an object as the *Orientation*. By default, the orientation is Global, but if you have created the object under a different coordinate system, that CS will be shown for the orientation. You can click on the current orientation to see a drop-down menu of other orientations that you can assign for an object.

User-defined CSs are saved with the active project. When you open a project, the CS designated as working CS when you last saved it is active.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Working Coordinate System

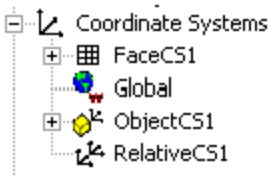
The working coordinate system (CS) is the current CS with which objects being drawn are associated. The working CS can be the Global CS or a user-defined Relative, Object, or Face CS. Select the working CS by clicking its name in the History Tree, or follow this procedure:

1. Click **Modeler> Coordinate System> Set Working CS**.

   The *Select Coordinate System* dialog box appears.

2. Click a CS in the list.
3. Click **Select**.

   A red *W* appears at the lower-left corner of the CS name in the History tree, indicating that it is the working CS. In this following figure the Global is the working CS.



   Objects that you draw hereafter will be associated with the CS you selected.

Creating a Relative Coordinate System

When creating a relative CS, you have the following options:

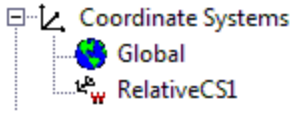
- You can create an offset relative CS, that is, a relative CS whose origin lies a specified distance from another CS's origin. By moving a CS's origin, you can enter coordinates relative to an existing object, without having to add or subtract the existing object's global coordinates.
- You can create a rotated relative CS, that is, a relative CS whose axes are rotated away from another CS's axes. By rotating the axes of a CS, you can easily add an object that is turned at an angle relative to another object.
- You can also create a relative CS that is both offset and rotated.
- You can click **Tools> Options> General Options** to display the *Options* dialog box.

  Under the *3D Modeler> Drawing* options, choose your preferred *Relative Coordinate System Creation Mode*. The **Axis/Position** option uses the cursor or coordinate entry boxes in the Status bar to define the CS. The **Euler angle** option causes the *Set Origin, Set Euler Angles,* or *Set Origin and Euler Angles* dialog box to display (depending on the type of Offset CS you are creating). You define the origin and angles numerically in the dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also press **F3** or **F4** after clicking a *RelativeCS* command to switch between the two entry modes on the fly.

After you have created a relative CS it appears in the History Tree:

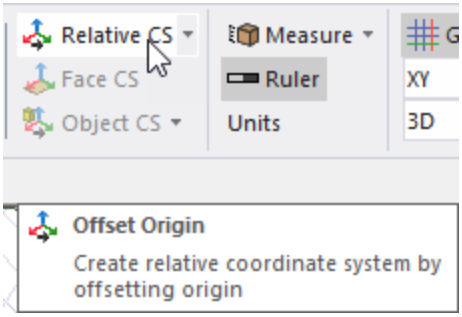


Selecting the CS in the History Tree causes the docked *Properties* window to show the CS properties.

Creating an Offset Relative CS

To create a relative CS with an origin that lies a specified distance from another CS's origin:

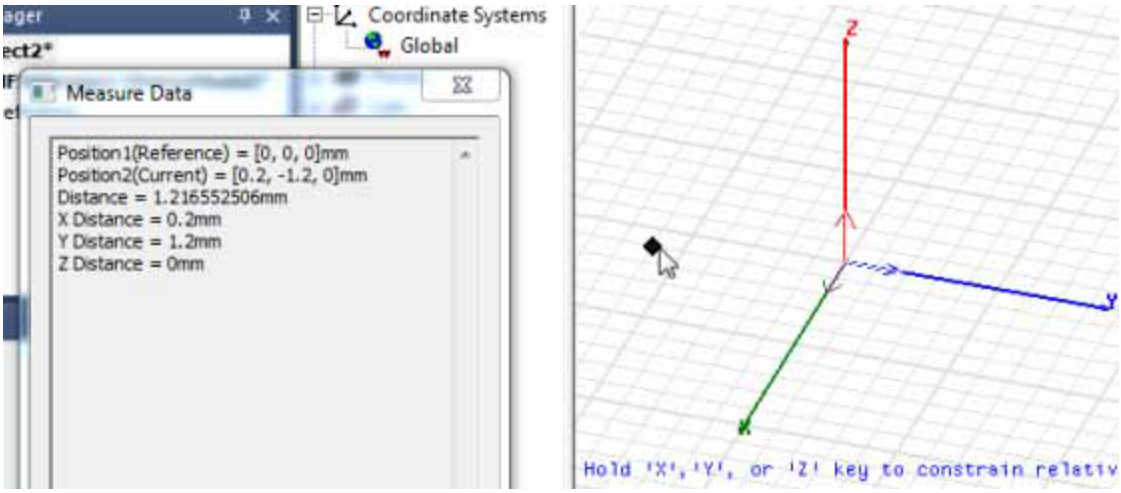
1. In the History Tree, click the CS upon which you want to base the new relative CS, making it the working CS.
2. From the menu bar, click **Modeler> Coordinate System> Create> Relative CS> Offset** or, on the **Draw** ribbon tab, click **Relative CS**. (You do not have to access the drop-down menu because *Offset* is the default option for a *Relative CS* in the ribbon:



3. Depending on your *Relative Coordinate System Creation Mode* choice (**Tools> Options> General Options**, *3D Modeler> Drawing*), or whether you press **F3** or **F4**, you can select the origin in one of the following two ways:
   - **Axis/Position** option selected, or **F3** pressed – Use one of the following two coordinate entry methods:
     - Specify the coordinates graphically (click to select):

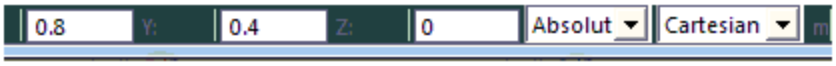
       Use the cursor to click the point. You can see the coordinate information in the *Measure Data* window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To select a point that does not lie in the current plane, use the *Movement Mode* commands or choose a different drawing plane.

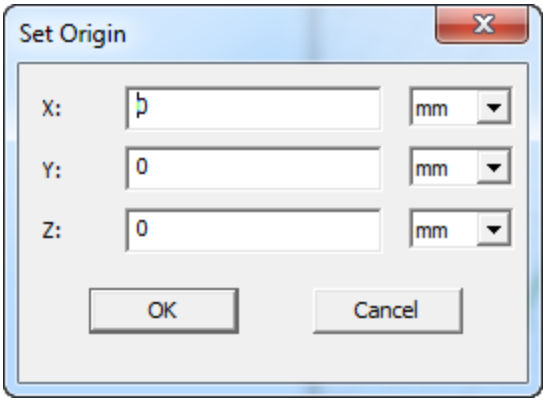
- Specify the coordinates numerically (using the coordinate entry text boxes in the Status Bar):



  a. In the Status Bar, choose the type of coordinate system from the rightmost drop-down menu (**Cartesian, Cylindrical,** or **Spherical**).
  b. Select either Relative or Absolute coordinates from the other drop-down menu.
  c. Type the CS origin coordinates in the **X, Y** and **Z** (or **dX, dY,** and **dZ**) text boxes.

- **Euler angle** option selected, or **F4** pressed:

  In the *Set Origin* dialog box, specify the **X, Y,** and **Z** coordinates and units and click **OK**.



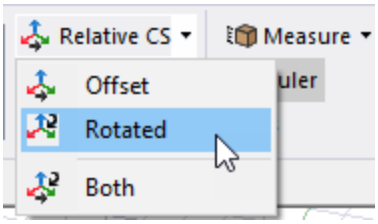
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The new relative CS is created. Its origin has moved from the previous working CS, but its axes remain the same. It is listed in the History Tree under **Coordinate Systems**. It automatically becomes the working CS; objects that you draw hereafter will be based on the coordinates of this relative CS. Default planes are created on its xy, yz, and xz planes.

Creating a Rotated Relative CS

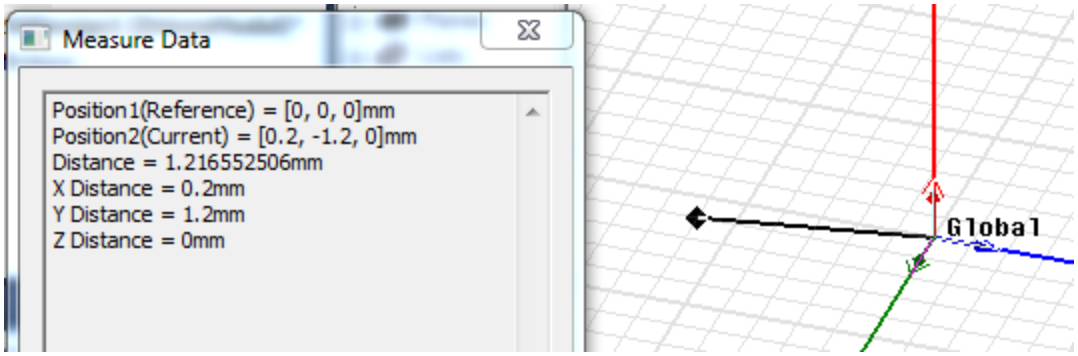
To create a new relative CS with its axes rotated away from another CS's axes:

1. In the History Tree, select the CS upon which you want to base the new relative CS, making it the working CS.
2. Click **Modeler> Coordinate System> Create> Relative CS> Rotated** or, on the **Draw** ribbon tab, click **Relative CS> Rotated**:



.

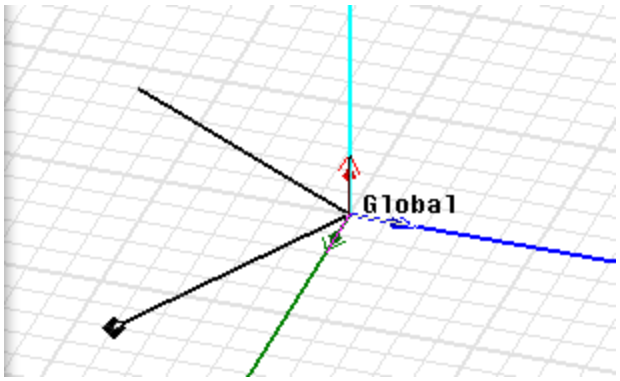
3. Depending on your *Relative Coordinate System Creation Mode* choice (**Tools> Options> General Options** – *3D Modeler> Drawing*), or whether you press **F3** or **F4**, you can specify the axes in one of the following two ways:
   - **Axis/Position** option selected, or **F3** pressed – Use one of the following coordinate entry methods:
     - Specify the X-axis and XY plane graphically (click to select), as follows:
       a. Use the cursor to click a point along the desired X-axis. You can see the coordinate information in the *Measure Data* window.



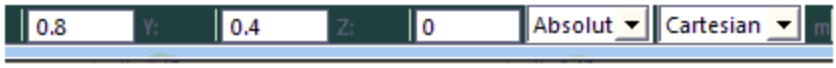
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To select a point that does not lie in the current plane, use the *Movement Mode* commands or choose a different drawing plane.

b. In the same manner, select any point lying on the XY plan that's not on the X-axis.



- Enter the coordinates numerically in the Status Bar, as follows:



a. In the Status Bar, choose the type of coordinate system from the rightmost drop-down menu (**Cartesian, Cylindrical,** or **Spherical**).

b. Select either Relative or Absolute coordinates from the other drop-down menu.

c. Type the coordinates of a point on the desired X-axis in the X, Y and Z (or dX, dY, and dZ) text boxes and press **Enter**.

d. In the same manner, type the coordinates of a point lying on the XY plane, but not on the X-axis, and press **Enter** again.

You do not need to specify the Z-axis. It is automatically calculated to be perpendicular to the new XY plane.

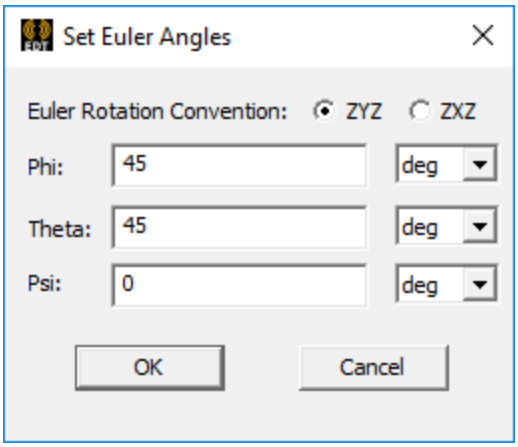
- **Euler angle** option selected, or **F4** pressed:

a. In the *Set Euler Angles* dialog box, select the preferred **Euler Rotation Convention** (**ZYZ** or **ZXZ**).

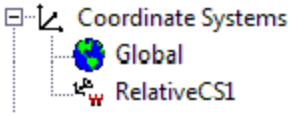
b. Specify the **Phi, Theta,** and **Psi** values and select units for each from the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

drop-down menus.



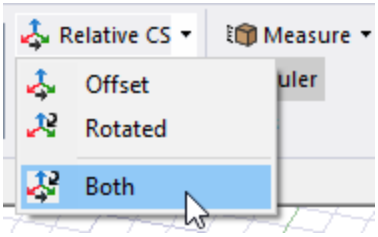
The new relative CS is created. It has the same origin as the previous working CS, but its axes are rotated. It is listed in the History Tree under *Coordinate Systems*. It automatically becomes the working CS; objects that you draw hereafter will be based on the coordinates of this relative CS. Default planes are created on its XY, YZ, and XZ planes.



Creating an Offset and Rotated Relative CS

To create a new relative CS that is both offset and rotated from an existing CS:

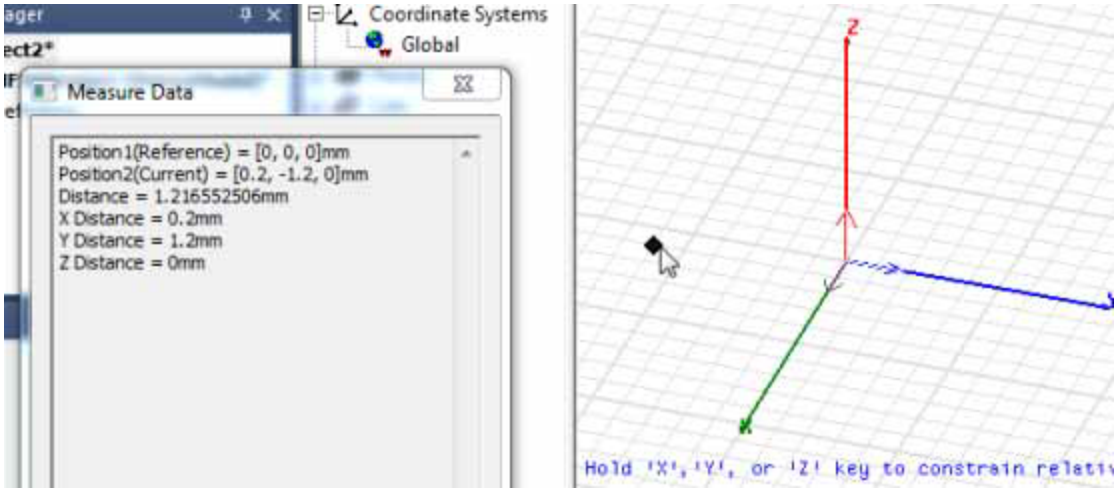
1. In the History Tree, select the CS upon which you want to base the new relative CS, making it the working CS.
2. Click **Modeler> Coordinate System> Create> Relative CS>** **Both** or, on the **Draw** ribbon tab, click **Relative CS> Both**:



3. Depending on your *Relative Coordinate System Creation Mode* choice (**Tools> Options> General Options** – *3D Modeler> Drawing*), or whether you press **F3** or **F4**, you can specify the origin and axes in one of the following two ways:

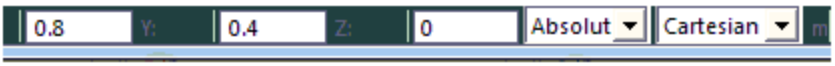
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Axis/Position** option selected, or **F3** pressed – Use one of the following coordinate entry methods:
  - Specify the Origin, X-axis, and XY plane graphically (click to select), as follows:
    a. Use the cursor to click the Origin point. You can see the coordinate information in the *Measure Data* window.



    To select a point that does not lie in the current plane, use the *Movement Mode* commands or choose a different drawing plane.

    b. In the same manner, click a point to define the X-axis.
    c. Select a third point, one lying on the XY plan that's not on the X-axis, to complete the CS definition.
  - Specify the Origin, X-axis, and XY plane numerically in the Status Bar, as follows:

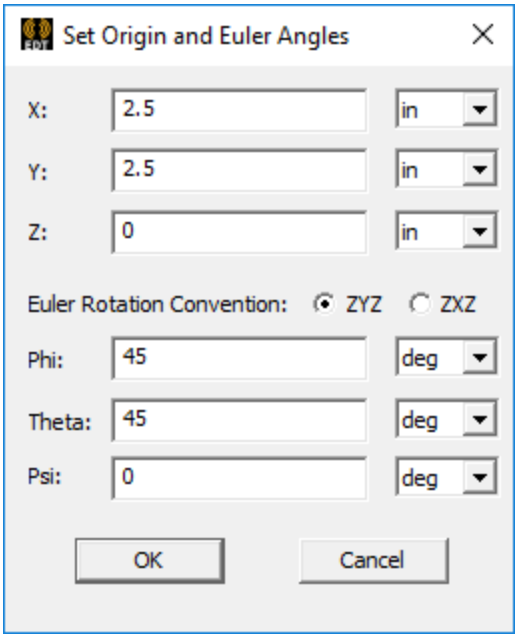


    a. In the Status Bar, choose the type of coordinate system from the rightmost drop-down menu (**Cartesian, Cylindrical,** or **Spherical**).
    b. Select either Relative or Absolute coordinates from the other drop-down menu.
    c. Type the CS origin coordinates in the **X, Y** and **Z** (or **dX, dY,** and **dZ**) text boxes.
    d. In the same manner, type the coordinates of a point on the X-axis and press **Enter**.
    e. Type the coordinates of a third point, one lying on the XY plan that's not on the X-axis, and press **Enter** to complete the CS definition.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You do not need to specify the Z-axis. It is automatically calculated to be perpendicular to the new XY plane.

- **Euler angle** option selected, or **F4** pressed:
  a. In the *Set Origin and Euler Angles* dialog box, specify the **X, Y,** and **Z** coordinates of the Origin and select units for each from the drop-down menus.
  b. Select the preferred **Euler Rotation Convention** (**ZYZ** or **ZXZ**).
  c. >Specify the **Phi, Theta,** and **Psi** values and select units for each from the drop-down menus.

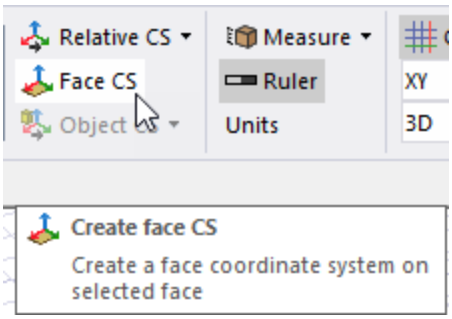


The new relative CS is created. It is listed in the History Tree under *Coordinate Systems*. It automatically becomes the working CS; objects that you draw hereafter will be based on the coordinates of this relative CS. Default planes are created on its XY, YZ, and XZ planes.

Creating a Face Coordinate System

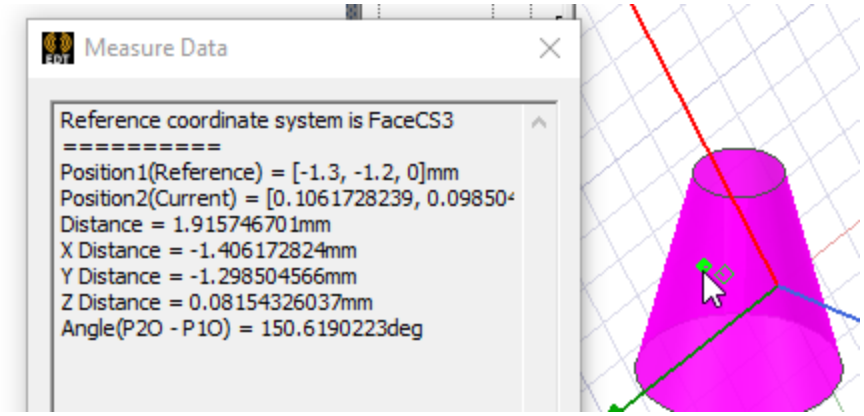
1. Select the object face upon which you want to create the face CS. The CS face can be planar or curved.
2. Click **Modeler> Coordinate System> Create> Face CS** or, on the **Draw** ribbon tab, click the **Create Face CS** icon.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



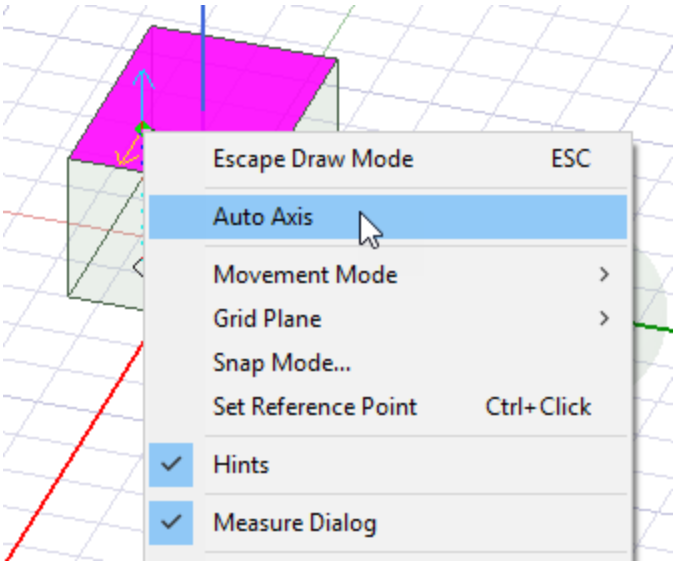
3. Select the Origin in one of the following ways:
   - Click a point on the selected face to define the Origin. If the point is not on the face, you will receive an error message.

     The *Measure Data* window shows the current cursor coordinates. The cursor shape will change to identify grid points, vertices, edge center, face center, quadrant, and arc center points, but you can click any point on the selected face.



   - Type the point's coordinates in the **X, Y,** and **Z** text boxes on the Status Bar.
4. Specify the positive X-axis direction in one of the following ways:
   - Click a point on the selected face. If the point is not on the face, you will receive an error message.
   - Type the coordinates of a point that is relative to the previously selected point in the **dX, dY,** and **dZ** text boxes, where **d** is the distance from the previously selected point.

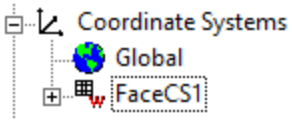
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Right-click in the Modeler window and select **Auto Axis** from the shortcut menu. The modeler selects the X-axis based on the principal curvature of the face or the dimensions of a flat face. The *axial* direction of a curved face or the long dimension of a flat face is selected as the X-axis direction.

You do not need to specify the Y or Z axes. The modeler assumes that the Z-axis is normal to the object face and the Y-axis is automatically calculated to be perpendicular to the XZ plane.

The new face CS is listed in the History Tree under *Coordinate Systems*. It automatically becomes the working CS; objects that you draw hereafter will be referenced to the coordinates of this face CS. Default planes are created on its XY, YZ, and XZ planes.



Only operations listed in the History Tree *before* the face CS's creation will affect the face CS and, in turn, affect objects dependent upon that face CS. A face CS, or any object created on it, is *not* affected by operations that occur after the face CS is created. Also see the Move CS to End command.

For example, suppose you create a box, then a face CS on a face of the box, and then a cylinder on the face CS. If you then edit the box's dimensions in the docked *Properties* window, the cylinder will move accordingly. But if you rotate the box using the **Edit> Arrange> Rotate** command, the box will move, but the cylinder will not move (because the rotation operation occurs later in the History Tree).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automatically Creating Face Coordinate Systems

You can instruct the modeler to automatically create a new face CS every time you draw on an object's face.

1. Click **Tools> Options> General Options**.

   The *Options* dialog box appears.

2. In the tree on the left side of the dialog box, select **3D Modeler> Operation**.
3. In the *Coordinate System* section, select **Automatically switch to face coordinate system**.
4. Click **OK**.

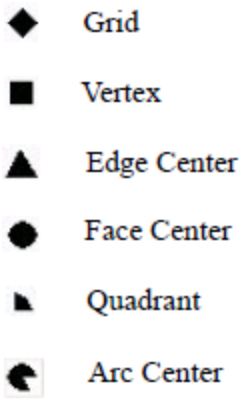
Now, when you select a face, and then click a drawing command, a new face CS will be created on the selected face. The modeler automatically sets the new face CS as the working CS, and the object you draw is oriented according to the new face CS.

> **Note:**
>
> The modeler will not automatically create a new face CS if a face CS has already been assigned to the selected face.

Creating an Object Coordinate System

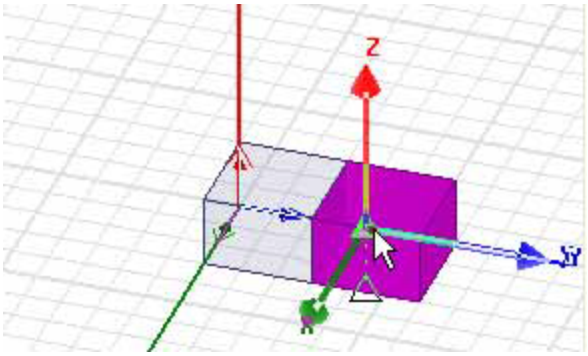
You can create coordinate systems based on any object of solid, sheet, or wire type. The **Modeler> Coordinate System> Create> Object CS** command, and the equivalent ribbon command, is enabled when you select an object. An Object CS can be Offset, Rotated or Both. Executing one of the Object CS commands changes the cursor to the selection marker mode.

As you drag the selection marker over an object, it follows the 3D surfaces of the object, dropping a dashed reference line to a point on the current plane. The cursor changes shape to provide information about the object at the corresponding coordinate:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, in this case, the cursor shows Edge Center triangles as valid selection points for a **Modeler> Coordinate Systems> Create Object CS> Offset** command.



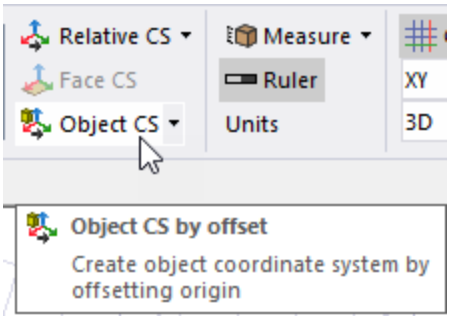
Only operations listed in the History Tree *before* the Object CS's creation will affect the Object CS, and in turn, affect objects dependent upon that Object CS. An Object CS, or objects created on it, is *not* affected by operations that occur after it is created. Also see the Move CS to End command.

For example, suppose you create a box, then an Object CS on a face of the box, and then a cylinder on the Object CS. If you then edit the box's dimensions in the docked *Properties* window, the cylinder will move accordingly. But if you rotate the box using the **Edit> Arrange> Rotate** command, the box will move, but the cylinder will not move because the operation occurs later in the History Tree.

Creating an Offset Object CS

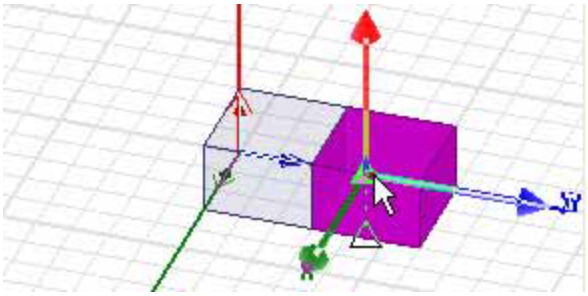
To create an Offset Object coordinate system (CS):

1. Select the working CS to use as the basis of the new Object CS.
2. Select the object to use as the basis of the new Object CS.
3. From the menu bar, click **Modeler> Coordinate System> Create> Object> Offset** or, from the **Draw** ribbon tab, click **Object CS**. (You do not have to access the *Object CS* drop-down menu because *Offset CS by offset* is the default type of Object CS):



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click a point on the selected object to locate the Origin for the new CS.
    - You can select any snap point on the object (depending on the current snapping mode options) to select origin of the Object CS. When you hover the mouse over a valid point, a coordinate system preview is shown.

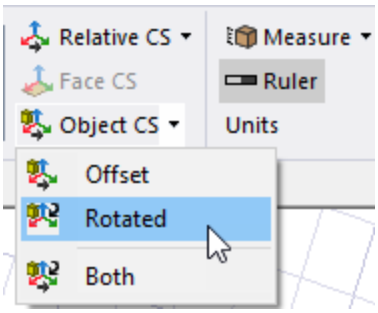


    - The point must be on the selected object.
    - When you select the Origin point, the location is validated, and an appropriate message indicates when a point is invalid. Points where a CS preview appears are always valid.
    - The X-axis {1,0,0}, Y-axis {0,1,0} and Z-axis {0,0,1} are parallel to the corresponding axes of the working CS that was active when you executed the Object CS command, but the Origin has been relocated.
    - The axis coordinates can be later edited through the docked *Properties* window or the *Properties* dialog box.

Creating a Rotated Object CS

To create a Rotated Object coordinate system (CS):

1. Select the working CS to use as the basis of the new Rotated Object CS.
2. Select the object to use as the basis of the new Rotated Object CS.
3. From the menu bar, click **Modeler> Coordinate System> Create> Object> Rotated** or, on the **Draw** ribbon tab, click **Object CS> Rotated**:



You are prompted to *Select X axis*. A preview of X axis is shown for valid selections.

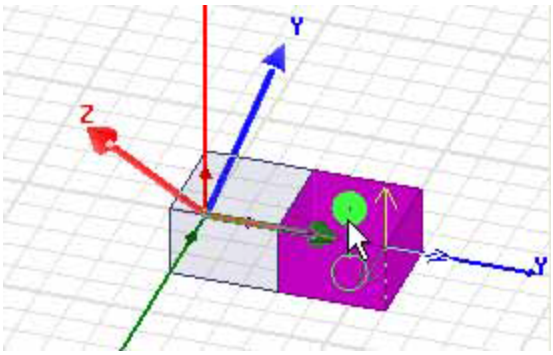
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click a point on the selected object to locate the X-axis direction for the new CS.
   - The Origin {0,0,0} remains at the location defined for the working CS that was active when you executed the Rotated Object CS command. You can edit the coordinates later through the docked *Properties* window or the *Properties* dialog box.
   - The clicked point must be on the selected object.

   You are prompted to *Select to define XY plane*.

5. For 3D models, click a second point on the selected object to define the XY plane.

   A CS preview displays as the cursor hovers over valid selections.



   In the 2D modeler, you are prompted to select only the X axis. The 2D model lies on the XY plane, and the Y-axis is 90 degrees from the X-axis.

Creating an Object CS that is Both Offset and Rotated

To create an Object CS that is both offset and rotated:

1. Set the working CS to use as the basis of the new Object CS.
2. Select the object to use as the basis of the new Object CS.

   The requirements to define this type of Object CS are a combination of *Offset* and *Rotated* Object CS definitions. You are first prompted to select the Origin (refer to the Creating an Offset Object CS page).

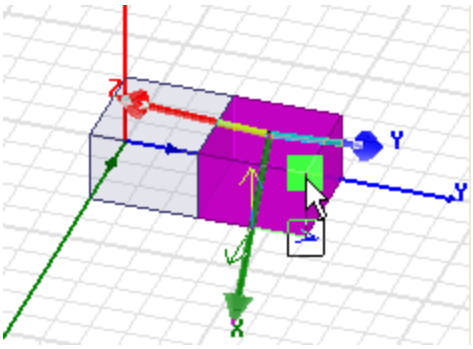
3. Select a point on the object to define the Origin.

   You are then prompted to define the XY plane (refer to the Creating a Rotated Object CS page.

4. For 3D models, select a second point on the object to define the XY plane.

   A CS preview appears when you hover over valid snapping points:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



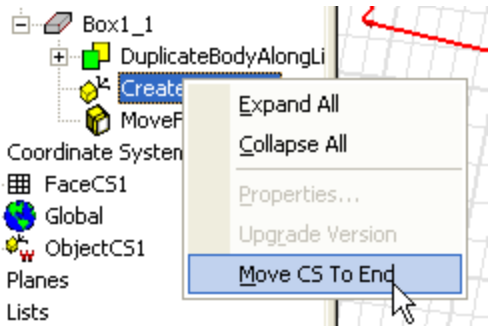
In the 2D modeler, you are asked only to select the origin and the X axis. The 2D model lies on the XY plane, and the Y-axis is 90 degrees from the X-axis.

Move CS to End Command for History Tree

Only operations listed in the History Tree *before* the Object CS's creation will affect the Object CS and, in turn, affect objects dependent upon that Object CS. An Object CS, or objects created on it, is *not* affected by operations that occur after it is created. It is sometimes useful to have the coordinate system affected after any other operations that might have edited, moved, or rotated the object.

If you have at least one operation after a *CreateFaceCS* or *CreateObjectCS* in the History Tree:

Then, selecting a Face or Object CS enables the **Move CS to End** command in the **Modeler> Coordinate System** submenu and in the shortcut menu that appears when you right-click the operation in the History Tree:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Executing this command moves the selected CreateObject CS to the end position in the History Tree and updates associated items (other CS, object history, any dependent parts, and so on).

Modifying Coordinate Systems

Keep in mind that when you edit a CS, the following will also be affected:

- All objects drawn on the CS.
- All CSs that were defined relative to that CS.
- All objects drawn on a CS that were defined relative to that CS.

There are two ways to modify a coordinate system. You can select the coordinate system in the History Tree and edit its parameters in the docked *Properties* window. This approach does not allow you to change whether the coordinate system is Absolute or Relative. Nor does it allow you to change how you express the coordinates (as Cartesian, Cylindrical, or Spherical).

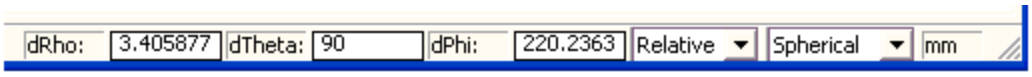
If you want to also modify the whether the coordinate system is Absolute or Relative, and to change how your express the coordinates, do the following:

1. From the menu bar, click **Modeler> Coordinate System> Edit**.

   The *Select Working CS* dialog box appears.

2. Select the CS you want to modify.
3. Click **Select**.

   This action places you in the graphical selection mode for redefining the CS (origin or X-axis) and enables the editable text boxes in the Status Bar at the bottom right area of the Ansys Electronics Desktop application.



4. You can click points to redefine the CS. Alternatively, you can specify the type of coordinate definition (Absolute or Relative) and tab between the coordinate entry text boxes in the Status Bar to numerically define new coordinates.
   - If you selected a Relative CS, follow the directions for creating a relative CS.
   - If you selected a Face CS, follow the directions for creating a face CS.
   - If you selected an Object CS, follow the directions for creating an object CS.
5. Select the type of coordinate system from rightmost drop-down menu in the Status Bar (Cartesian, Cylindrical, or Spherical).

   The values you specify are translated to Cartesian coordinates in the *Properties* of the revised coordinate system.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

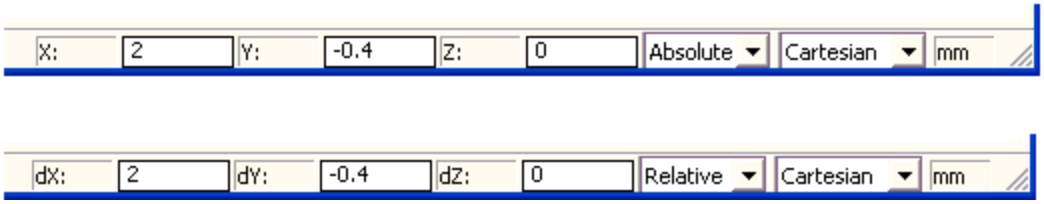
An Object CS can be edited in the same way as it was created. For example, if the CS was created in Offset mode, it will be edited in Offset mode only.

- You can edit the reference CS of an object CS through the docked *Properties* window or the *Properties* dialog box. Doing so will impact object CSs created in *Offset* or *Rotated* mode. It will not impact an object CS created in *Both* (offset and rotated) mode, as the CS fully depends on the object.
- The Origin will be editable when an Object CS was created in *Rotated* mode. Otherwise the Origin coordinates will be read-only text.
- The X Axis and Y Axis will be editable when an Object CS was created in *Offset* mode. Otherwise it will be axis coordinates will be read-only text.
- Either the X Axis or Y axis can be reversed through the docked *Properties* window or the *Properties* dialog box. When the X Axis is reversed, the Y Axis does not change and vice versa.
- When origin or axis properties are defined from snapping points, the point ID is listed in the properties instead of the coordinates. Some examples of snapping point IDs are: *Vertex_10, Face_7 center, Edge_9 midpoint, Edge_8 quadrant,* and *Edge_17 arc center*.

Expressing Cartesian, Cylindrical, or Spherical Coordinates

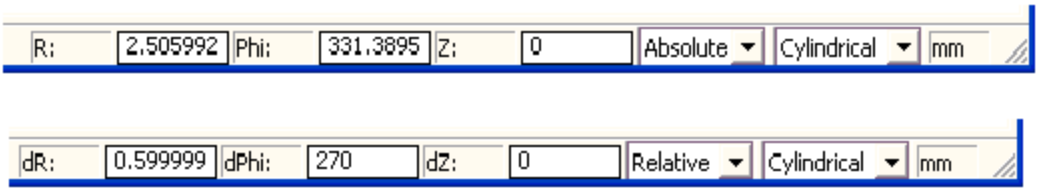
You have choices for expressing coordinates, specifically as Cartesian, Cylindrical, or Spherical by using the editable fields in the Status Bar. In each case, you can also specify whether to enter the coordinates as Absolute or Relative to a reference point. Click the first text box, or press **Tab** to begin entering values or variables. **Tab** to the next text box or **Ctrl+Tab** to the previous one. Press **Enter** when you are done. After you enter values or variables in the text boxes, they are evaluated as Cartesian for the coordinate system *Properties*, regardless of the way they were defined.

- **Cartesian:** Coordinates represent the point's distance from the origin in the x, y, and z directions. The values are specified in the **X, Y,** and **Z** text boxes.

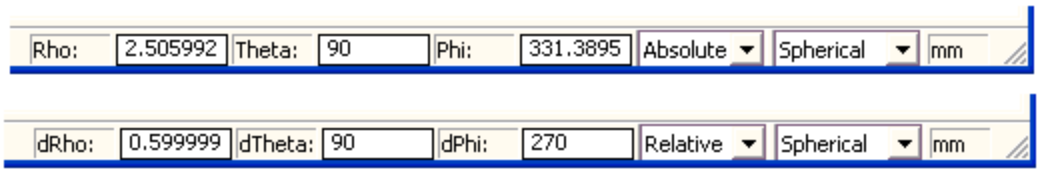


- **Cylindrical:** Specify the point's radius, measured from the origin, is the **R** text box, the angle from the x-axis in the **Theta** text box, and the z-distance from the origin in the **Z** text box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Spherical:** Specify the point's radius, measured from the origin, in the **Rho** text box, the angle from the x-axis in the **Theta** text box, and the angle from the origin in the z direction in the **Phi** text box.



Deleting Coordinate Systems

1. In the *Coordinate System* branch of the History Tree, click the name of the CS you want to delete.
2. Delete the CS using one of the following methods:
   - Press **Delete**.
   - From the menu bar, click **Edit**> ✕ **Delete**.
   - On the **Desktop** ribbon tab, click ✕ **Delete**.

   The CS will be deleted and, all objects drawn on it will be deleted. Furthermore, any CS that was dependent upon the deleted CS will be deleted, and any objects that were drawn on the dependent CS will also be deleted.

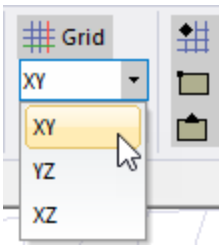
Setting the Drawing Plane

The *Drawing plane* options are available from a drop-down menu on the *Draw* ribbon tab and from the *Grid Plane* submenu of the *Modeler* menu. The XY, YZ, and XZ drawing planes correspond to the active coordinate system, not the global planes (unless *Global* is the currently active CS).

To set the drawing plane, do one of the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- On the **Draw** ribbon tab, select **XY, YZ,** or **XZ** from the **Drawing plane** drop-down menu:



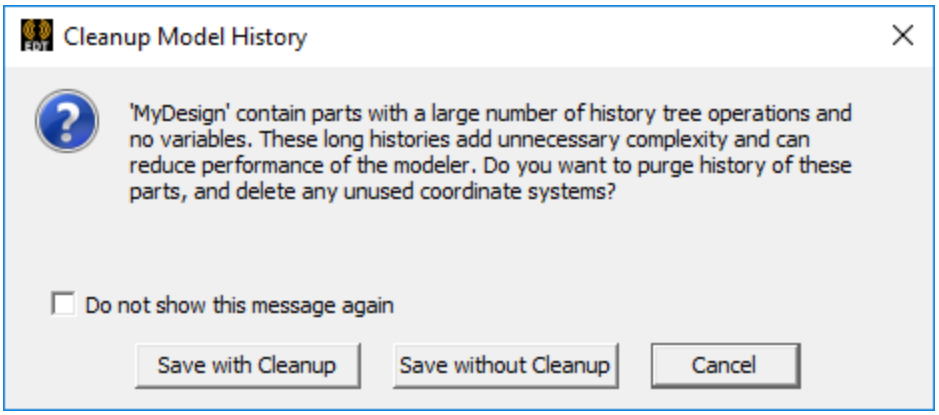
- From the menu bar, click **Modeler> Grid Plane>** and select one of the planes **XY, YZ,** or **XZ**.

Cleaning Up Model History

Ansys Electronics Desktop can clean up history tree operations in order to improve performance.

When saving your project, you will be prompted to clean up your model's history if some parts meet two conditions:

- No variables assigned to the part
- A history greater than a specified number of actions



The number of actions that triggers this prompt can be set in 3D Modeler Operations Options.

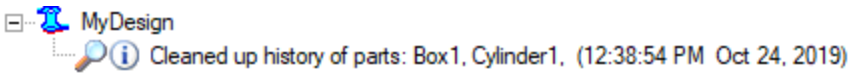
When promoted, you can choose **Do not show this message again** to hide the prompt for the active project.

To clean up the model history manually:

- Select **Modeler** > **Cleanup Model History**.

After the model has been cleaned up, you can view cleaned up parts in the **Messages** window:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

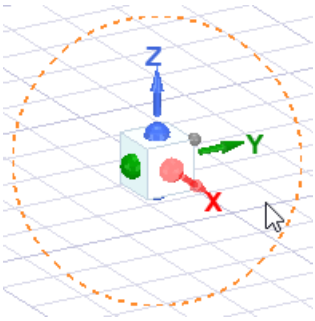
7 - Modifying the 3D Model View

You can modify the view of objects in the 3D **Modeler** window without changing their actual rotation or position within the coordinate system. You vary the viewing direction and the relative location of the "camera" (that is, the viewpoint) relative to the model geometry. The tools allow you to zoom, pan, or rotate the view orientation and to quickly switch to predefined or user-defined view orientations. Additionally, you can choose different object visualization options, show/hide objects, and customize the window background.

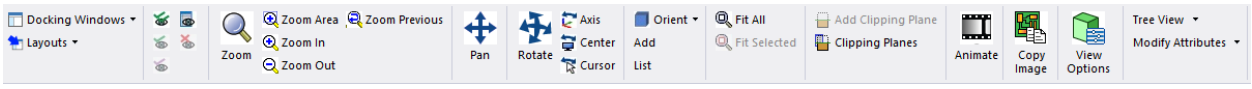
What do you want to do?

Change the View	Show or Hide Design Objects
Change Object Visualization	Change the Background

In addition to various menu bar, ribbon, and shortcut menu commands, you can manipulate the view orientation of 3D models and 3D plots using the Orientation Gadget:



Finally, you can also access many of the Modify View features via the **View Ribbon**. The following image has hot links for each area.



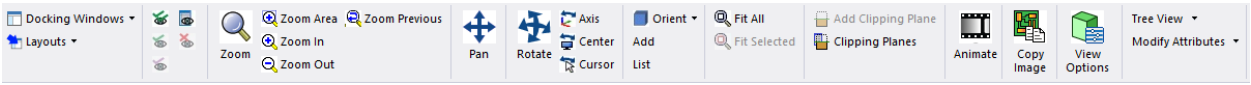
Change the View

- Using the Orientation Gadget
- Changing the Model View with Alt+Double-Click Areas in the Modeler Window
- Apply an Orientation to the Current View

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Set the View Options
- Rotate the View
- Pan the View
- Zoom In or Out
- Zoom to Selected Excitation
- Zoom In or Out Using a Mousewheel
- Fit Contents in the View Window
- Use Clip Planes

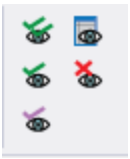
You can also access many of the Modify View features via the **View Ribbon**. The following image has hot links for each area.



Show or Hide Design Objects

- Show or Hide Objects
- Showing Only Selected Objects in All or Active Views
- Show or Hide Boundaries or Excitations
- Visualize or Select Nets

You can also access the Show or Hide features by using the following icons on the **View, Draw,** or **Model** ribbon tabs:



Change Object Visualization

- Render Objects as Wireframes, Flat-Shaded, or Smooth-Shaded Solids
- Set the Enhanced Display of Material Color and Transparency
- Set the Surface Visualization
- Modify the View Orientation
- Modify the Lighting
- Set the Projection View

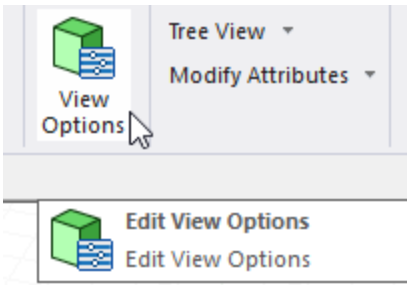
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Change the Background

- Setting the Background Color
- Modify the appearance of the Coordinate System Axes
- Modify the appearance of the Grid

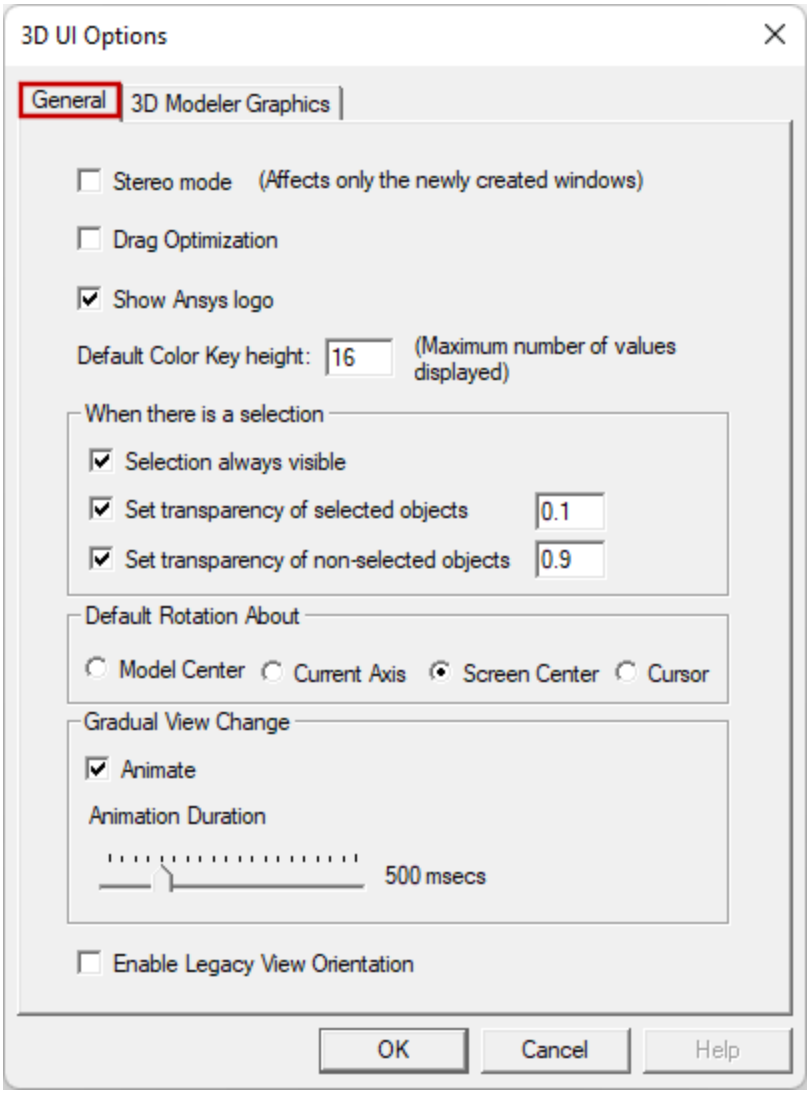
View Options: 3D UI Options

To access View Options, use the **View** > **Options** command, or click **View Options** on the **View** ribbon.



The **3D UI Options** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **General** tab lets you set defaults for the following view options:

- Stereo Mode (default, disabled) – This option is only applicable to systems with support for stereographic viewing for greater 3D realism.
- Drag Optimization (default, disabled)
- Show Ansys logo (default, enabled)
- Default Color Key height (default = 16)

When there is a selection option:

- Selection always visible (default, enabled)
- Set transparency of selected objects
- Set transparency of non-selected objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Default Rotation About options:

- Model Center
- Current Axis
- Screen Center (default)
- Cursor

  When you select one of these as the default, the **View** > **Rotate** selection menu changes to show that the **Alt+drag** combination attaches to your selection.

Gradual View Change settings:

- Animate
- Animation Duration

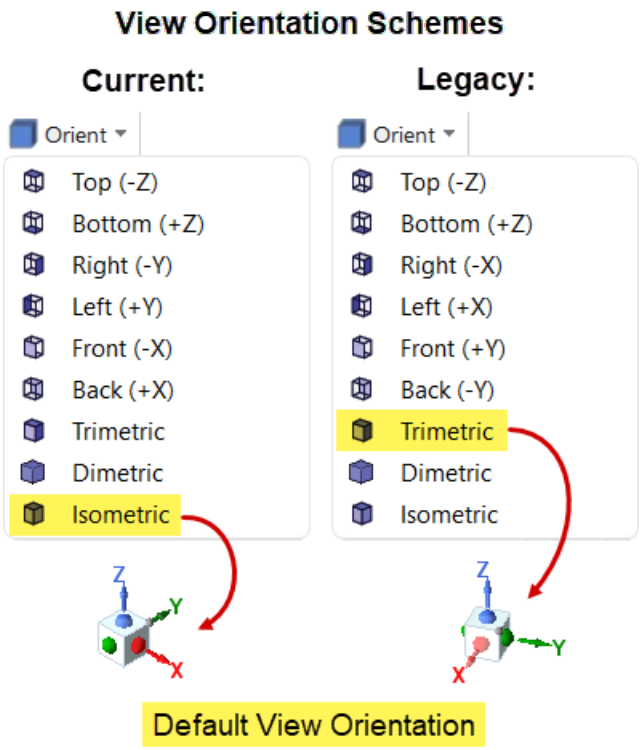
  For operations such as Fit Selection, Fit All, and Zoom, these settings determine whether Electronics Desktop animates the transition between images and how smoothly it does so.

Enable Legacy View Orientations option:

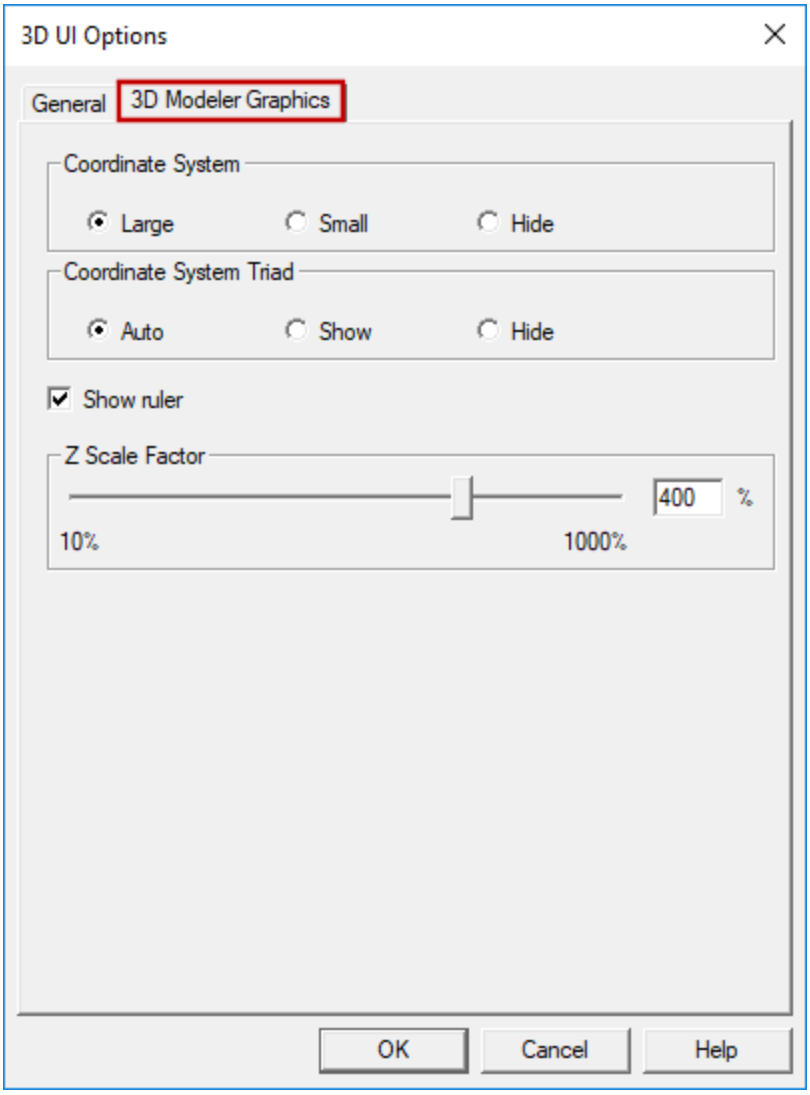
- When selected, the view orientations scheme and the axis directions for named views follow the legacy conventions (software version 2023 R2 or older):
- When cleared, the view orientations scheme and the axis directions for named views follow the current view orientation scheme

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following view is a comparison of the two options:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **3D Modeler Graphics** tab lets you set the following options which are invoked for the next Project and design created.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

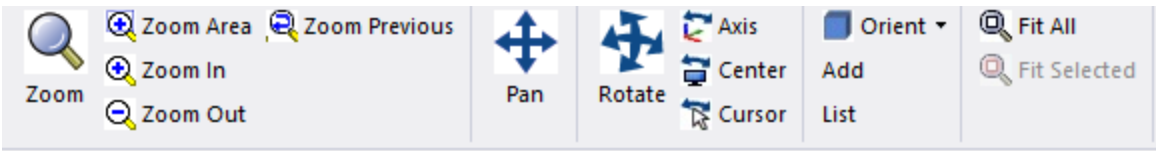
Settings	Description of Choices
Coordinate System	• **Large** (extends to the window edges) • **Small** (centered, not extending toward edges with smaller arrows) • **Hide** (do not display)
Coordinate System Triad (A secondary, smaller coordinate system appearing near the lower right corner of the Modeler window)	• **Auto** • **Show** • **Hide**
Show Ruler (To indicate the scale of the model, based on current units. The ruler appears at the lower center of the Modeler window)	Select to enable; clear to disable
Note: You can also toggle the ruler visibility by clicking **Ruler** on the **Draw** ribbon tab or by right-clicking in the *Modeler* window and choosing **View** > **Ruler** from the shortcut menu.	
Z Scale Factor	The default is 400%. You can use the slider or the text field to specify a different value.

Panning the View

To move (pan) the view:

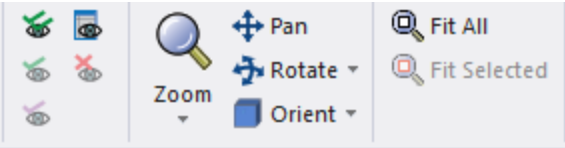
1. Click **View** > **Interaction** > **Pan** or click the **Pan** icon on the **View, Draw,** or **Model** ribbon tab.

   View ribbon tab:



   Draw or **Model** ribbon tabs:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click and drag the mouse in the direction you want to pan the view.

   The view will pan until you release the left mouse button.

3. To exit **Pan** mode, click **Pan** on the **View** menu again or press **Esc**.

> **Tip:**
>
> Alternatively, pan the view using one of the following methods:
>
> - Hold down **Ctrl** as you click and drag using the **middle mouse button**.
> - If you chose to select *Enable Legacy View Navigation* in the General > User Interface options, you can also use the legacy combination **Shift** + **drag** (with the left mouse button) to pan the view.
> - Press **Ctrl** + ***arrow keys*** to pan vertically or horizontally. The movement is 10 screen pixels per press of the arrow key.
> - In addition to the menu bar and ribbon tabs, you can access *Pan* from a shortcut menu. Right-click in the Modeler window and click **View** > **Pan**. Click the command again or press **Esc** to terminate panning.

Zooming in and Out

You can magnify (zoom in) or shrink (zoom out) the contents in the view window using the following methods:

- Hot Keys (that work without the mouse)
- Hot Keys (used in conjunction with clicking and dragging the mouse)
- **Zoom In** and **Zoom Out** commands, which produce 5% zoom increments
- **Zoom Previous** to undo recent zoom commands, as described below
- Zoom into a Rectangular Area
- Zoom in or out Using a Mouse Wheel
- Zoom to a Selected Boundary or Excitation

> **Note:** Some international keyboards may not support the following hotkeys. If the Shift key is required for a character, such as the plus sign (+), the hotkey may not function.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To zoom in using hotkeys:

- Press **Ctrl** + "**+**"

  If using the "=\+" key (not the "+" key on the numeric keypad) do **not** press the *Shift* key.

  The view zooms in 5 percent.

To zoom out using hotkeys:

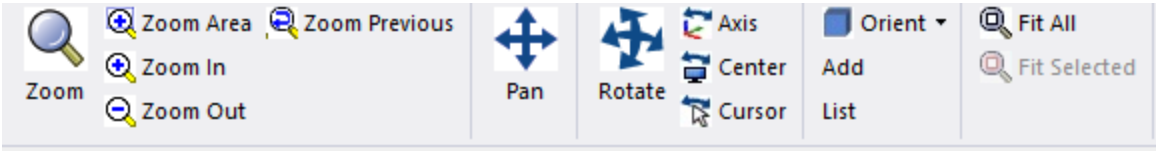
- Press **Ctrl** + "**-**"

  The view zooms out 5 percent.

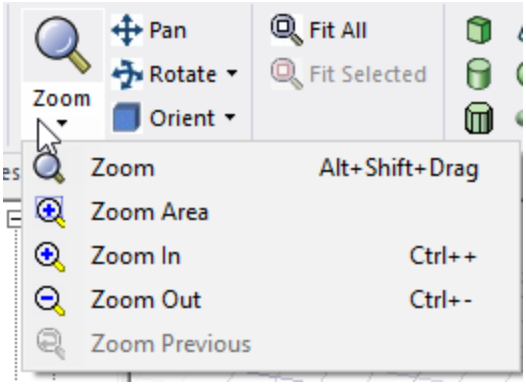
To zoom by dragging the mouse.

1. Click **View** > **Interaction** > **Zoom**, click **View** > **Zoom** on the shortcut menu, or click the **Zoom** icon on the **View, Draw,** or **Model** ribbon tabs.

   View ribbon tab with **Zoom** commands.



   Draw or **Model** ribbon tabs with **Zoom** icon, and drop-down menu for Zoom commands:



2. To zoom in, drag the mouse toward the top of the view window. The objects in view expand as you drag.

   To zoom out, drag the mouse toward the bottom of the view window. The objects in view decrease in size as you drag.

   When zooming on a view of model objects the absolute size of the model does not change.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When zooming on a 2D report, axis labels and ticks will adjust automatically during the zoom operation and will rescale to their final value after the zoom operation is complete.

3. To end **Zoom** mode, click **View** > **Interaction** > **Zoom** again, or click the Zoom icon, or press **Esc**.

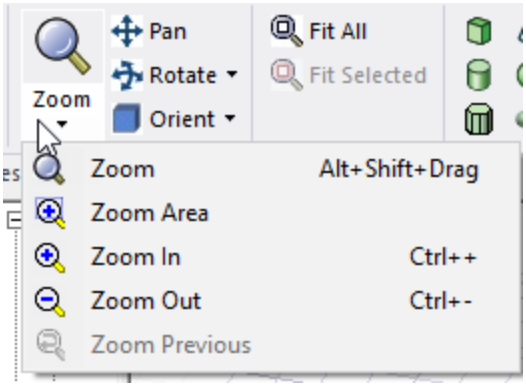
Zoom In and Zoom Out Commands

To use **Zoom In**, **Zoom Out** to zoom in or out in 5% increments:

1. Click **View** > **Interaction** > **Zoom**, click **View** > **Zoom In** or **Zoom Out** on the shortcut menu, or click the **Zoom** icon on the **View, Draw,** or **Model** ribbon tabs.

   View ribbon tab with **Zoom In** and **Zoom Out**.

   Draw or **Model** ribbon tabs with **Zoom** icon, and drop-down menu for **Zoom In** and **Zoom Out**:



2. Click **Zoom In** or **Zoom Out** for a 5% increment zoom in or out respectively.

   To zoom out, drag the mouse toward the bottom of the view window. The objects in view decrease in size as you drag.

   When zooming on a view of model objects the absolute size of the model does not change.

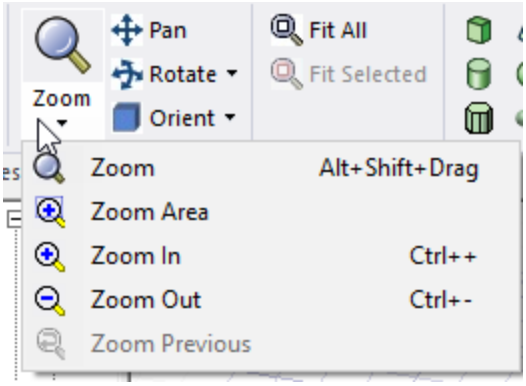
   When zooming on a 2D report, axis labels and ticks will adjust automatically during the zoom operation and will rescale to their final value after the zoom operation is complete.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. To end **Zoom** mode, click **View** > **Interaction** > **Zoom** again, or click the **Zoom** icon, or press **Esc**.

Zoom Previous Command

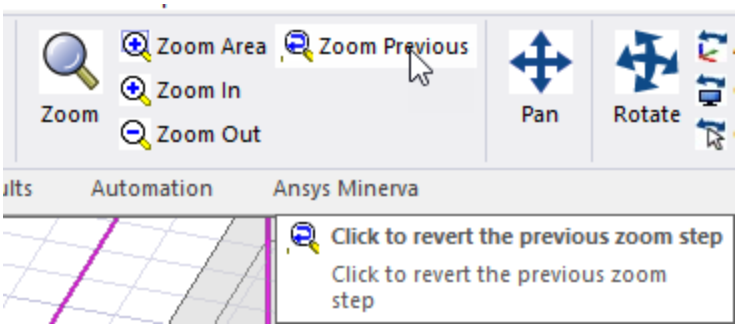
The **Zoom Previous** command is not enabled until at least one **Zoom** command has been used.



It operates on any existing sequence of **Zoom** commands. That is, it undoes the last previous **Zoom** command, and then, another exists, the one before that. To use **Zoom Previous**.

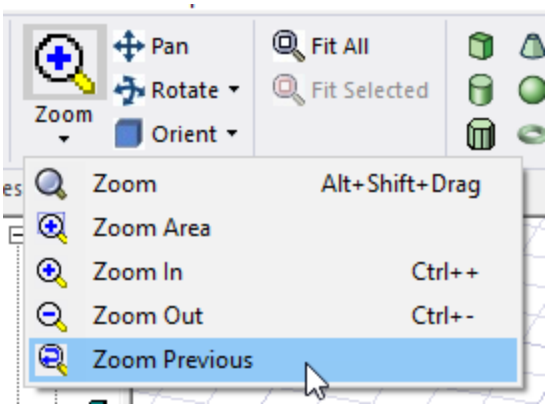
1. Click **View** > **Interaction** > **Zoom Previous**, click **View** > **Zoom Previous** or Zoom Out on the shortcut menu, or click the **Zoom** icon on the **View, Draw,** or **Model** ribbon tabs.

   View ribbon tab with .



   Draw or **Model** ribbon tabs with **Zoom** icon, and drop-down menu for **Zoom Previous**:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. **Zoom Previous** undoes the last executed Zoom command, and then, if others exist, can be used to undo them in reversed order of execution.

   When zooming on a view of model objects the absolute size of the model does not change.

   When zooming on a 2D report, axis labels and ticks will adjust automatically during the zoom operation and will rescale to their final value after the zoom operation is complete.

3. To end **Zoom** mode, click **View** > **Interaction** > **Zoom** again, or click the **Zoom** icon, or press **Esc**.

Tip:

Alternatively, zoom in or out using one of the following methods:

- Hold down **Shift** as you click and drag using the **middle mouse button**. Moving the cursor upward zooms in and moving it downward zooms out.
- If you chose to select *Enable Legacy View Navigation* in the General > User Interface options, you can also use the legacy mouse-button and hotkey for zooming. Hold down **Alt** + **Shift** as you click and drag (using the left mouse button).
- Roll the mouse wheel. Rolling the top of the wheel forward (away from you) moves the model view further away from you (zooming out). Conversely, rolling the top of the wheel rearward (towards you) zooms in.
- Right-click in the Modeler window and then click **View** > **Zoom** on the shortcut menu. Click and drag using the left mouse button to zoom in or out. Repeat the command or press **Esc** to cancel zooming.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

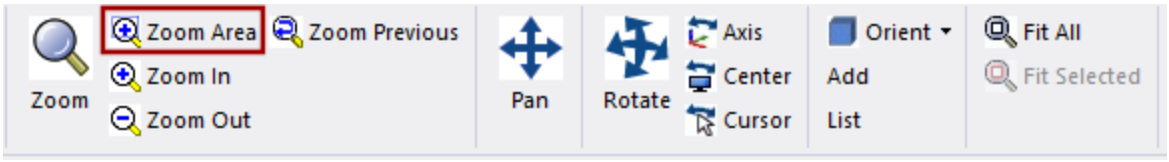
If zooming is slow, especially for complex models, for some graphics cards, you can improve performance by setting **NVIDIA Control Panel** > **3D Settings** > **Manage 3D Settings** > **Global Settings** > **Global Presets: Workstation App - Dynamic Streaming**

For more information about graphics cards, navigate to the appropriate Windows or Linux Ansys Installation Guide and search for **OpenGL**.

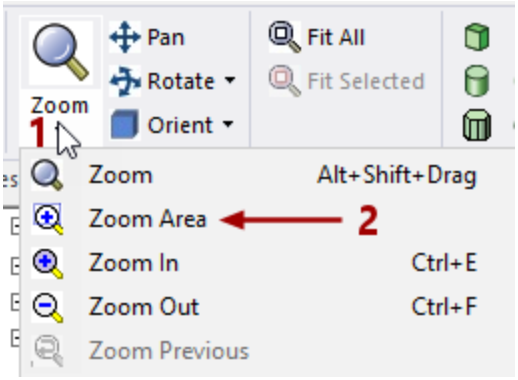
Zooming into a Rectangular Area

Zoom to expand a specified rectangular area to fill the view window, as follows:

1. Use one of the following methods to begin the *Zoom Area* mode:
   - Using the menu bar, click **View** > **Interaction** > **Zoom Area**.
   - From the **View** ribbon tab, click **Zoom Area**:



   - From the **Draw** or **Model** ribbon tab, click **Zoom** > **Zoom Area**:



   - Right-click in the Modeler window and choose **View** > **Zoom Area** from the shortcut menu.
2. Click and drag the mouse to draw a rectangle enclosing the area that you want to zoom into.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you release the mouse button, the rectangular area will come close to filling the display area of the window.

The absolute size of the zoomed geometry does not change, only the scaling of the model view is affected.

3. To end the **Zoom Area** mode, click the command again or press **Esc**.

   The rotate and zoom drop-down menu icons on the ribbon update dynamically to indicate the current rotate/zoom mode. These ribbon buttons are also "sticky." That is, the last active rotate or zoom mode becomes the button default until a different command is chosen or the application is restarted. Default commands are restored the next time you launch the Ansys Electronics Desktop program.

   For 2D or 3D reports, axis labels and tick marks are adjusted after the zoom operation is complete.

Zooming in or out Using a Mouse Wheel

You can zoom the active model view in and out by rotating the mouse wheel. This action works in the Modeler window, for 2D and 3D plots, and more. The cursor location is the center point whenever using the mouse wheel for zooming, and this point remains fixed. The grid and ruler, when visible, are adjusted dynamically as you zoom in and out.

Rotating the wheel forward, away from you, moves the model away from you (zooming out). Rotating the wheel backward, towards you, pulls the model closer (zooming in).

> **Note:**
>
> If Zooming is slow, especially for complex models, for some graphics cards, you can improve performance by setting **NVIDIA Control Panel** > **3D Settings** > **Manage 3D Settings Global Settings** > **Global Presets: Workstation App - Dynamic Streaming**
>
> For more information about graphics cards, navigate to the appropriate Windows or Linux Ansys Installation Guide and search for **OpenGL**.

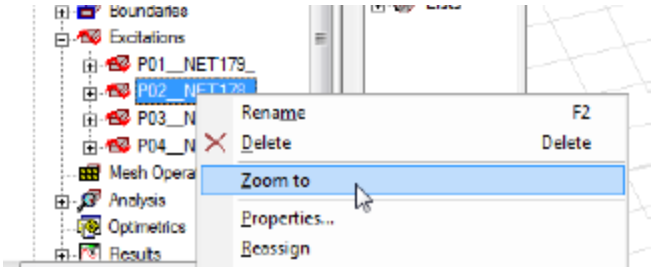
Zoom to a Selected Boundary or Excitation

Zoom to is an option that can enlarge the view of a boundary or excitation. This command can be very useful for locating small items with boundaries or excitations assigned or to inspect any problem areas.

You can use the **Zoom to** command as follows.

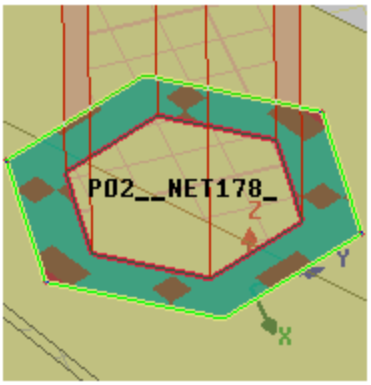
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. If needed, expand the *Boundary* or *Excitation* branch in the Project Manager to see the previously assigned conditions.
2. Right-click the boundary or excitation of interest to highlight it in the model view and to access the shortcut menu, as shown below:



3. Select **Zoom to**.

   Notice that the command instantly increases magnification, filling the screen with the object(s) or face(s) to which the boundary or excitation is assigned, as shown in the following figure:

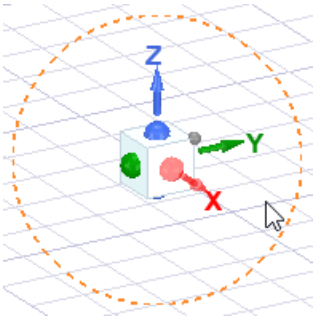


Modifying the View Orientation

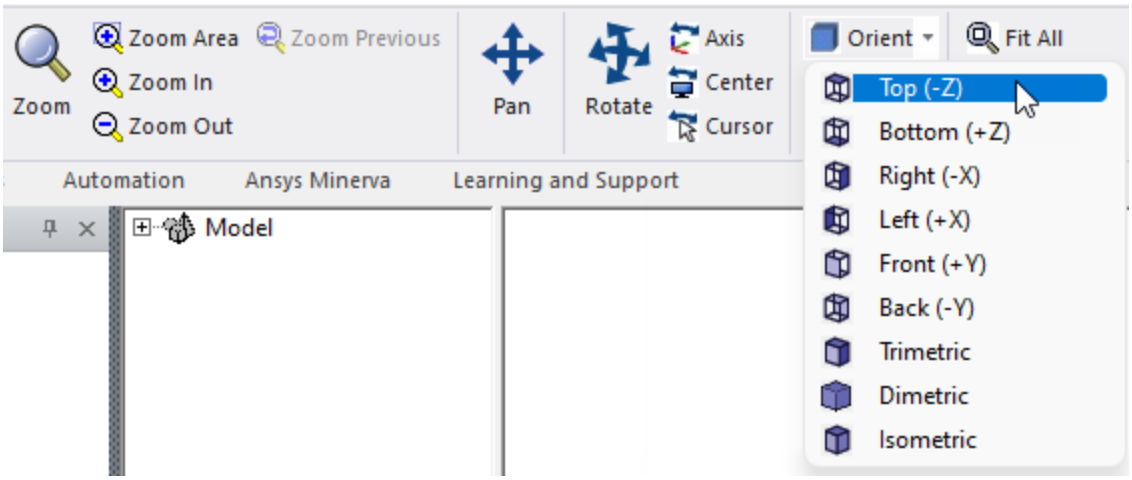
There are several places you can access commands to change the view orientation in a *Modeler* or *3D Plot* window:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the View Orientation Gadget:



- Select the **Orient** drop-down Menu on the **View** ribbon tab and choose one of the predefined view orientations:



  The **Orient** drop-down menu can also be located on the **Draw** and **Model** ribbon tabs.

  The sign and letter in parentheses that follows each standard view indicates the viewing direction, not the model side. For example, consider a simple box aligned with the global planes. The right side of the box is the +X side, but the viewing direction is -X (you look leftward to see the right side of an object).

- Right-click in the Modeler window and, from the shortcut menu that appears, point to **View** > **Apply Orientation**, and select the desired viewing direction from the submenu. The choices include those shown in the preceding image plus any custom views you have defined (see the next bullet).
- From the menu bar, click **View** > **Modify Attributes** > **Orientation List**.

  A dialog box with orientation settings appears. It includes a table with the names of nine basic orientations (Top, Bottom, Right, Left, Front, Back, Trimetric, Dimetric, and Isometric), any additional orientations that you have added, and a section for managing orientations (adding/removing orientations or setting the default view).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The table includes columns that show the viewing angles relative to the global axes and the equivalent vector components. These vector components are listed based on the global coordinate system and any applicable local coordinate systems.

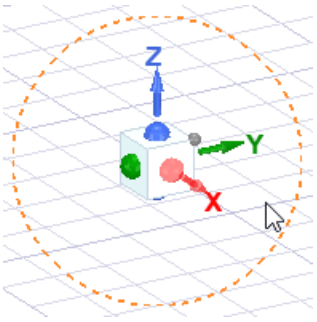
1. Select a default orientation or create a new orientation.
2. Click **Apply to View** for the selected view to appear in the window.
3. Optionally, click the **Reset View Orientation** button to restore the selected view to the original angles.
4. Click **Make Default** if you want the selected viewing direction to be the initial viewing direction when a **3D Modeler** window is opened, either in the current project or future projects.
5. Click **Close** to dismiss the dialog box.

   The orientation you set will be saved with the design. New orientations assigned to other designs after this point will not affect this orientation.

Apply an Orientation to the Current View

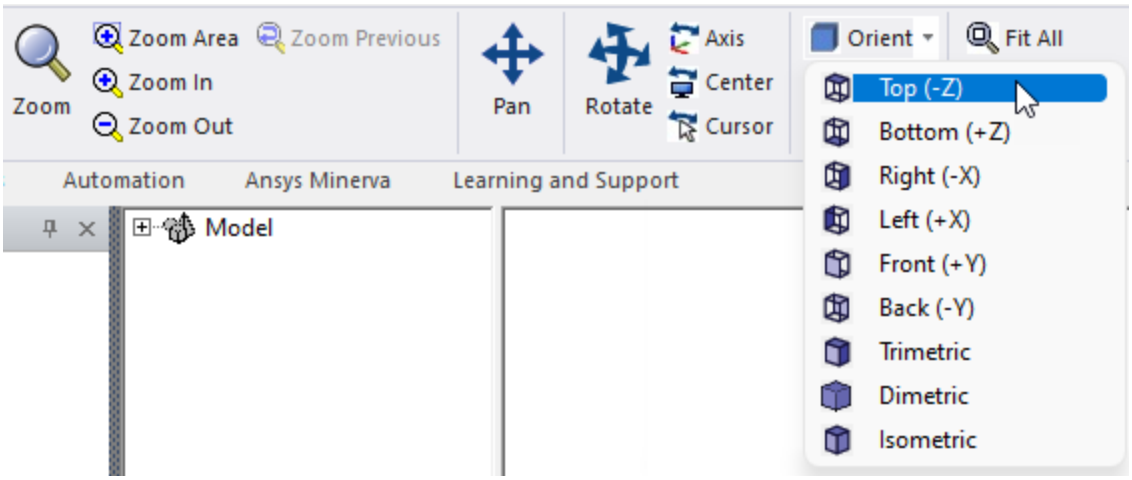
Apply view orientation settings using any one of several available methods:

- Use the Orientation Gadget:

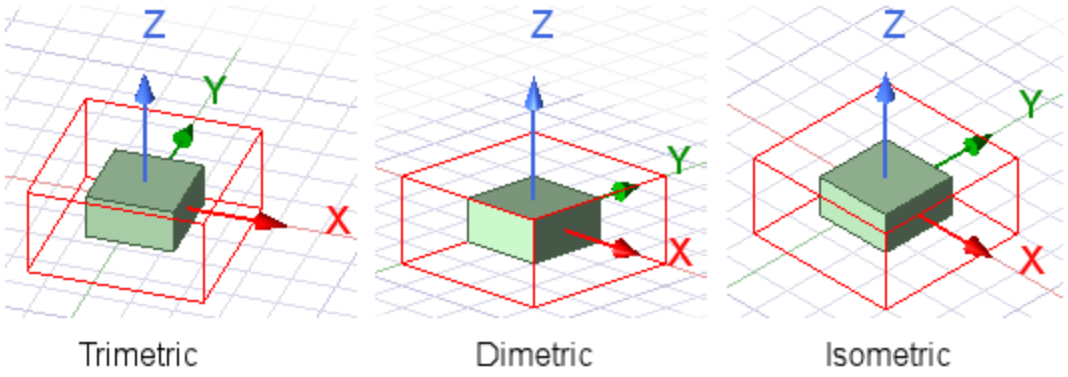


- Use the commands in the **Orient** drop-down menu located on the **View, Draw,** or **Model** ribbon tabs:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

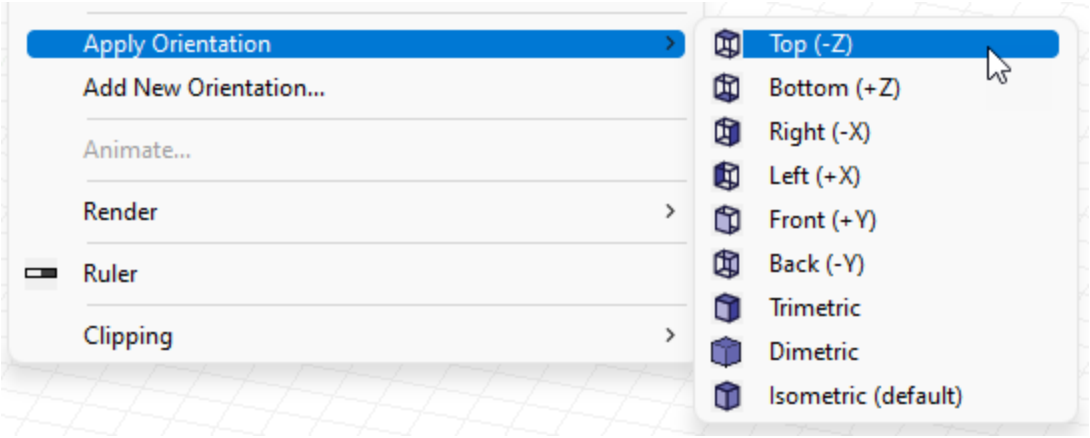


Possible orientations are Top, Bottom, Right, Left, Front, Back, Trimetric, Dimetric, and Isometric. These last three differ as shown below:



- Right-click in the Modeler window. In the shortcut menu that appears, go to the **View** > **Apply Orientation** submenu:

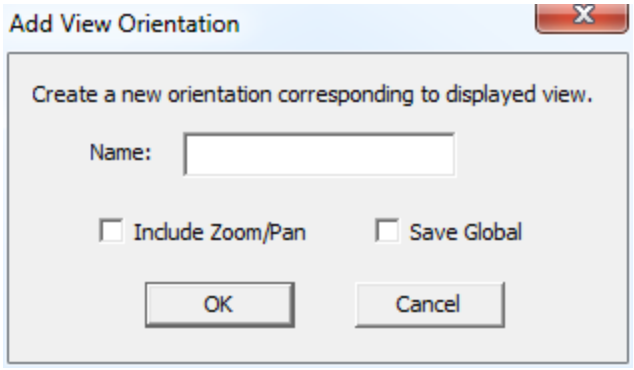
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In addition to the standard views, any custom views that you have defined will appear in this submenu.

After applying an orientation, you can click **View** > **Undo View : *orientation***.

You can also right-click in the Modeler window and choose **View** > **Add New Orientation** to create a custom viewpoint.

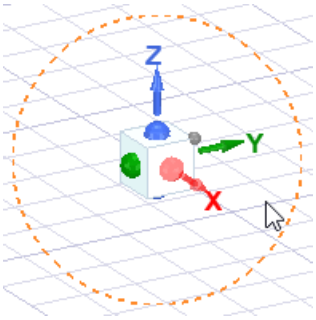


The **Name** that you assign will appear in the **View** > **Apply Orientation** submenu within the Modeler window's shortcut menu. You can specify whether to **Include Zoom/Pan** (in addition to the viewpoint orientation) and whether to **Save Global**, which makes the custom orientation available for all projects.

Using the Orientation Gadget

The **Orientation Gadget** is a tool for quickly manipulating the view orientation of 3D models and 3D plots graphically. It is a convenient alternative to using the **Orient** drop-down menu on the **View** ribbon tab. The Orientation Gadget is enabled by default and appears near the lower left corner of the display area in *Modeler* and *3D Plot* windows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The orange dashed circle appears only when the cursor is near the gadget, and its purpose is discussed later in this topic.

There is a colored dot at the center of each face of the cube and red (X), green (Y), and blue (Z) axes, indicating the axis positive directions for the currently active coordinate system. Additionally, there is a smaller gray dot at the corner where the +X, +Y, and +Z faces meet.

The following list summarizes the results of clicking different parts of the gadget (and optionally dragging the mouse while clicking):

- **Click the large dot at the center of any of the six cube faces:**

  This action rotates the view orientation to place the side of the model that the clicked face represents towards you, parallel with the screen. For example, if you click the dot on the -Y face, the -Y side of the model faces you, which is the *Left* view (+Y viewing direction).

  > **Note:**
  >
  > If the current viewing direction corresponds to one of the global or active user-defined coordinate system axes (±X/Ux, ±Y/Uy, or ±Z/Uz), clicking the axis label or dot at the center of the Orientation Gadget (on the face that is towards you) reverses the view orientation. So, with a single click of the mouse, you can switch from front-to-back, right-to-left, and top-to-bottom (or the inverses of these view orientation changes).

- **Click on one of the gadget axes:**

  This action is equivalent to clicking the dot at the center of the same positive cube face. The axis orientation becomes perpendicular to the screen and towards you. Therefore, the resulting three views are looking at the +X (front), +Y (right), or +Z (top) side of the model (-X, -Y, or -Z viewing direction, respectively).

- **Click the small gray dot at the cube's +X, +Y, +Z corner:**

  This action produces the standard *Isometric* view orientation.

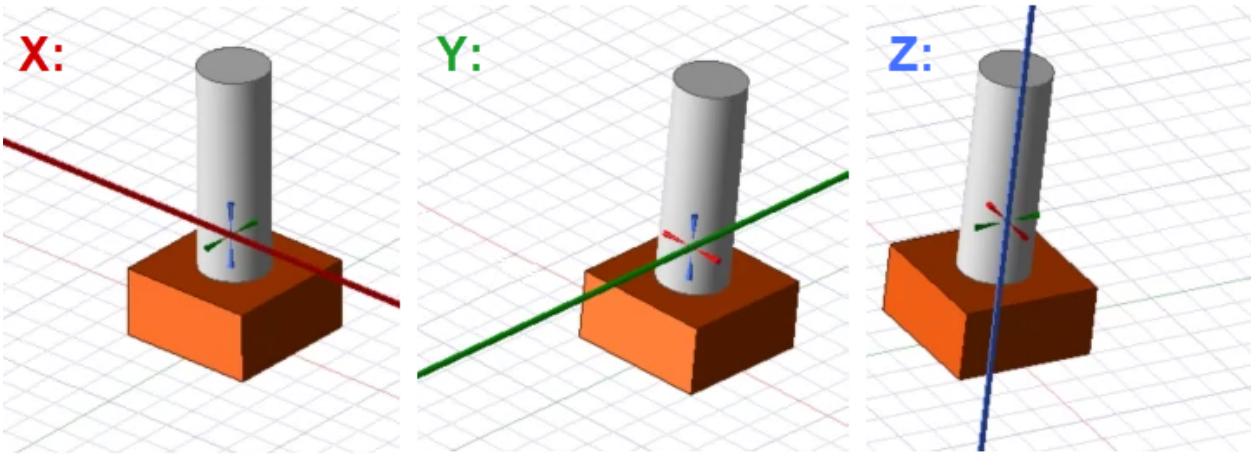
- **Click on or very close to the cube (but not on an axis or dot) and drag the mouse:**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

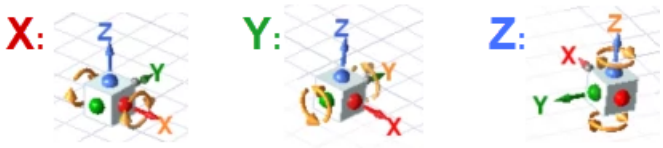
This action freely rotates the view orientation. The rotation center depends on the **Default Rotation About** option selected in the *3D UI Options* dialog box. The behavior is equivalent to clicking and dragging in the display area using the middle button / wheel but is performed with the left mouse button. Additionally, you can constrain rotation to be about any of the three active coordinate system axes by pressing one of the following keys while clicking and dragging the mouse:

- **X + click and drag** to rotate the view orientation about the active **X** axis.
- **Y + click and drag** to rotate the view orientation about the active **Y** axis.
- **Z + click and drag** to rotate the view orientation about the active **Z** axis.

During constrained rotation, a heavy red (X), green (Y), or blue (Z) axis line appears in the Modeler window to provide a clear visual indication of the axis of rotation. These axes pass through the center of rotation, and this center is marked by crosshairs. When dragging the view orientation gadget to rotate the view, the center of rotation is the centerpoint of the drawing canvas. The following image shows the appearance of the rotation axes and center markers:



Additionally, a pair of gold, circular arrows appear on the view orientation gadget indicating the axis of constrained rotation:



While the same constrained rotation functionality for 3D models is also available for 3D reports, the red, green, and blue axes do not appear in 3D report windows during axis-constrained rotation. However, the pair of circular, gold arrows appear on the view

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

orientation gadget in both the Modeler and 3D Report windows to indicate the axis of constrained rotation.

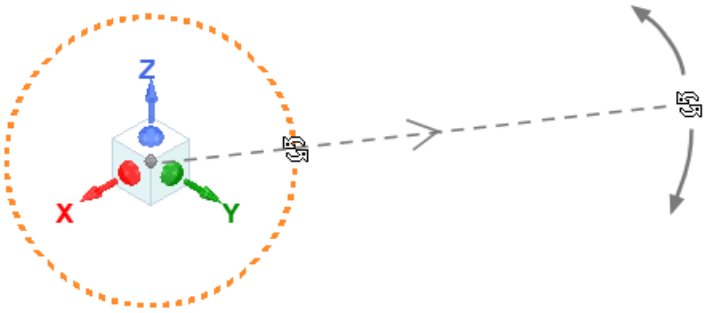
For more information and tips about the usage of this feature, see Rotating or Spinning the View.

- **Orange dashed circle:**

  This circle is used to rotate the current view about an axis normal to the screen. That is, the view rotation is constrained to the plane of the screen regardless of the current viewing direction. The line weight of the dashed orange circle becomes heavier when the cursor is touching it, indicating a correct clicking point. There are three actions, as follows:

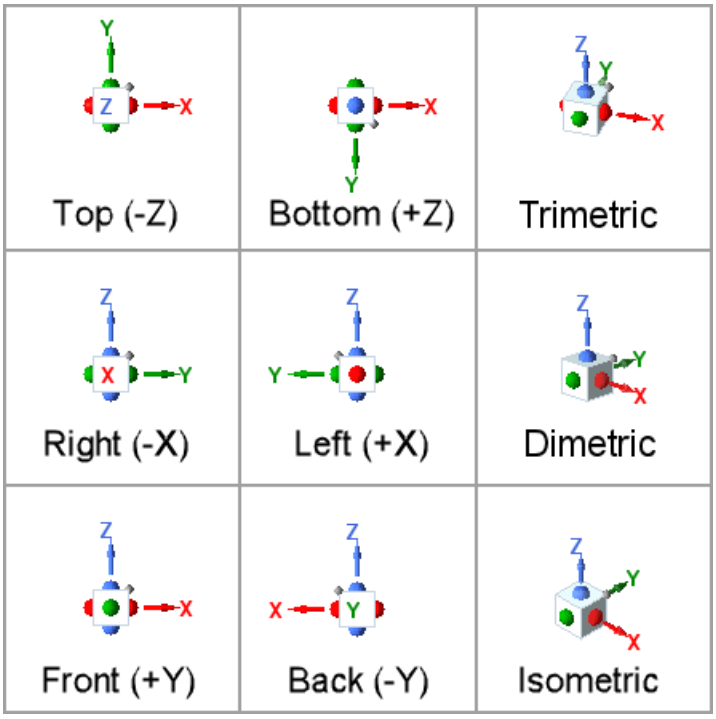
  - **Click anywhere on the circle:** The model rotates **90 degrees clockwise**.
  - **Ctrl+click anywhere on the circle:** The model rotates **90 degrees counterclockwise**. This action and the preceding one are particularly useful for changing the in-plane positive axis directions for standard top, bottom, left, right, front, or back views.
  - **Click and drag:** Freely rotates the model in either direction. Release the mouse button when the desired orientation is reached. While the mouse button remains pressed, the cursor changes to a rotation icon:

    You do not have to keep the cursor on the circle while dragging. You can often achieve more precise rotations with the cursor further away from the orientation gadget. Moving the cursor in a radial direction away from the center of the orientation gadget produces minimal rotation. Then, move it tangentially for precise rotation. The greater the distance from the gadget, the finer the rotational control.



The appearance of the Orientation Gadget for each of the standard view orientation is as shown in the following image:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

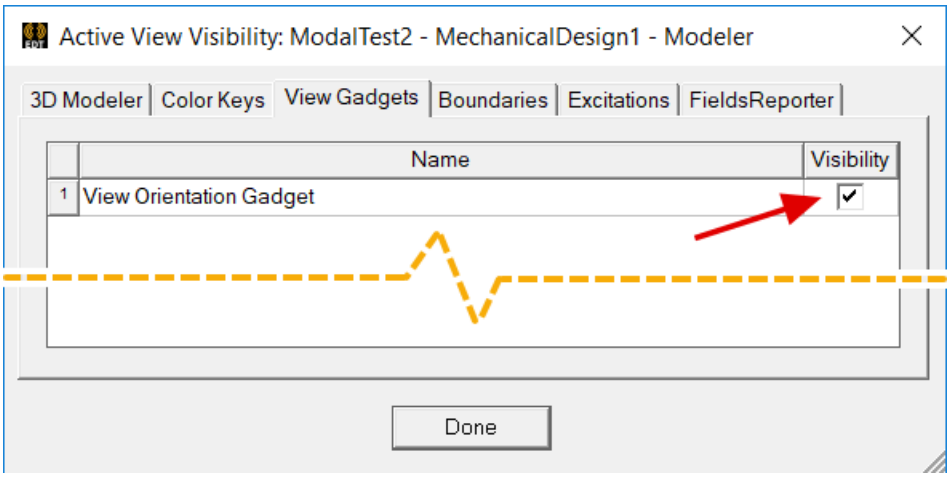
The orientation, colored dots, and axes of the Orientation Gadget correspond to the active coordinate system, whether **Global** or **user-defined**. All standard views are defined in the same manner as they are for the global CS, but Ux, Uy, and Uz directions are used instead of global X, Y, and Z.

Controlling Visibility of the Orientation Gadget

You can enable or disable the Orientation Gadget visibility as follows:

1. Access the *Active View Visibility* dialog box using one of the following two methods:
   - From the menu bar, click **View > Visibility > Active View Visibility**.
   - On the **View, Draw,** or **Model** ribbon tab, click **Hide/Show overlaid visualization in the active view**.
2. Select the **View Gadgets** tab.
3. Select or clear the checkbox in the **Visibility** column to the right of **View Orientation Gadget** to control the gadget visibility.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click **Done** to close the dialog box.

Rotating or Spinning the View

You can rotate the view relative to the Model Center, the Screen Center, the Current Axis (that is, the active coordinate system origin), or the Cursor location. You can activate the rotation modes via the menu bar, ribbon tabs, or shortcut menu. When a *Rotation* command is active, you can click and drag using the **left mouse button** to manipulate the model viewpoint.

Rotation Using Only the Mouse: You can use the **middle mouse button** to rotate the view *without* selecting a *Rotation* command. From the menu bar, click **View** > **Options** to access the *General* tab of the *3D UI Options* dialog box. Here, you can choose the desired **Default Rotation About** option. The view rotates about the **Screen Center** by default. The other available options are rotation about the **Model Center**, **Current Axis** (active coordinate system origin), or **Cursor** location. This setting also controls the rotation center for the **Spin** command.

Rotation using Only the Keyboard: You can use the key combination **Alt** + ***arrow keys*** to rotate the model view about a vertical or horizontal axis. Similar to using the middle mouse button, the center of rotation, through which the vertical or horizontal rotation axis runs, is determined by the **Default Rotation About** selection in the *3D UI Options* dialog box.

For each active *Rotation* command, or when using the middle mouse button, the behavior differs depending on where you click and drag, as follows:

- **Click and drag the cursor near any border of the Modeler window's display area:**

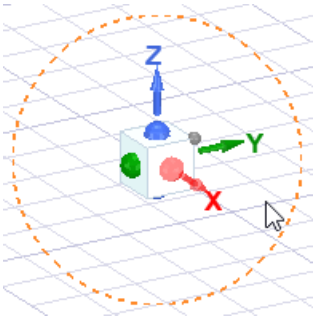
  The view is constrained to rotate about an axis that is perpendicular to the screen. The location of the axis, which is the rotation centerline, depends on the active rotation mode (Current Axis, Model Center, Screen Center, or Cursor) or, when no command is active, the *Default Rotation About* option previously described.

- **Click and drag the cursor within the interior of the display area (not close to a border):**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this case, the view rotates freely about any axis, depending on the direction you drag and the relative positions of the cursor and center of rotation. The rotation center depends on the active rotation mode (Current Axis, Model Center, Screen Center, or Cursor) or, when no command is active, the *Default Rotation About* option previously described.

- **Click and drag on or near the View Orientation Gadget to freely rotate the view:**



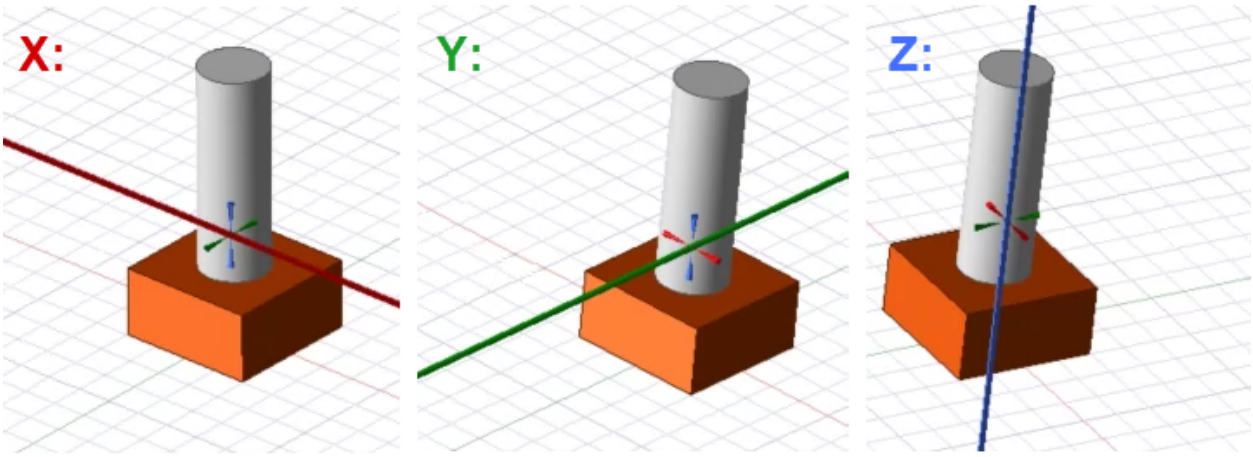
- **Constrained rotation about the X, Y, or Z axis**:

  Use the following hotkeys to constrain rotation of the model viewpoint to be about a coordinate system axis (global or user-defined). These hotkeys work whether you perform the rotation using the **middle mouse button**, any of the **Rotate** commands available on the ribbon (under the *View, Draw,* or *Model* tabs), or any of the **Rotate** commands in the menu bar (**View** > **Interactions** > **...**).

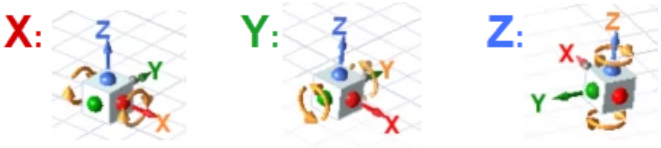
  - **X + click and drag** to rotate the view orientation about the active **X** axis.
  - **Y + click and drag** to rotate the view orientation about the active **Y** axis.
  - **Z + click and drag** to rotate the view orientation about the active **Z** axis.

  During constrained rotation, a heavy red (X), green (Y), or blue (Z) axis line appears in the Modeler window to provide a clear visual indication of the axis of rotation. These axes pass through the center of rotation, and this center is marked by crosshairs. When using the middle mouse button, the center of rotation is the centerpoint of the drawing canvas. For menu and ribbon commands, you can choose alternative rotation centerpoints (model center, cursor location, or origin), which are covered further down in this help topic. The following image shows the appearance of the rotation axes and center markers:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Additionally, a pair of gold, circular arrows appear on the view orientation gadget, indicating the axis of constrained rotation:



While the same constrained rotation functionality for 3D models is also available for 3D reports, the red, green, and blue axes do not appear in 3D report windows during axis-constrained rotation. However, the pair of circular, gold arrows appear on the view orientation gadget in both the Modeler and 3D Report windows to indicate the axis of constrained rotation.

This feature provides a very convenient means of achieving the ideal model viewpoint for setup operations and for presentation purposes, giving you much more control than free rotation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Tip:**
>
> - For best results, drag the cursor in a roughly circular path around the rotation centerpoint. The model orientation will follow the cursor movement, with the angle of rotation matching the angle of the cursor's arc about the same centerpoint.
> - Hold down the X, Y, or Z key throughout the constrained rotation operation. The axial rotation constraint is canceled as soon as you release the key, though the rotation mode remains active. Depending on how you initiated rotation, cancel it by either releasing the middle mouse button, pressing **Esc**, or clicking the command again.
> - During an active constrained rotation operation, you can release the current hotkey and press one of the other hotkeys to choose a different rotation axis, without first canceling the operation.

- **Rotating the view about an axis normal to the screen:**

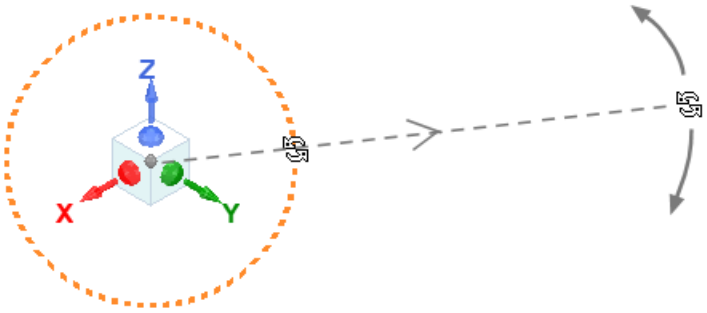
  This orange dashed circle that appears around the view orientation gadget when the cursor is nearby is used to rotate the current view about the screen-normal axis. That is, the view rotation is constrained to the plane of the screen regardless of the current viewing direction. The line weight of the dashed orange circle becomes heavier when the cursor is touching it, indicating a correct clicking point. There are three actions, as follows:

  - **Click anywhere on the circle:** The model rotates **90 degrees clockwise**.
  - **Ctrl + click anywhere on the circle:** The model rotates **90 degrees counterclockwise**. This action and the preceding one are particularly useful for changing the in-plane positive axis directions for standard top, bottom, left, right, front, or back views.
  - **Click and drag:** Freely rotates the model in either direction. Release the mouse button when the desired orientation is reached. While the mouse button remains pressed, the cursor changes to a rotation icon:

    You do not have to keep the cursor on the circle while dragging. You can often achieve more precise rotations with the cursor further away from the orientation gadget. Moving the cursor in a radial direction away from the center of the orientation gadget produces minimal rotation. Then, move it tangentially for precise

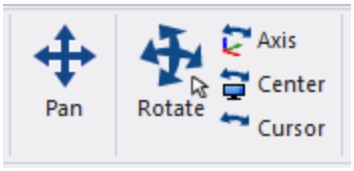
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

rotation. The greater the distance from the gadget, the finer the rotational control.



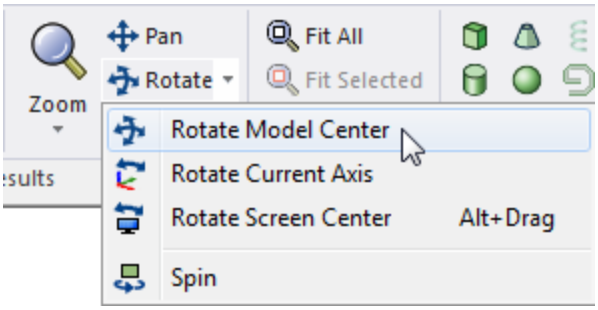
To rotate the view about the **Model Center**:

1. Choose the *Model Center* rotation mode using one of the following four methods:
   - From the **View** ribbon tab, choose **Rotate**:



> **Note:**
>
> The *Center* icon on the *View* ribbon tab is for rotation about the *screen center*, not the model center.

   - From the **Draw** or **Model** ribbon tab, choose **Rotate** > **Rotate Model Center**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Tip:**
>
> After selecting a rotation mode, the selected option becomes the default operation for the remainder of the application session or until a different rotation option is selected. To repeat the last rotation command, you do not have to access the drop-down menu; just click **Rotate**:

- From the menu bar, click **View** > **Interaction** > **Rotate Model Center**.
- Right-click in the Modeler window and choose **View** > **Rotate** > **Rotate Model Center** from the shortcut menu.

2. Click and drag the mouse in the direction you want to rotate the view.
3. To exit the rotation mode, click the command again (per step 1) or press **Esc**.

Current Axis option – To rotate the view about the origin of the currently active coordinate system axes:

1. Choose the *Current Axis* rotation mode using one of the following four methods:
   - From the **View** ribbon tab, click **Axis**.
   - From the **Draw** or **Model** ribbon tab, choose **Rotate** > **Rotate Current Axis**.
   - From the menu bar, click **View** > **Interaction** > **Rotate Current Axis**.
   - Right-click in the Modeler window and choose **View** > **Rotate** > **Rotate Current Axis** from the shortcut menu.
2. Click and drag the mouse in the direction you want to rotate the view.
3. To exit the rotation mode, click the command again (per step 1) or press **Esc**.

To rotate the view around the **Screen Center**:

1. Choose the *Screen Center* rotation mode using one of the following four methods:
   - From the **View** ribbon tab, click **Center**.
   - From the **Draw** or **Model** ribbon tab, choose **Rotate** > **Rotate Screen Center**.
   - From the menu bar, click **View** > **Interaction** > **Rotate Screen Center**.
   - Right-click in the Modeler window and choose **View** > **Rotate** > **Rotate Screen Center** from the shortcut menu.
2. Click and drag the mouse in the direction you want to rotate the view.
3. To exit the rotation mode, click the command again (per step 1) or press **Esc**.

To rotate the view around the **Cursor** location:

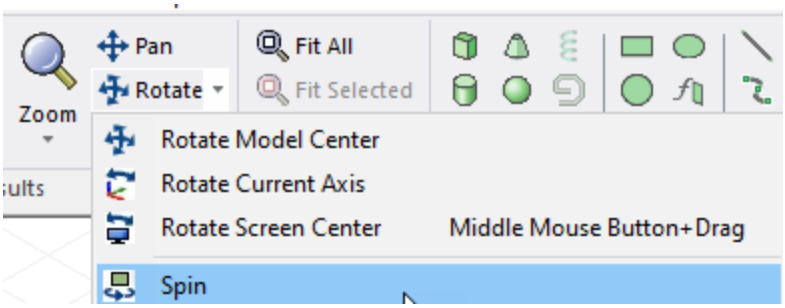
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Choose the *Cursor* rotation mode using one of the following three methods:
   - From the **View** ribbon tab, click **Cursor**.
   - From the menu bar, click **View** > **Interaction** > **Rotate Cursor**.
   - Right-click in the Modeler window and choose **View** > **Rotate** > **Rotate Cursor** from the shortcut menu.
2. Click and drag the mouse in the direction you want to rotate the view.
3. To exit the rotation mode, click the command again (per step 1) or press **Esc**.

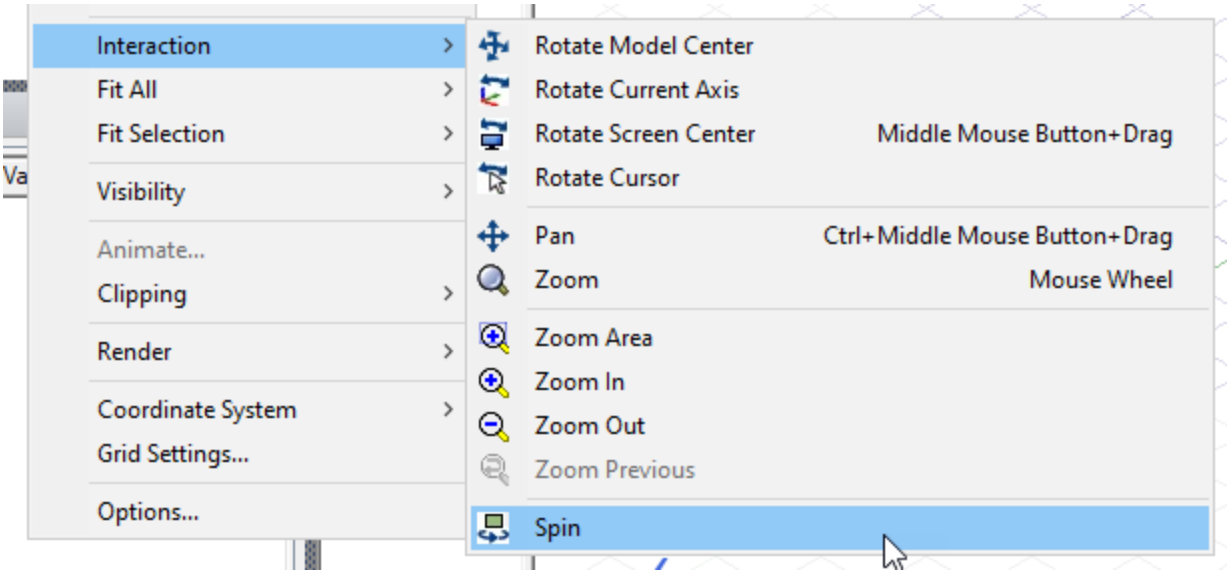
Spin Mode: This tool is useful for capturing animations (for presentation purposes) that show a 360 degree view of the model geometry, mesh, or simulation results.

To **Spin** the model view (that is, to keep it rotating after you release the mouse button):

1. Choose the *Spin* mode using one of the following two methods:
   - From the **Draw** or **Model** ribbon tab, choose **Rotate** > **Spin**.



   - From the menu bar, click **View** > **Interaction** > **Spin**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

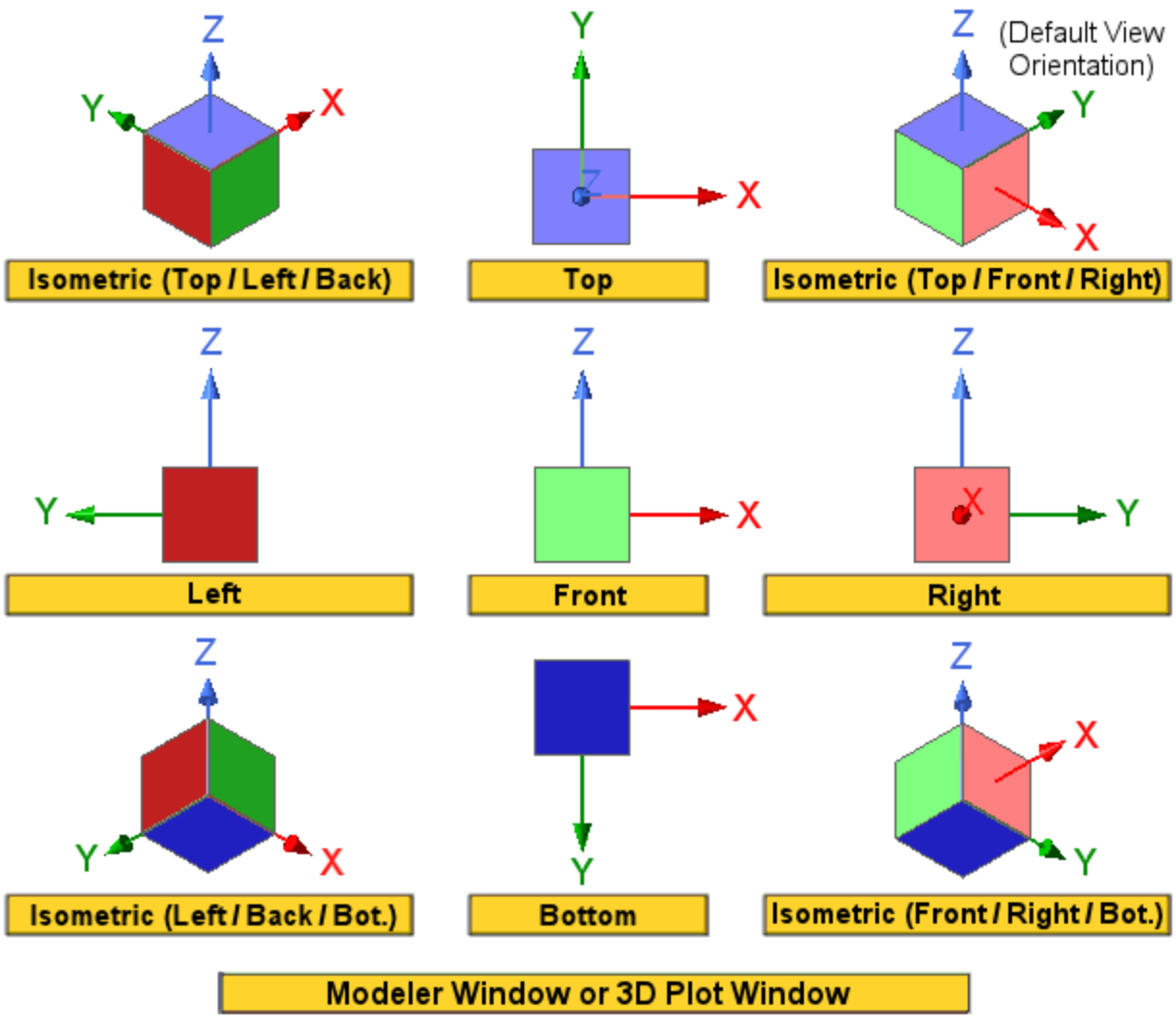
2. Click and drag the mouse in the direction you want to spin the view. Release the mouse button while still dragging.
    - The model will continue to spin in the specified direction. The spinning speed will be proportional to how fast you were dragging the cursor when you released the mouse button.
    - The center of rotation for the *Spin* mode is controlled by the *Default Rotation About* option selected in the 3D UI Options dialog box.
    - The *X, Y,* and *Z* keys also work as modifiers for the *Spin mode* (for constrained spinning about one of the active coordinate system axes).
3. To exit the *Spin* mode, click the command again (per step 1) or press **Esc**.

> **Note:**
>
> If rotation response is slow, especially for complex models:
>
> - For some graphics cards, you can improve the performance by setting **NVIDIA Control Panel** > **3D Settings** > **Manage 3D Settings** > **Global Settings** > **Global Presets: Workstation App - Dynamic Streaming**.
>
>   For more information about graphics cards, navigate to the appropriate Windows or Linux Ansys Installation Guide and search for **OpenGL**.

Changing the Model View with Alt+Double-Click Areas

In the following figure, the colored boxes show the nine view orientations you can obtain by using **Alt + double-click** in the corresponding areas of the *Modeler* window. Each face is a unique color and is named according to the standard view orientation names. The **Alt + double-click** view navigation method is also applicable to report windows containing *3D Plots* (such as a 3D Polar Gain plot).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The *Back, Trimetric,* and *Dimetric* standard view orientations are not available via *Alt + double-click* zones. For these three views, use other available ribbon commands or shortcut menus. For user-defined views, use the *Update View Orientation* dialog box or the Modeler window's shortcut menu (*Right-click* > *View* > *Apply Orientation*).

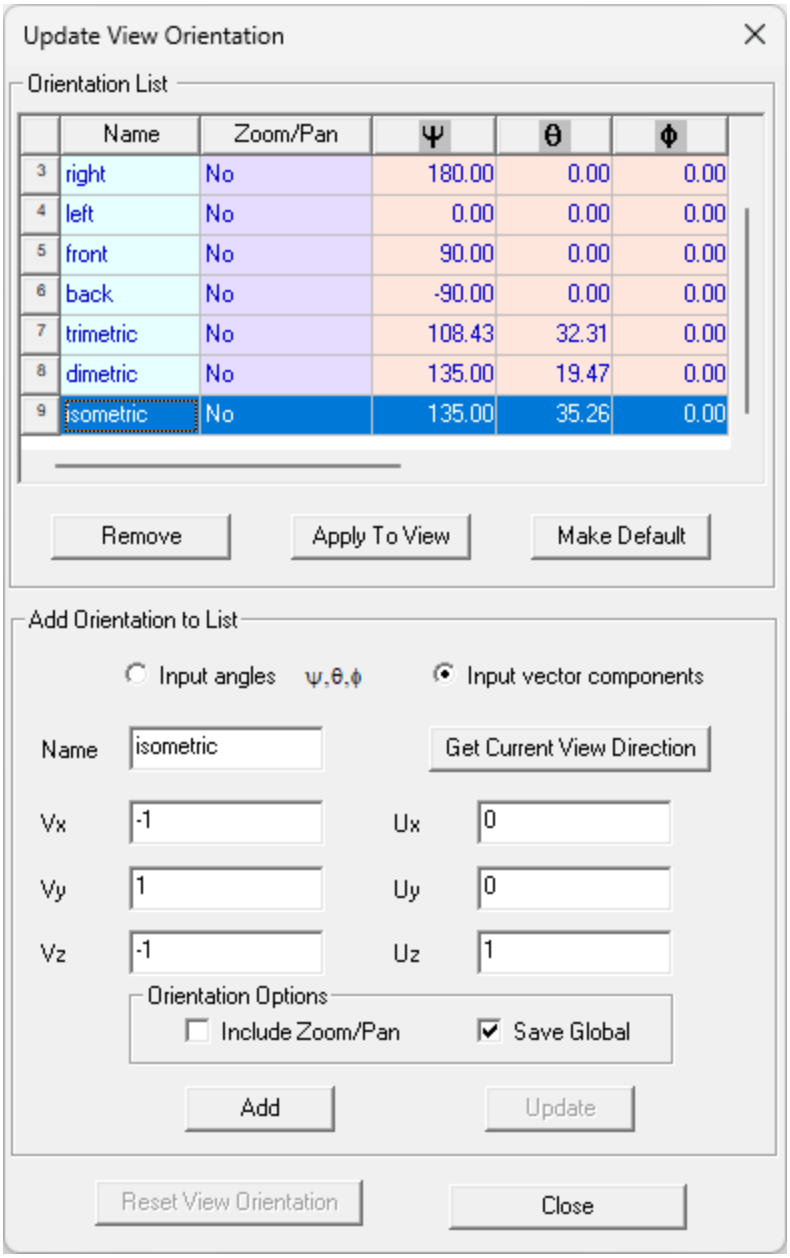
Applying a Default View Orientation

To apply a default viewing direction to the active view window:

1. Access the *Update View Orientation* dialog box using one of the following two methods:
   - From the **View** ribbon tab, click **List**.

     No icon is associated with this command.
   - From the menu bar, click **View** > **Modify Attributes** > **Orientation List**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click one of the orientation names listed in the orientation list.
3. Click one of the orientation names listed in the viewing directions list.
4. Click **Apply**.

   The viewing direction will be applied to the active view window.

   You can use the **Reset View Orientation** button to restore the view to the original angle.
5. Click **Close**.

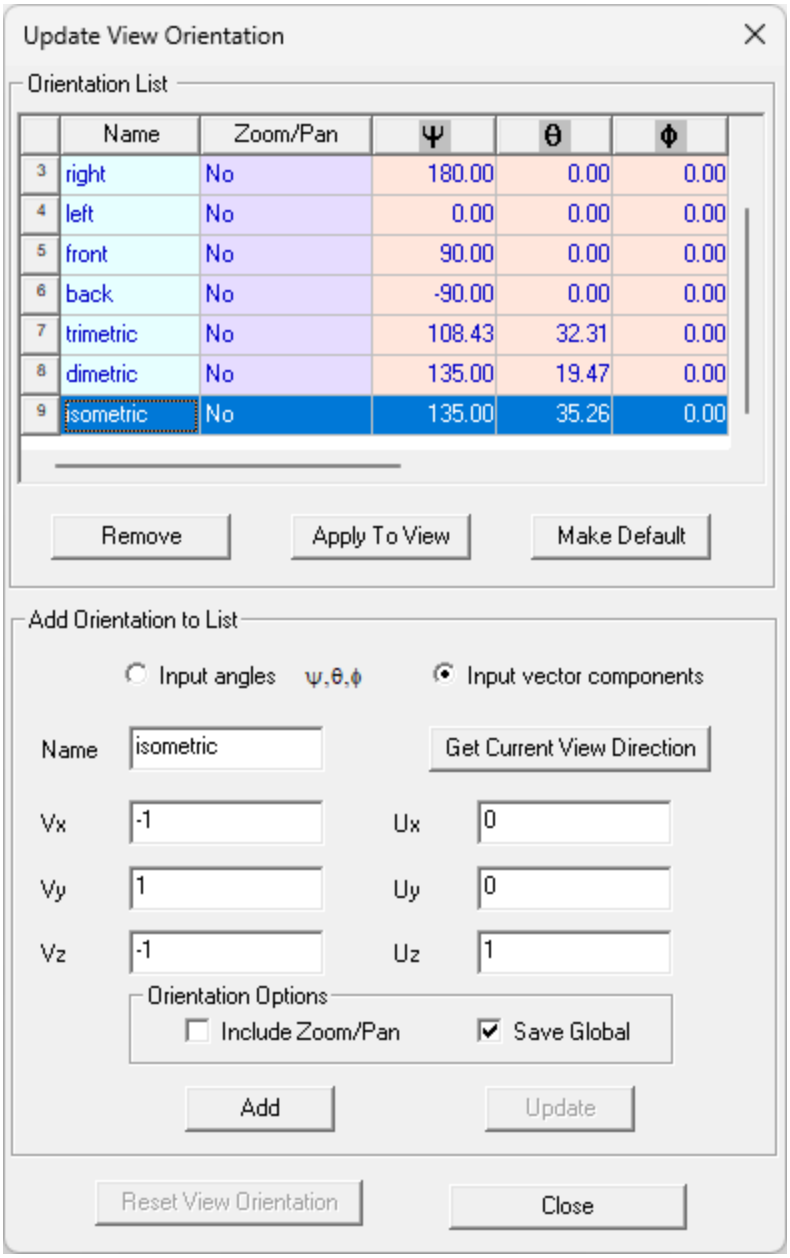
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Applying a New View Orientation

To create and apply a new viewing direction to the active view window:

1. Access the *Update View Orientation* dialog box using one of the following two methods:
   - From the **View** ribbon tab, click **List**.

     No icon is associated with this command.
   - From the menu bar, click **View** > **Modify Attributes** > **Orientation List**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. To create a viewing direction that is based on a default viewing direction, click the existing orientation name in the viewing directions list.

   To create a viewing direction based on the current view in the **Modeler** window, click **Get Current View Direction**.

   - To modify the selected orientation's vector components, select **Input vector components** under **Add Orientation to List** and then modify the values in the **Vx, Vy**, or **Vz** text boxes, and the **Up vector** boxes for **Ux, Uy,** and **Uz**.
   - To modify the selected orientation's input angles, select **Input angles** under **Add Orientation to List** and then modify the values in the **psi, phi** and **theta** text boxes.
3. Type a name for the new orientation in the **Name** text box.
4. Click **Add/Edit**.

   The new orientation is added to the list of viewing directions.
5. Click **Make Default** if you want the new viewing direction to be the initial viewing direction when a **3D Modeler** window is opened in the current project or future projects.
6. Click **Close**.

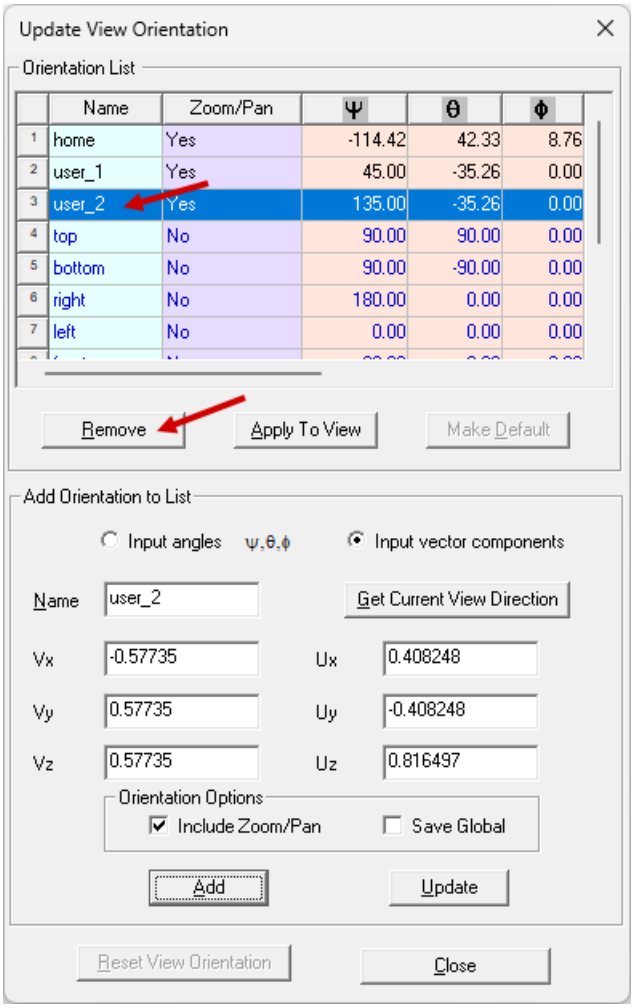
Removing an Orientation

To remove a viewing direction from the list in the orientation settings dialog box:

1. Access the *Update View Orientation* dialog box using one of the following two methods:
   - From the **View** ribbon tab, click **List**.

     No icon is associated with this command.
   - From the menu bar, click **View** > **Modify Attributes** > **Orientation List**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click the viewing orientation you want to delete from the list of names.
3. Click **Remove**.

   The viewing direction is removed from the list.

> **Note:**
>
> This operation cannot be undone.

Fitting Objects in the View Window

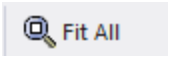
What do you want to do?

- Fit all objects or traces in a view window
- Fit selected objects in a view window

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fitting All Objects in a View Window

To fit all the views: click **View** > **Fit All** > **All Views** or use the equivalent icons in the **Draw** or **Model** ribbon tabs.



All view windows displaying the active design change to include all model objects.

To fit only the active view: click **View** > **Fit All** > **Active View**.

The view in the active Modeler window changes to include all model objects.

> **Tip:**
>
> Alternatively, fit all objects in the active view window using one of the following methods:
>
> - Press **Ctrl+D**
> - Right-click in the Modeler window and then click **View** > **Fit All** on the shortcut menu.

When **Fit All** is used in a **Report** view, the window is automatically rescaled to fit all traces in the window and the axis label and ticks are rescaled.

Fitting Selected Objects in a View Window

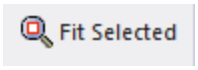
To fit one or more selected objects, 3D Components, or User Defined Models in the Modeler window or traces selected in the Reporter:

1. When you are working on a model view, select the objects you want to fit in the view. When you are working on a report, select the traces you want to fit.
   - To fit the selection in the active view window: Click **View** > **Fit Selection** > **Active View**.
   - To fit the selection in every open view window of the active design: Click **View** > **Fit Selection** > **All Views.**

     > **Tip:**
     >
     > Alternatively, fit the selection in the active view window by clicking **View** > **Fit Selection** on the shortcut menu.

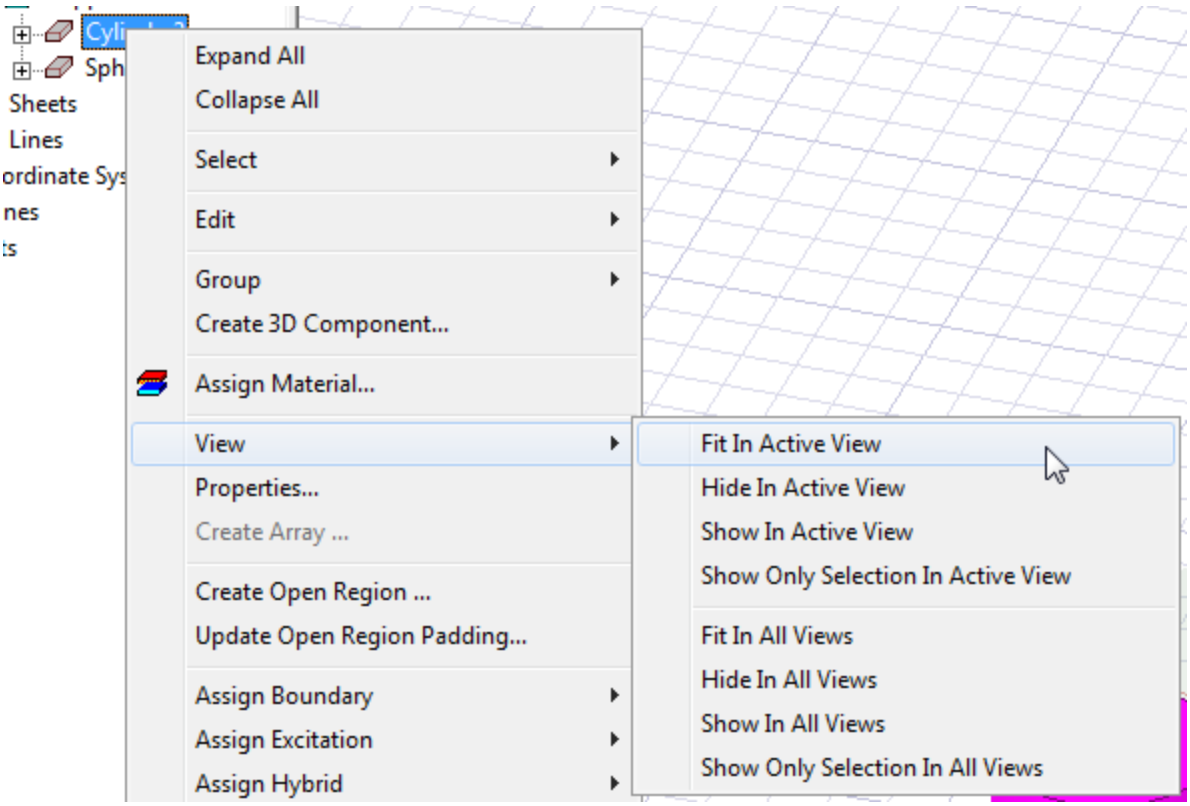
   You can also use the Fit Selected icon in the **Draw** and **Model** ribbon tabs:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To fit one or more objects or components selected in the History Tree:

1. Select the objects or components of interest and right-click to display the shortcut menu.
2. Select **View** > **Fit in Active View** or **Fit in All Views**:



.The view adjusts to fit the select objects.

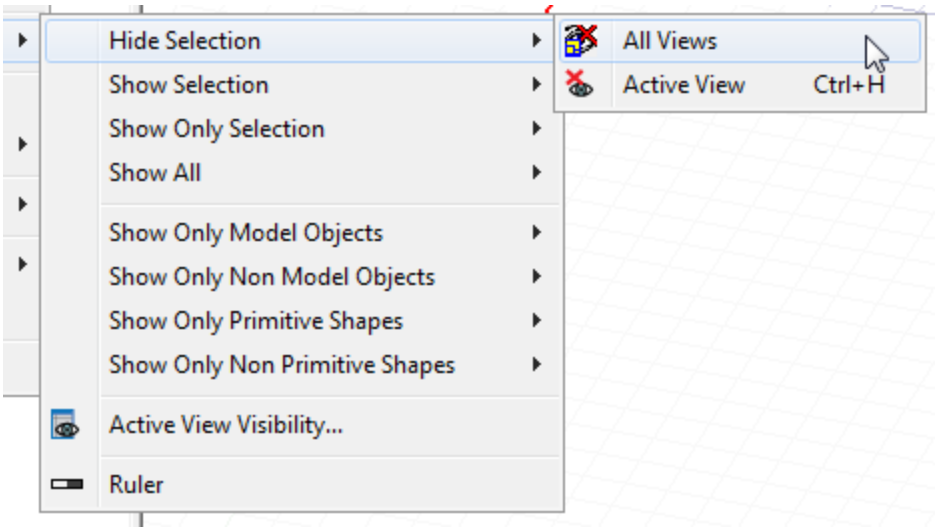
Hiding Objects from View

To hide selected objects.

1. Select the object you want to hide from view. This enables the menu commands and **View, Draw,** and **Model** ribbon tabs icons for hiding objects.
2. Click **View** > **Visibility** > **Hide Selection** and one of the following commands:
   - **All Views** to hide the selected object in every open view window.
   - **Active View** to hide the selected object in the active view window. You can also use **Ctrl+H** to hide a selected object.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also right-click in the **Modeler** window for a short-cut menu and use the **View** commands for **Hide Selection** or **Show Only Selection**.



You can also use the **Hide selected objects** icon in the **View, Draw,** or **Model** ribbon tabs to hide selected objects in the active view.

The objects you selected are hidden.

If there are many objects, you may find it easier to hide objects using the Active View Visibility dialog box. You can also choose to Show only selected objects in all or active views, effectively hiding all unselected objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

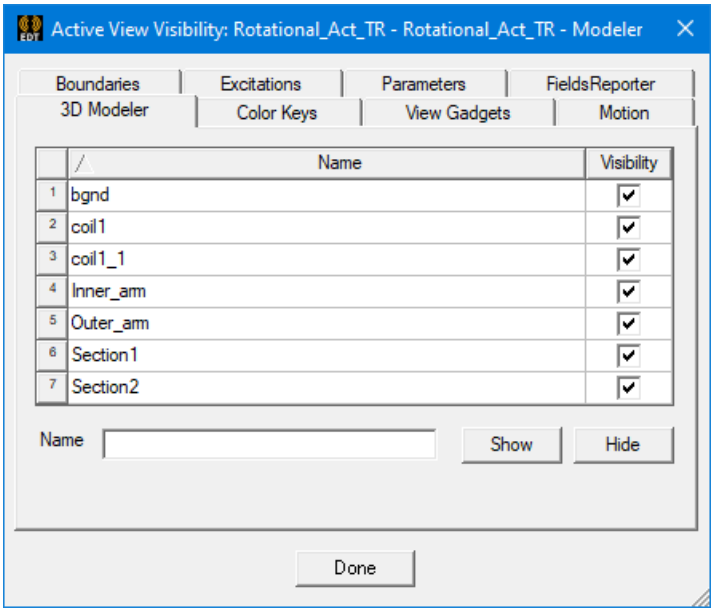
> **Note:**
>
> Hiding boundaries also turns off a check for boundary overlaps during boundary assignment. In the case of very large models with many boundaries, hiding boundaries can prevent delays during boundary assignment. Full model validation will subsequently check for boundary overlaps.

Object visibility is saved with the project.

Showing Objects

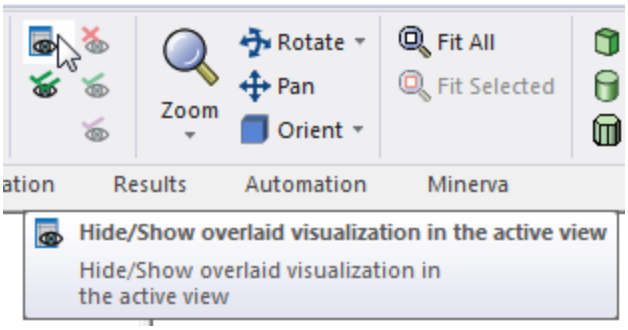
You can show or hide objects in several ways, using the **Active View Visibility** dialog box for individual objects, or **Visibility** menu selections for All Objects, Only Selection, Model Objects, Non-model Objects, Primitive Shapes, or Non-primitive shapes.

The **Active View Visibility** dialog box contains tabs for 3D Modeler, Color Keys, Array Setup, Boundaries, Excitations, and Fields Reporter. Select the tab for the objects you want to show or hide, and select or deselect **Visibility**.

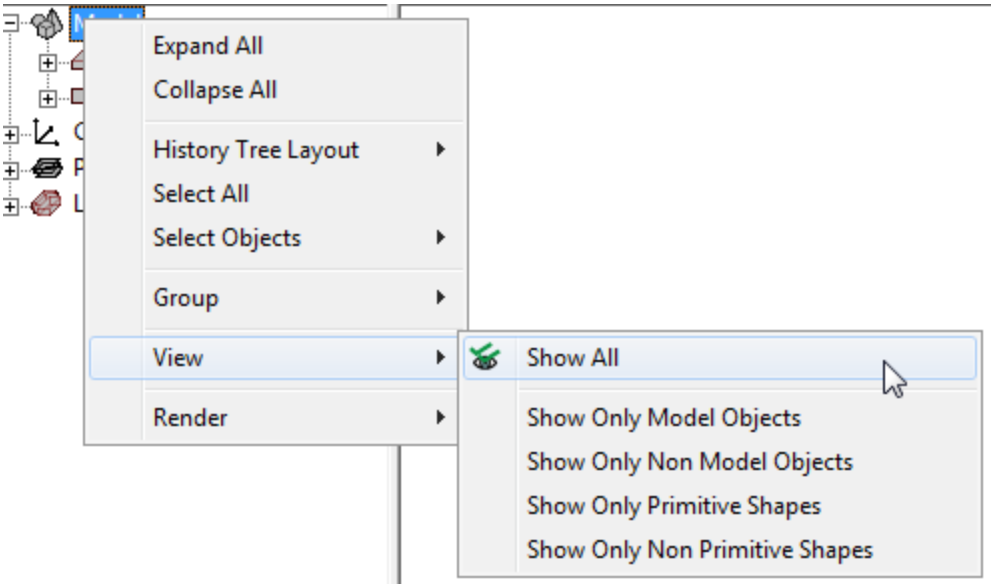


You can access the dialog box from the menu bar (**View** > **Visibility** > **Active View Visibility...**) or by clicking the Hide/Show visualization icons in the **View, Draw,** or **Model** Ribbon tabs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Visibility** menus are available on the **View** menu and as shortcut menus in the *Modeler* window and in the History Tree when you right-click the **Model** icon:



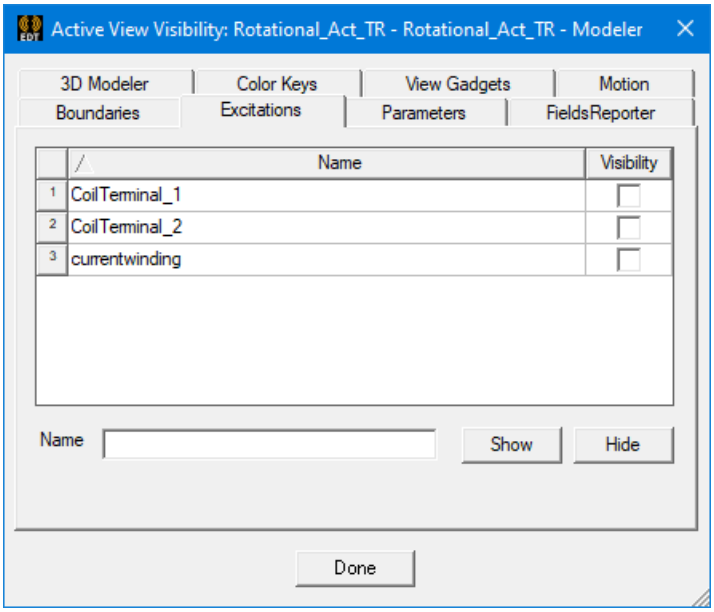
To show *one or more objects* that are currently hidden by using the *Active View Visibility* dialog box:

1. Click **View** > **Visibility** > **Active View Visibility...**, or click the visibility icons on the **View, Draw,** or **Model** ribbon tabs.

   The **Active View Visibility** dialog box appears.

2. Select the tab for the objects you want to show or hide. The dialog box contains tabs for 3D Modeler objects: Color Key objects, Boundaries, Excitations, and Fields Reporter objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Under the tab you need, select the **Visibility** option for the objects you want to show in the active view window.
    - For designs with large numbers of objects, you can resize the dialog box for easier selection.
    - By default, objects are listed in alphabetical order. You can invert the order by clicking the Name bar above the Name fields. A triangle in the bar indicates the direction of the listing.
    - You can also use the **Name** field to type in an object name and apply the visibility via the **Show** and **Hide** buttons.

    The objects you select and designate as Visible (by selecting the property or using Show) reappear.

> **Note:**
>
> Hiding boundaries also turns off a check for boundary overlaps during boundary assignment. In the case of very large models with many boundaries, hiding boundaries can prevent delays during boundary assignment. Full model validation will subsequently check for boundary overlaps.

To show *all objects* that are currently hidden by using the **Visibility** menu:

1. Click **View** > **Visibility** > **Show All** or use the shortcut menu in the Modeler window or History Tree and one of the following commands:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **All Views** to show all objects in every open view window
- **Active Views** to show all objects in the active view window.

The selected objects reappear.

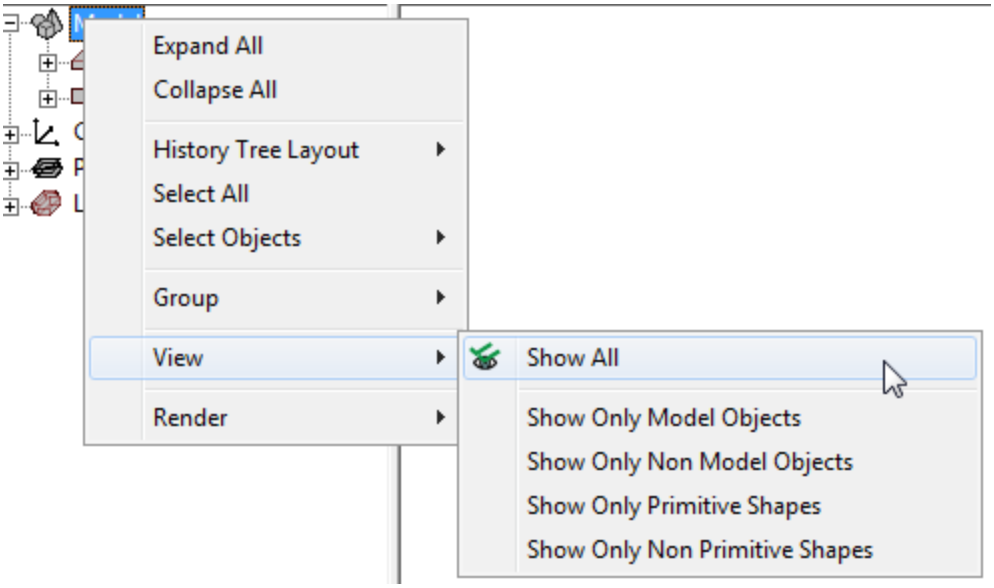
To show *selected objects* that are currently hidden:

1. Select the object. Hidden items are selected once the node corresponding to them is clicked in the history tree.
2. Click **View** > **Visibility** > **Show Selection** and one of the following:
   - **All Views** to show selected objects in every open view window
   - **Active Views** to show selected objects in the active view window.

   You can also use the **View, Draw,** or **Model** ribbon tab icons to **Show selected objects in active views**. Or, open the **Active View Visibility** dialog box.

To show one or more *3D components* that are currently hidden:

1. In the History Tree, select one or more 3D components. This activates the **Hide/Show** icons on the **View,Draw,** and **Model** ribbon tabs and the **View** menu selections.
2. You can right-click to display the shortcut menu and select **View** > **Show All**.



Object visibility is saved with the project.

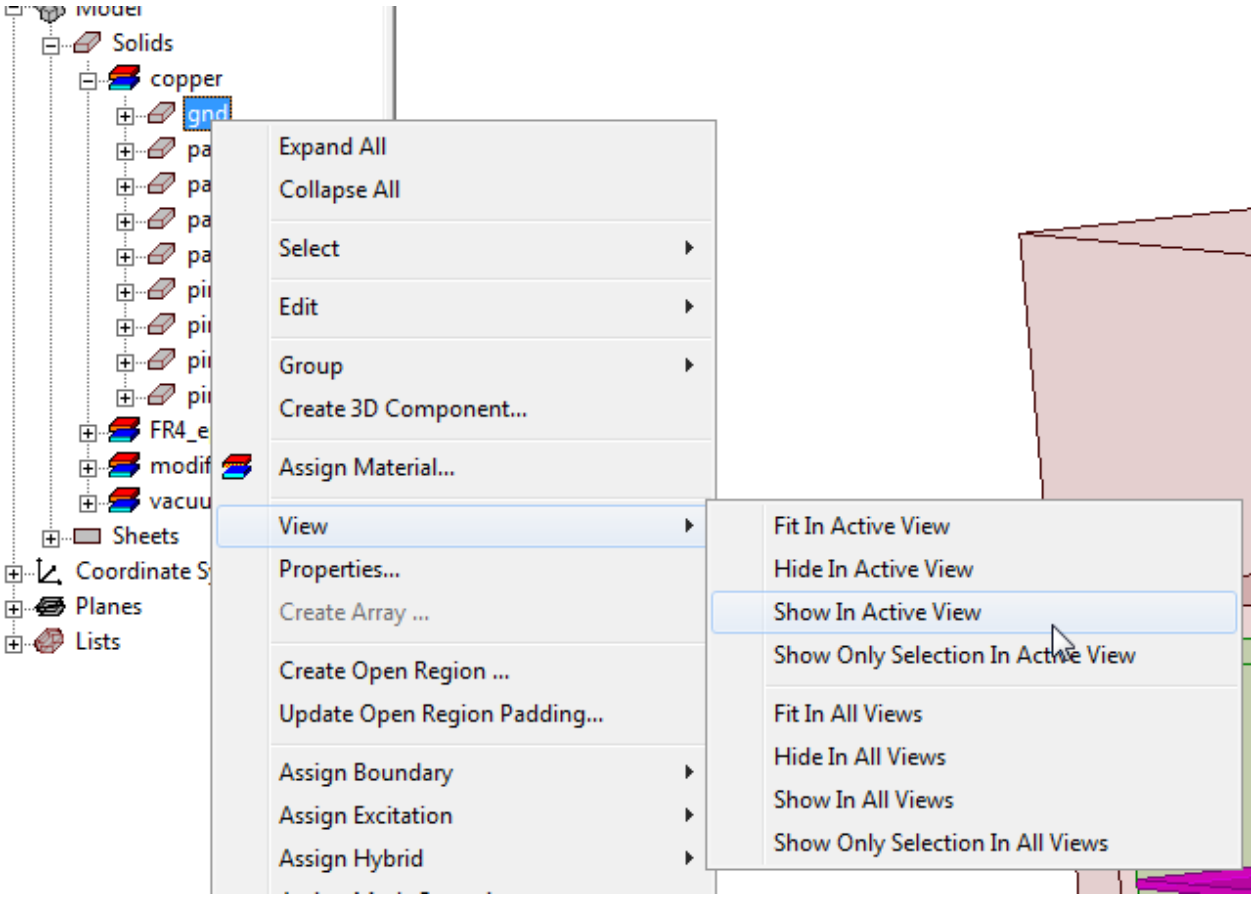
Showing Only Selected Objects in All or Active Views

To show only *selected objects*:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

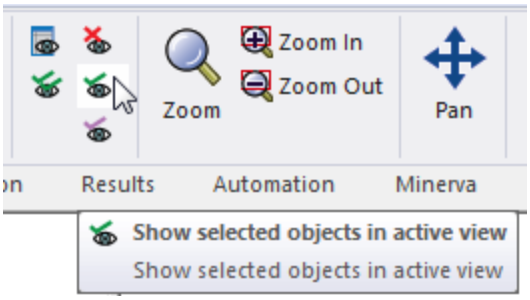
1. Select the object from the Project Manager or History Tree. Hidden items are selected once the node corresponding to them is clicked in the History Tree.
2. Click **View** > **Visibility** > **Show Selection** and one of the following:
   - **All Views** to show selected objects in every open view window
   - **Active Views** to show selected objects in the active view window

   You can also right-click an object in the History Tree and use the shortcut menu:
3. You can right-click to display the shortcut menu and select **View** > **Show in Active View, Show in All Views,** or **Show Only Selection In Active View**.



You can also use the **View/Draw/Model** ribbon icons to **Show selected objects in all views** and **Show selected objects in active views**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The selected objects reappear.

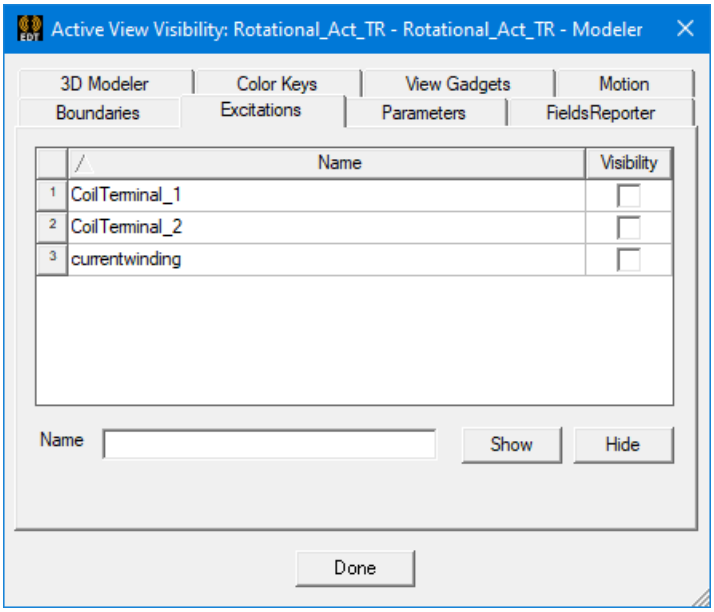
Active View Visibility Dialog Box

If there are many objects, it may be easier to show or hide objects using the **Active View Visibility** dialog box.

1. Click **View** > **Visibility** > **Active View Visibility**, or click the equivalent **Hide/Show** icon on the **View, Draw,** or **Model** ribbon tabs.

The **Active View Visibility** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select the tab for the objects you want to show or hide. The dialog contains tabs for 3D Modeler objects, Color Key objects, Boundaries, Excitations, and Fields Reporter objects.
    - For designs with large numbers of objects, you can resize the dialog for easier selection.
    - By default, objects are listed in alphabetical order. You can invert the order by clicking the Name bar above the Name fields. A triangle in the bar indicates the direction of the listing.
    - You can also use the **Name** field to type in an object name and apply the visibility via the **Show** and **Hide** buttons.
3. Under the tab, clear the **Visibility** option for the objects you want to hide in the active view window.

    The objects you designate are hidden.

> **Note:**
>
> Hiding boundaries also turns off a check for boundary overlaps during boundary assignment. In the case of very large models with many boundaries, hiding boundaries can prevent delays during boundary assignment. Full model validation will subsequently check for boundary overlaps.

Object visibility is saved with the project.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rendering Objects as Wireframes or Solids

To render (display) *all* objects in the view window as wireframe outlines, flat-shaded solids, or smooth-shaded solids:

1. Click **View**> **Render** and click one of the following:
    - **Wireframe**:

      The objects in the view window are displayed as skeletal structures, enabling you to see all sides of the objects at one time.

      You can also use the **F6** key to display a wireframe view.

    - **Smooth Shaded**:

      The objects in the view window are displayed as shaded objects with smooth edges.

      You can also use the **F7** key to display a smooth shaded view.

To render a *single* object in the view window as a *wireframe outline*:

1. Select the object you want to render as a wireframe.
2. In the **Attribute** tab of the docked **Properties** window, select **Display Wireframe**.

Setting the Default View Rendering Mode

To set a default rendering mode for all objects created in the active design and in future designs:

1. Click **Tools** > **Options** > **Modeler Options**.
2. Click the **Display** tab.
3. Select one of the following options from the **Default View Render Mode** drop-down menu:
    - **Wireframe**.

      The objects in the view window will be displayed as skeletal structures, enabling you to see all sides of the objects at one time.

    - **Smooth Shaded**.

      The objects in the view window will be displayed as shaded objects with smooth edges.

4. Click **OK**.

   The rendering mode will be applied to all new objects you create.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Enhanced Display of Material Color and Transparency

You can specify the use of material color and transparency display by using the **View** > **Render** > **Enhanced Display** command. Any changes reset the default.

To set the Enhanced Display for the active modeler window:

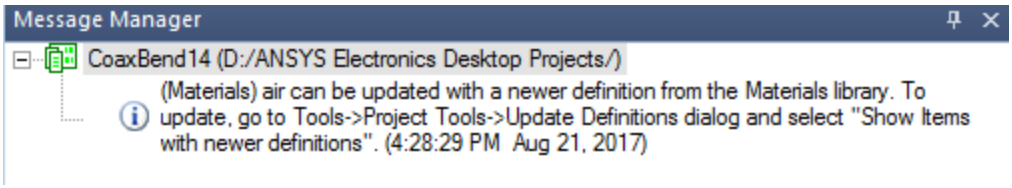
1. Click **View** > **Render** > **Enhanced Display** or press **F8**.

Enhanced Display overrides part color and transparency properties with material settings for all the parts visible in the active view. By default it is off (unchecked on the **View> Render** menu) and part attribute color and transparent values are applied.

It is possible for the view to be in the “Enhanced Display” mode but some of the parts in the model could have “Use Material Appearance” unchecked. In this case you can edit the **Color** or **Transparent** values, or select **Use Material Appearance** in the docked **Properties** window, but they are not applied to the active view visualization. This resembles how the **Wire Frame** setting is handled currently.

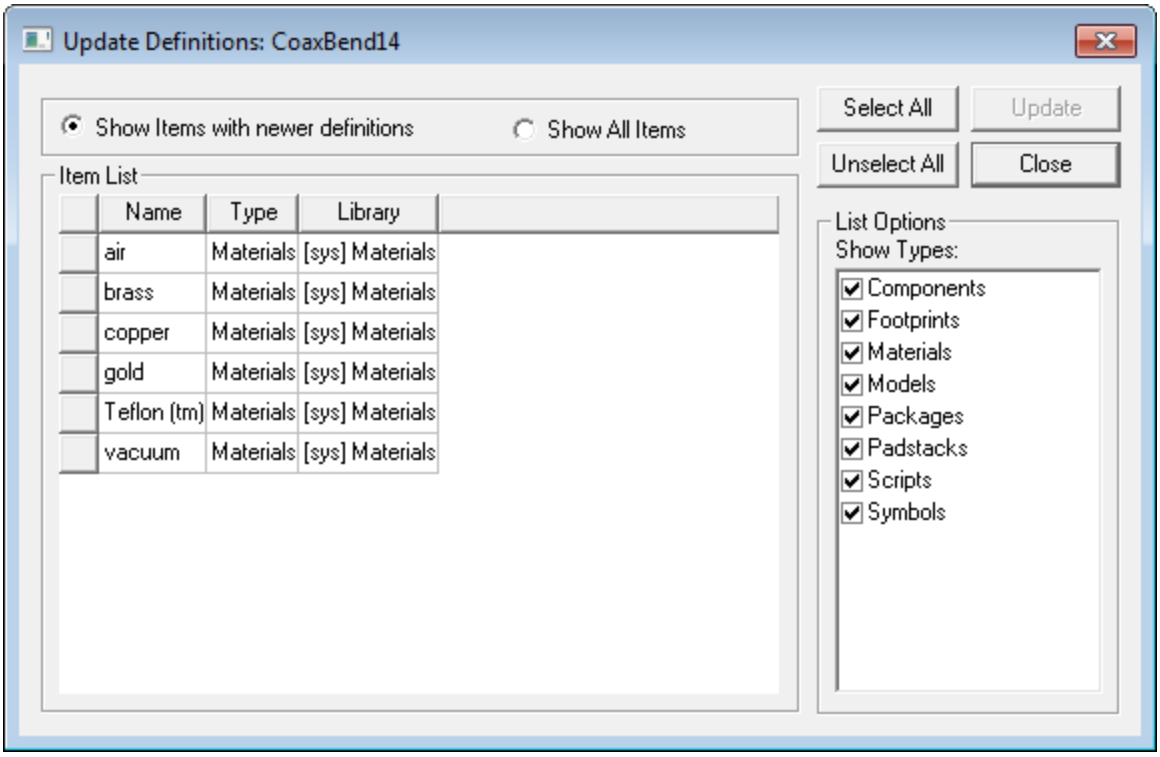
Sheet objects in the 3D modeler do not have a material assignment, but they could have a boundary assignment with material assigned to the boundary. In this case sheet object visualization resembles solid object visualization, where the material appearance from the assignment is used.

When you are working with a legacy project and use **View** > **Render** > **Enhanced Display**, if there are updates to the materials definitions for appearance, you will receive an info message informing you that the material definition was updated.



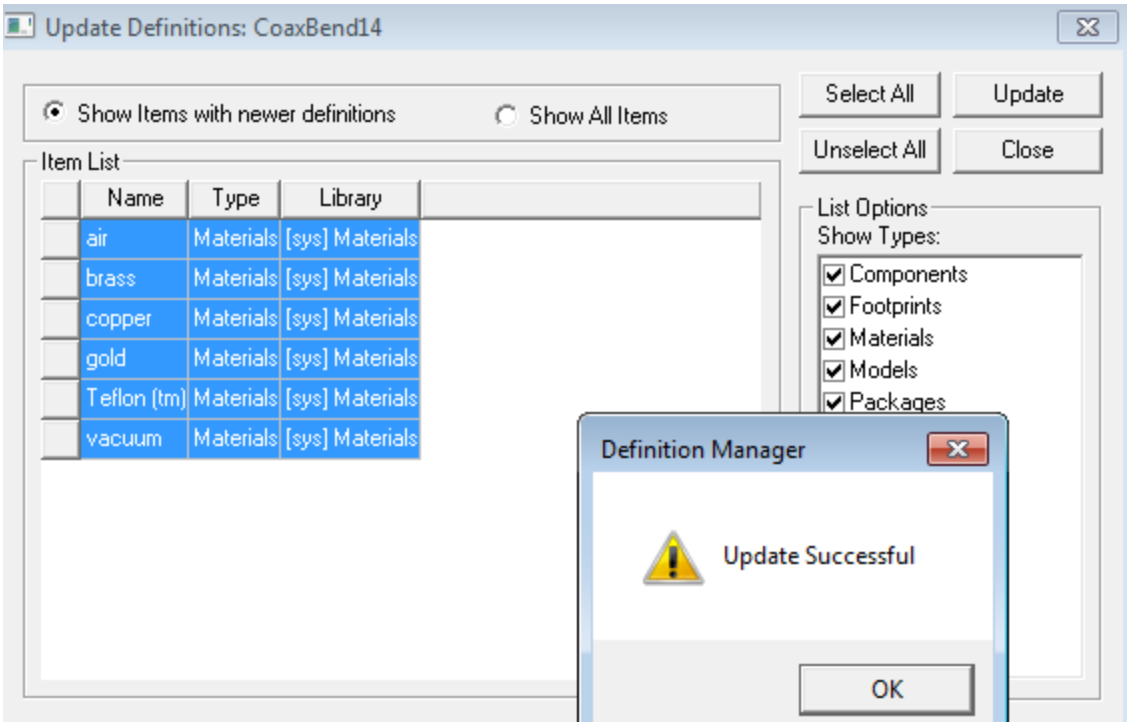
If you want to update, click **Tools** > **Project Tools** > **Update Definitions** to open the **Update Definitions** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Here you can multi-select rows of materials to update them. In case of updates available from multiple libraries, you can choose the library location. Selecting material rows enables the **Update** button. Selecting **Update** then produces a message on a successful update.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Setting the Curved Surface Visualization

You can specify the faceting for rendering true curves by using the **View** > **Render** > **Curved Object Visualization** command. There are two options for control--Maximum surface deviation and Maximum normal deviation. This resembles the Mesh surface approximation settings. Reduce either or both of the allowed deviations to improve the image quality. Improved image quality comes at the cost of increased CPU consumption. Changes apply to the current model until they are changed again. Any changes reset the default. The default gives satisfactory results (cpu/memory consumption vs. graphical display) for various model complexities.

When you change Curved Object Visualization settings and apply them to a design, those settings are saved with design unless you change it again. That means when you open the design again, it will apply saved visualization settings and NOT the default settings. Because this affects the CPU and memory required to open the project, typically, you should not save a project with other than the default settings.

To set the Curved Object Visualization for the active modeler window:

1. Click **View** > **Render** > **Curved Object Visualization**.

   This command displays the **Curved Object Visualization**dialog for the active modeler window. The dialog contains areas for setting the Maximum Deviation, and the Maximum Normal deviation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Set the Maximum Deviation by first selecting from the radio buttons for **Ignore**, set as **Relative Deviation** or set as **Absolute Deviation**. Selecting the later two radio buttons enables the value field.

   When set as **Relative Deviation**, the actual surface deviation depends on the model size. For example, sphere with a radius of 10 has same number of facets as a sphere with a radius of 1. This means that CPU cost does not increase based on the model dimension.

   When set as **Absolute Deviation**, the maximum surface deviation for both the spheres will be approximately same since a bigger sphere has more facets than a smaller one. This means that the most CPU cost applies to the larger objects.

3. If you selected the radio buttons for Relative or Absolute Deviation for Maximum Deviation, enter a value in the field.
4. To change the **Maximum Normal Deviation**, enter a value in the text field. Units are degrees.

   > **Note:**
   >
   > Wire bodies cannot be rendered with a Maximum Normal Deviation value less than 1 degree. When using a setting less than 1 degree all wire bodies will be rendered with a setting of 1 degree and all closed bodies will be rendered with the dialog box setting.

5. The **Save As Default** button lets you Save any values you change to the drop-down menus for the fields.
6. The **Restore Default**s button lets you return to the original values. Any values you provided through **Save As Default** remain on the drop-down menus for the fields for surface and normal deviations.
7. Click **Apply** to apply the current values to the active modeler window, and **Close** or **Cancel** to close the dialog box without changing settings.

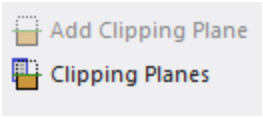
Using Clip Planes

The **View** > **Clipping** > **Clip Plane** command lets you define a clip plane that you can use to interactively make any desired cut-away view of a model. If you use **Edit** > **Copy Image** or **Modeler** > **Export**> **[image format]** with the clip plane active, the image shows the clipped plane. When parts of the model are hidden by a clip plane, model selection works as though only the visible parts are present.

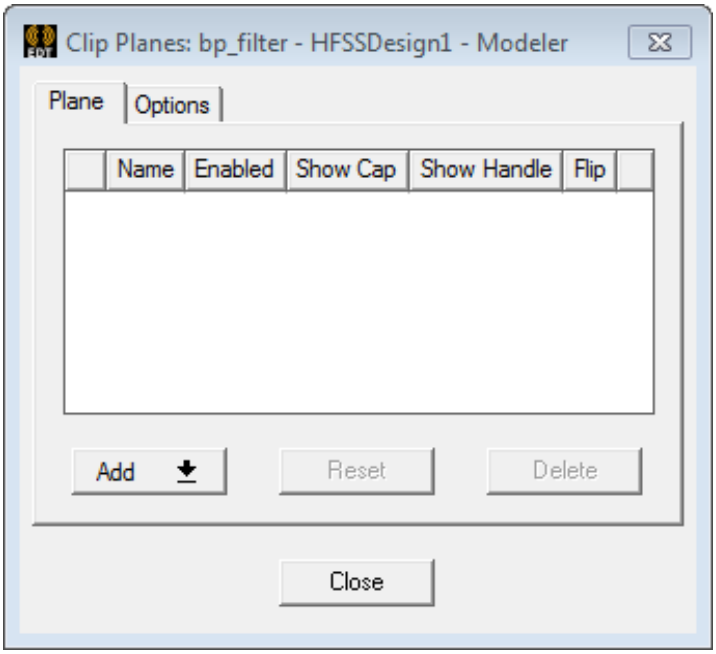
To add a clip plane:

1. Click **View** > **Clipping** > **Clip Plane** or click the **Clipping Planes** icon on the **View** ribbon tab.

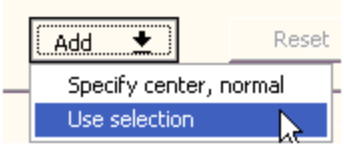
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This displays the **Clip Plane** dialog box with the **Plane** tab selected.

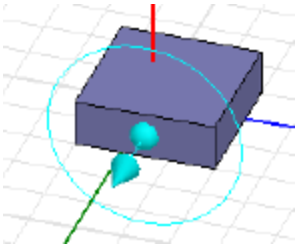


2. The **Add** button contains a drop-down menu with choices for **Specify center, normal**, and **Use selection**. If you want to use a selection, you must first select a face or an existing cut plane.

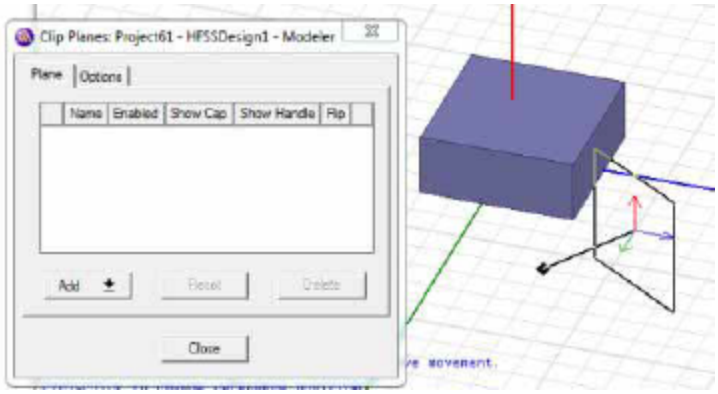


- If you first select a face or cut plane, and then click **Add** > **Use Selection**, the clip plane is added on that face.

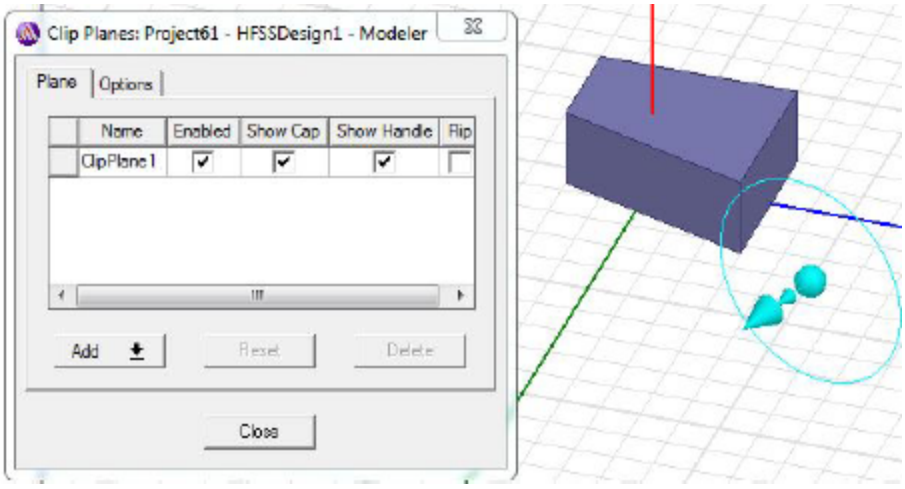
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- If you select **Add** > **Specify center, normal**, this launches a Measure dialog box and enters a mode for you to click to define the start location (shown as a triad).
  a. When you move the cursor, a rectangle represents the clip plane, and a vector the current direction.



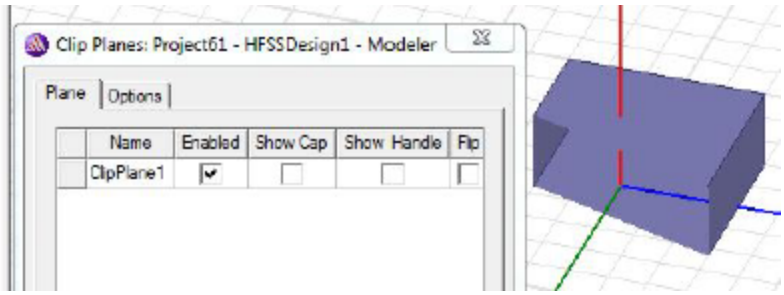
  b. Click again to set the reference position.



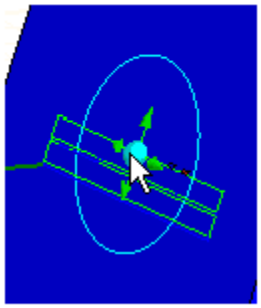
  After the second click, the clip plane is active. The handle is visible as a circle with a sphere at the center, and an arrow pointing the normal for the plane.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

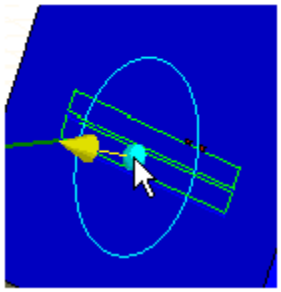
The **Clip Planes** dialog box shows the clip plane name, that it is enabled, the cap (which is the plane surface), and the handle. **Flip selection** lets you reverse the direction of the clip plane. If you deselect **Show cap** and/or **show handle**, they disappear from the display.



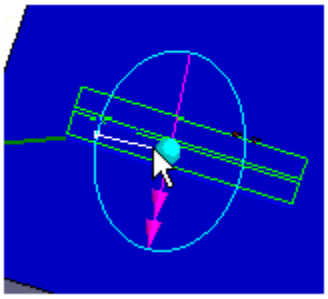
3. With **Show Handle** enabled, you can use the handle to manipulate the location and orientation of the clip plane. The handle changes appearance and function relative to the position of the cursor. Dragging the cursor makes use of the current function.



Move Handle independently of clip plane. This lets you move the handle away from the model.



Move Handle while dragging clip plane



Rotate clip plane by dragging the cursor. The white line is parallel to the clip plane rotation. The double arrow line is the axis.

Move the cursor around the center of the handle to change the axis.

4. The **Options** tab of the **Clip Planes** dialog box contains four options:
   - Force opaque for the unclipped portion.
   - Disable clip plane when drawing a new clip plane.
   - Plane handle color

     The button shows the current color. Click the button to display a color selection dialog box. Select a default or custom color and click OK.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Plane handle radius.

  This slider lets you resize the radius of the handle to the most convenient size. The radius resizes dynamically. When you close and reopen the modeler window, the last selected size persists.

You can save your choices as new defaults.

Modifying the Lighting

You have the option to emit the following types of light on a design:

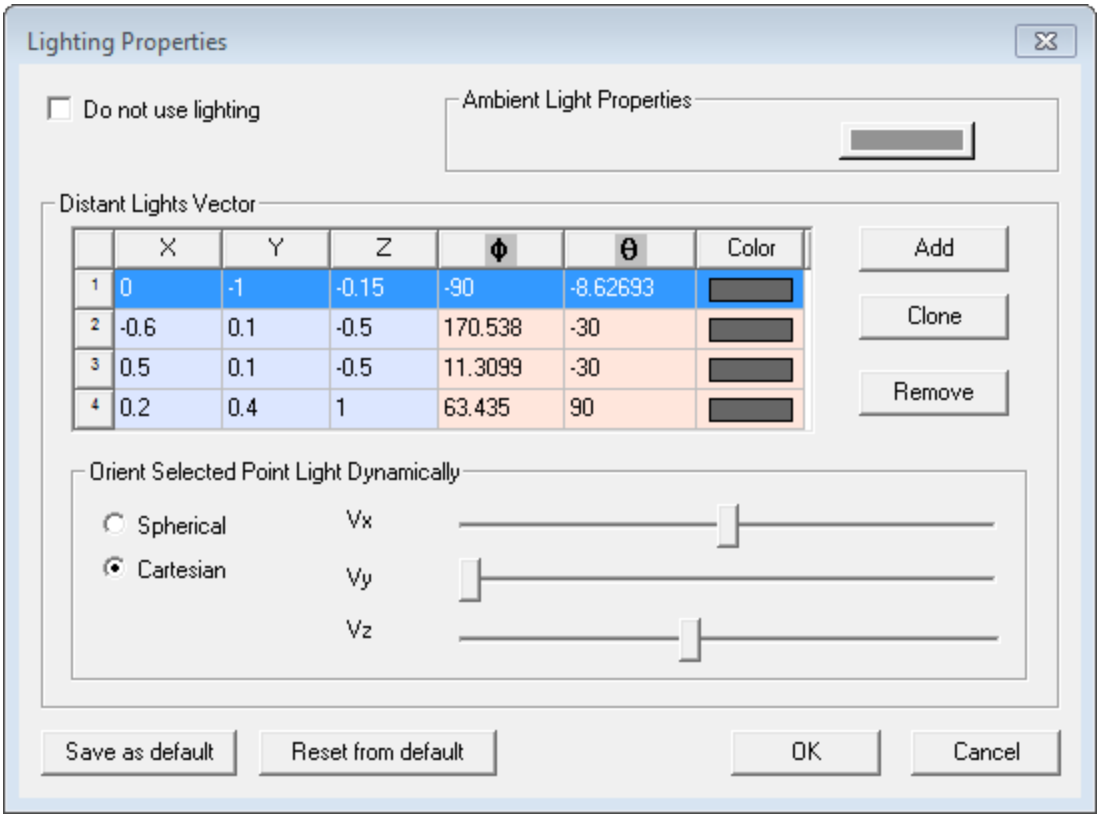
- *Ambient* lighting surrounds the model evenly with light. All objects are lit evenly in every direction by a color of light that you specify.
- *Distant* lighting directs a ray of light at the model in a direction you specify. By default, two distant light vectors are in effect for every new view window.

To modify the lighting:

1. Click **View** > **Modify Attributes** > **Lighting** or, on the **View** tab of the ribbon, click on **Modify Attributes** and select **Lighting** from the drop-down menu.

   The **Lighting Properties** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. To turn off ambient and distant lighting, select **Do Not Use Lighting**.
3. To surround the model with light, click the **Ambient Light Properties** color button and then select a color for the surrounding light from the *Color* palette.
4. To modify the distant light on a model, do one of the following:
   - Add a new distant light by clicking **Add**.
   - Copy an existing distant light that you intend to modify by first selecting it in the **Distant Light Vectors** table and then clicking **Clone**.
   - Select a default distant light to modify by selecting it in the **Distant Light Vectors** table.
5. For the selected distant light vector, specify the vector direction:
   a. To modify the direction by specifying Cartesian coordinates, do one of the following:
      - Enter the new Cartesian coordinates in the **X**, **Y**, and **Z** boxes.
      - Use the **Vx**, **Vy**, and **Vz** sliders to specify the Cartesian coordinates dynamically.
   b. To modify the direction by specifying the spherical coordinates, do one of the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enter the new spherical coordinates in the Φ (phi) and θ (theta) boxes.
- Use the Φ and θ sliders to specify the spherical coordinates dynamically.

6. To revert to the default ambient and distant light settings, click **Reset**.
7. Click **Save As Default** if you want the new lighting settings to be the defaults for all **3D Modeler** windows, either in the current project or future projects.
8. Click **OK** to dismiss the dialog box.

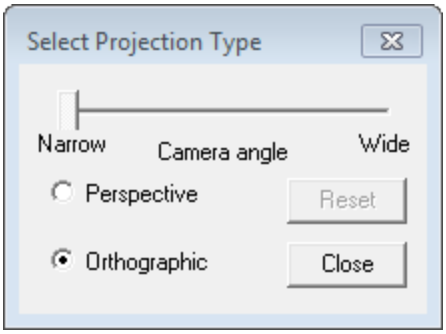
The lighting settings will be saved with the design. New lighting applied to other designs after this point, including new default settings, will not affect these lighting settings.

Setting the Projection View

To modify the projection of model objects (the camera angle) in the view window:

1. Click **View**> **Modify Attributes**> **Projection** or, on the **View** tab of the ribbon, click **Modify Attributes** and select **Projection** from the drop-down menu.

   The *Select Projection Type* dialog box appears:



2. Select **Perspective** to enable the slider to change the angle of the view.
   - Move the slider to the right to increase the proximity, or widen, the view. Move the slider to the left to decrease the proximity, or flatten, the view.

   Objects that are closer appear larger relative than objects that are farther away.

3. Select **Orthographic** to view the model without distortion.

   The slider is disabled because a distortion scale is no longer applicable.

4. Click **Reset** to return the model to its original view.
5. Click **Close** to accept the projection setting and dismiss the window.

   The *Select Projection Type* dialog box closes. The last view you specified in the projection window remains visible in the view window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

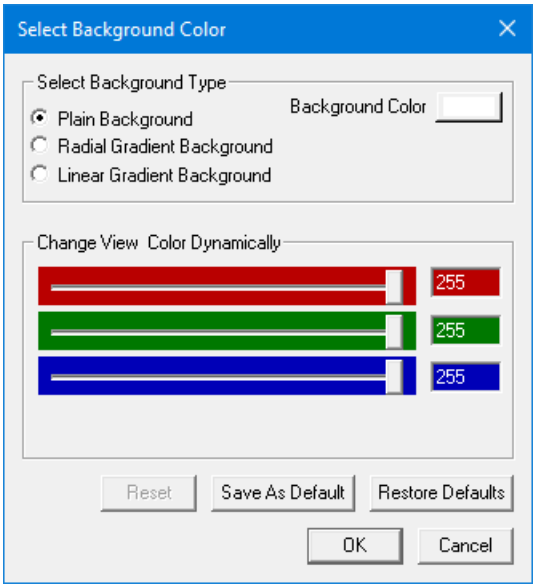
The projection view you set will be saved with the design. New projection views assigned to other designs after this point will not affect this projection setting.

Setting the Background Color

To set the color of the background in the view window:

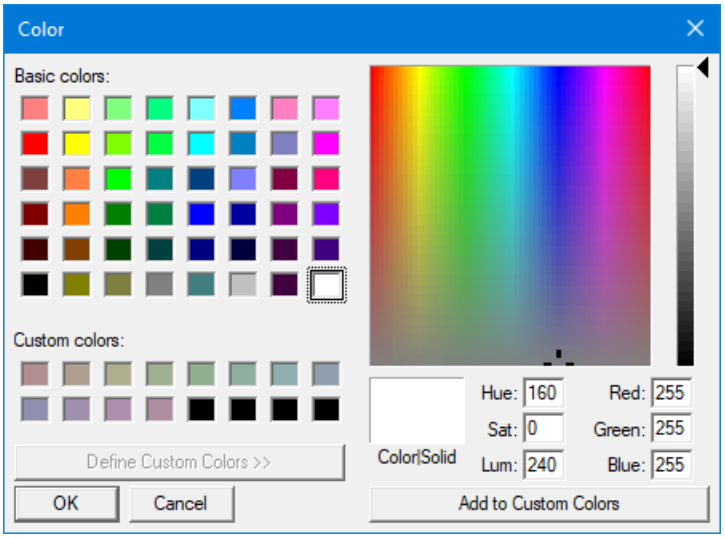
1. Click **View** > **Modify Attributes** > **Background color** or, on the **View** tab of the ribbon, click **Modify Attributes** and select **Background Color...** from the drop-down menu.

   The **Select Background Color** window appears.



2. To assign a solid background color, do the following:
   a. Select **Plain Background**.
   b. Modify the background color in one of the following ways:
      - Click the **Background Color** button and then select a color from the **Color** palette.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Use the RGB sliders under **Change View Color Dynamically** to specify the color's red, green, and blue values.

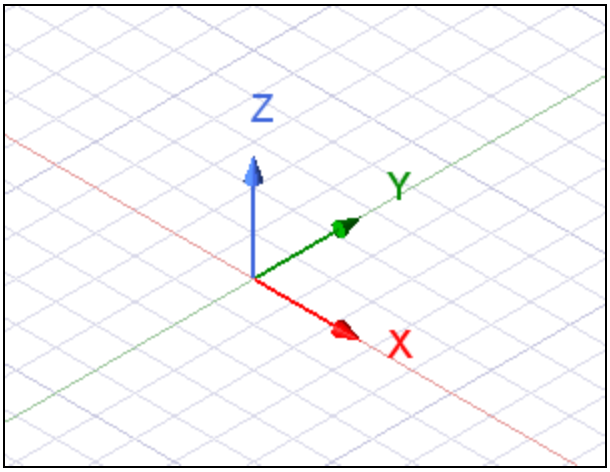
3. To assign a background color that gradually changes from one color to another, do the following:
   a. Select **Gradient Background**.
   b. Specify the background color at the top and bottom of the view window in one of the following ways:
      - Under **Select Background Type**, click the **Top Color** button and select a color from the **Color** palette. Then click **Bottom Color** and select a color from the **Color** palette.
      - Under **Change View Color Dynamically**, click **Top Color** or **Bottom Color** and use the RGB sliders to specify the color's red, green, and blue values.
4. Click **Reset** to revert to the default background colors.
5. Click **Save As Default** if you want the new background color to be the background color for all *3D Modeler* windows in either the current project or future projects.
6. Click **OK**.

The background color you set will be saved with the design. New background color settings assigned to other designs after this point, including new default settings, will not affect this design.

Modifying the Coordinate System Axes View

The coordinate system axes displays the x, y, z orientation from the origin point for the current working coordinate system. The axes use an XYZ = RGB coloring scheme.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



By option, you can also use a triad which is a secondary depiction of the coordinate system that appears at the lower right of the Modeler window. It shows the orientation of the currently selected working coordinate system.

What do you want to do?

- Show or hide the coordinate system axes
- Show the coordinate system axes for selected objects.
- Enlarge or shrink the size of the coordinate system axes
- Show or hide the triad axes

Showing or Hiding the Axes

1. Click **View** > **Coordinate System**, then click one of the following:
   - **Hide** to hide the x-, y-, and z-axes in the active view window.
   - **Show** to display the x-, y-, and z-axes in the active view window.

Show the Axes for Selected Objects

1. Click **Tools** > **Options** > **General Options**.

   This command displays the *Options* dialog box.
2. Expand the **3D Modeler** branch of the options tree.
3. Expand the **Display** sub-branch.
4. Select the **General** subgroup.
5. Select **Show orientation of selected objects**.
6. Click **OK** to close the dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enlarging or Shrinking the Axes

1. Click **View**> **Coordinate System**, then click one of the following:
    - **Large** to display the x-, y-, and z-axes as extending to the edges of the active view window.
    - **Small** to display the x-, y-, and z-axes in a smaller size in relative to the edges of the active view window.

Showing or Hiding the Triad Axes

The triad is a secondary depiction of the coordinate system that appears at the lower right of the Modeler window. It shows the orientation of the currently selected working coordinate system. It can be shown or hidden separately from the selected coordinate system.

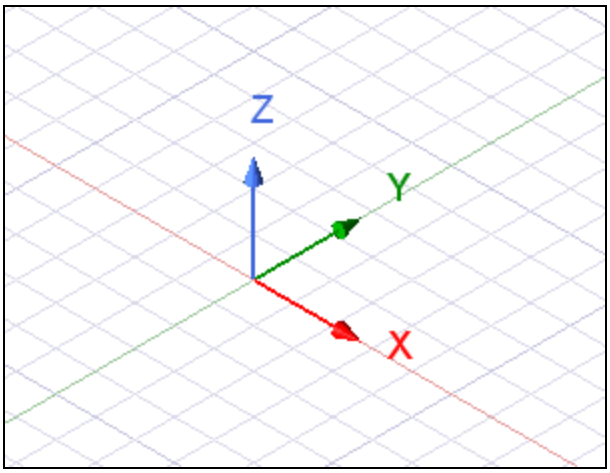
To show the triad:

1. Click **View**> **Coordinate System**> **Triad**, then click one of the following:
    - **Hide** to hide the triad x-, y-, and z-axes at the lower right of the active view window.
    - **Show** to display the triad x-, y-, and z-axes in the lower right active view window.
    - **Auto** to generally hide the triad axes.

Choosing Grid Settings

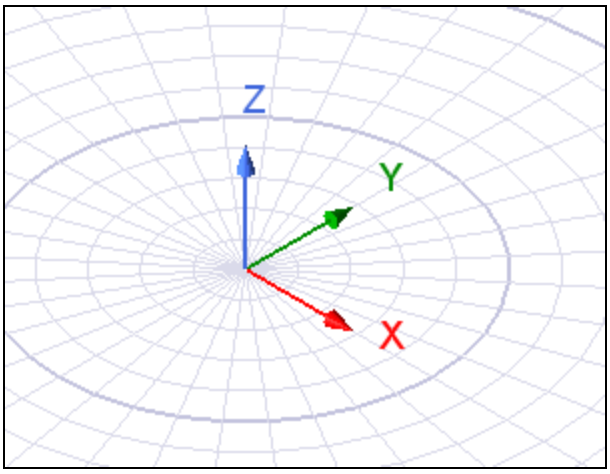
The grid displayed in the **Modeler** window is a drawing aid that helps to visualize the location of objects.

For Cartesian grids, the location of points on the grid are defined by intersections of planes that are perpendicular to and along the x-, y-, and z-axes. The division (the distance between neighboring parallel planes perpendicular to the same axis) can be set.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For polar grids, the location of points on the grid are defined by intersections of planes that are perpendicular to the local radius and angle coordinates. The division (the distance between neighboring parallel planes perpendicular to the same radius and angle) can be set.



Grid spacing is set according to the current project's drawing units. You can control the following aspects of the grid:

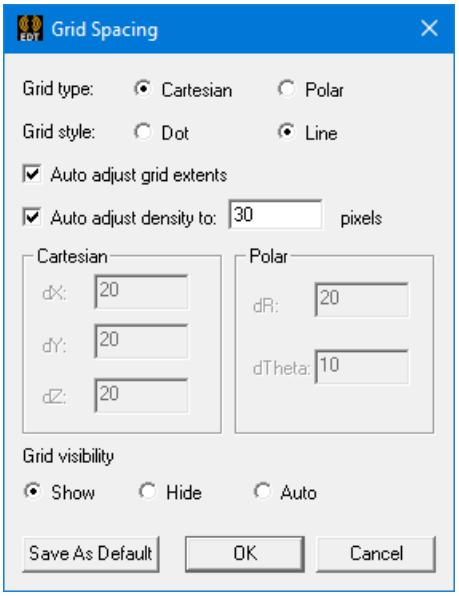
- Type (rectangular or circular)
- Style (dots or lines)
- Grid Extent (minimal for existing objects or as a plane)
- Density
- Spacing
- Visibility
- Snap settings
- Grid plane

Setting the Grid Type

1. Click **View** > **Grid Settings**.

   The **Grid Spacing** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select a grid type for the active Modeler window: **Cartesian** for a rectangular grid or **Polar** for a circular grid.

   The grid in the active view window is centered at the origin of the working coordinate system.

For Cartesian grids, you define a coordinate by specifying its distance from the origin along each axis in the **X**, **Y**, and **Z** text boxes or its relative distance from the previously selected point in the **dX**, **dY**, and **dZ** text boxes.

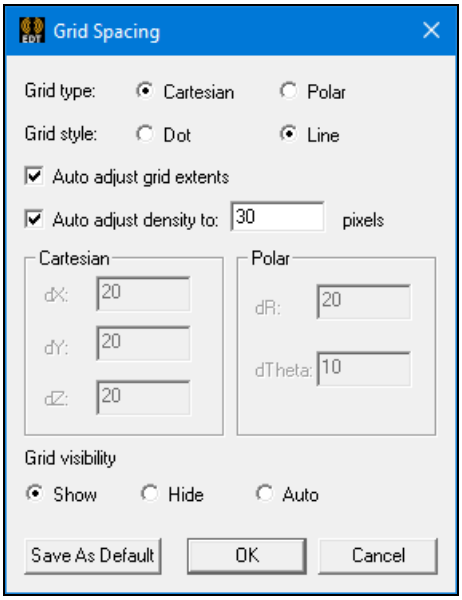
For polar grids, you define a coordinate by specifying its radius from the origin in the **R** text box and its angle from the x-axis in the **Theta** text box or its relative distance from the previously selected point in the **dR** and **dTheta** text boxes.

Setting the Grid Style

1. Click **View** > **Grid Settings**.

   The **Grid Spacing** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select one of the following grid styles for the active view window:

   Dot Displays each grid point as a dot.

   Line Displays lines between grid points.

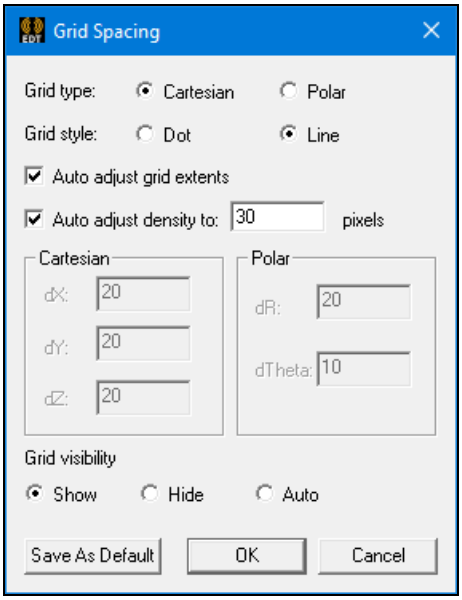
3. To show a Minimal grid around existing objects, check **Auto adjust grid extends**. To display a grid as a plane, deselect **Auto adjust grid extends**.

Setting the Grid Density and Spacing

1. Click **View** > **Grid Settings**.

   The **Grid Settings** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. If you want to change the density of the grid in the active Modeler window as you zoom in or out on objects, do the following:
    a. Select **Auto adjust density to**.
    b. Specify a distance between grid points by typing a value in the **pixels** box.

    The default is set to 30 pixels, which is generally the best setting for displaying objects.

3. If you do not want the grid density to change when you zoom in or out, but instead want to specify a constant grid spacing, do the following:
    a. Clear the **Auto adjust density to** option.
    b. Specify the grid's spacing in the active design's units.

    If you selected a Cartesian grid, type the values of **dX**, **dY**, and **dZ**. These values represent the difference between one grid point and the next in the x, y, and z directions, respectively.

    If you selected a polar grid, type the values for **dR** and **dTheta**. **dR** represents the difference between each radius. **dTheta** is the difference between angles.

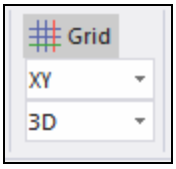
    The distance between grid points will increase and decrease proportionately as you zoom in and out in the active Modeler window.

4. Click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Grid's Visibility

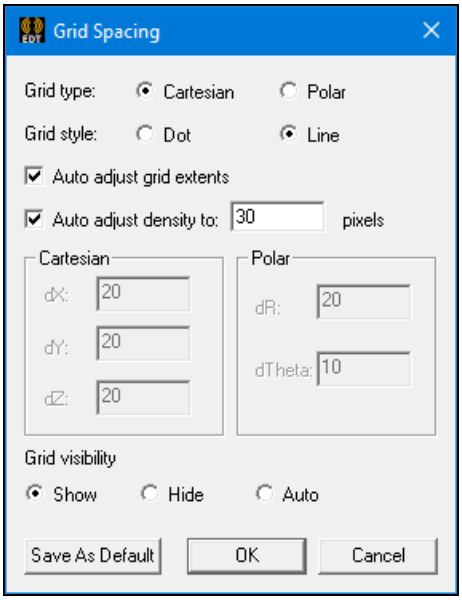
To hide the grid in the Modeler window, on the **Draw** ribbon tab, click the Grid toolbar icon. Click it again to show the grid.



Alternatively:

1. Click **View** > **Grid Settings**.

   The **Grid Spacing** dialog box appears.



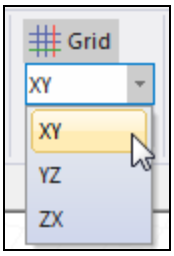
2. Select **Show** to make the grid always visible in the active **Modeler** window. Select **Hide** to make the grid always invisible in the active **Modeler** window. Select **Auto** so the grid appears only while you are drawing an object in the **Modeler** window.

Setting the Grid Plane

To specify the plane on which you want to display the grid in the active view window, do one of the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click **Modeler** > **Grid Plane** and then select a grid plane: **XY**, **YZ**, or **XZ**.
- Click a grid plane on the drop-down menu under Grid on the **Draw** ribbon tab:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8 - Optimetrics

Optimetrics enables you to determine the best design variation among a model's possible variations. You create the original model, the *nominal design*, and then define the design parameters that vary, which can be nearly any design parameter assigned a numeric value in HFSS. For example, you can parameterize the model geometry or material properties. You can then perform the following types of analyses on your nominal design:

optiSLang	optiSLang simulations can be integrated with Ansys Electronics Desktop, such that you can create a setup very much like an Optimetrics setup to run through an optiSLang installation.
Parametric	In a parametric analysis, you define one or more *variable sweep definitions,* each specifying a series of variable values within a range. For example, you can parameterize component values. (See Variables for more information.) Optimetrics solves the design at each variation. You can then compare the results to determine how each design variation affects the performance of the design. Parametric analyses are often used as precursors to optimization solutions because they help to determine a reasonable range of variable values for the optimization analysis.
Design of Experiments	Design of Experiments (DOE) is a technique used to scientifically determine the location of sampling points and is included as part of the Response Surface, Goal Driven Optimization, and Analysis systems.
DesignXplorer	An optimization tool for studying a range of design variations, used with Design of Experiments. This permits you to manage an Optimetrics simulation from the Ansys Workbench.
Optimization	For an optimization analysis, you identify the cost function and the optimization goal. Optimetrics changes the design parameter values to meet that goal. The cost function can be based on any solution quantity that can be computed.
Sensitivity	In a sensitivity analysis, you use Optimetrics to explore the vicinity of the design point to determine the sensitivity of the design to small changes in variables.
Tuning	Tuning allows you to change variable values interactively while monitoring the performance of the design. If you want to ensure that tuning does not resolve variations already solved by parametric setup, you must check **Save Fields Mesh** in the **Options** tab of the Optimetrics setup.
Statistical	In a statistical analysis, you use Optimetrics to determine the distribution of a design's performance, which is caused by a statistical distribution of variable values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Sensitivity, Statistical, Design Xplorer, and some Optimizers have been placed in legacy mode because they are no longer under active development. Go to **Tools** > **Options** > **General Options** > **Miscellaneous** and toggle the **Enable Legacy Optimetrics Tools** check box to show or hide these legacy tools.

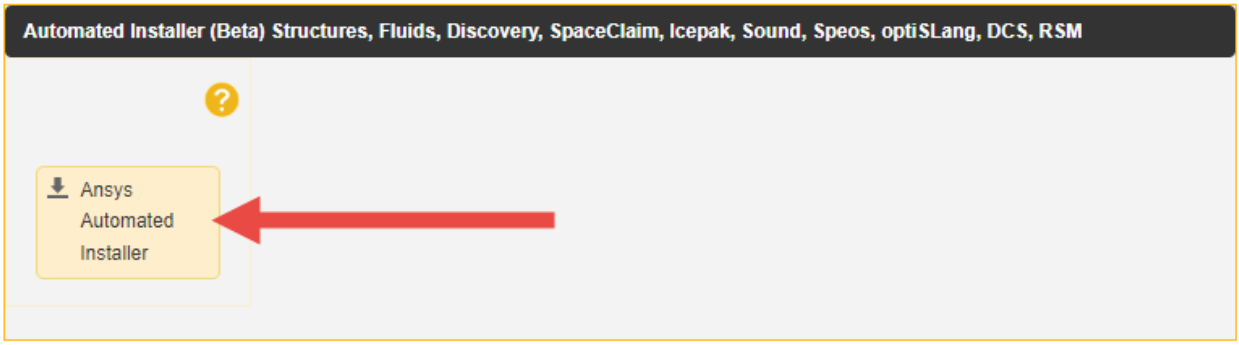
Note:

Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

The HPC and Analysis Options dialog can be accessed from the setup dialog for each type of Optimetrics analysis.

optiSLang Integration with Electronics Desktop

optiSLang is a tool for graphical programming, process integration, and automation that can be integrated with Ansys Electronics Desktop (AEDT) for Optimetrics analysis. It is available as part of the Ansys Automated Installer package on the Ansys Customer Portal.



Comprehensive online help for optiSLang is available from the optiSLang Help menu, or at:

https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v251/en/opti_ug/opti_ug_intro.html

This section covers only its integration with Ansys Electronics Desktop.

optiSLang in AEDT allows AEDT and optiSLang to stay in continuous interaction by exchanging API commands between each other. optiSLang algorithms can be executed without exiting AEDT because optiSLang setups follow Optimetrics norms and have an explicit

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

association with a specific analysis setup. AEDT stays continuously open while optiSLang algorithms demand set after set of designs.

Each optiSLang setup resides in one specific design of an AEDT project and is linked to one specific analysis setup. When an optiSLang setup is executed within AEDT, optiSLang runs in the background in batch mode and executes the algorithm that the optiSLang setup represents. optiSLang administers an optiSLang project file (*.opf) and a normal optiSLang project directory (*.opd), stored relative to the AEDT project.

The **standalone optiSLang GUI** allows for further editing of the optiSLang project structure, with each analysis setup represented as a single Ansys EDT node:

If the original Ansys EDT node and its copies are all kept in the optiSLang Setup run mode, they will remain linked to the specific optiSLang setup existing in the referenced AEDT project. The optiSLang project can contain and execute multiple algorithm systems, which means that the linked optiSLang setup in the AEDT project represents a connector, not an algorithm, in that case.

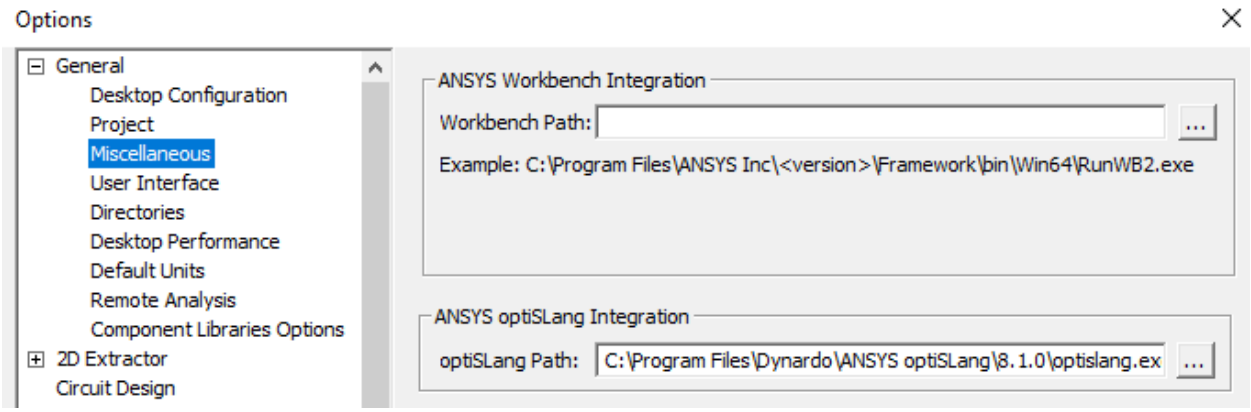
The topics in this section cover:

- optiSLang User Workflow
- Creating an optiSLang Setup in AEDT
- Solving an optiSLang Setup in AEDT
- optiSLang Menu Options
- Viewing optiSLang Postprocessing Results

Prerequisites

Before working with optiSLang in AEDT, specify the path to the optiSLang installation using **Tools** > **General Options** > **General** > **Miscellaneous**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Additionally, ensure that the following are true:

- The project is solved
- Parameters exist (See: Parametrization for optiSLang Integration)
- Results reports exist (See: Results and Reports for optiSLang Integration)

optiSLang User Workflow

This topic explores the general workflow for creating an optiSLang project from Ansys Electronics Desktop (AEDT).

1. Prepare the AEDT project by ensuring:
    - Parameters exist.

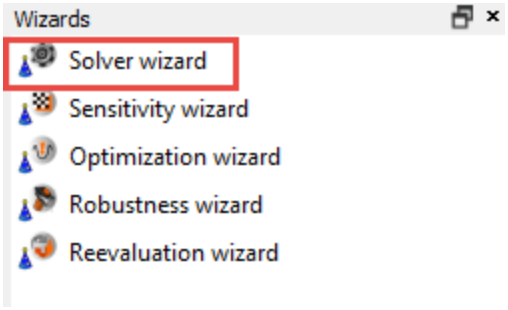
      > **Note:**
      >
      > optiSLang scans parameters at both the project and design level. Hidden parameters are also scanned. See: Parametrization for optiSLang Integration.

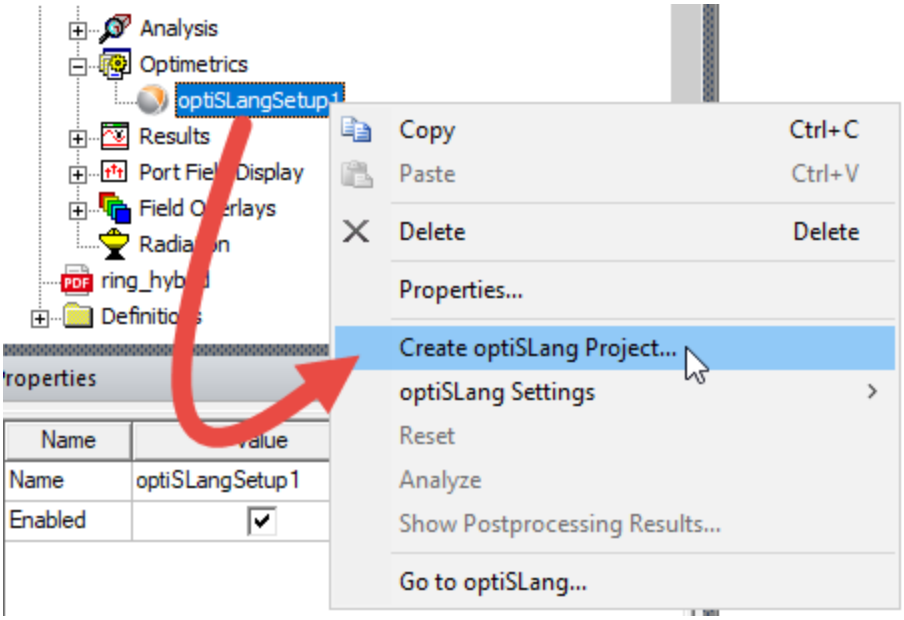
    - Project is solved.
    - Results reports exist and are prepared for the specific use case
        - For a standard use case, set all variables to nominal.
        - For a use case with a user-defined sweep, display all variations.

      See: Results and Reports for optiSLang Integration.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

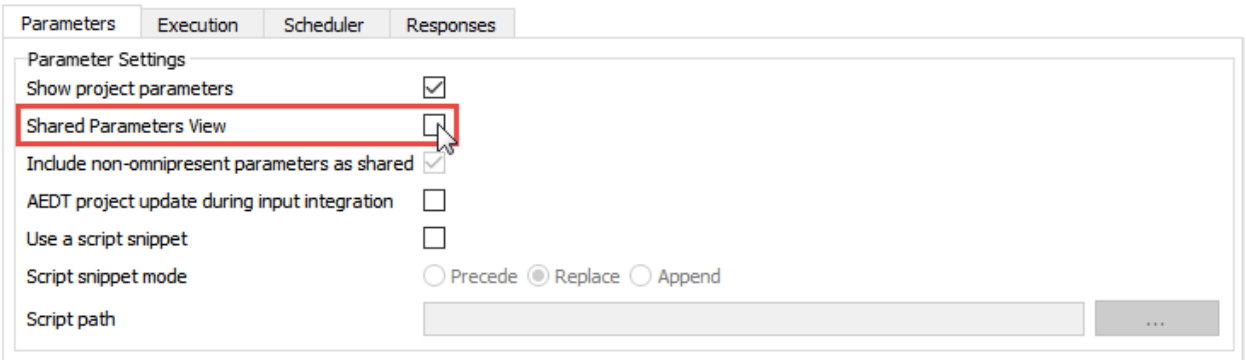
2. Invoke the Solver Wizard in optiSLang or create an optiSLang project directly from AEDT.





3. Create a one-node setup or a two-node setup.
    - A one-node setup registers both parameters and responses. This is the type of setup created by the wizard.
    - A two-node setup registers parameters in the first node and responses in the second. This is a more complicated setup that requires cloning the integration node, setting function switches, and setting absolute or relative paths. Consult the optiSLang help for more information.
4. View results in either tabular or tree view.
5. Enable **Shared Parameters View**. By default, parameters from different models are treated separately and a prefix based on the model name ensure uniqueness of names. In shared mode, sub-models are kept in sync and prefixes are unnecessary.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

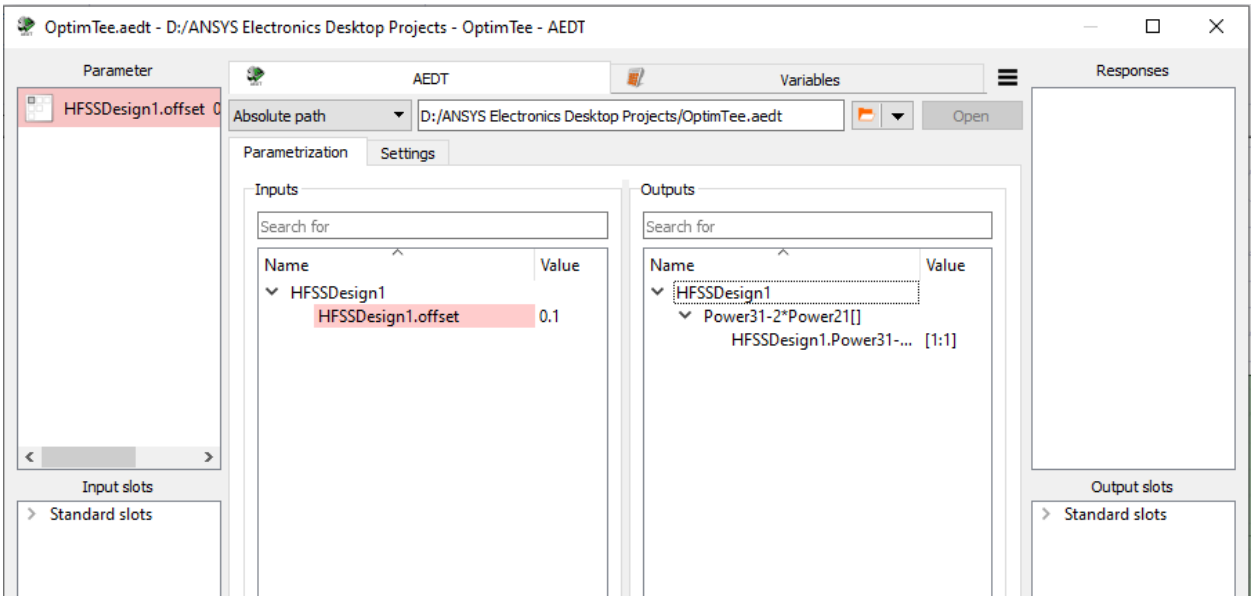


6. Reload the AEDT project in optiSLang to see the effects of changed settings.

Parametrization for optiSLang Integration

Certain AEDT data must exist before optiSLang can use it.

Variables act as **Inputs** in optiSLang, while simulation results (reports) act as **Outputs**.



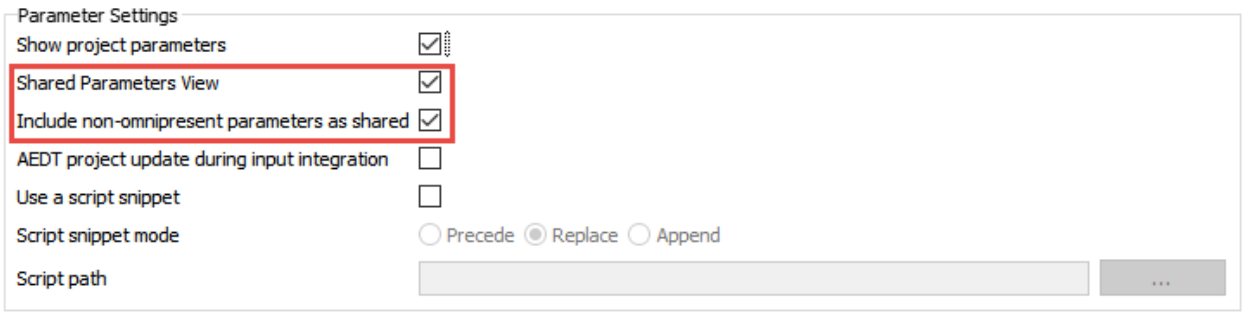
Inputs

When integrated with AEDT, optiSLang automatically detects project variables and design variables for every design in the project. Note that:

- Dependent parameters are skipped.
- Any parameter marked as "hidden" is always included.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Users decide whether to treat parameters from different models in the project as independent or communal (kept synchronized). In optiSLang, the AEDT integration has a **Shared Parameters View** with an additional **Include non-omnipresent parameters as shared** option.



In AEDT, project variables can be used to steer model parameters.

It is best to leverage features on both sides of the optiSLang-AEDT integration for managing overlapping (that is, partially synced) parameter sets. optiSLang's standard parametrization management tools offer the parameter type Dependent for aligning or otherwise controlling parameters.

In the context of bundling optiSLang-demanded designs for simultaneous execution as DSO jobs, remember that DSO design variation tables are hosted in the Optimetrics section of each design, and that there is no Optimetrics branch at the project level. So, even varying only project variables will still yield separate DSO tables and thus separate jobs for each model in the project.

A project walker mechanism inquires a given AEDT project and stores the generated project structure info in a JSON file for optiSLang. If the same reference project is linked for use in a different AEDT node even from a different optiSLang project, the stored project inquiry results can be recalled and the parametrization can be instantly displayed. This mechanism spares users wait time for the project inquiry call.

Outputs

The framework for result data transfer from AEDT to optiSLang is very simple: each report trace shows up in optiSLang response listings as a signal object. You can register the signals directly or apply math functions offered by optiSLang's calculator with each integration node. The transfer vessels are CSV files.

Even such plots as Smith charts or 3D polar plots can be collected as groups of signal objects using the uniform CSV file export function offered in AEDT.

Parameterization with optiSLang Analysis Goals in Mind

The parametrization scheme should connect to the goals of the optiSLang analysis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bad parametrization might hide the design optimum from the best optimizer or destroy the connection of a robustness sampling with reality. Thoughtfully improved parametrization can achieve simulation goals with a fraction of the designs that would be needed for a task within the most simple ad hoc parametrization scheme.

The algorithms being applied in a specific scenario should also guide your parametrization scheme. For example, it makes sense to give a robustness sampling algorithm the chance to examine a parameter space where the simulation can represent the relevant physical effects happening in the real world. An optimizer algorithm, meanwhile, needs the chance to hit feasible designs frequently.

Some tips:

- Think ahead to devise a stable and meaningful parametrization
- Be attentive to the kinds of errors encountered by failing designs
- Be quick to eliminate the origins of errors that appear most frequently in error logs
- Learn to use optiSLang post-processing as a diagnostic toolbox
- Apply feedback and lessons learned to your parametrization schemes

Results and Reports for optiSLang Integration

AEDT can export simulation reports as CSV files that can be read in optiSLang.

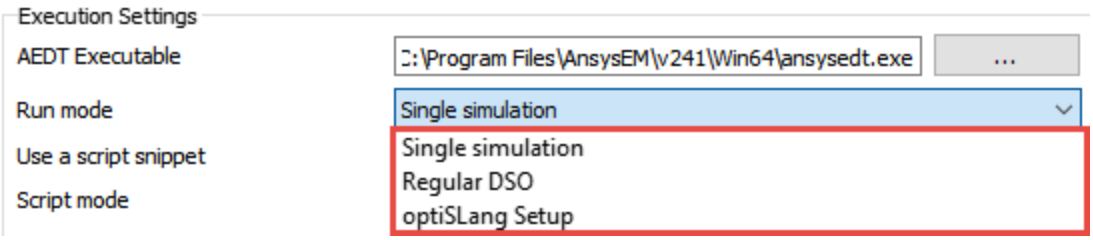
Additionally, all Optimetrics setups in AEDT (including optiSLang setups) have a **Calculations** tab, allowing for additional results. See: Setting up Calculations for Optimetrics.

When the scripted AEDT integration machinery within optiSLang exports reports, it is just as if a user had exported a CSV file from within the AEDT UI. Any reports that can be exported as CSV (rectangular plots, data tables, polar plots, etc.) are sent in CSV format, and the reports' traces can be registered as responses in optiSLang. As a general rule, each data column of a CSV file is exposed as a signal trace.

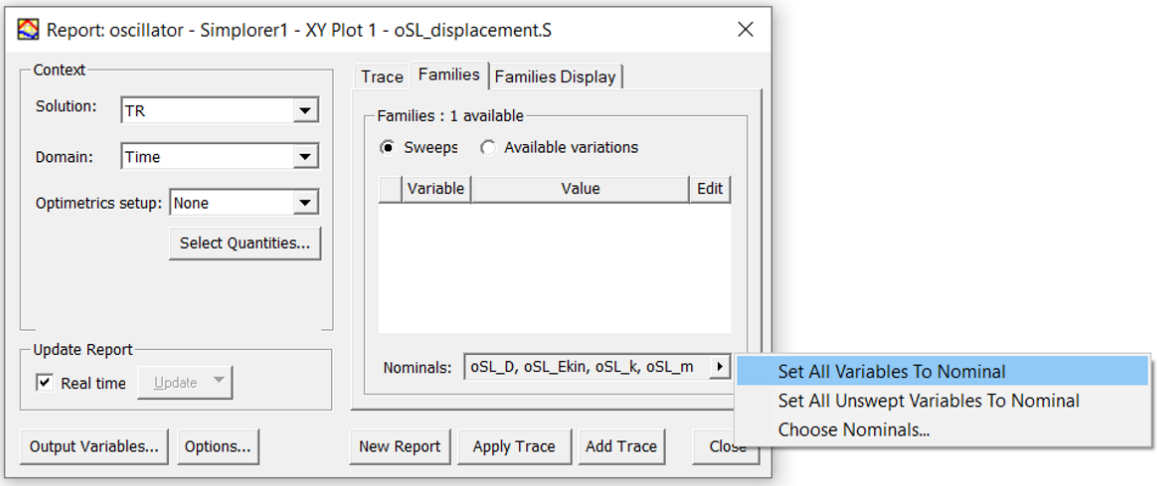
> **Note:**
>
> For 3D Polar Plots, this can cause a small disadvantage by yielding, for example, 180 signal objects instead of a matrix. However, the simplicity of the routine achieves good performance and robustness, and allows the AEDT integration node in optiSLang to cover a large array of use cases.

Below, find considerations for setting up reports for use in optiSLang's different run modes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



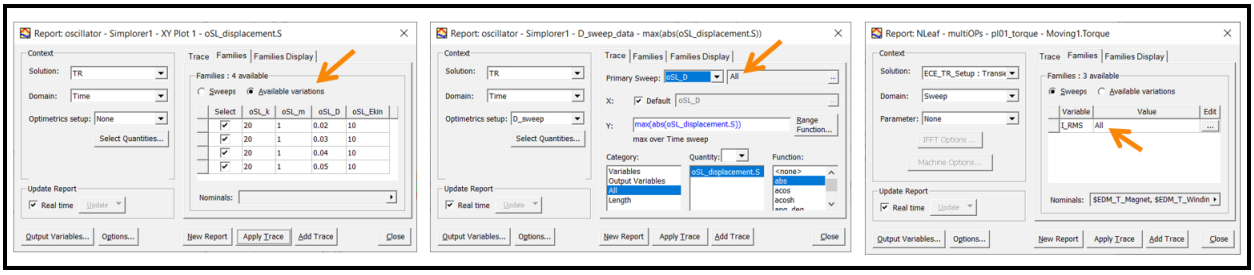
In optiSLang's **Single Simulation** run mode, optiSLang modifies the parameters of the nominal design and triggers the solution process. Therefore, reports used in this way should be configured to display the nominal design if they are to contain traces after modification and solution update. To ensure all report parameters are set to nominal, open the report settings and select the **Families** tab. From the **Nominals** field, select **Set All Variables to Nominal**.



In optiSLang's **Regular DSO** mode, the result export script cycles over all design variations of the Optimetrics setup created by previous script commands for holding the set of design variations demanded by optiSLang. In the loop, it applies the next design to be displayed as the nominal design, forces a refresh of all reports, and exports all reports. This loop is repeated for all design variations and the CSV files are stored each time in the corresponding design folder. Therefore, reports used in this mode should also be configured to display the nominal design. To ensure all report parameters are set to nominal, open the report settings and select the **Families** tab. From the **Nominals** field, select **Set All Variables to Nominal**.

In optiSLang's **optiSLang Setup** mode, the software solves a pre-existing Optimetrics setup (either serially or as a DSO job). This is commonly used to represent a parametric sweep or a prescribed set of operating points. In these use cases, reports should contain all data from all design variations to support a single export report call. To ensure the reports are adequate, open the report settings and enable all available variations:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

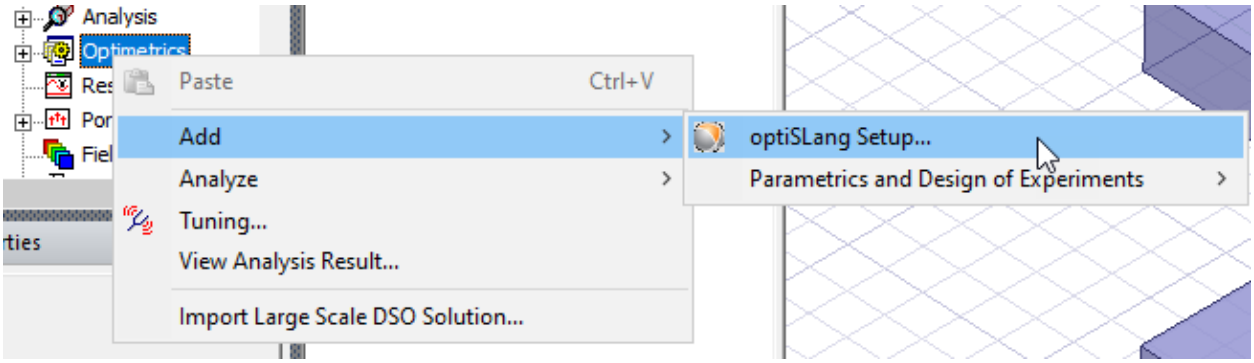


Creating an optiSLang Setup in AEDT

optiSLang setups are defined similarly to Optimetrics setups.

Create an optiSLang setup one of two ways:

- From the **Project Manager**, right-click **Optimetrics** and select **Add** > **optiSLang Setup**.
- Click **HFSS** > **Optimetrics Analysis** > **Add optiSLang Setup**.

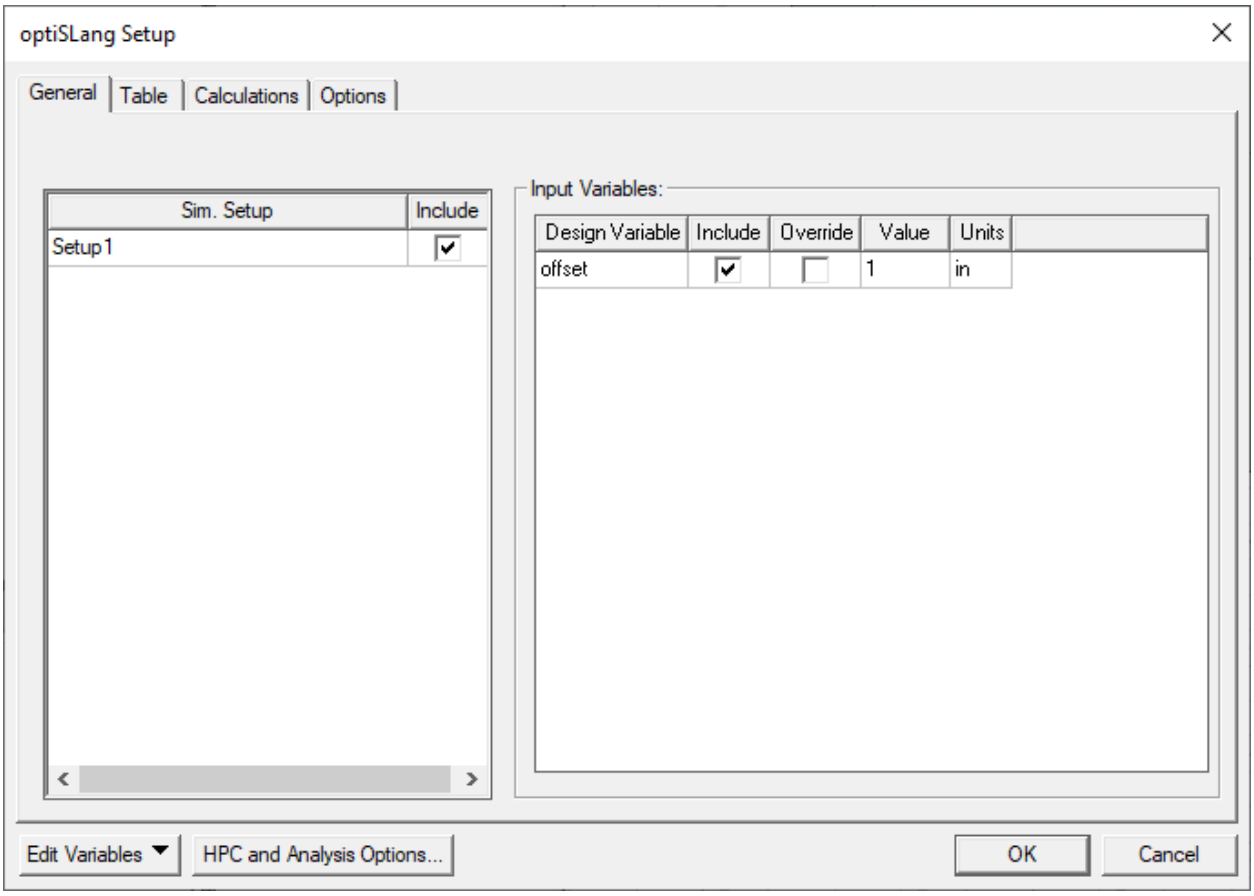


The **optiSLang Setup** window appears, with content based on the current design.

This window contains several tabs:

- **General** – lists solution setups and design variables associated with the current design, and allows you to specify which setups and variables to include, as well as to override a variable's value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Table** – lists the values of the any variable(s) selected for inclusion.
- **Calculations** – allows for the setup of optiSLang calculations, similar to Optimetrics or Parametric setup. See: Setting up Calculations below.
- **Options** – lists mesh options. See: Setting Options below.

Each tab of the window also provides access to Edit Variables and HPC and Analysis Options.

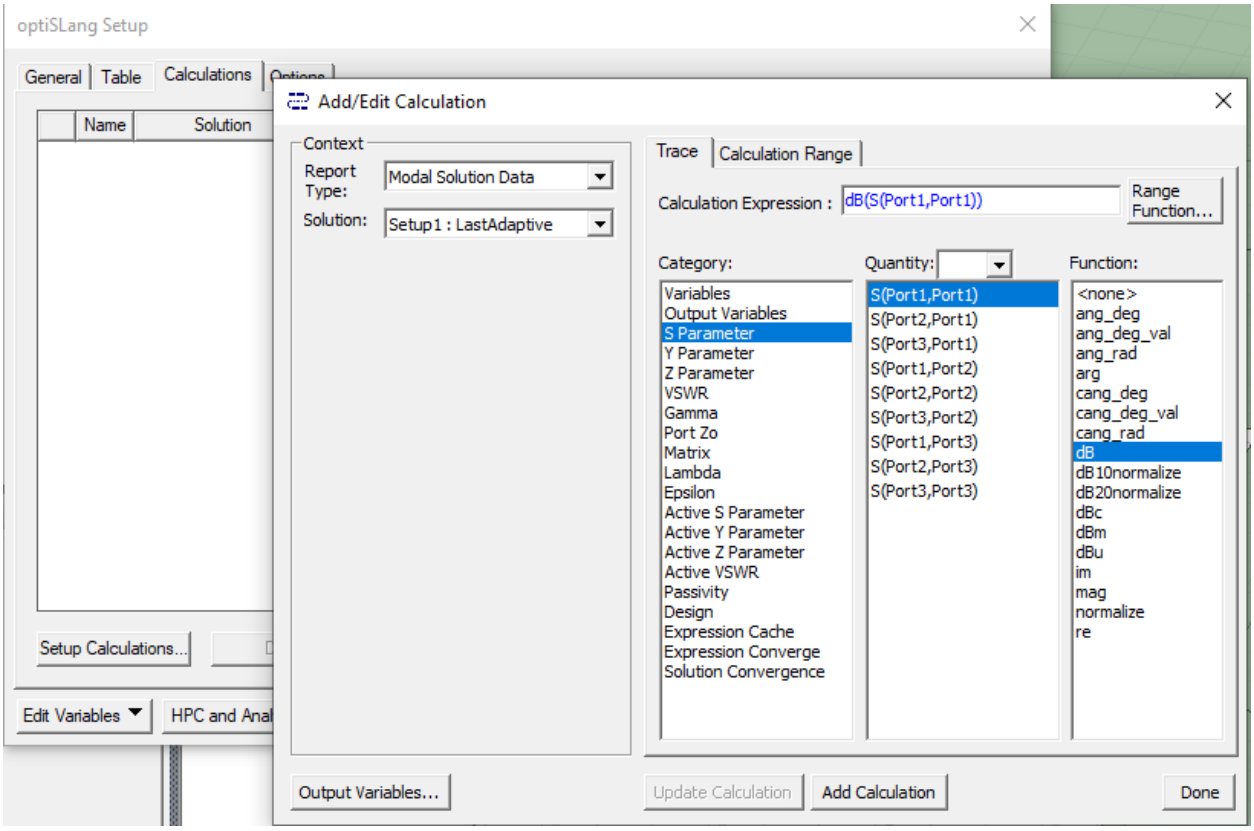
Setting up Calculations

Initially, the **Calculations** tab is empty.

To set up a calculation, click **Setup Calculations**.

The **Add/Edit Calculation** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This window contains **Context**, **Category**, **Quantity**, and **Function** fields that can be used to build an optimization calculation expression, just as for other optimization or parametric calculations.

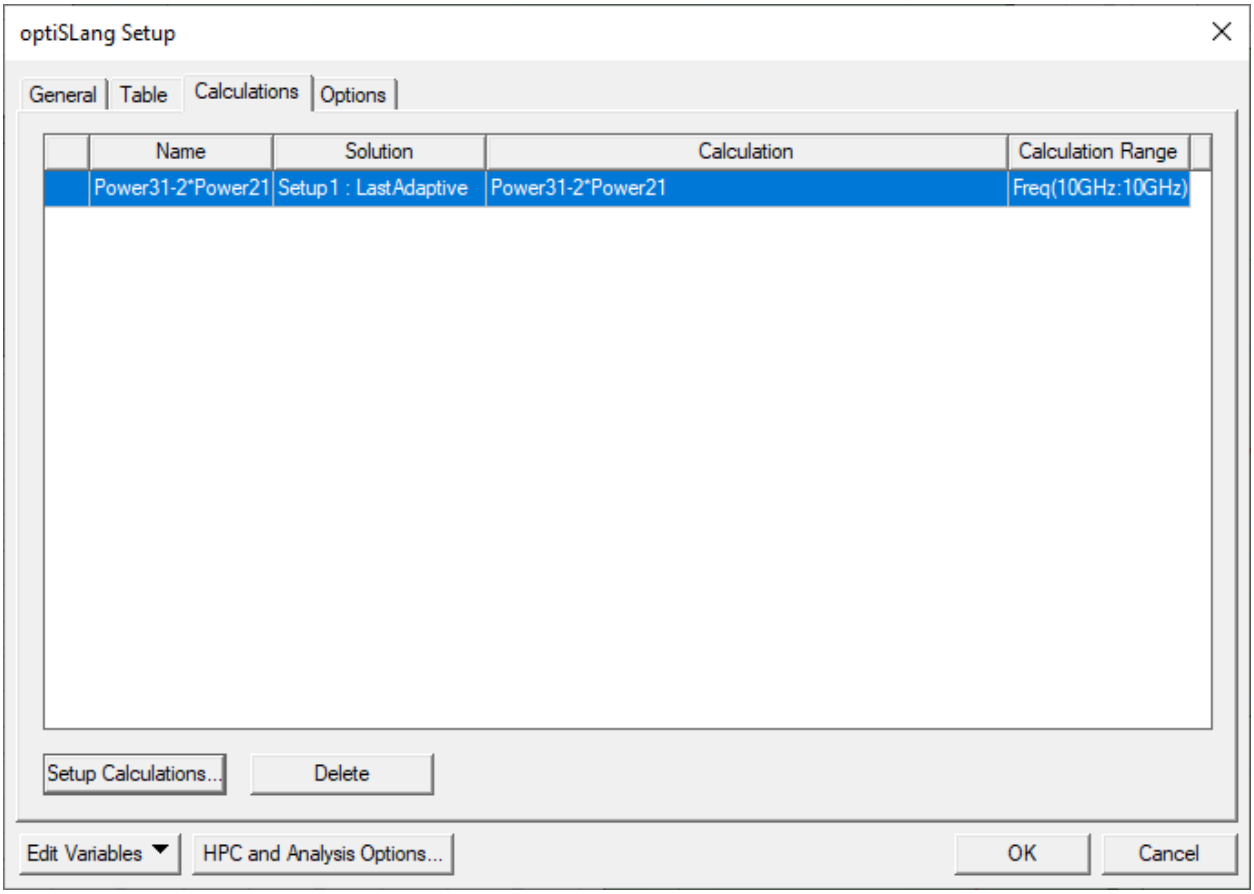
See: Setting Up Calculations for Optimetrics.

Once you have created an expression, click **Add Calculation** to add it to the **optiSLang** setup.

Click **Done** to close the **Add/Edit** window and return to the **Calculations** tab.

The new calculation is listed:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



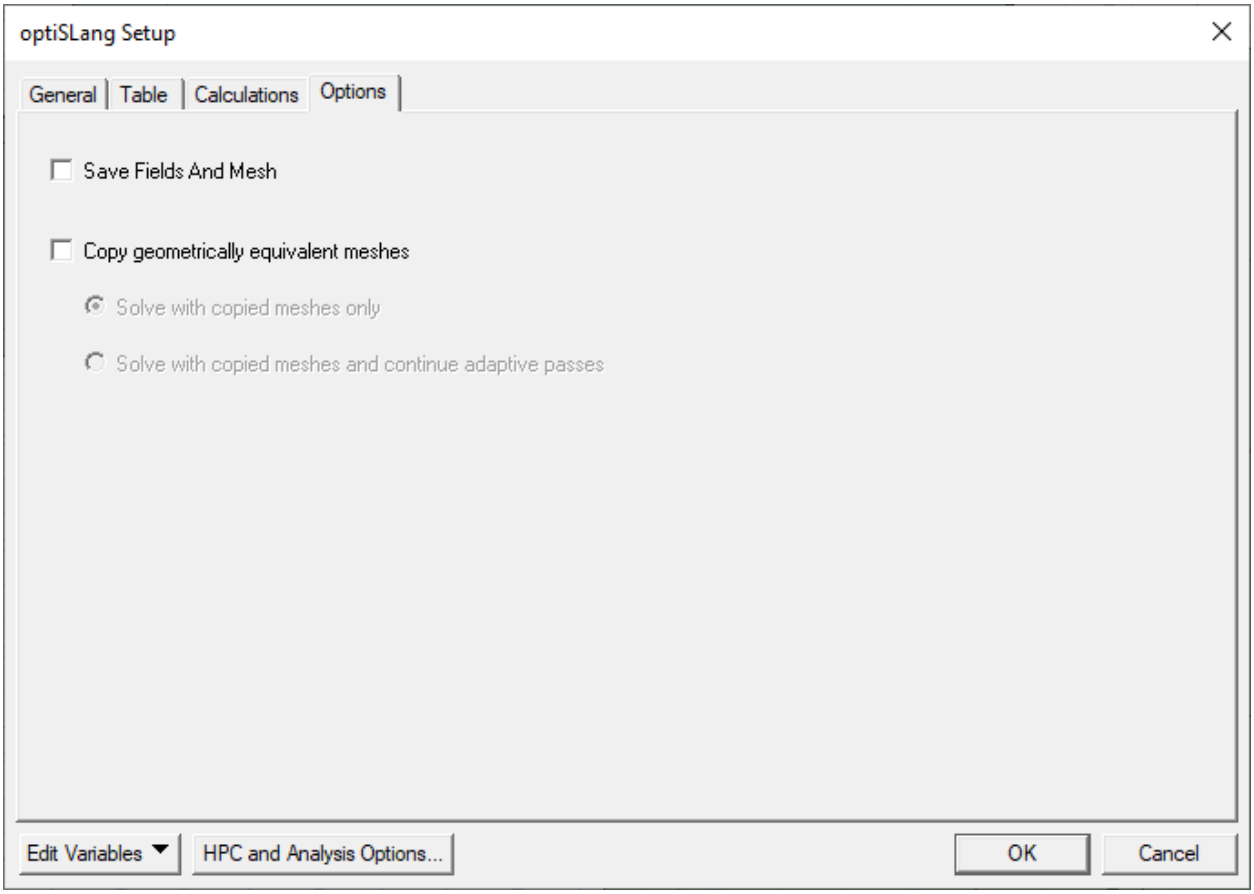
Setting Options

The **Options** tab contains options determining whether to **Save Fields and Mesh** and whether to **Copy geometrically equivalent meshes**.

In order to preserve disk space, by default AEDT does not save field solution data for every solved design variation. **Save Fields and Mesh** overrides this behavior.

Copy geometrically equivalent meshes directs AEDT to copy a mesh that was calculated for one sweep variation for reuse on a geometrically equivalent sweep variation. See: Copying Meshes in Optimetrics Sweeps.

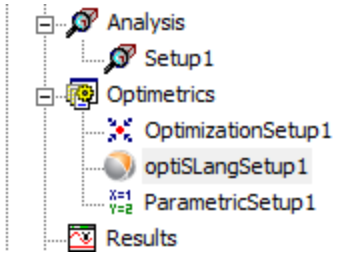
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Finishing Setup

Click **OK** when you have completed the setup.

The optiSLang setup appears in the **Project Manager**, under **Optimetrics**:



Example optiSLang Setups

The following are example scenarios illustrating different considerations that might be taken when preparing an optiSLang setup. See: Solving an optiSLang Setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Settings in these examples are handled both in AEDT and in optiSLang.

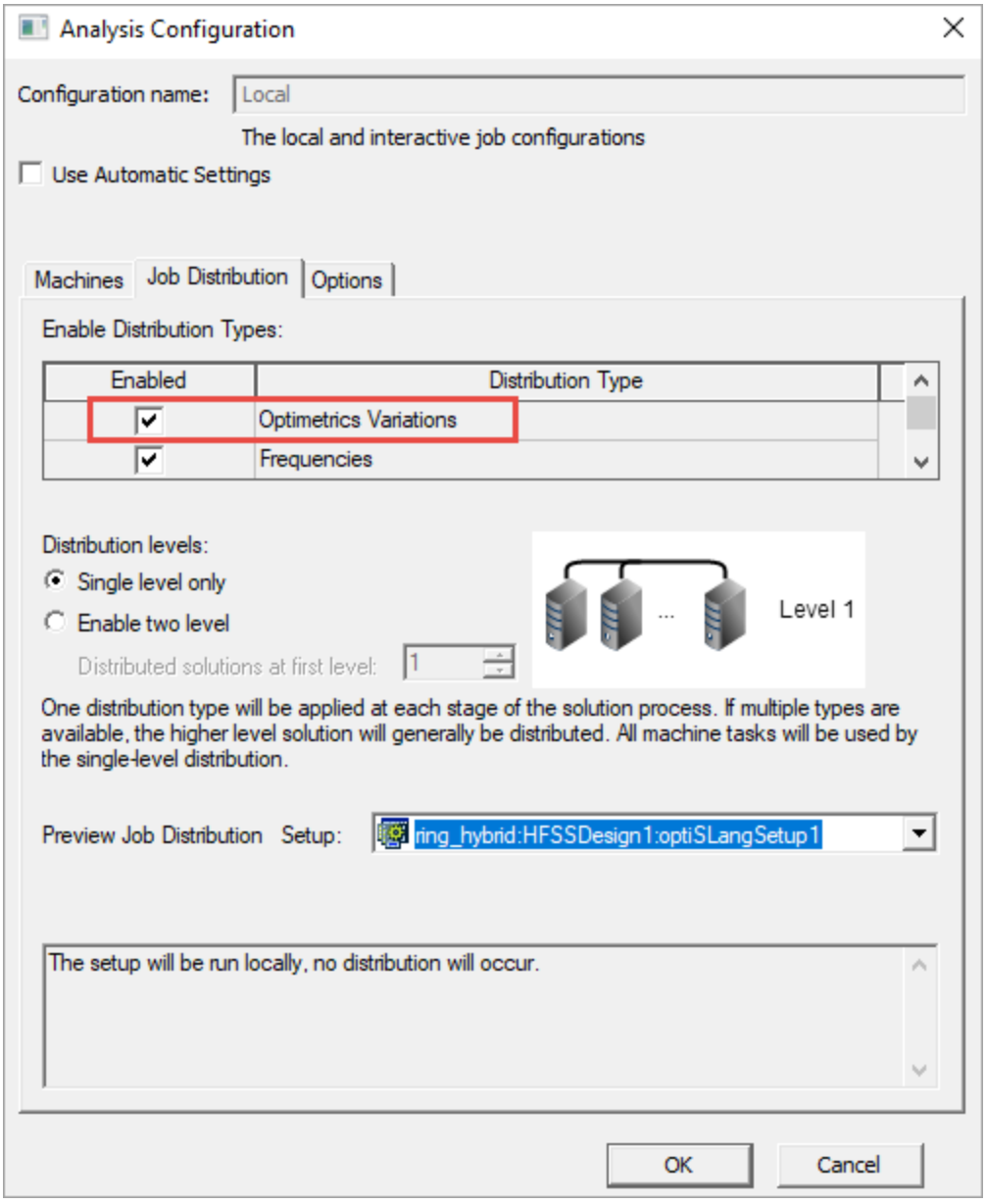
Example 1: Solving a User-created, Pre-existing optiSLang Setup

Recall that optiSLang setups are handled as Optimetrics setups in AEDT.

By preparing an optiSLang setup in an AEDT project, you can create regular DSO jobs even though the AEDT integration node is left in the single simulation run mode. In every design folder, the clone of the reference project contains the optiSLang setup and its solution is triggered under the **Analyze All** umbrella. Depending on the HPC settings active in AEDT for the model type in question, this yields optiSLang DSO jobs.

A key setting is the check box for enabling **Optimetrics Variations** in the **Job Distribution** tab of the **Analysis Configuration** window:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

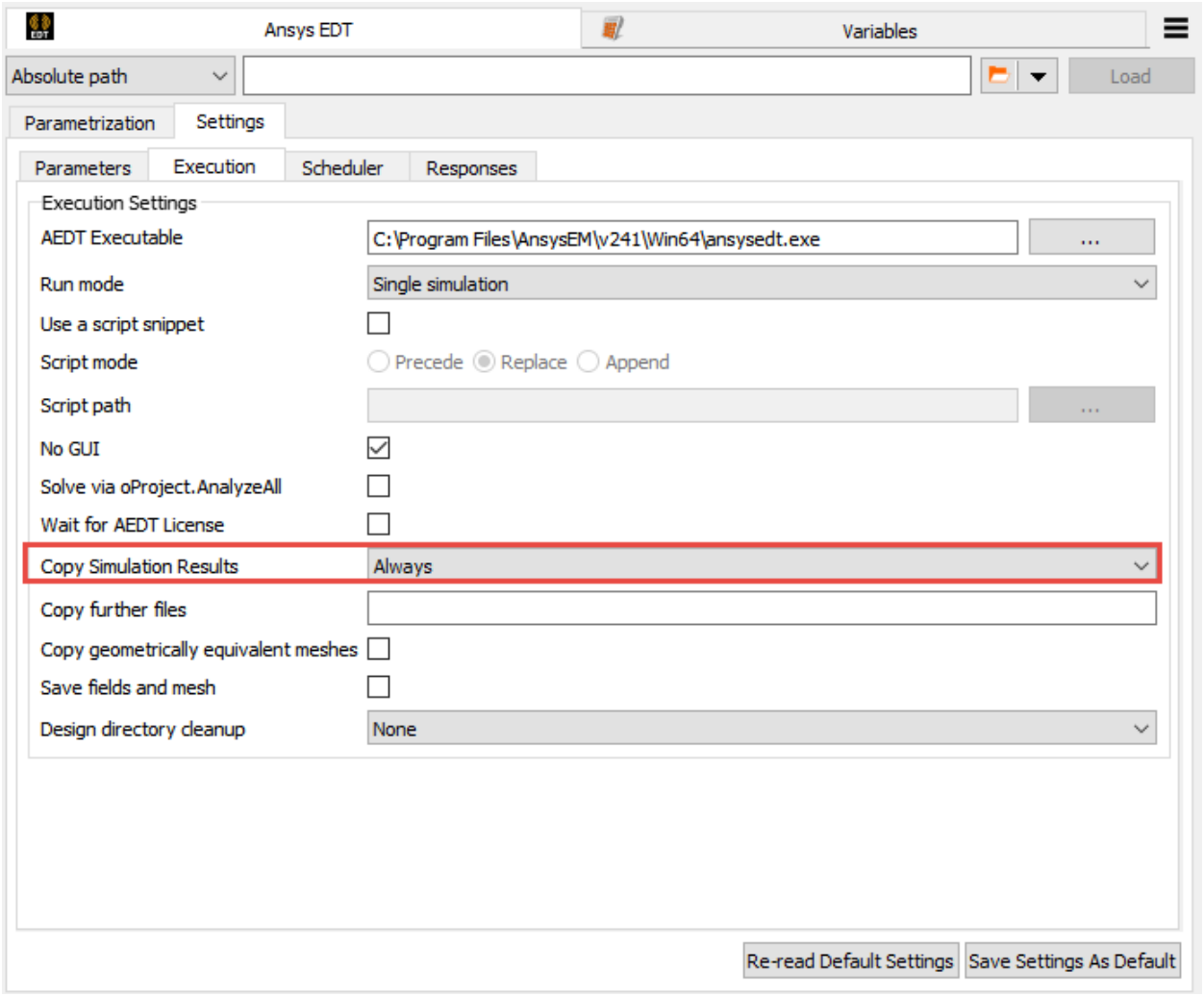
The **Use Automatic Settings** check box must be disabled to view this tab.

The working points being solved completely independently (simultanization) enables nice speed-up degrees for all designs within the optiSLang setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example 2: Increasing Performance by Avoiding Wasteful Solution of the Nominal Design

In Example 1, performance drag can exist if the analysis setup of the nominal design is also solved in every optiSLang design folder in series with a desired Optimetrics setup. Keeping the optiSLang setting **Copy Simulation Results** on the value **Always** can avoid redundant solution of the nominal design at the cost of the time for copying the *.aedtresults folder of the reference design repeatedly into every design folder.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example 3: Avoiding Redundant Computations while Exploiting AEDT Features around Postprocessing Variables

In HFSS, different excitations can be evaluated from one single set of FEM solution data based on linear superposition of field components. The FEM solution takes time and consumes large compute resources, whereas the linear superposition evaluation represents a comparatively tiny calculation.

In optiSLang, an AEDT node can be used to copy a reference project along with its associated *.aedtresults directory of solution data. When this node is used for evaluating different combinations of postprocessing variables, AEDT does not call any solver and that the evaluation time per design is accordingly fast. The only drag consists in loading a fresh instance of the AEDT program for every new design.

Example 4: Parametrization of an AEDT Design for optiSLang Integration

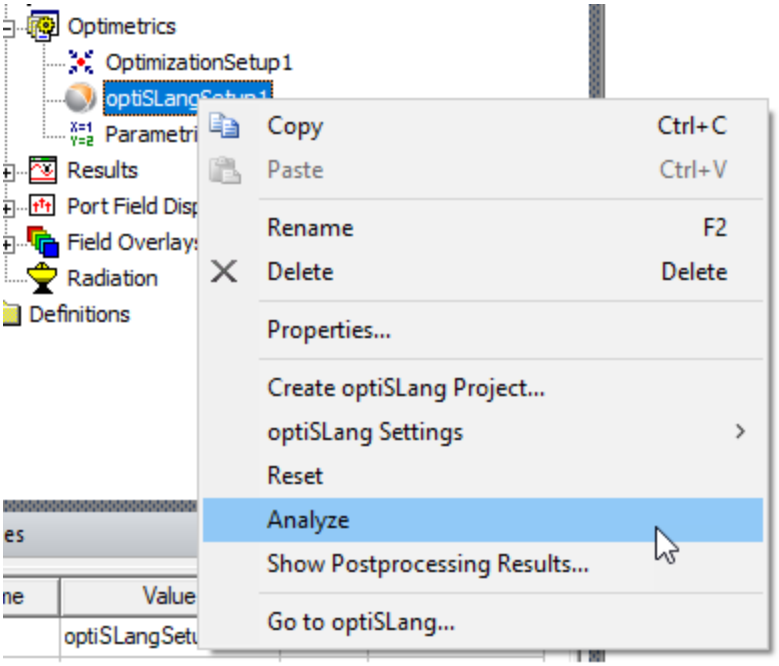
Consider a 2D Maxwell model of a permanent magnet synchronous motor. For design optimization, the magnet positions and angles can be the crucial parameters and due to symmetry a quarter model can represent the entire machine. For questions of robustness, reliability, tolerances, and achieving specs, it might be necessary to parametrize eccentricity and ellipticity. If magnet quality scatter is important, this might be another aspect requiring a full 360 degree model because it would be drastically unrealistic to assume that 16 worst-case magnets can find their way into one single machine. See: Parametrization for optiSLang Integration.

Solving an optiSLang Setup

To analyze an optiSLang setup within Electronics Desktop:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From the **Project Manager**, right-click a setup and select **Analyze**.



Considerations for Using the Analyze All Command

The **HFSS** > **Analyze All** command solves all projects in a design, including Optimetrics setups. See: Running More Than One Simulation.

> **Warning:** Before using the **Analyze All** command, remove any unnecessary analysis setups from the project.

Some reasons for doing so:

- During regular DSO run mode, optiSLang creates Optimetrics DSO design tables. By leaving just one setup for analysis, there is no ambiguity about which analysis setup is associated with any new table.
- Numerical convergence settings steer the performance trade-off between speed and accuracy. If a small number of designs take exceptionally long to converge, it can make a substantial performance difference whether designs are solved independently or in groups (DSO jobs). In group jobs, one slowly converging simulation can delay the entire group. This increases the importance of upper bounds on iterations for mesh refining, interpolation point addition, and so on.
- The unintended presence of Optimetrics setups (including optiSLang setups) can lead to long execution times. When used intentionally, however, Optimetrics setups can be used to define parametric sweeps. These sweeps are easy to set up and represent the sweep

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

per design as a nested system in optiSLang. Note that reports must be adjusted accordingly, so that data is drawn from the entire sweep and not a single point, in order to reach the optiSLang database.

- Analysis setups contain settings that drastically impact the resulting file and folder sizes (for example, the **Save Fields** check box). If saved fields consume disk space in the range of gigabytes for a single project, then the creation of hundreds of design folders with solution data by optiSLang can quickly fill a hard drive.

Considerations for Distributed Analysis

There are additional considerations for using Distributed Analysis and Large-Scale DSO to analyze optiSLang setups.

- Using the AEDT node's run mode "regular DSO" is not the only way to create DSO jobs. Also in run mode "single simulation" DSO jobs can be spawned. For example, due to the **Analyze All** command, a user-created pre-existing Optimetrics setup could be solved for every one of optiSLang's design evaluations.
- The integration node's **Designs Per Execution** setting can work with HPC and DSO settings for AEDT solvers in many different ways. If set to 4, optiSLang can send 4 designs per AEDT call when AEDT is geared at solving 4 design variations simultaneously. If **Designs Per Execution** is set to 9999, however, each optiSLang algorithm will always send all designs waiting in the pipeline at once (a robustness sampling of 1000 designs can be cast into one single DSO job). Using HPC settings, AEDT can still be made to break it down with ntasks=ncores=4. The drawback is that the success/failure info becomes visible in optiSLang only at the very end of the job. A large value for **Designs Per Execution** can make sense for optimization algorithms demanding design sets of fluctuating size.
- With population-based optimizers, it can be useful to match the population size with compute resources to avoid job load fluctuations.
- Several optiSLang nodes offer the feature to list files to be copied from the reference location into each design folder. However, this may not work for directories. A simple solution is to use a Python node and the shutil.copytree command.
- Avoid redundant solve actions. It may be helpful to copy not only the bare *.aedt file but also the *.aedtresults directory into each design folder. When the AEDT node issues the **Analyze All** command on the project level, pre-existing solution data can reduce the scope of analysis setups that are triggered to be solved. Compare the time for copying the data of solved projects against the time gain due to less computation. Avoid disc space concerns by activating design directory purging rules to be applied to every design folder after finished evaluation.
- AEDT offers a special type of postprocessing variables. When they are varied while other parameters stay constant, a new design variation can be evaluated with very little computational burden as compared to solving from scratch. The AEDT integration node

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

provides copy functionalities for *.aedtresults folders allowing to benefit from AEDT features around postprocessing variables in certain use cases.

optiSLang Menu Options in AEDT

Once an optiSLang setup has been created in AEDT, additional menu options become available.

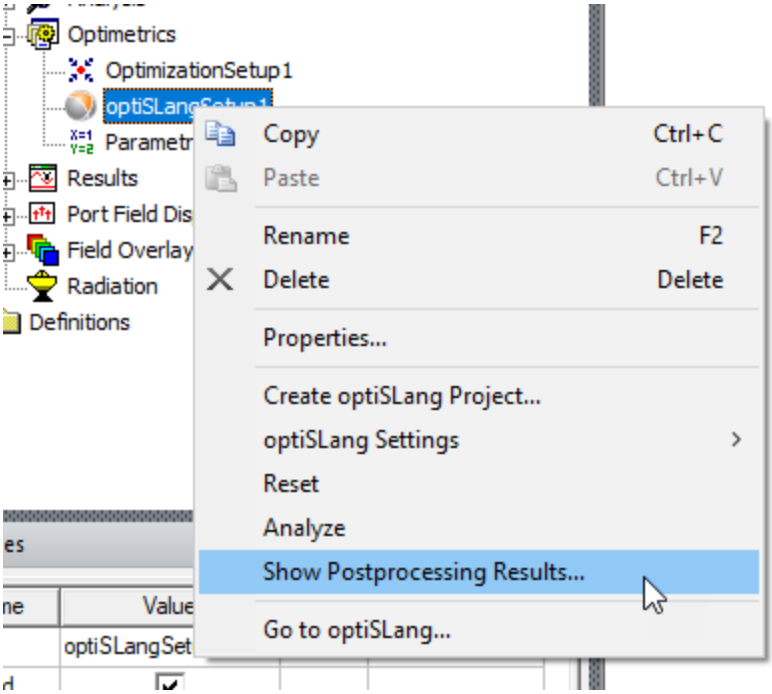
Right-click options include:

- **Create optiSLang Project** – creates an optiSLang project from the current design.
- **Edit AEDT Node Settings** – launches optiSLang with the AEDT project connected, on the Parametrization tab. For more information on node settings, consult the optiSLang help.
- **Edit Algorithm settings** – launches optiSLang and opens the Analysis Wizard, with options to analyze for Sensitivity, Optimization, or Robustness/Reliability. For more information on the Analysis Wizard, consult the optiSLang help.
- **Reset** – resets the optiSLang project.
- **Analyze** – launches the analysis of the optiSLang Optimetrics setup. When it has finished, post-processing results are available.
- **Show Postprocessing Results** – launches an optiSLang postprocessing results window. This menu option only available after analysis has been performed.
- **Go to optiSLang** – launches optiSLang.

Viewing optiSLang Postprocessing Results

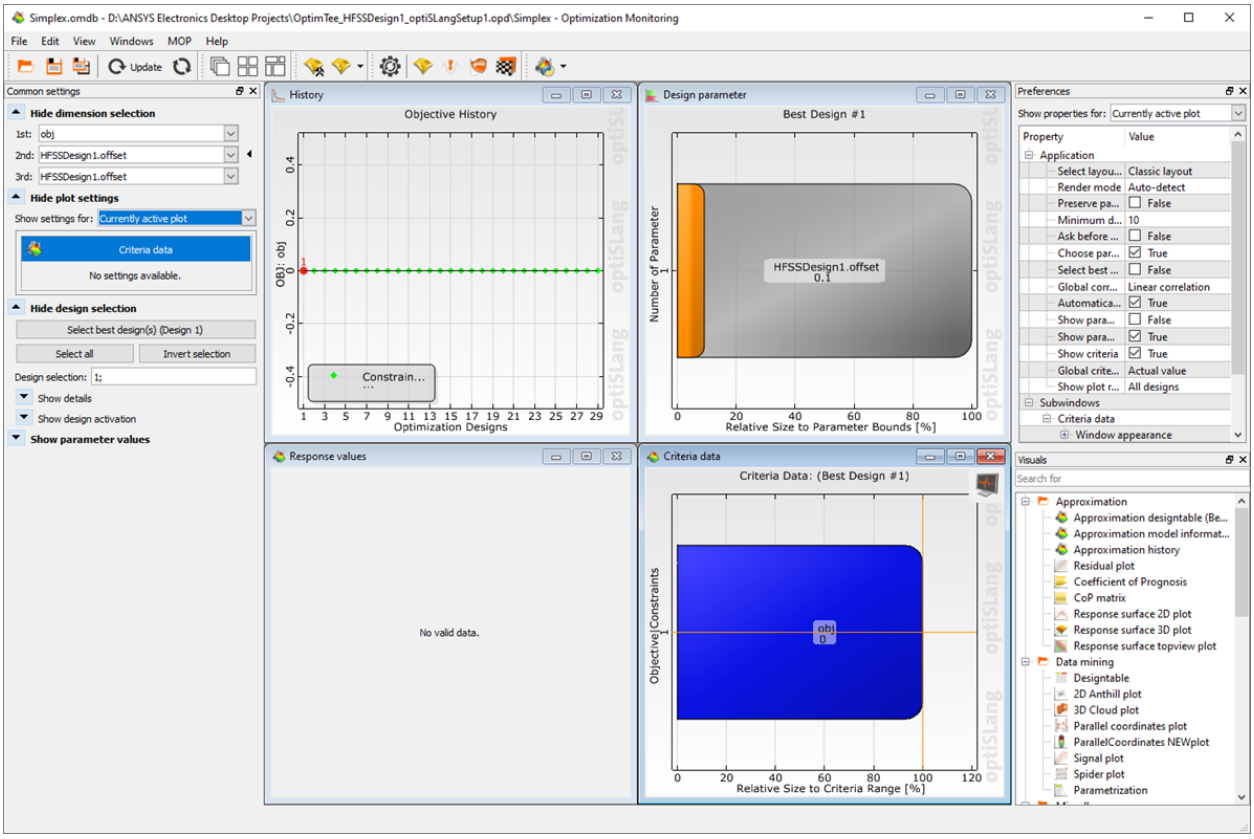
Once you have created and run an optiSLang setup, right-click it and select **Show Postprocessing Results**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



An optiSLang window opens, showing postprocessing features.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For more information, use optiSLang's **Help** menu to access the optiSLang online help.

Parametric Overview

Running a parametric analysis enables you to simulate several design variations using a single model. You define a series of variable values within a range, or a variable sweep definition, and HFSS generates a solution for each design variation. You can then compare the results to determine how each design variation affects the performance of the design.

You can vary design parameters that are assigned a quantity, such as geometry dimensions, material properties, and boundary and excitation properties (See the help topic for the specific parameter you want to vary). The number of variations that can be defined in a parametric sweep setup is limited only by your computing resources.

To perform a parametric analysis, you first create a nominal design. A nominal design is created like any other design, except that variables are assigned to those aspects of the model you want to change. You can create a parametric setup before defining variables but all variables must be defined before you start the parametric analysis. Although you are not required to solve the nominal design before performing a parametric analysis, doing so helps ensure that the model is

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

set up and operates as intended. Alternatively, you can perform a validation check on the nominal design before performing a parametric analysis.

> **Note:**
>
> For **HFSS** and **HFSS 3D Layout** designs only, you can validate all design variations in a parametric setup prior to performing the parametric analysis. See: Validating a Parametric Setup.

Parametric analyses are often used as precursors to optimization analyses because they enable you to determine a reasonable range of variable values within which optimal conditions will occur.

Setting Up a Parametric Analysis

A *parametric setup* specifies all of the design variations that Optimetrics drives the software to solve. A parametric setup is made up of one or more variable sweep definitions, which are a set of variable values within a range that you want the software to solve when you run the parametric setup.

You can define more than one parametric setup per design.

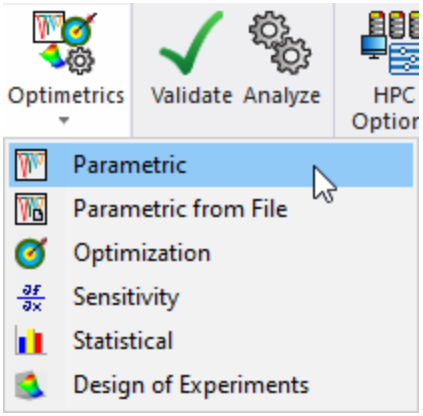
> **Note:**
>
> Once you have created a parametric setup, you can copy and paste it, and then make changes to the copy, rather than redoing the whole process for minor changes.

To add a parametric setup to a design:

1. Click **HFSS**> **Optimetrics Analysis**> **Add Parametric** .
    - Alternatively, right-click **Optimetrics** in the Project Manager and then click **Add** > **Parametric** on the shortcut menu.
    - Select the **Simulation** tab in the ribbon, and select **Parametric** from the drop-down

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

menu under the Optimetrics icon:



The **Setup Sweep Analysis** dialog box appears.

2. Add a variable sweep definition.

   After you define a parametric sweep, a shortcut menu becomes available when you right-click the setup name in the Project Manager.

3. Optionally, validate the parametric design variations before solving the parametric analysis.

   The **Validate** command is located in the shortcut menu that appears when you right-click the parametric setup name. The results of the validation process can be viewed by selecting **View Analysis Result** from the same shortcut menu. During validation, messages appear in the Message Manager window, and the validation progress is shown in the Progress window.

> **Note:**
>
> Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

Adding a Variable Sweep Definition

A parametric setup is made up of one or more *variable sweep definitions*. A variable sweep definition is a set of variable values within a range that Optimetrics drives to solve when the parametric setup is analyzed. You can add one or more sweep definitions to a parametric setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Sweeping a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

1. Click **HFSS** > **Optimetrics Analysis** > **Add Parametric** .
   - Alternatively, right-click **Optimetrics** in the project tree, and then click **Add** > **Parametric** on the shortcut menu.

   The **Setup Sweep Analysis** dialog box appears.
2. Under the **Sweep Definitions** tab, click **Add**.

   The **Add/Edit Sweep** dialog box appears.

   All the independent variables associated with the design are listed in the **Variable** drop-down menu of the **Add/Edit Sweep** dialog.
3. Click the variable for which you are defining the sweep definition from the **Variable** drop-down menu.

   If you do not define a sweep definition for a variable in the list, the variable's current value in the nominal design is used in the parametric analysis.
4. Specify the variable values to be included in the sweep.
5. Click **Add** and then click **OK**.

   You return to the **Setup Sweep Analysis** dialog box. The variable sweep is listed in the top half of the window.
6. View the design variations that are to be solved in table format under the **Table** tab. Viewing the sweep definition in table format enables you to visualize the design variations that are to be solved and manually adjust sweep points if necessary.
7. Click **OK**.

Specifying Variable Values for a Sweep Definition

To specify the variable values to include in a sweep definition:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select one of the following in the **Add/Edit Sweep** dialog box:

Single value	Specify a single value for the sweep definition.
Linear step	Specify a linear range of values with a constant step size.
Linear count	Specify a linear range of values and the number, or count of points within this range.
Decade count	Specify a logarithmic (base 10) series of values, and the number of values to calculate in each decade.
Octave count	Specify a logarithmic (base 2) series of values, and the number of values to calculate in each octave.
Exponential count	Specify an exponential (base e) series of values, and the number of values to calculate.

2. If you selected **Single value**, type the value of the sweep definition in the **Value** box.

   If you selected another sweep type, do the following:

   a. Type the starting value of the variable range in the **Start** text box.
   b. Type the final value of the variable range in the **Stop** text box.

   > **Warning:**
   >
   > Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

3. If you selected **Linear step** as the sweep type, type the step size in the **Step** box.

   The step size is the difference between variable values in the sweep definition. The step size determines the number of design variations between the start and stop value. The model is solved at each step in the specified range, including the start and stop values. The step size can be negative, when the **Stop** value is less than the **Start** value

   If you selected another sweep type, type the number of points, or variable values, in the sweep definition in the **Count** text box. For **Decade count** and **Octave count**, the **Count** value specifies the number of points to calculate in every decade or octave. For **Exponential count**, the **Count** value is the total number of points. The total number of points includes the start and stop values.

Synchronizing Variable Sweep Definitions

By default, variable sweep definitions are nested. Alternatively, you can synchronize the variable sweep definitions if they have the same number of sweep points.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, if you synchronize a sweep definition that includes values of 1, 2, and 3 inches with a second sweep definition that includes values of 4, 5, and 6 inches, 3 design variations are solved. The first variation is solved at the variable values of 1 and 4; the second variation is solved at the variable values 2 and 5; and the third variation is solved at the final variable values 3 and 6.

To synchronize variable sweep definitions:

1. Under the **Sweep Definitions** tab of the **Setup Sweep Analysis** dialog box, select the rows containing the sweep definitions you want to synchronize.
2. Click **Sync**.

   The synchronized sweeps are given a group number, which is listed in the **Sync #** column.

   Optionally, view the design variations that are to be solved in table format under the **Table** tab.

Modifying a Variable Sweep Definition Manually

You can manually modify the variable values that are solved for a parametric setup by explicitly changing, adding, or deleting existing points in a variable sweep definition under the **Table** tab of the **Setup Sweep Analysis** dialog box.

To manually modify a variable sweep definition:

1. Click the **Table** tab of the **Setup Sweep Analysis** dialog box.

   The design variations that will be solved for the parametric setup are listed in table format.

2. Do one of the following:
   - To modify a variable value, click a value text box in the table and type a new value.
   - To delete a variable value from the sweep definition, click the row you want to delete, and then click **Delete**.
   - To add a new variable value to the sweep definition, click **Add**. Then click in the value text box and type a new value.

   > **Warning:**
   >
   > Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

Your modifications are tracked and available for viewing at the bottom of the **Setup Sweep Analysis** dialog box under the **Sweep Definitions** tab. The operations you performed are listed with descriptions.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Warning:**
>
> If you modify an original sweep definition using the **Add/Edit Sweep** dialog box after you have manually modified its table of design variations, your manual modifications become invalid and are removed. A warning is displayed to inform you that your manual values are about to become invalid, so you can decide whether or not to proceed.

Overriding a Variable's Current Value in a Parametric Setup

If you choose not to sweep a variable, the variable's current value set for the nominal design is used when it solves the parametric setup. To override the current variable value for a parametric setup:

1. In the **Setup Sweep Analysis** dialog box, click the **General** tab.

   Under **Starting Point**, all of the current independent design variable values are listed.

2. Click the **Override** box of the design variable with the value you want to override for the parametric setup.
3. Type a new value in the **Value** box, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered will be used for the parametric setup. For this parametric setup, the new value will override the current value in the nominal design.

   > **Note:**
   >
   > Alternatively, you can select the **Override** option first, and then type a new variable value in the **Value** box.

4. Optionally, click a new unit in the **Units** box.

To revert to the current variable value, clear the **Override** option.

> **Warning:**
>
> Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

Specifying a Solution Setup for a Parametric Setup

To specify the solution setup that the software analyzes when it solves a parametric setup:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Setup Sweep Analysis** dialog box, click the **General** tab.
2. Select the solution setup you want the software to use when it solves the parametric setup.

   The parametric setup is solved using the solution setup you select. If you select more than one, results are generated for all selected solution setups.

Specifying the Solution Quantity to Evaluate for Parametric Analysis

When you add a parametric setup, you can identify one or more solution quantities to be presented in the *Post Analysis Display* dialog box. The solution quantities are specified by mathematical expressions that are composed of basic quantities, such as output variables. When you view the results, HFSS extracts the solution quantities and lists them in the results table.

1. In the *Setup Sweep Analysis* dialog box, click the **Calculations** tab.

   This displays a table that will show Solutions and associated Calculations. Below the table, are control buttons to **Setup Calculations...** and **Delete**.

2. Click **Setup Calculations**.

   This displays the *Add/Edit Calculation* dialog box. The dialog contains panes to set the **Context**, the **Trace** tab for the **Calculation Expression**, and the **Calculation Range** tab for the **Calculation Range**.

   Follow the procedure to Setup Calculations for Optimetrics.

3. Click **Add Calculation** to add the expression in the *Add/Edit Calculation* dialog box's **Calculation Expression** field to the Calculations tab of the *Setup Sweep Analysis* dialog box.
4. Click **Done** to close the *Add/Edit Calculation* dialog box.

Setting up Calculations for Optimetrics

The **Setup** dialog boxes for each of the Optimetrics types include a **Setup Calculations** button. Clicking this button displays the **Add/Edit Calculation** dialog box, which contains distinct panes and tabs to set the **Context**, the **Calculation Expression**, and the **Calculation Range**.

The **Context** pane contains fields for the Report Type to use, the Solution, and depending on the Report Type selection, the Geometry.

The **Trace** tab contains fields for the Calculation expression and, to build the expression, a Category list, a Quantity list with a Text Filter field, and a list of Functions available for the selected Category. The Range function button opens a dialog box in which you can define a range function to apply a function to the expression.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Category list for the **Trace** tab includes Variables and Output Variables. An Output Variables... button lets you open a dialog box to define and edit the Output Variables.

To set up an Optimetrics calculation:

1. Click **Setup Calculations** to open the **Add/Edit/Calculation** dialog box.
2. In the **Report Type** text field in the **Context** pane, select from the drop-down menu of available types.

   Selecting Fields as the Report type causes the **Geometry** field to display.
3. In the **Solution** text box, select from the drop-down menu of available solutions.
4. If the **Geometry** field is available, select from the drop-down menu.
5. In the **Trace** tab, specify the solution Category, a Quantity, and Functions. The resulting expression will be displayed in the *Calculation Expression* field.
   a. Select the **Category** from the list.

      The selection appears in the **Calculation Expression** field, and the Quantity and Function fields list what is available for the corresponding selection.

   b. Select the **Quantity** from the list.

      The selected quantity appears in the **Calculation Expression** field.

      If the **Quantity** list is long, you can filter it for easier selection by typing in the text filter field. Only quantities that contain those alphanumeric characters anywhere in their name will remain visible in the list.

      If you want to create an output variable that represents the solution quantity, do the following:

      i. Click **Output Variables**.

         The **Output Variables** dialog box appears.

      ii. Add the expression you want to evaluate, and then click **Done**.

         The recently created output variable appears in the **Quantity** list.

      iii. Click a new output variable in the **Quantity** drop-down menu.

      > **Note:**
      >
      > The calculation you specify must be able to be evaluated into a single, real number.

      The selected Quantity appears in the **Calculation Expression** field.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Select the **Function** from the list.

   The selected function is applied to the **Quantity** in the **Calculation Expression** field.

6. To apply a **Range function** to the **Calculation Expression**, see Setting a Range function.
7. Click **Add Calculation** to add the expression in the **Add/Edit Calculation** dialog box's **Calculation Expression** field to the Calculations tab of the **Setup Sweep Analysis** dialog box.
8. Click **Done** to close the **Add/Edit Calculation** dialog box.

Specifying a Solution Quantity's Calculation Range

The calculation range of a solution quantity determines the value of intrinsic variables such as frequency (F) at which the solution quantity will be extracted. For a parametric setup, the calculation range must be a single value. For a Driven Modal or Driven Terminal design, if you selected to extract the solution from the last adaptive solution, Optimetrics uses the adaptive frequency defined in the solution setup. If you selected to extract the solution quantity from a frequency sweep solution, Optimetrics by default will use the starting frequency in the sweep.

1. In the **Setup Sweep Analysis** dialog box, click the **Calculations** tab.
2. Click the **Setup Calculations** button.

   The **Add/Edit/Calculation** dialog box appears.

3. Select the **Calculation Range** tab.
4. In the **Variable** list, click an intrinsic variable.

   Depending on the variable and Report Type, either a single value or All appears in the Value field.

5. In the **Value** box, click the ellipsis button to display a dialog box that lets you select particular values, or specify a range of values (for Time sweeps). By default, **Use all values** is selected.
6. Click **Update**, and then click **Edit**.

Viewing Results for Parametric Solution Quantities

1. In the project tree, right-click the parametric setup for which you want to view the results calculated for the solution quantities, and then click **View Analysis Result** on the shortcut menu.

   The **Post Analysis Display** dialog box appears.

2. Select the parametric setup with the results you want to view from the drop-down menu at the top of the dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. If it is not already selected, select **Table** as the view type.

   The results for the selected solution quantities are listed in table format for each solved design variation. The variation column in the table lists the entries in order. Clicking the Vision header inverts the order. Clicking other headers sorts the entries by value, and clicking again inverts the order.

4. Optionally, select **Show complete output name**.

   The complete name of the solution for which the results are being displayed will be listed in the column headings.

5. Optionally, click a design variation in the table, and then click **Apply** (at the far right side of the dialog box).

   The design displayed in the **Modeler** window is changed to represent the selected design variation.

Using Distributed Analysis

If you have purchased the appropriate license, the Electronics Desktop supports distributed solve, which involves distributing rows of a parametric table during Optimetrics solve.

If you do a distributed solve, the Electronics Desktop launches solver engines on multiple machines, assuming that you have configured your HPC and Analysis Options correctly. Also see Large Scale DSO for Parametric Analysis.

To run a distributed analysis:

1. Under **Optimetrics** in the project tree, right-click the specific parametric setup.

   A shortcut menu appears.

2. Select **Analyze** from the shortcut menu.

   > **Note:**
   >
   > After you define a parametric sweep, a shortcut menu becomes available when you right-click the setup name.

   While the analysis is running, you can access parent and child progress bars. By default, only the main progress bar is displayed, while the child progress bars (or subtasks) remain hidden. You can toggle between showing and hiding the child progress bars.

   To show the child progress bars:

   - Right-click the progress window, and select **Show Subtask Progress Bars**.

     To hide the child progress bars:

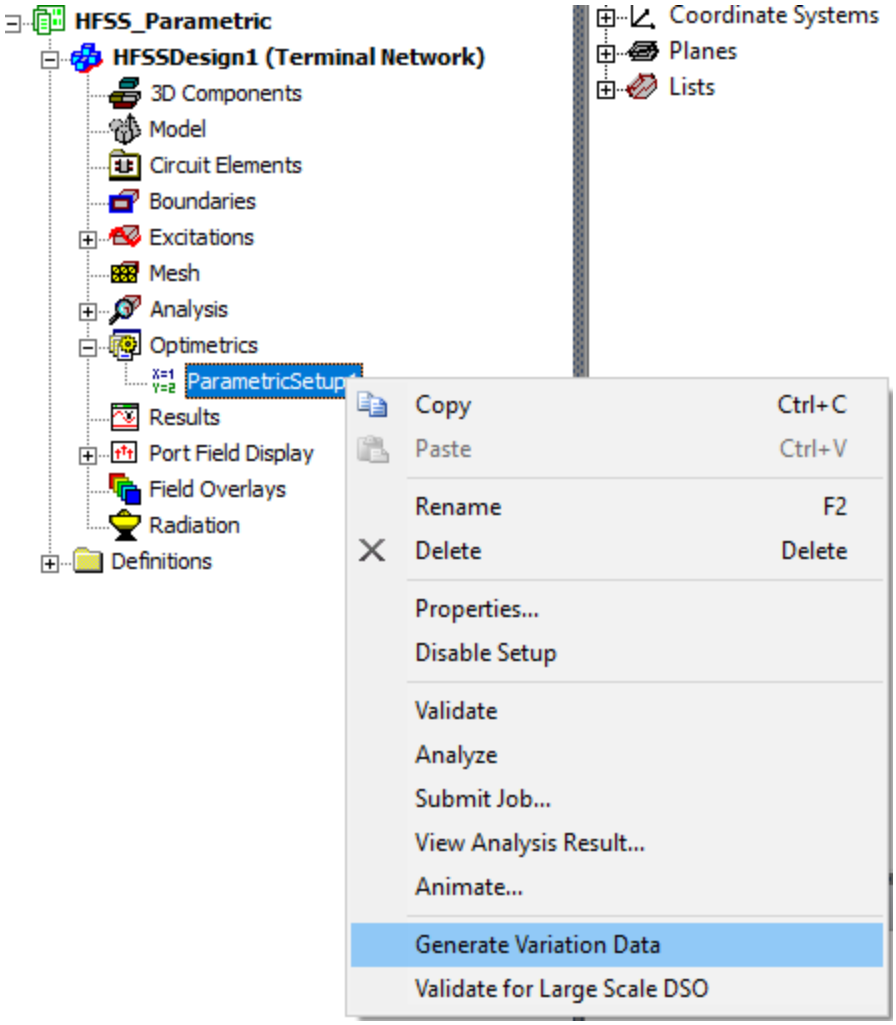
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Right-click the progress window, and select **Hide Subtask Progress Bars**.

Solving a Parametric Analysis on Another Machine

If you would like to run a parametric analysis on another machine, for example, to run a Discovery-linked model on Linux, you can generate the variation data and then export an archive of the project.

To generate the variation data, right-click on the **ParametricSetup** entry in the Project Manager tree, and select **Generate Variation Data**.



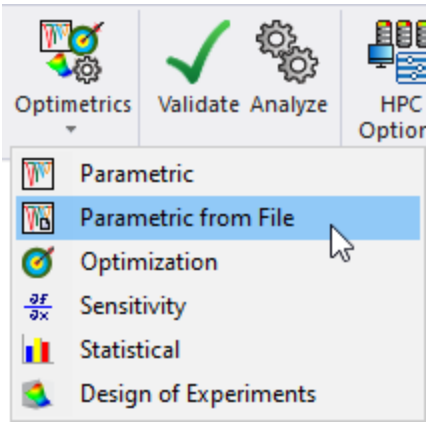
When this option is selected, HFSS will generate geometry for all variations and cache it in the results folder. Users can archive all the data, and then solve on other machines.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** If the parametric analysis uses a Discovery-linked model, the machine that the archive is moved to does not have to have Discovery installed.

Adding a Parametric Sweep from a File

You can specify the parameters for a parametric sweep in a spreadsheet that uses either a .csv (comma delimited) or .txt (tab delimited) format. You can then import the parametric sweep using the **HFSS** > **Optimetrics Analysis** > **Add Parametric from File** command, or with the **Simulation** tab selected, use the drop-down menu under the Optimetrics icon and select **Parametric from File**:



These methods open a file browser for a comma delimited file (.csv) or a tab delimited .txt file.

For example, a .txt spreadsheet file could resemble the following:

```
a $b $c[in] d[m] $e $f
0.1 mil 2mm 11 21 0.6in 8
0.2mil 3mm 1.3 2.6mm 3 9cm
...
```

The first row lists the Project and Design Variable names, and when followed by parentheses, the units. When unit not present, SI unit is assumed. The following rows provide the variable values and units, where SI is assumed if not specified. Project or Design variables must be defined before they are accepted from a file. The characters in variable names are not case sensitive. Consecutive separators are treated as one separator.

The header row also takes units in ( ) as well as the conventional [ ].

After you have imported a valid file, the detailed information regarding sweep values can be found at the lower portion of the Parametric **Setup Sweep Analysis** dialog**Sweep Definitions**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tab (Operation/Description panel). It is treated as: first row as single point. then each time a new sweep point is added.

Validating a Parametric Setup

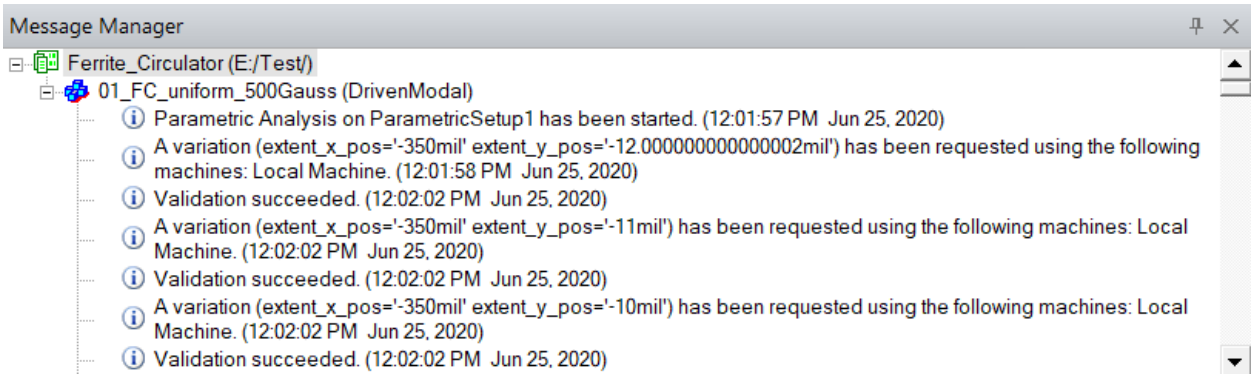
For HFSS and HFSS 3D Layout designs, you can validate all design variations in a parametric setup before solving the parametric analysis. This process is separate from the **HFSS> Validation Check** command or the equivalent **Validate** icon on the **Simulation** ribbon tab, which only validate the nominal solution setup.

After creating a parametric analysis setup, validate the design variations using the following procedure:

1. Under *Optimetrics* in the Project Manager, right-click the ***parametric setup name*** and choose **Validate** from the shortcut menu.

   The design variations are validated one at a time until all have been checked.

2. Optionally, display the **Progress** window to see a visual indication of the validation progress.
3. Optionally, display the **Message Manager** window to see the validation messages for each design variation:



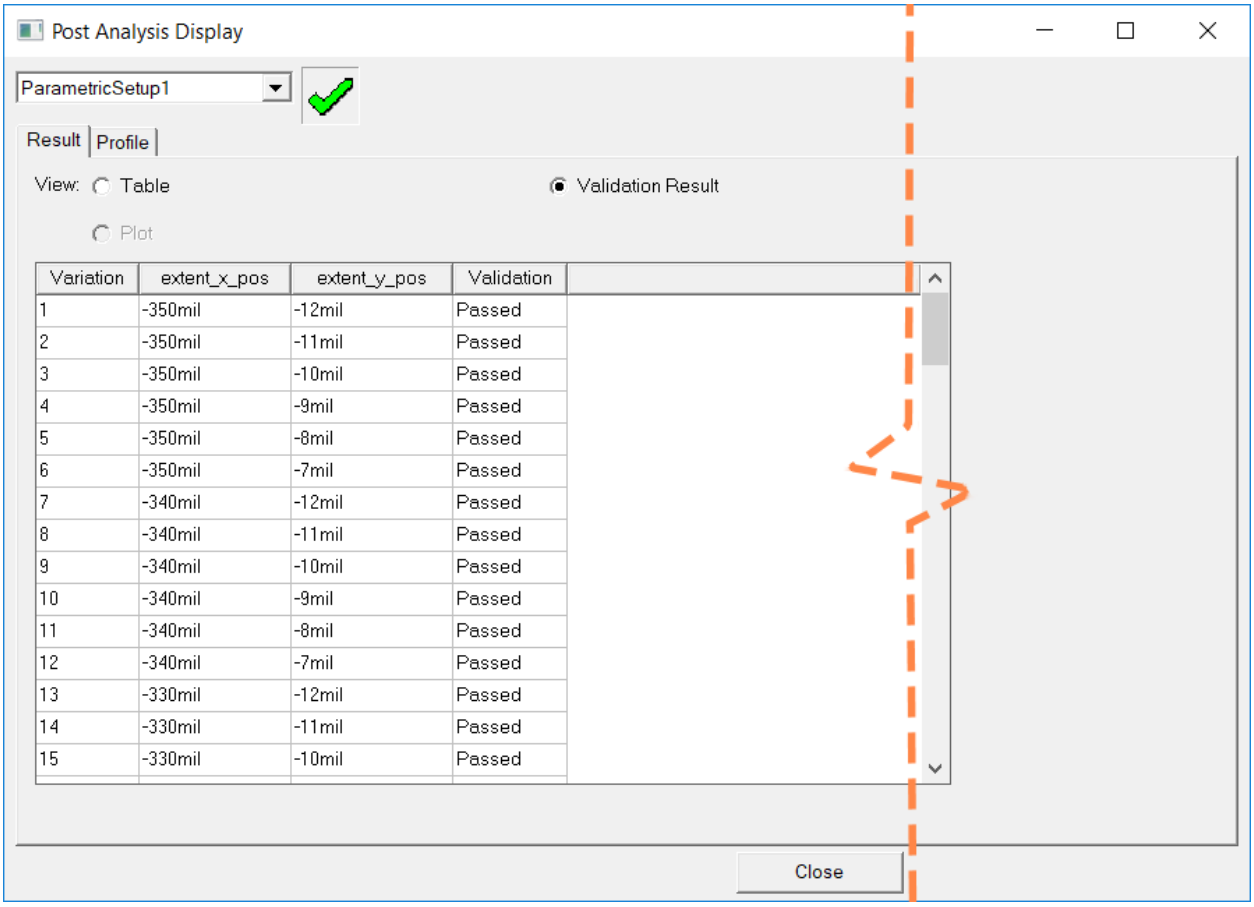
4. Under *Optimetrics* in the Project Manager, right-click the ***parametric setup name*** and choose **View Analysis Result**.

   The *Post Analysis Display* dialog box appears with the **Result** tab selected and with **Validation Result** selected as the *View* option.

5. If you have added more than one parametric setup to the design, select the ***setup name*** you wish to review from the drop-down menu in the upper-left corner of the dialog box.

   The following example shows a validation result where all variations passed:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



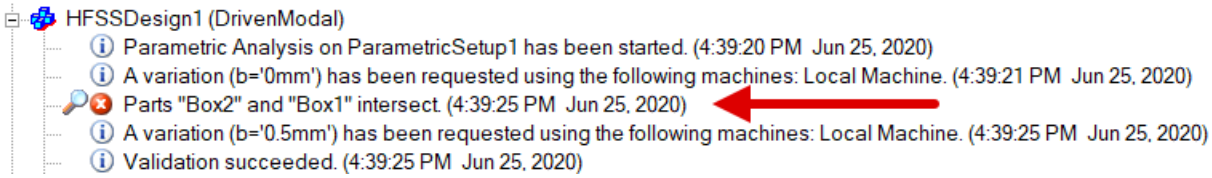
Variation	extent_x_pos	extent_y_pos	Validation
1	-350mil	-12mil	Passed
2	-350mil	-11mil	Passed
3	-350mil	-10mil	Passed
4	-350mil	-9mil	Passed
5	-350mil	-8mil	Passed
6	-350mil	-7mil	Passed
7	-340mil	-12mil	Passed
8	-340mil	-11mil	Passed
9	-340mil	-10mil	Passed
10	-340mil	-9mil	Passed
11	-340mil	-8mil	Passed
12	-340mil	-7mil	Passed
13	-330mil	-12mil	Passed
14	-330mil	-11mil	Passed
15	-330mil	-10mil	Passed

If a design variation fails the validation check, that row of the table is rendered with red text, and the *Validation* column shows "Failed":

Variation	b	Validation
1	0mm	Failed
2	0.5mm	Passed
3	1mm	Failed
4	-0.5mm	Passed

The details of the failure are not given in the *Post Analysis Display* dialog box. Look in the **Message Manager** window for those details:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the preceding example, two of the four design variations specified a dimensional variable (*b*) that caused two parts to intersect. That is, geometric interference was detected for two design variations.

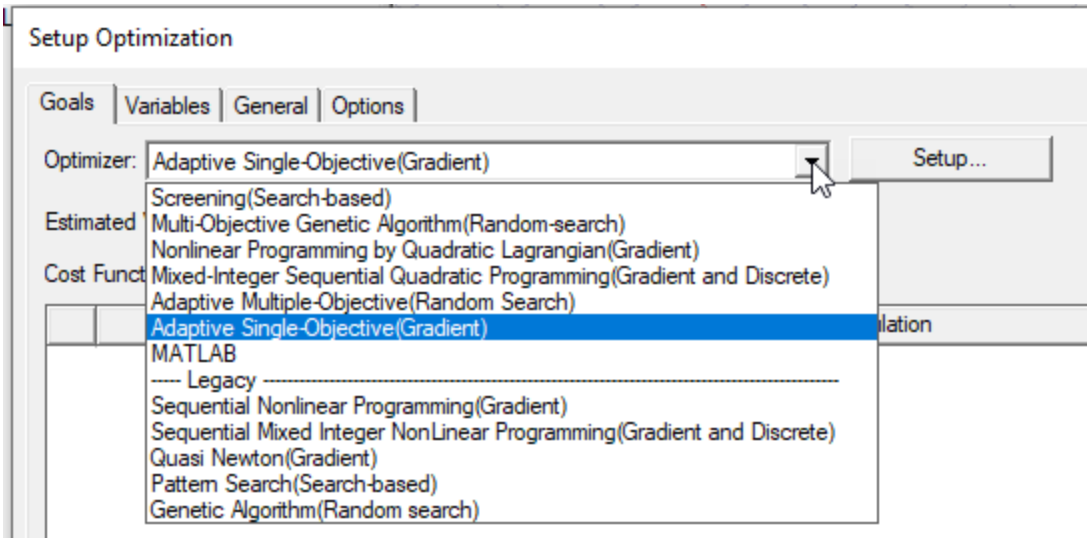
6. Click **Close** to dismiss the *Post Analysis Display* dialog box.

Optimization Overview

Optimetrics interfaces with Ansys Electromagnetics products to help you optimize a wide variety of design parameters based on variable geometry, materials, excitations, component values, etc. Optimization is the process of locating the minimum of a user-defined cost function. Optimetrics modifies the variable values until the minimum is reached with acceptable accuracy.

Choosing an Optimizer

Conducting an optimization analysis allows you to determine an optimum solution for your problem. In optimization analyses, there are many available optimizers.



Goal Driven Optimizers

These use a Decision Support Process (DSP) based on satisfying criteria as applied to the parameter attributes using a weighted aggregate method. In effect, the DSP can be viewed as a

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

post-processing action on the Pareto fronts as generated from the results of the various optimization methods.

- Screening (Search based) – a non-iterative direct sampling method that uses a quasi-random number generator based on the Hammersley algorithm. You can start with Screening to locate the multiple tentative optima and then refine with NLPQL or MISQP to zoom in on the individual local maximum or minimum value. Usually Screening is used for preliminary design, which can lead you to apply one of the other approaches for more refined optimization results.
- Multi-Objective Genetic Algorithm – an iterative random search algorithm that can optimize problems with continuous input parameters. It is better for calculating the global optima. You can start with MOGA to locate the multiple tentative optima and then refine with NLPQL or MISQP to zoom in on the individual local maximum or minimum value.
- Nonlinear Programming by Quadratic Lagrangian (Gradient) – a gradient-based, single-objective optimizer based on quasi-Newton methods. Ideally suited for local optimization.
- Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete) – a gradient-based, single-objective optimizer that solves mixed-integer non-linear programming problems by a modified sequential quadratic programming (SQP) method. Ideally suited for local optimization.
- Adaptive Multiple Objective – an iterative, multi-objective optimizer that employs a Kriging response surface and MOGA. In this method, the use of a Kriging response surface allows for a more rapid optimization process because all design points are not evaluated except when necessary and part of the population is simulated by evaluations of the Kriging response surface, which is constructed of all design points submitted by Multi-Objective Genetic Algorithm (MOGA).
- Adaptive Single Objective (Gradient) – a gradient-based, single-objective optimizer that employs an OSF (Optimal Space-Filling) DOE, a Kriging response surface, and MISQP.
- MATLAB

All optimizers assume that the nominal problem you are analyzing is close to the optimal solution; therefore, you must specify a domain that contains the region in which you expect to reach the optimum value.

All optimizers allow you to define a maximum limit to the number of iterations to be executed. This prevents you from consuming your remaining computing resources and allows you to analyze the obtained solutions. From this reduced range, you can further narrow the domain of the problem and regenerate the solutions.

All optimizers also allow you to enter a coefficient in the **Add Constraints** window to define the linear relationship between the selected variables and the entered constraint value. For the SNLP and SMINLP optimizers, the relationship can be linear or nonlinear. For the Quasi Newton and Pattern Search optimizers, the relationship must be linear.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cost functions can be quite nonlinear. As a result, during the function evaluations of the algorithm, the cost function can vary significantly. Also, it is important to understand the relationship between optimization function evaluation and iteration. Every iteration, depending on the number of parameters to be optimized, performs several function evaluations. These function evaluations, depending on how nonlinear the cost function is, could show drastic changes. The presence of drastic changes has no bearing on whether the optimization algorithm converged or not.

In the case of non-gradient search-based optimization algorithms, such as "pattern search," which are entirely based on function evaluations, one could see drastic changes in the function evaluations depending on how nonlinear the cost function is. This could seem misleading as if the algorithm did not converge since in theory one expects the cost function to decrease from one iteration to the next. The optimetrics, however, reports function evaluations and not necessarily the optimizer performance per iteration.

> **Note:**
>
> The MATLAB optimizer displays function evaluation when the **Show all functions evaluation** check box is selected. If the check box is not selected, it displays iteration.

Legacy Optimizers

These include

- Sequential Nonlinear Programming (Gradient)
- Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete)
- Quasi-Newton (Gradient)
- Pattern Search (Search-based)
- Genetic Algorithm (Random Search)

Quasi Newton (Gradient)

If the Sequential Non Linear Programming Optimizer has difficulty, and if the numerical noise is insignificant during the solution process, use the Quasi Newton optimizer to obtain the results. The Quasi Newton optimizer works on the basis of finding a minimum or maximum of a cost function which relates variables in the model or circuit to overall simulation goals. The user marks one or more variables in the project and defines a cost function in the optimization setup. The cost function relates the variable values to field quantities, design parameters like force or torque, power loss, etc. The optimizer can then maximize or minimize the value of the design parameter by varying the problem variables.

Sir Isaac Newton first showed that the maximum or minimum of any function can be determined by setting the derivative of a function with respect to a variable (x) to zero and solving for the

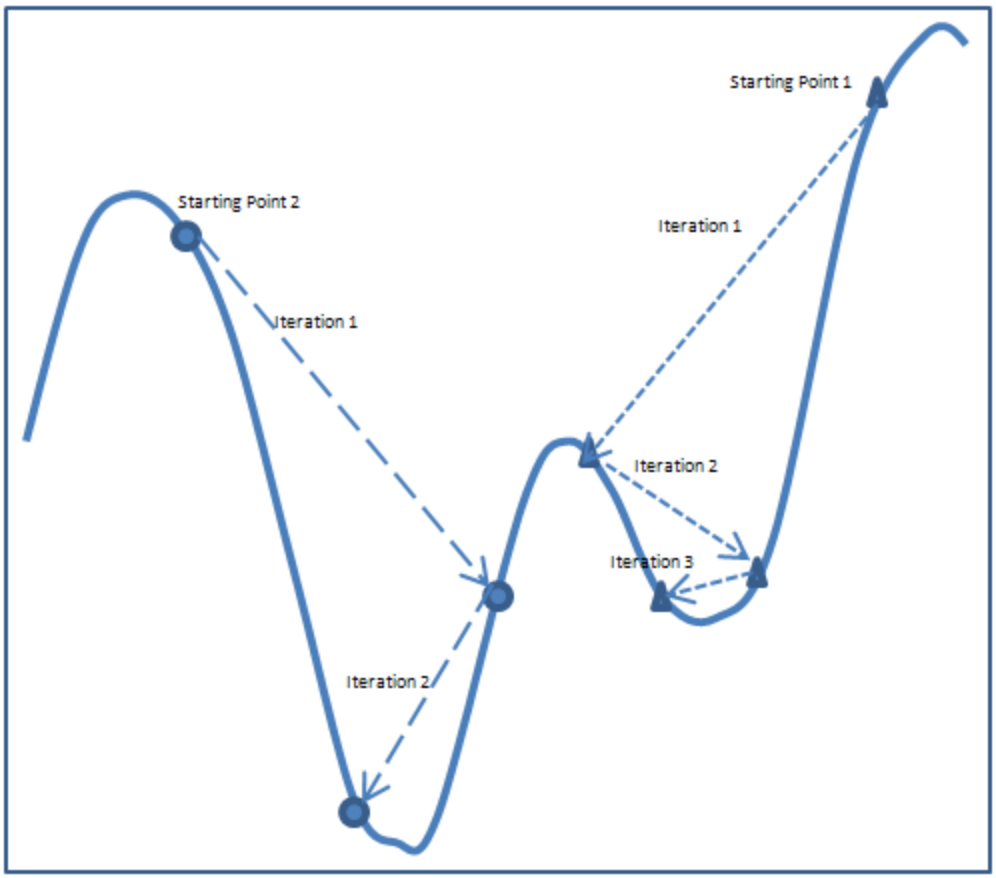
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

variable. This approach leads to the exact solution for quadratic functions. However, for higher order functions or numerical analysis, an iterative approach is commonly taken. The function is approximated locally by a quadratic and the approximation is solved for the value of x. This value is placed back into the original function and used to calculate a gradient which provides a step direction and size for determining the next best value of x in the iteration process.

In the Quasi-Newton optimization procedure, the gradients and Hessian are calculated numerically. Essentially, the change in x and the change in the gradient are used to estimate the Hessian for the next iteration. The ratio of the change in cost to the change in the values of x provides the gradient, whereas, the ratio of the change in the gradients to the change in the values of x provides the Hessian for the next step and is know as the quasi-Newton condition. In order to perform the Quasi-Newton optimization, at least three solutions are required for each parameter being varied. This can have a significant computational cost depending upon the type of analysis being performed.

There are numerous methods described in the literature for solving for the Hessian and the details of the method used by Optimetrics are beyond the scope of this document. However, as the Quasi-Newton method is, at its heart, a gradient method, it suffers from two fundamental problems common to optimization. The first is the possible presence of local minima. The following figure illustrates the problem of local minima.

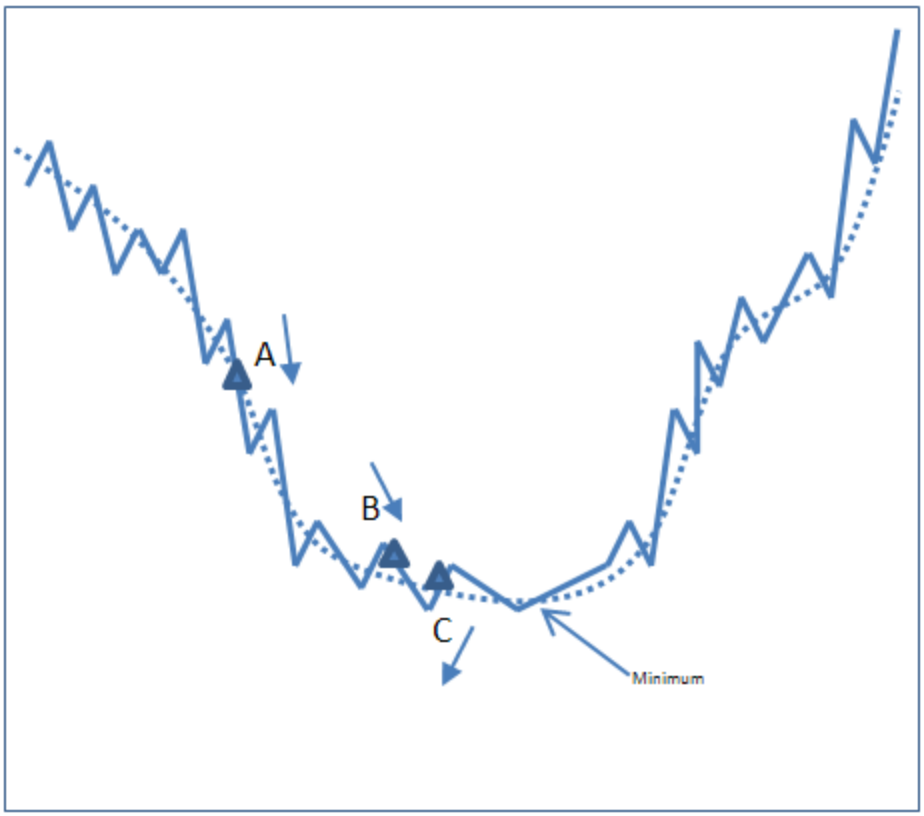
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In this scenario, you can see that in order to find the minimum of the function over the domain, a number of factors will determine the overall success including the initial starting point, the initial set of gradients calculated, the allowable step size, etc. Once the optimizer has located a minimum, the Quasi-Newton approach will locate the bottom and will not search further for other possible minima. In the example shown, when the optimizer begins at the point labeled "Starting Point 1" the minima it finds is a local minima and not a good global solution to the problem.

The second basic issue with Quasi-Newton optimization is numerical noise. In gradient optimization, the derivatives are assumed to be smooth, well behaved functions. However, when the gradients are calculated numerically, the calculation involves taking the differences of numbers that get progressively smaller. At some point, the numerical imprecision in the parameter calculations becomes greater than the differences calculated in the gradients and the solution will oscillate and may never reach convergence. To illustrate this, consider the figure shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In this scenario, the optimizer is looking for the point labeled "minimum". Three possible solutions are labeled A, B and C, with each arrow indicating the direction of the derivative of the function at that point. If points A and B represent the last two solution points for the parameter, then it is easy to see that the changes in the magnitude and the consistent direction of the derivatives will serve to push the solution closer to the desired minimum. If, however, points A and C are the last two solution points respectively, the magnitude indicates the proper direction of movement, but the derivatives are opposite, possibly causing the solution to move away from the minimum, back in the direction of point A.

In order to use the Quasi-Newton optimizer effectively, the cost function should be based on parameters that exhibit a smooth characteristic (little numerical noise) and a starting point of the optimization should be chosen somewhat close to the expected minimum based on an understanding of the physical problem being optimized. This becomes increasingly difficult, however, when multiple parameters are being varied or when multiple parameters are to be optimized. In addition, the computational burden of multivariate optimization with Quasi-Newton increases geometrically with the number of variables being optimized. As a result, this method should only be attempted when 1 or 2 variables are being optimized as a time.

For more information regarding Quasi-Newton optimization methods, see the following reference:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Schoenberg, Ronald. *Optimization with the Quasi-Newton Method*. Aptech Systems, Inc. 2001.

Pattern Search (Search-Based)

If the noise is significant in the nominal project, use the Pattern Search optimizer to obtain the results. It performs a grid-based simplex search, which makes use of simplices: triangles in 2D space or tetrahedra in 3D space. A simplex is a Euclidean geometric spatial element having the minimum number of boundary points, such as a line segment in one-dimensional space, a triangle in two-dimensional space, or a tetrahedron in three-dimensional space.

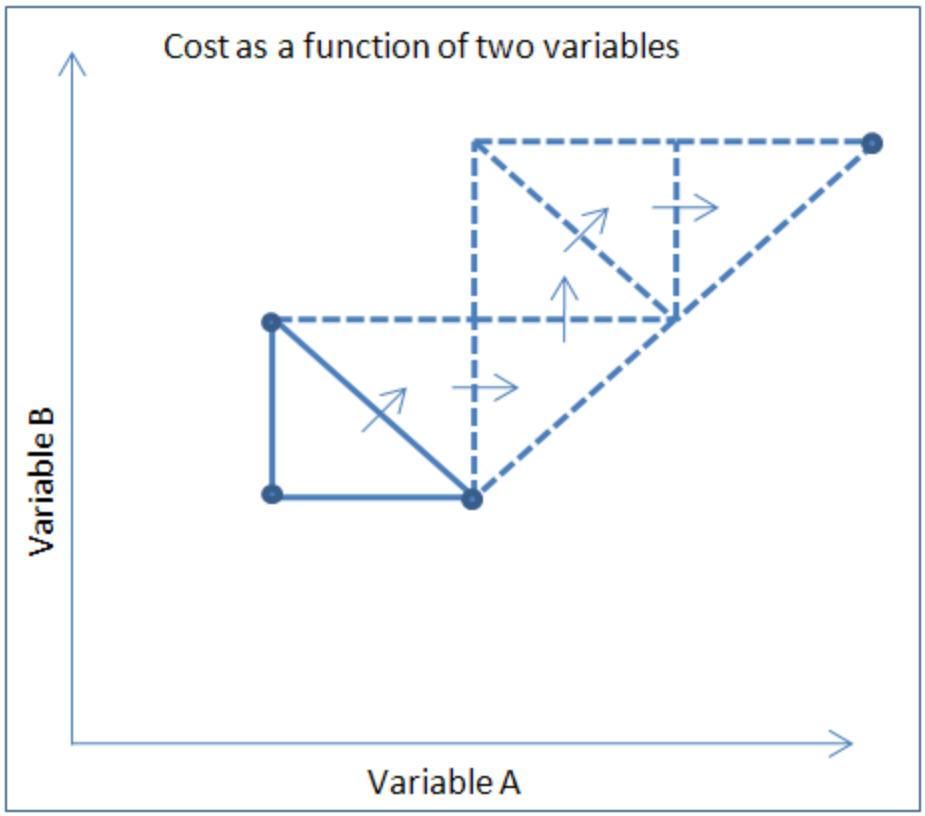
The cost value is calculated at the vertices of the simplex. The optimizer mirrors the simplex across one of its faces based on mathematical guidelines and determines if the new simplex provides better results. If it does not produce a better result, the next face is used for mirroring and the pattern continues. If no improvement occurs, the grid is refined. If improvement occurs, the step is accepted and the new simplex is generated to replace the original one. The figures below illustrate a triangular simplex mirrored several times to demonstrate the pattern search approach in two variables and the simplices superimposed on a 2D cost function to demonstrate the convergence toward a minimum in the cost function.

Cost functions can be quite nonlinear. As a result, during the function evaluations of the algorithm, the cost function can vary significantly. Also, it is important to understand the relationship between optimization function evaluation and iteration. Every iteration, depending on the number of parameters to be optimized, performs several function evaluations. These function evaluations, depending on how nonlinear the cost function is, could show drastic changes. The presence of drastic changes has no bearing on whether the optimization algorithm converged or not.

In the case of non-gradient search-based optimization algorithms, such as "pattern search," which are entirely based on function evaluations, one could see drastic changes in the function evaluations depending on how nonlinear the cost function is. This could seem misleading as if the algorithm did not converge since in theory one expects the cost function to decrease from one iteration to the next. The optimetrics, however, reports function evaluations and not necessarily the optimizer performance per iteration.

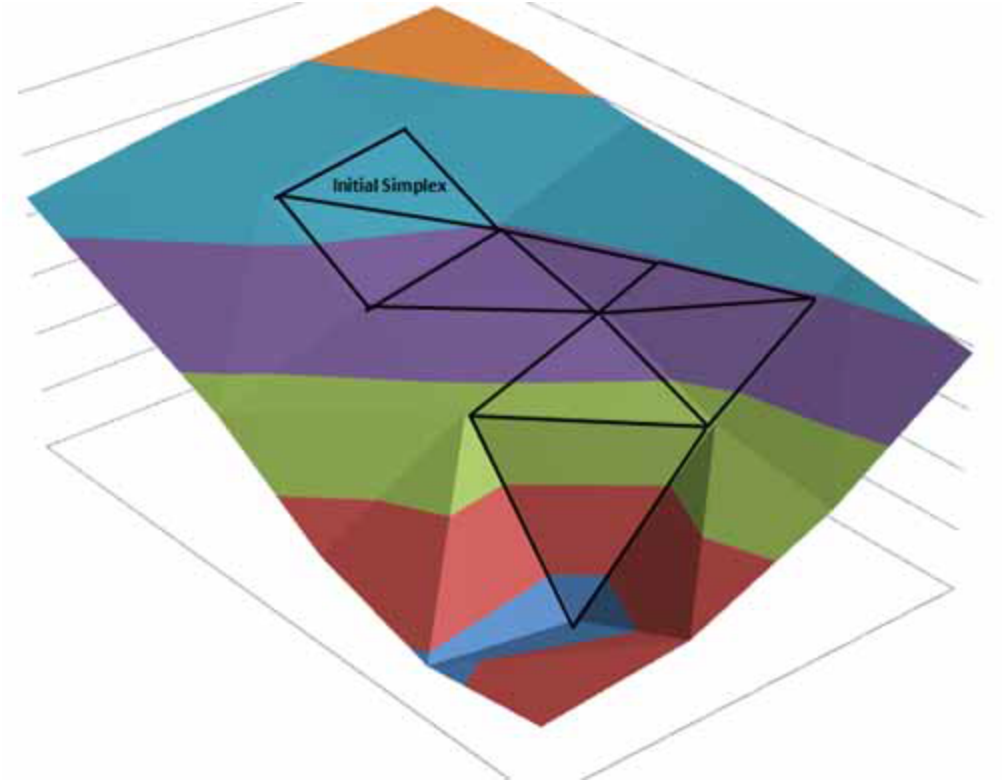
> **Note:**
>
> The MATLAB optimizer displays function evaluation when the **Show all functions evaluation** check box is selected. If the check box is not selected, it displays iteration.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Pattern Search algorithms are extensible to three variable optimization by using tetrahedral simplices, however, they are not easily represented in graphical form. Generally, Pattern Search algorithms are not used when more than three variables are used in the optimization.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When there is no improvement in the cost function regardless of the direction the simplex is mirrored, then the simplex is subdivided into smaller simplices and the process restarted.

Pattern Search algorithms have several advantages over Quasi-Newton algorithms. First, they are less sensitive to noise because the cost function is evaluated at all node points on the simplex and the numerical noise averages out over the simplex. The second advantage is that the number of initial solutions is generally smaller. However, since the pattern search does not use gradient information to locate the minimum the process converges more slowly toward the true minimum, taking more steps to successively divide the simplices as the minimum is approached.

Sequential Nonlinear Programming (Gradient)

The main advantage of SNLP (Gradient) over Quasi Newton (Gradient) is that it handles the optimization problem in more depth. This optimizer assumes that the optimization variables span a continuous space. As a result, there is no Minimum Step Size specified in this optimizer and the variables may take any value within the allowable constraints and within the numerical precision limits of the simulator. Like Quasi Newton, the SNLP optimizer assumes that the noise is not significant. It does reduce the effect of the noise, but the noise filtering is not strong.

The SNLP optimizer approximates the FEA characterization with Response Surfaces (RS). With the FEA-approximation and with light evaluation of the cost function, SNLP has a good

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

approximation of the cost function in terms of the optimization variables. This approximation allows the SNLP optimizer to estimate the location of improving points. The overall cost approximations are more accurate. This allows the SNLP optimizer a faster practical convergence speed than that of quasi Newton.

The SNLP Optimizer creates the response surface using a polynomial approximation from the FEA simulation results available from past solutions. The response surface is most accurate in the local vicinity. The response surface is used in the optimization loop to determine the gradients and calculate the next step direction and distance. The response surface acts as a surrogate for the FEA simulation, reducing the number of FEA simulations required and greatly speeding the problem. Convergence improves as more FEA solutions are created and the response surface approximation improves.

The SNLP method is similar to the Sequential Quadratic Programming (SQP) method in two ways: Both are sequential, updating the optimizer state to the current optimal values and iterating. Sequential optimization can be thought of as walking a path, step by step, toward an optimal goal. SNLP and SQP optimizers are also similar in that both use local and inexpensive surrogates. However, in the SNLP case, the surrogate can be of a higher order and is more generally constrained. The goal is to achieve a surrogate model that is accurate enough on a wider scale, so that the search procedures are well lead by the surrogate, even for relatively large steps. All functions calculated by the supporting finite element product (for example, Maxwell 3D or HFSS) is assumed to be expensive, while the rest of the cost calculation (for example, an extra user-defined expression) — which is implemented in Optimetrics — is assumed to be inexpensive. For this reason, it makes sense to remove inexpensive evaluations from the finite element problem and, instead, implement them in Optimetrics. This optimizer holds several advantages over the Quasi Newton and Pattern Search optimizers.

Most importantly, due to the separation of expensive and inexpensive evaluations in the cost calculation, the SNLP optimizer is more tightly integrated with the supporting FEA tools. This tight integration provides more insight into the optimization problem, resulting in a significantly faster optimization process. A second advantage is that the SNLP optimizer does not require cost-derivatives to be approximated, protecting against uncertainties (noise) in cost evaluations. In addition to derivative-free state of the RS-based SNLP, the RS technique also proves to have noise suppression properties.

Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete)

The Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) optimizer is equivalent to the SNLP (Gradient) optimizer with only one difference. Many problems require variables take only discrete values. One example might be to optimize on the number of turns in a coil. To be able to optimize on number of turns or quarter turns, the optimizer must handle discrete optimization variables. The SMINLP optimizer can mix continuous variables among the integers, or can have only integers, and works if all variables are continuous. The setup

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

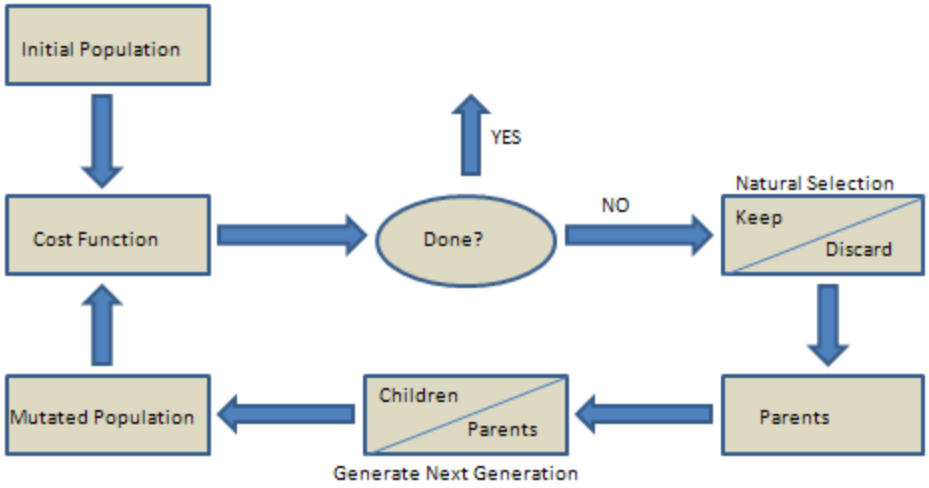
resembles the setup for SNLP, except that you must flag the integer variables.supporting integer variables. You can set up internal variables based on the integer optimization variable.

For example, consider N to be an integer optimization variable. By definition it can only assume integer values. You can establish another variable, which further depends on this one: K = 2.345 * N, or K = sin(30 * N). This way K has a discrete value, but is not necessarily integer. Or, one can use N directly as a design parameter.

Genetic Algorithm (Random Search)

Genetic Algorithm (Random Search) optimizers are part of a class of optimization techniques called stochastic optimizers. They do not use the information from the experiment or the cost function to determine where to further explore the design space. Instead, they use a type of random selection and apply it in a structured manner. The random selection of evaluations to proceed to the next generation has the advantage of allowing the optimizer to jump out of a local minima at the expense of many random solutions which do not provide improvement toward the optimization goal. As a result, the GA optimizer will run many more iterations and may be prohibitively slow.

The Genetic Algorithm search is an iterative process that goes through a number of generations (see picture below). In each generation some new individuals (Children / Number of Individuals) are created and the grown population participates in a selection (natural-selection) process that in turn reduces the size of the population to a desired level (Next Generation / Number of Individuals).



When a smaller set of individuals must be created from a bigger set, the GA selects individuals from the original set. During this process, better fit (in relation to the cost function) individuals are preferred. In the elitist selection, simply the best so many individuals are selected, but if you turn on the roulette selection, then the selection process gets relaxed. An iterative process starts

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

selecting the individuals and fills up the resulting set, but instead of selecting the best so many, we use a roulette wheel that has for each selection-candidate divisions made proportional to the fitness level (relative to the cost function) of the candidate. This means that the fitter the individual is, the larger the probability of his survival will be.

MATLAB Optimizer

The MATLAB optimizer option lets you pass a script to MATLAB to perform the optimization. When the optimization is analyzed, MATLAB is launched and a script is passed in to MATLAB to perform the optimization. During the optimization, MATLAB will call back into our application to perform the solve and compute the cost. The cost will be reported back to MATLAB, and MATLAB's optimization will determine the next step in the optimization.

The optimization script is specified as part of the optimization setup. By modifying the optimization script, users can change the optimization parameters and optimization method as well as use the full power of MATLAB in their optimization.

Running the Optimization

The MATLAB optimization is launched just like any other optimization. The Message Window will display status messages when MATLAB is being launched, and status messages will be generated for each solve that is being performed.

In most cases, MATLAB will terminate when the optimization has been completed. Some reasons why MATLAB would not terminate are:

- The user has modified the MATLAB script to not terminate MATLAB after the optimization.
- A syntax error or some other has occurred.
- The user has added some other code which runs after the optimization has completed.

System Requirements

In order to use MATLAB to perform optimizations from your application:

- A version of MATLAB must be installed on your system.
- The computing platform (e.g., 32/64 bit or Linux) of MATLAB MUST match the platform of the Ansys application you are using it with.
- You must have the MATLAB Optimization Toolkit installed.
- The MATLAB installation must include the MATLAB Optimization Toolbox. In addition, the MATLAB license must support using the Optimization Toolbox.

To see if the optimization toolbox is installed, users can type the "ver" command at the command prompt of a running MATLAB instance. For example:

```
>> ver
-----------------------------------------------------------------------
---------------------------------
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
MATLAB Version: 8.1.0.604 (R2023a)
MATLAB License Number: 162684
Operating System: Microsoft Windows 10
Java Version: Java 1.6.0_17-b04 with Sun Microsystems Inc. Java
HotSpot(TM) 64-Bit Server VM mixed mode
--------------------------------------------------------------------
--------------------------------
MATLAB Version 8.1 (R2023a)
Simulink Version 8.1 (R2023a)
Optimization Toolbox Version 6.3 (R2023a)
```

To see if the optimization toolbox is licensed, you can use the "license('test','optimization_ toolbox')" command at the MATLAB command prompt:

```
>> license('test','optimization_toolbox')
ans =
1
```

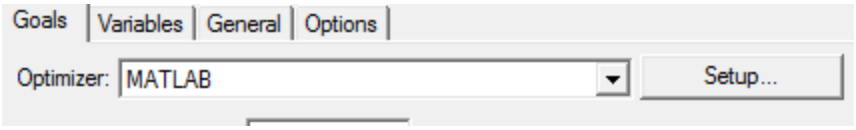
The answer will be 1 if the MATLAB Optimization Toolbox is licensed or 0 otherwise.

Specifying the MATLAB Location

The Tools> General Options: Miscellaneous group contains a setting for the MATLAB location. This setting must to point to the version of MATLAB to be used for performing the optimization. The platform (32/64 bit or Linux) of the specified version of MATLAB must match the platform of this application.

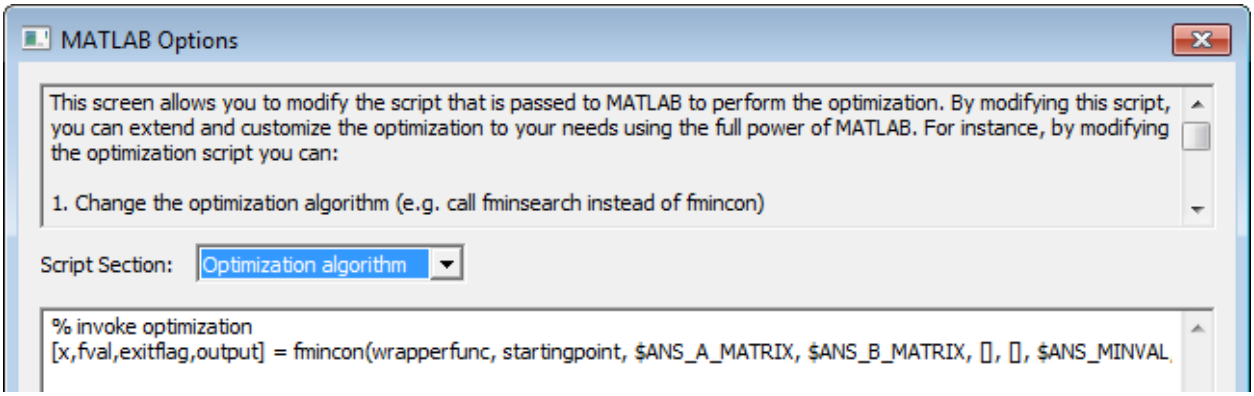
MATLAB Optimization Setup

MATLAB optimization starts by creating an optimization and selecting MATLAB from the optimizer drop-down menu. If you select MATLAB as the optimizer, the *Setup Optimization* dialog box displays a **Setup...** button.



Select **Setup...** to open the *MATLAB Options* dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The upper text panel is informative. The Script section drop-down menu lets you select a lower panel display for Optimization algorithm, Options, or the Full script template.

This screen allows you to modify the script that is passed to MATLAB to perform the optimization. The complete script contains all the instructions necessary for MATLAB to connect to our application and perform the optimization, and a lot of that code is unimportant to users. We have addressed this issue by displaying a dropdown to let you view only the portion of code they are interested in without having to view the full script. The choices are:

- **Optimization algorithm**: displays only the line of code invoking the actual optimization function. By changing this line, users can use a different MATLAB function for optimization. By default we use fmincon() which is a derivative based constrained optimization. By modifying this line, users could replace the fmincon() call with fminsearch () to use an unconstrained pattern searching optimizer or another optimization function. See the MATLAB documentation for details about available optimization functions.
- **Options**: Each optimization function contains a multitude of options and parameters which are set in the MATLAB script prior to actually calling the optimization function. By modifying these options, the optimization can be customized as desired. For instance, options can be set for fmincon() to specify the algorithm that it uses internally. See the MATLAB documentation for details about options available for each optimization function.
- **Full script template**: This choice displays the full optimization script that is passed to MATLAB.

The initial Script Section display for the Optimization algorithm shows the following:

```
% invoke optimization
[x,fval,exitflag,output] = fmincon(wrapperfunc, startingpoint, [],
[], [], [], $ANS_MINVAL, $ANS_MAXVAL, nlcon, options)
```

The initial Script Section Options display shows the following:

```
% customers can add their own options below
options = optimset(options, 'display', 'iter')
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
options = optimset(options, 'Algorithm', 'interior-point')
% options = optimset(options, 'PlotFcns', @optimplotfval)
```

You can modify the script to extend and customize the optimization to your needs. You must ensure that the script follows MATLAB syntax. For instance, by modifying the optimization script you can:

- Change the optimization algorithm (e.g., call fminsearch instead of fmincon).
- Change the parameters/options of the optimization algorithm (see the MATLAB documentation for details).
- Specify a plot function to provide graphical output during optimization.
- Specify a user defined output function to be called at completion or per iteration.

Symbols

When modifying the MATLAB code, users can use symbols to represent values from the optimization setup. The symbols and their definitions are listed below.

$ANS_VARIABLE_LIST:	list of variables we are optimizing
$ANS_STARTING_POINT:	vector of starting values of variables used in the optimization
$ANS_MAXITERATIONS:	maximum number of iterations specified in optimization setup
$ANS_MINVAL:	vector of minimum values from optimization setup
$ANS_MAXVAL:	vector of maximum values from optimization setup
$ANS_MINSTEP:	vector of minimum step sizes from optimization setup
$ANS_MAXSTEP:	vector of maximum step sizes from optimization setup
$ANS_A_MATRIX	matrix of linear constraint coefficients (left-hand side) generated from optimization setup
$ANS_B_MATRIX	matrix of linear constraint bounds (right-hand side) generated from optimization setup

Note:

The linear constraints as generated for MATLAB have the form [A][x] <= [B], where [A] is the coefficient matrix, [x] is the variable list matrix (column vector), and [B] is the bounds matrix (column vector).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> While modifying the script, please ensure that the script follows MATLAB syntax.

MATLAB Optimization Script Template

The script template shown in the Script Section is as follows:

```
% make sure platform matches
if strcmp(computer, '$ANS_EXPECTED_PLATFORM') ~= 1
   h = msgbox('32/64 platform does not match calling application,
  exiting')
   uiwait(h)
   exit
end
% add installation dir to search path so .mex file can be found
originalpath = addpath('$ANS_EXEDIR')
% connect back to opticomengine
callbackinterface = optimex('connect', '$ANS_CONNECTIONSTRING')
% set up optimization
% variables are: $ANS_VARIABLELIST
startingpoint = $ANS_STARTINGPOINT
options = optimset('MaxIter', $ANS_MAXITERATIONS)
iterationCallbackWrapper = @(x, optimValues, state) optimex
('notifyiterationcomplete', callbackinterface, x, optimValues.fval,
state)
options = optimset(options, 'OutputFcn', iterationCallbackWrapper)
% halt execution so debugger can be attached
% h = msgbox('attach debugger if desired')
% uiwait(h)
% attributes that user can pass to optimization algorithm
% variables are: $ANS_VARIABLELIST
% this is the objective function which returns cost
wrapperfunc = @(x)optimex('eval', callbackinterface, x)
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
% this is our non linear constraint function, returns no constraints
returnempty = @(x)[];
nlcon = @(x) deal(returnempty(x), returnempty(x));
% DO NOT EDIT THIS LINE - START OPTIONS SECTION
% customers can add their own options below
options = optimset(options, 'display', 'iter')
options = optimset(options, 'Algorithm', 'interior-point')
% options = optimset(options, 'PlotFcns', @optimplotfval)
% DO NOT EDIT THIS LINE - END OPTIONS SECTION
% DO NOT EDIT THIS LINE - START OPTIMIZATION ALGO SECTION
% invoke optimization
[x,fval,exitflag,output] = fmincon(wrapperfunc, startingpoint, $ANS_
A_MATRIX, $ANS_B_MATRIX, [], [], $ANS_MINVAL, $ANS_MAXVAL, nlcon,
options)
% DO NOT EDIT THIS LINE - END OPTIMIZATION ALGO SECTION
% write exit message to Ansoft message window (warning=0,error=1,info=2)
optimex('postansoftmessage', callbackinterface, 2, output.message)
% notify opticomengine that optimization is finished
optimex('optimizationfinished', callbackinterface, exitflag)
% restore original path
path = originalpath
% note: comment below line if you want MATLAB to remain
% running after optimization
exit
```

Multi-Objective Genetic Algorithm (MOGA)

The Multi-Objective Genetic Algorithm (MOGA) used in GDO is a hybrid variant of the popular NSGA-II (Non-dominated Sorted Genetic Algorithm-II) based on controlled elitism concepts. It supports all types of input parameters. The Pareto ranking scheme is done by a fast, non-dominated sorting method that is an order of magnitude faster than traditional Pareto ranking methods. The constraint handling uses the same non-dominance principle as the objectives.

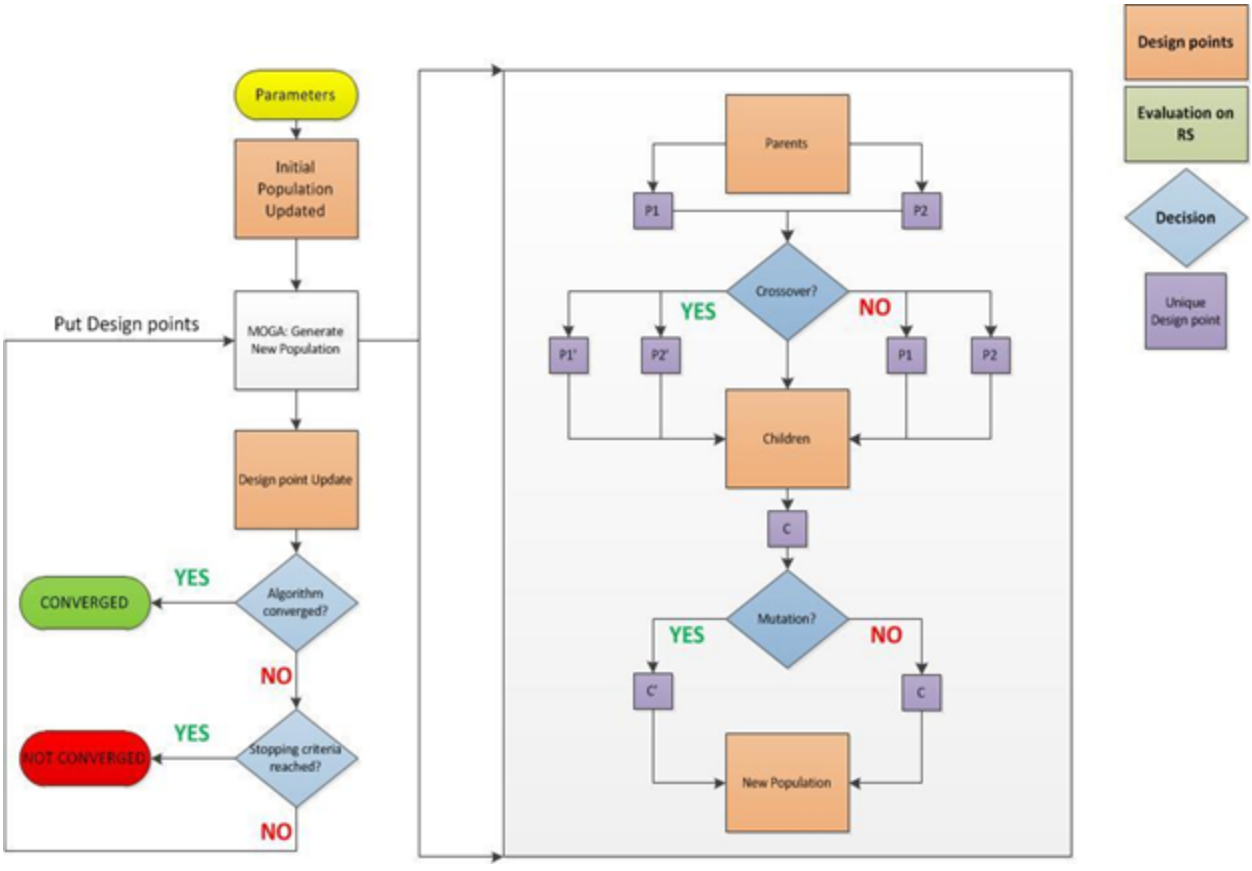
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Therefore, penalty functions and Lagrange multipliers are not needed. This also ensures that the feasible solutions are always ranked higher than the infeasible solutions.

The first Pareto front solutions are archived in a separate sample set internally and are distinct from the evolving sample set. This ensures minimal disruption of Pareto front patterns already available from earlier iterations. You can control the selection pressure (and, consequently, the elitism of the process) to avoid premature convergence by altering the Maximum Allowable Pareto Percentage property. For more information about this and other MOGA properties, see Setup Multi-Objective Genetic Algorithm.

MOGA Workflow

The MOGO workflow follows:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MOGA Steps

1. **First Population of MOGA**

   The initial population is used to run MOGA.

2. **MOGA Generates a New Population**

   MOGA is run and generates a new population via cross-over and mutation. After the first iteration, each population is run when it reaches the number of samples defined by the Number of Samples Per Iteration property. For details, see **MOGA Steps to Generate New Population** below.

3. **Design Point Update**

   The design points in the new population are updated.

4. **Convergence Validation**

   The optimization is validated for convergence.

   - **Yes: Optimization Converged**

     MOGA converges when the Maximum Allowable Pareto Percentage or the Convergence Stability Percentage has been reached.

   - **No: Optimization Not Converged**

     If the optimization is not converged, the process continues to the next step.

5. **Stopping Criteria Validation**

   If the optimization has not converged, it is validated for fulfillment of stopping criteria.

   - **Yes: Stopping Criteria Met**

     When the Maximum Number of Iterations criterion is met, the process is stopped without having reached convergence.

   - **No: Stopping Criteria Not Met**

     If the stopping criteria have not been met, MOGA is run again to generate a new population (return to Step 2).

6. **Conclusion**

   Steps 2 through 5 are repeated in sequence until the optimization has converged or the stopping criteria have been met. When either of these things occurs, the optimization concludes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MOGA Steps to Generate a New Population

The process MOGA uses to generate a new population has two main steps: **Cross-over** and **Mutation**.

1. **Cross-over**

   Cross-over combines (mates) two chromosomes (parents) to produce a new chromosome (offspring). The idea behind cross-over is that the new chromosome can be better than both of the parents if it takes the best characteristics from each of the parents. Cross-over occurs during evolution according to a user-definable cross-over probability.

   - **Cross-over for Continuous Parameters**

     A cross-over operator that linearly combines two parent chromosome vectors to produce two new offspring according to the following equations:

     $$\text{Offspring1}=\mathbf{a}*\text{Parent1}+\left(1-\mathbf{a}\right)*\text{Parent2}$$
     $$\text{Offspring2}=\left(1-\mathbf{a}\right)*\text{Parent1}+\mathbf{a}*\text{Parent2}$$

     Consider the following two parents (each consisting of four floating genes), which have been selected for cross-over:

     $$\text{Parent 1: }(0.3)(1.4)(0.2)(7.4)$$
     $$\text{Parent 2: }(0.5)(4.5)(0.1)(5.6)$$

     If a = 0.7, the following two offspring would be produced:

     $$\text{Offspring1: }(0.36)(2.33)(0.17)(6.86)$$
     $$\text{Offspring2: }(0.402)(2.981)(0.149)(6.842)$$

   - **Cross-over for Discrete Parameters and Continuous Parameters with Manufacturable Values**

   Each discrete parameter or continuous parameter with manufacturable values is represented by a binary chain corresponding to the number of levels. For example, a parameter with two values (levels) is encoded to one bit, a parameter with seven values is encoded to three bits, and an -bits chain represents a parameter with values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The concatenation of these chains forms the chromosome, which crosses over with another chromosome.

Three different kinds of cross-over are available:

- One-Point

  A one-point cross-over operator that randomly selects a cross-over point within a chromosome then interchanges the two parent chromosomes at this point to produce two new offspring.

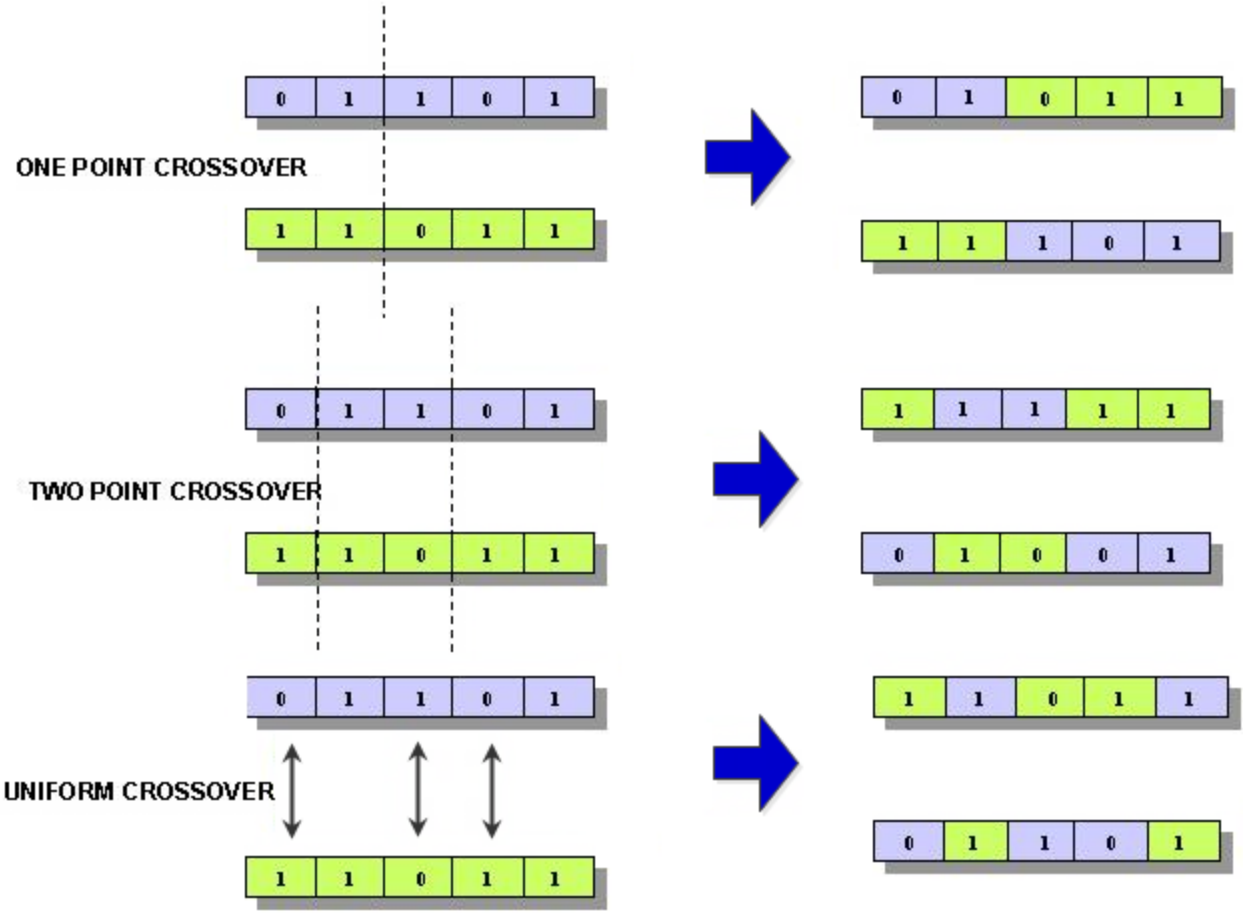
- Two-Point

  A two-point cross-over operator randomly selects two cross-over points within a chromosome then interchanges the two parent chromosomes between these points to produce two new offspring.

- Uniform

  A uniform cross-over operator decides (with some probability, which is known as the "mixing ratio") which parent contributes each of the gene values in the offspring chromosomes. This allows the parent chromosomes to be mixed at the gene level rather than the segment level (as with one and two-point cross-over). For some problems, this additional flexibility outweighs the disadvantage of destroying building blocks.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. **Mutation**

   Mutation alters one or more gene values in a chromosome from its initial state. This can result in entirely new gene values being added to the gene pool. With these new gene values, the genetic algorithm might be able to arrive at a better solution than was previously possible. Mutation is an important part of the genetic search, as it helps to prevent the population from stagnating at any local optima. Mutation occurs during evolution according to a user-defined mutation probability.

   - **Mutation for Continuous Parameters**

     For continuous parameters, a polynomial mutation operator is applied to implement mutation.

$$C = P + (\text{UpperBound} - \text{LowerBound})\delta$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where C is the child, P is the parent, and δ is a small variation calculated from a polynomial distribution.

- **Mutation for Discrete Parameters and Continuous Parameters with Manufacturable Values**

  For discrete parameters or continuous parameters with manufacturable values, a mutation operator simply inverts the value of the chosen gene (0 goes to 1 and 1 goes to 0) with a probability of 0.5. This mutation operator can only be used for binary genes. The concatenation of these chains forms the chromosome, which crosses over with another chromosome.

Convergence Rate % and Initial Finite Difference Delta % in NLPQ and MISQP

Typically, the use of Nonlinear Programming by Quadratic Lagrangian (NLPQL) or Mixed-Integer Sequential Quadratic Programming (MISQP) optimizers is suggested for continuous problems when there is only one objective function. The problem might or might not be constrained and must be analytic. This means that the problem must be defined only by continuous input parameters and that the objective functions and constraints should not exhibit sudden "jumps" in their domain.

The main difference between these algorithms and Multi-Objective Genetic Algorithm (MOGA) is that MOGA is designed to work with multiple objectives and does not require full continuity of the output parameters. However, for continuous single objective problems, the use of NLPQL or MISQP gives greater accuracy of the solution as gradient information and line search methods are used in the optimization iterations. MOGA is a global optimizer designed to avoid local optima traps, while NLPQL and MISQP are local optimizers designed for accuracy.

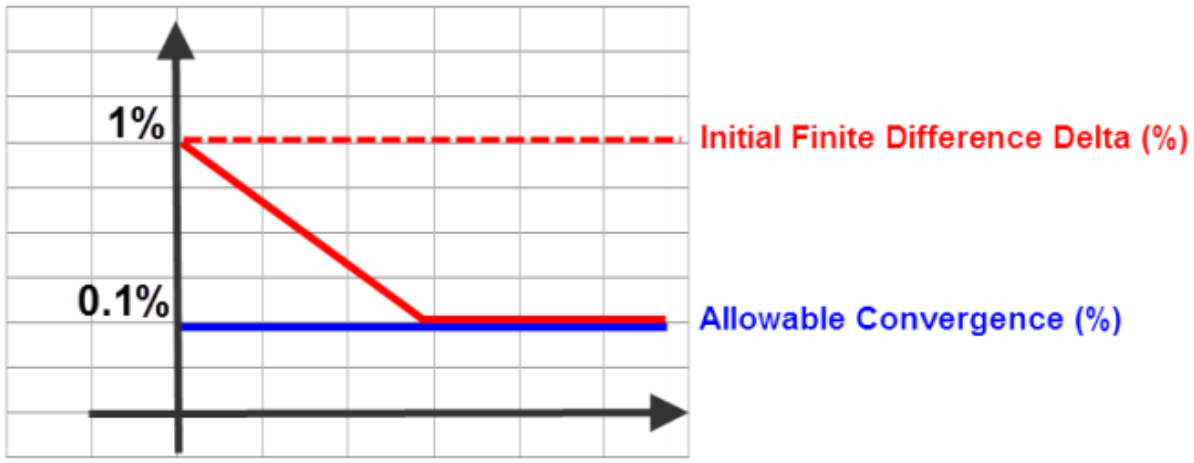
For NLPQL and MISQP, the default convergence rate, which is specified by the Allowable Convergence (%) property, is set to 0.1% for a Direct Optimization system. The maximum value for this property is 100%. This is computed based on the (normalized) Karush-Kuhn-Tucker (KKT) condition. This implies that the fastest convergence rate of the gradients or the functions (objective function and constraint) determine the termination of the algorithm.

The default convergence rate is used in conjunction with the initial finite difference delta percentage value, which is specified by the Initial Finite Difference Delta (%) advanced property. This property defaults to 1% for a Direct Optimization system. You use this property to specify a percentage of the variation between design points to ensure that the Delta use in the calculation of finite differences is large enough to be seen over simulation noise. The specified percentage is defined as a relative gradient perturbation between design points.

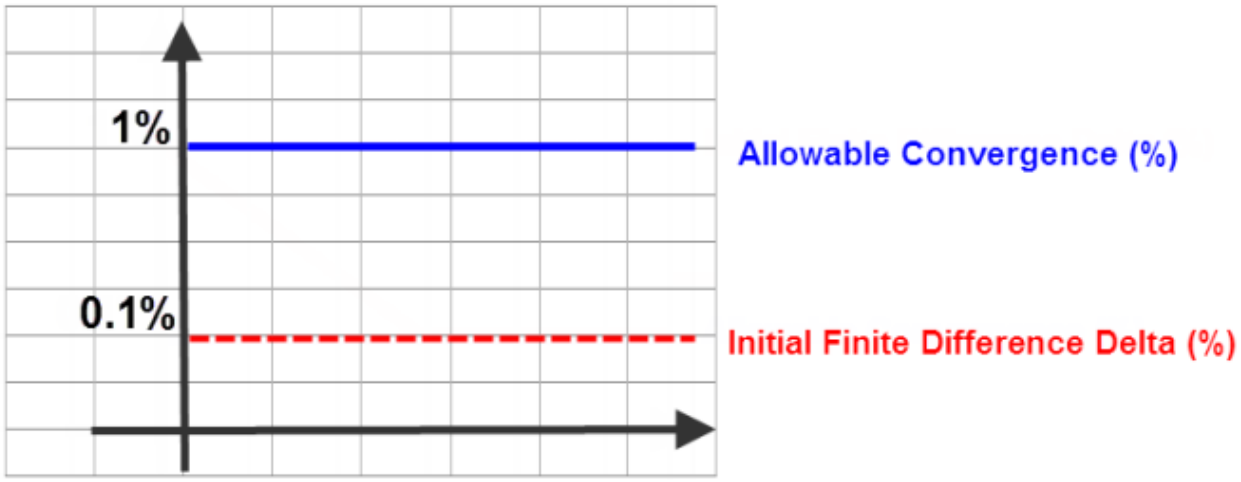
The advantage of this approach is that for large problems, it is possible to get a near-optimal feasible solution quickly without being trapped into a series of iterations involving small solution steps near the optima. To work most effectively with NLPQL and MISQP, keep the following guidelines in mind:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the Initial Finite Difference Delta (%) is greater than the Allowable Convergence (%), the relative gradient perturbation gets iteratively smaller, until it matches the allowable convergence rate. At this point, the relative gradient value stays the same through the rest of the analysis.



- If the Initial Finite Difference Delta (%) is less than or equal to the Allowable Convergence (%), the current relative gradient step remains constant through the rest of the analysis.



- Both the Initial Finite Difference Delta (%) and Allowable Convergence (%) should be higher than the magnitude of the noise in your simulation.

When setting the values for these properties, you have the usual trade-offs between speed and accuracy. Smaller values result in more convergence iterations and a more accurate (but slower) solution, while larger values result in fewer convergence iterations and a less accurate (but faster) solution. At the same time, however, you must be aware of the amount of noise in

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

your model. For the input variable variations to be visible in the output variables, both values must be greater than the magnitude of the simulation's noise.

In general, default values for Initial Finite Difference Delta (%) and Allowable Convergence (%) cover the majority of optimization problems. For example, if you know that the noise magnitude in your direct optimization problem is 0.0001, then the default values (Allowable Convergence (%) = 0.001 and Initial Finite Difference Delta (%) = 0.01) are good.

When the defaults are not a good match for your problem, of course, you can adjust the values to better suit your model and your simulation needs. If you require a more numerically accurate solution, you can set the convergence rate to as low as 1.0E-10% and then set the Initial Finite Difference Delta (%) accordingly.

Mixed-Integer Sequential Quadratic Programming

Mixed-Integer Sequential Quadratic Programming (MISQP) is a mathematical optimization algorithm as developed by Oliver Exler, Thomas Lehmann and Klaus Schittkowski (NLPQL). This method solves Mixed-Integer Non-Linear Programming (MINLP) of the form:

Minimize:

$$f(x,y)$$

Subject to:

$$g_j(x,y)=0,\ j=1,\ldots,m_e,$$
$$g_j(x,y)\geq 0,\ j=m_e+1,\ldots,m$$

Where:

$$x\in\mathbb{R}^{n_c}, y\in\mathbb{N}^{n_i}$$
$$x_l\leq x\leq x_u$$
$$y_l\leq y\leq y_u$$

The symbols and denote the vectors of the continuous and integer variables, respectively. It is assumed that problem functions and are continuously differentiable subject to all . It is not

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

assumed that integer variables can be relaxed. In other words, problem functions are evaluated only at integer points and never at any fractional values in between.

MISQP solves MINLP by a modified sequential quadratic programming (SQP) method. After linearizing constraints and constructing a quadratic approximation of the Lagrangian function, mixed-integer quadratic programs are successively generated and solved by an efficient branch-and-cut method. The algorithm is stabilized by a trust region method as originally proposed by Yuan for continuous programs. Second order corrections are retained. The Hessian of the Lagrangian function is approximated by BFGS updates subject to the continuous and integer variables. MISQP is able to solve also non-convex nonlinear mixed-integer programs.

Adaptive Multiple Objective Optimization

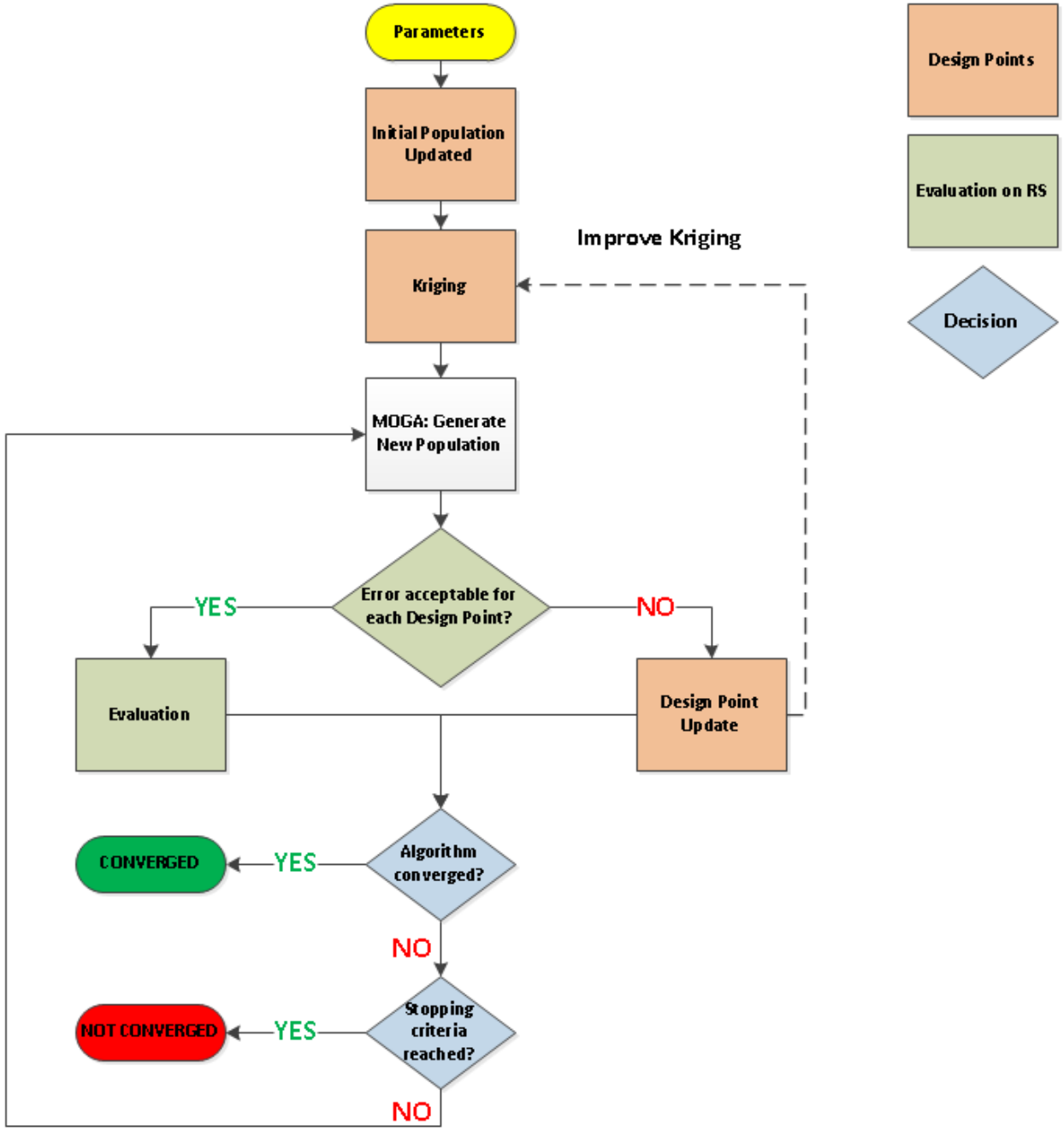
Adaptive Multiple-Objective (AMO) is a mathematical optimization that combines a Kriging response surface and MOGA. It allows you to either generate a new sample set or use an existing set, providing a more refined approach than the Screening method. Except when necessary, the optimizer does not evaluate all design points. The general optimization approach is the same as MOGA, but a Kriging response surface is used. Part of the population is "simulated" by evaluations of the Kriging response surface. The Kriging error predictor reduces the number of evaluations used in finding the first Pareto front solutions.

AMO supports multiple objectives and multiple constraints. It is limited to continuous parameters, including those with manufacturable values. It is available only for a Direct Optimization system. When discrete parameters are used, MOGA is the more efficient optimization method. For more information, see Multi-Objective Genetic Algorithm (MOGA).

AMO Workflow

The workflow of AMO follows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



AMO Steps

1. **First Population of MOGA**

   The initial population is used to run MOGA.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. **Kirging Generation**

   A Kriging response surface is created for each output, based on the first population and then improved during simulation with the addition of new design points.

3. **MOGA**

   MOGA is run, using the Kriging response surface as an evaluator. After the first iteration, each population is run when it reaches the number of samples defined by the Number of Samples Per Iteration property.

4. **Evaluate the Population**
5. **Error Check**

   The Kriging error predictor is checked for each point.

   - **Yes: Error Acceptable**

     Each point is validated for error. If the error for a given point is acceptable, the approximated point is included in the next population to be run through MOGA (return to Step 3).

   - **No: Error Not Acceptable**

   If the error is not acceptable, the points are promoted as design points. The new design points are used to improve the Kriging response surface (return to Step 2) and are included in the next population to be run through MOGA (return to Step 3)

6. **Convergence Validation**

   The optimization is validated for convergence.

   - **Yes: Optimization Converged**

     MOGA converges when the maximum allowable Pareto percentage has been reached. When this happens, the process is stopped.

   - **No: Optimization Not Converged**

   If the optimization is not converged, the process continues to the next step.

7. **Stopping Criteria Validation**

   If the optimization has not converged, it is validated for fulfillment of the stopping criteria.

   - **Yes: Stopping Criteria Met**

     When the maximum number of iterations has been reached, the process is stopped without having reached convergence..

   - **No: Stopping Criteria Not Met**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the stopping criteria have not been met, the MOGA algorithm is run again (return to Step 3).

8. **Conclusion**

   Steps 2 through 7 are repeated in sequence until the optimization has converged or the stopping criteria have been met. When either of these things occurs, the optimization concludes.

Adaptive Single Objective Optimization

Adaptive Single-Objective (ASO) is a mathematical optimization method that combines an OSF (Optimal Space-Filling) DOE, a Kriging response surface, and MISQP. It is a gradient-based algorithm based on a response surface, which provides a refined, global, optimized result.

ASO supports a single objective and multiple constraints. It is available for continuous parameters, including those with manufacturable values. It does not support the use of parameter relationships in the optimization domain and is available only for a Direct Optimization system.

Like MISQP, ASO solves constrained nonlinear programming problems of the form:

Minimize:

$$f = f(\{x\})$$

Subject to:

$$g_k(\{x\}) \leq 0, \forall k = 1,2,\ldots,K$$

$$h_l(\{x\}) = 0, \forall I = 1,2\ldots,L$$

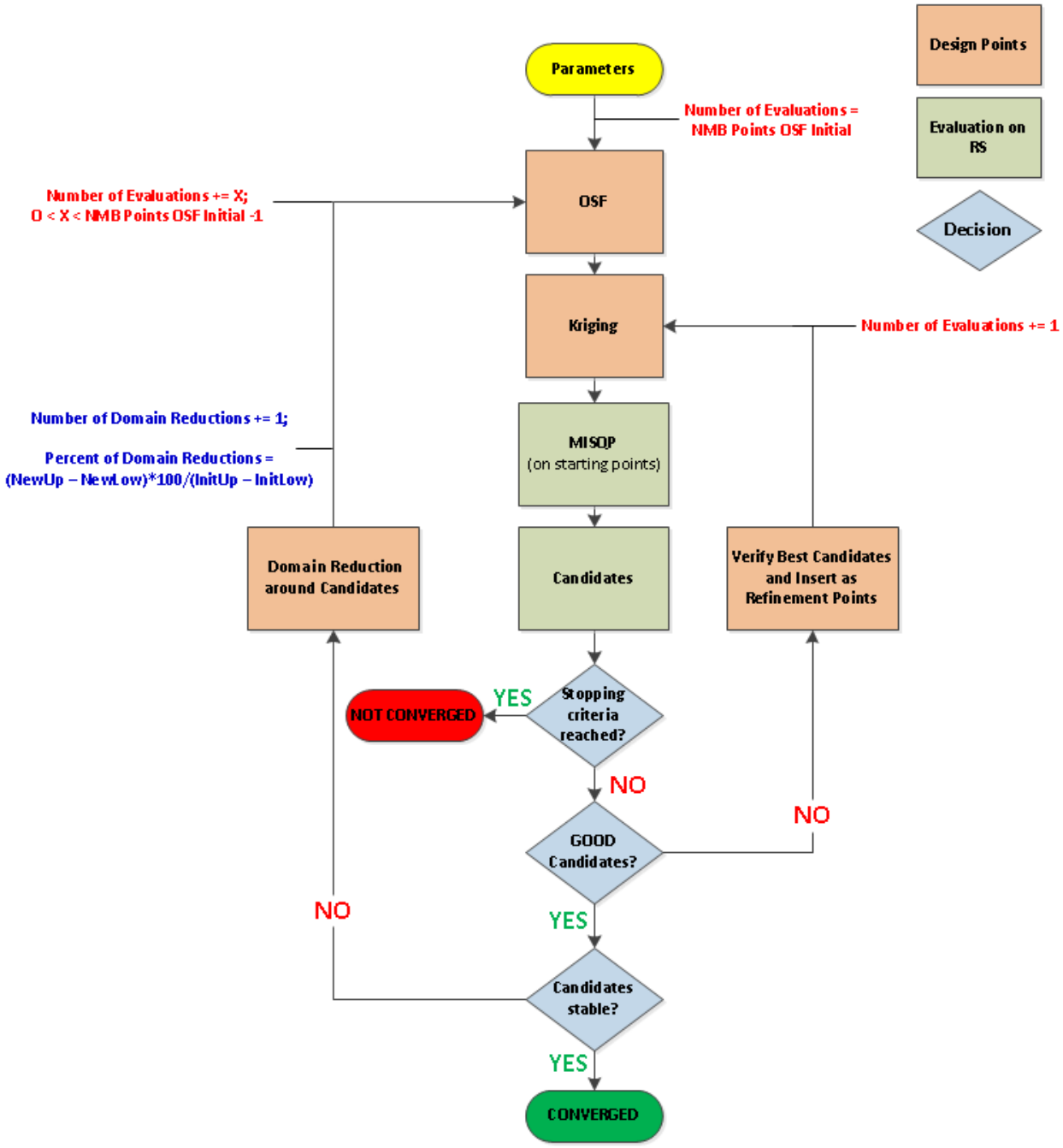
Where:

$$\{x_L\} \leq \{x\} \leq \{x_U\}$$

ASO Workflow

The workflow of ASO follows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

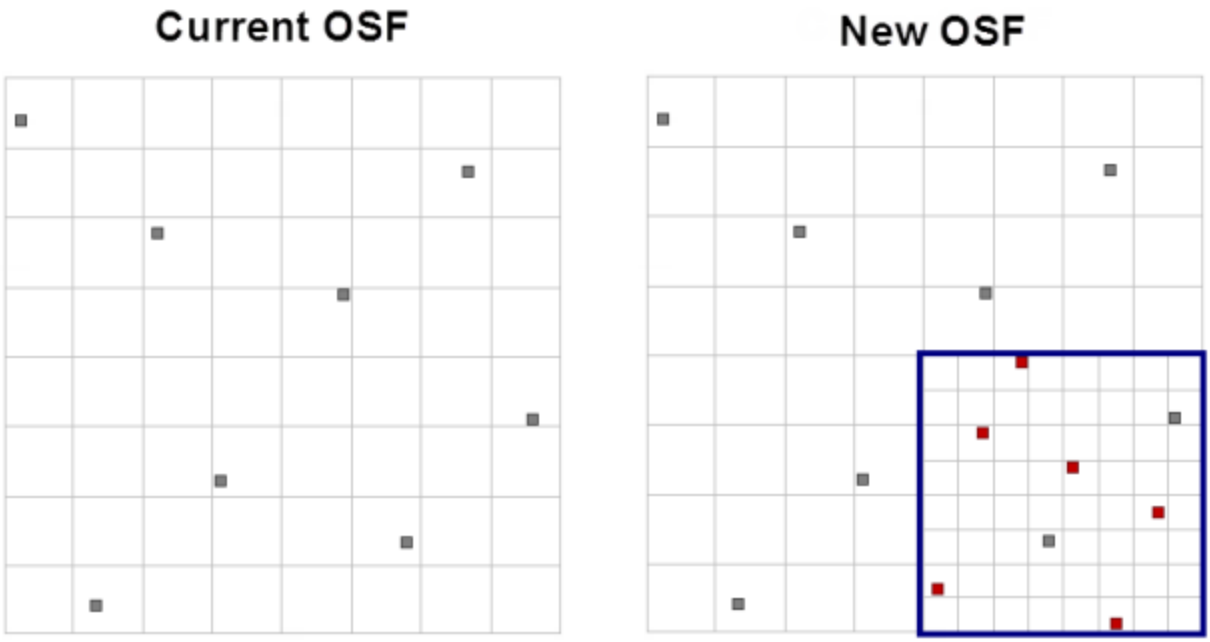
AMO Steps

1. **OSF Sampling**

   OSF (Optimal Space-Filling Design) is used for the Kriging construction. In the original OSF, the number of samples equals the number of divisions per axis and there is one sample in each division.

   When a new OSF is generated after a domain reduction, the reduced OSF has the same number of divisions as the original and keeps the existing design points within the new bounds. New design points are added until there is a point in each division of the reduced domain.

   In the following example, the original domain has eight divisions per axis and contains eight design points. The reduced domain also has eight divisions per axis and includes two of the original design points. To have a design point in each division, six new design points need to be added.



2. **Kirging Generation**

   A response surface is created for each output, based on the current OSF and consequently on the current domain bounds.

   A Kriging response surface is created for each output, based on the first population and then improved during simulation with the addition of new design points.

   For more information on Kriging algorithms, see Kriging Algorithms.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. **MISQP**

   MISQP is run on the current Kriging response surface to find potential candidates. A few MISQP processes are run at the same time, beginning with different starting points, and consequently, giving different candidates.

4. **Candidate Point Validation**

   All the obtained candidates are either validated or not, based on the Kriging error predictor. The candidate point is checked to see if further refinement of the Kriging surface changes the selection of this point. A candidate is considered as acceptable if there aren't any points, according to this error prediction, that call it into question. If the quality of the candidate is not called into question, the domain bounds are reduced. Otherwise, the candidate is calculated as a verification point.

   - **Refinement Point Creation** (If the selection is ***not*** to be changed)

     When a new verification point is calculated, it is inserted in the current Kriging response surface as a refinement point and the MISQP process is restarted.

   - **Domain Reduction**(If the selection is to be changed)

     When candidates are validated, new domain bounds must be calculated. If all of the candidates are in the same zone, the bounds are reduced, centered on the candidates. Otherwise, the bounds are reduced as an inclusive box of all candidates. At each domain reduction, a new OSF is generated (conserving design points between the new bounds) and a new Kriging response surface is generated based on this new OSF.

5. **Convergence and Stop Criteria**

   The optimization is considered to be converged when the candidates found are stable. This occurs when all of the MISQP processes run on the response surface converge to the same verified candidate point. However, there are four stop criteria that can stop the algorithm: Maximum Number of Evaluations, Maximum Number of Domain Reductions, Percentage of Domain Reductions, and Convergence Tolerance.

Optimization Variables and the Design Space

Once the optimization variables are specified, the optimizer handles each of them as an *n*-dimensional vector *x*. Any point in the design space corresponds to a particular *x*-vector and to a design instance. Each design instance may be evaluated via FEA and assigned a cost value;

therefore, the cost function is defined over the design space (cost(*x*): $R^n \rightarrow R$, where *n* is the number of optimization variables.

In practice, a solution of the minimization problem is sought only on a bounded subset of the Rn space. This subset is called the feasible domain and is defined via linear constraints.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Up an Optimization Analysis

Optimization allows you to vary predefined variables in the nominal design to search for the solution that best satisfies a set of user defined goals or cost functions. Optimetrics modifies the variable values until the minimum is reached with acceptable accuracy.

> **Note:**
>
> - You can define more than one optimization analysis setup per design.
> - You can create an Optimization setup before defining variables but all variables must be defined before you start the Optimization analysis.
> - Once you have created an optimization analysis setup, you can copy and paste it, and then make changes to the copy, rather than redoing the whole process for minor changes.

To provide a broad range of capability, Optimetrics incorporates the following types of numerical optimizers:

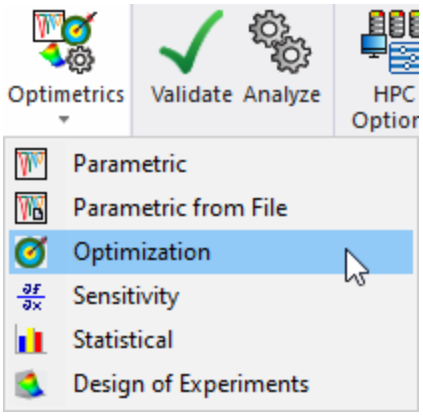
- Screening (Search based)
- Multi-Objective Genetic Algorithm
- Nonlinear Programming by Quadratic Lagrangian (Gradient)
- Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete)
- Adaptive Multiple Objective (Gradient)
- Adaptive Single Objective
- MATLAB

Legacy Optimizers include

- Sequential Nonlinear Programming (Gradient)
- Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete)
- Quasi Newton (Gradient)
- Pattern Search (Search-based)
- Genetic Algorithm (Random search)

Click on the links above to view the setup procedure for each optimizer. Options for the analysis are listed in the table. Besides setting up an Optimization analysis from the Optimization menu, you can also the **Simulation** tab of the ribbon, and select from the menu under the Optimetrics icon:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following *optional* optimization solution setup options can also be used:

- Modify the starting variable value.
- Edit the Calc. Range text field or use the Edit Calculation Range dialog box.
- Modify the minimum and maximum values of variables that will be optimized.
- Exclude variables from optimization.
- Modify the values of fixed variables that are not being optimized.
- Set the minimum and maximum step size between solved design variations (for the Quasi Newton (Gradient) and Pattern Search (Search based optimizers), **Variables** tab).
- Set the minimum and maximum focus size (for the SNLP Gradient and SMINLP Gradient and Discrete optimizers, **Variables** tab).
- Set Linear constraints.
- Request that Optimetrics solve a parametric sweep before an optimization analysis.
- Request that Optimetrics solve a parametric sweep during an optimization analysis.
- Automatically update optimized variables to the optimal values during an optimization or after an optimization analysis is completed.
- Change the norm used for the cost function calculation (Advanced Option)
- Open the HPC and Analysis Options window.

> **Note:**
>
> Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

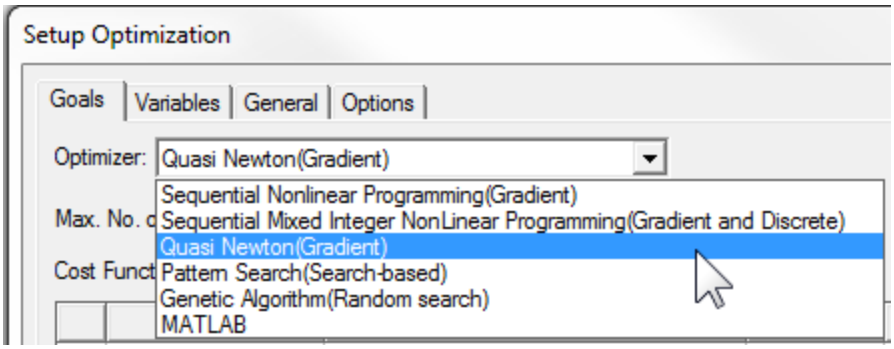
Optimization Setup for the Quasi Newton (Gradient) Optimizer

Following is the procedure for setting up an optimization analysis using the Quasi Newton (Gradient) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **HFSS**> **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.

3. Under the **Goals** tab, select the optimizer by selecting **Quasi Newton (Gradient)** from the **Optimizer** drop-down menu. Selecting Quasi Newton (Gradient) enables the **Acceptable Cost** and **Noise** fields.



4. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the **Max. No. of Iterations** text box.
5. Under **Cost Function**, add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.
6. Type the value of the cost function at which the optimization process should stop in the **Acceptable Cost** text box. Note that for Quasi Newton, if you specify 0 as the acceptable cost, the simulation stops after the first analysis.
7. Type the cost function noise in the **Noise** text box.
8. If you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.

   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.

9. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
10. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Step Size** for the analysis.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.
11. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

12. Under the **Options** tab, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

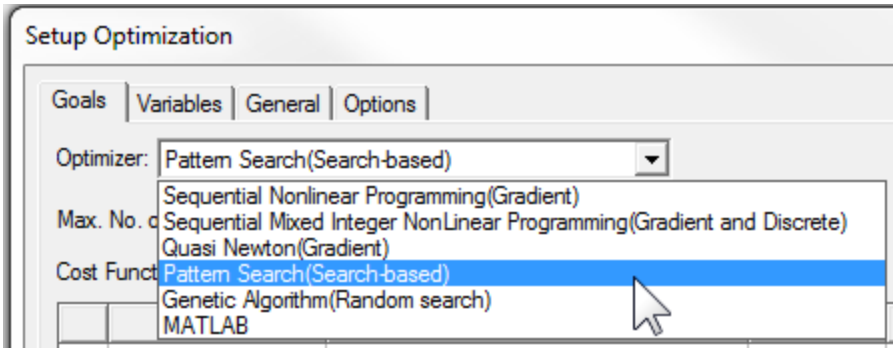
Optimization Setup for the Pattern Search (Search-based) Optimizer

Following is the procedure for setting up an optimization analysis using the Pattern Search (Search-based) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **HFSS**> **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.

3. Under the **Goals** tab, select the optimizer by selecting **Pattern Search (Search-based)** from the **Optimizer** drop-down menu.



   Selecting Pattern Search enables the **Acceptable Cost** and **Noise** fields.

4. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the **Max. No. of Iterations** text box.
5. Under **Cost Function**, add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.
6. Type the value of the cost function at which the optimization process should stop in the **Acceptable Cost** text box.
7. Type the cost function noise in the **Noise** text box.
8. If you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.

   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.

9. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
10. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Step Size** for the analysis.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.

    Select the **View all columns** check box to see all columns, including hidden columns.
11. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.
12. Under the **Options** tab, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Up Merit-based Sequential Quadratic Programming (Gradient) Optimizer

Following is the procedure for setting up an optimization analysis using the Merit-based Sequential Quadratic Programming(Gradient) Optimizer or MBSQ Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **HFSS**> **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Merit-based Sequential Quadratic Programming(Gradient)** from the **Optimizer** drop-down menu.
4. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the **Max. No. of Iterations** text box.
5. Under **Cost Function**, add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.
6. If you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
7. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
8. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Focus** for the analysis.
   - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
   - Optionally, modify the values of fixed variables that are not being optimized.
   - Optionally, set Linear constraints.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select the **View all columns** check box to see all columns, including hidden columns.

9. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

   Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

10. Use the **Options tab** if you want to enable use of a fast calculation-update algorithm to speed up Optimetrics and report updates during Optimetrics analyses, and to save the solution data for solved design variations in the analysis.

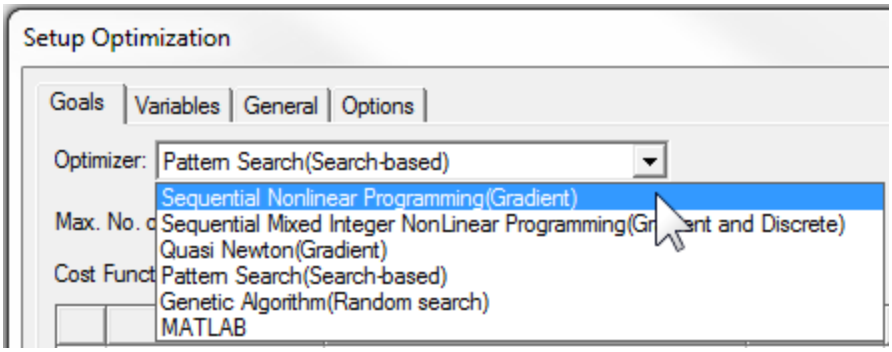
Optimization Setup for the Sequential Nonlinear Programming (Gradient) Optimizer

Following is the procedure for setting up an optimization analysis using the Sequential Nonlinear Programming (Gradient) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **HFSS** > **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.

3. Under the **Goals** tab, select the optimizer by selecting **Sequential Nonlinear Programming (Gradient)** from the **Optimizer** drop-down menu.



>

4. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the **Max. No. of Iterations** text box.
5. Under **Cost Function**, add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. If you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
7. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
8. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Focus** for the analysis.
   - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
   - Optionally, modify the values of fixed variables that are not being optimized.
   - Optionally, set Linear constraints.
   - Select the **View all columns** check box to see all columns, including hidden columns.
9. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

   Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.
10. Under the **Options** tab, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

> **Note:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

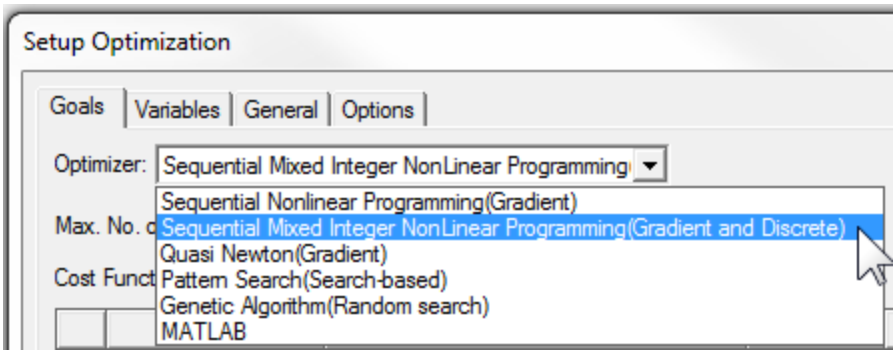
Optimization Setup for the Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) Optimizer

Following is the procedure for setting up an optimization analysis using the Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **HFSS** > **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.

3. Under the **Goals** tab, select the optimizer by selecting **Sequential Mixed Integer Nonlinear Programming (Gradient and Discrete)** from the **Optimizer** drop-down menu.



4. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the **Max. No. of Iterations** text box.
5. Under **Cost Function**, add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.
6. If you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.

   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.

7. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
8. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Focus** for the analysis.
   - Check the Integer box for integer variables.
   - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
   - Optionally, modify the values of fixed variables that are not being optimized.
   - Optionally, set Linear constraints.
   - Select the **View all columns** check box to see all columns, including hidden columns.
9. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

   Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

10. Under the **Options** tab, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

> **Note:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

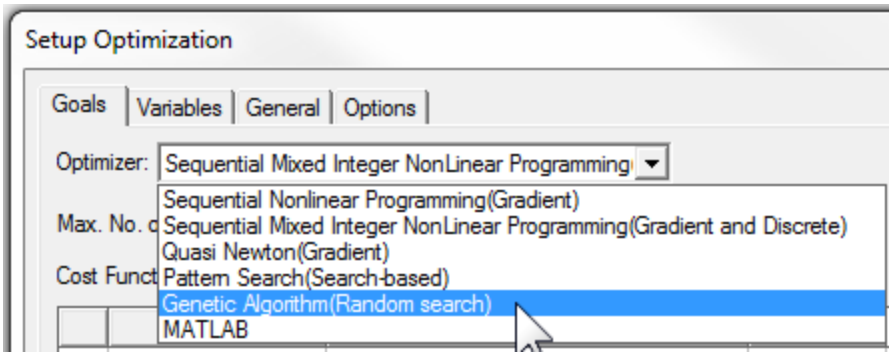
Optimization Setup for the Genetic Algorithm (Random search) Optimizer

Following is the procedure for setting up an optimization analysis using the Genetic Algorithm (Random search) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **HFSS** > **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.

3. Under the **Goals** tab, select the optimizer by selecting **Genetic Algorithm(Random search)** from the **Optimizer** pull-down list.



4. Click **Setup...** to modify the Advanced Genetic Algorithm Optimizer Options.
5. Under **Cost Function**, add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.
6. If you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.

   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.

7. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
8. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Focus** for the analysis.
   - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
   - Optionally, modify the values of fixed variables that are not being optimized.
   - Optionally, set Linear constraints.
   - Select the **View all columns** check box to see all columns, including hidden columns.
9. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

   Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

10. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Optimization Setup for the MATLAB Optimizer

Following is the procedure for setting up an optimization analysis using the MATLAB Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

1. Set up the variables you want to optimize in the *Design Properties* dialog box.
2. Click **HFSS**> **Optimetrics Analysis** > **Add Screening & Optimization**.

   The *Setup Optimization* dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Under the **Goals** tab, select the optimizer by selecting **MATLAB** from the **Optimizer** drop-down menu. Selecting **MATLAB** enables the **Acceptable Cost** and **Noise** fields.
4. Click **Setup...** to modify the MATLAB Optimizer Options.
5. Type the maximum number of iterations you want Optimetrics to perform during the optimization analysis in the Max. No. of Iterations text box.
6. Under Cost Function, add a cost function by selecting the Setup Calculations button to open the Add/Edit Calculation dialog box.
7. Type the value of the cost function at which the optimization process should stop in the Acceptable Cost text box.
8. Type the cost function noise in the Noise text box.
9. If you want to select a Cost Function Norm Type:
   - Check the Show Advanced Option check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select L1, L2, or Maximum.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors. (For more detail, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Min/Max Focus** for the analysis.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.
12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

> **Note:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

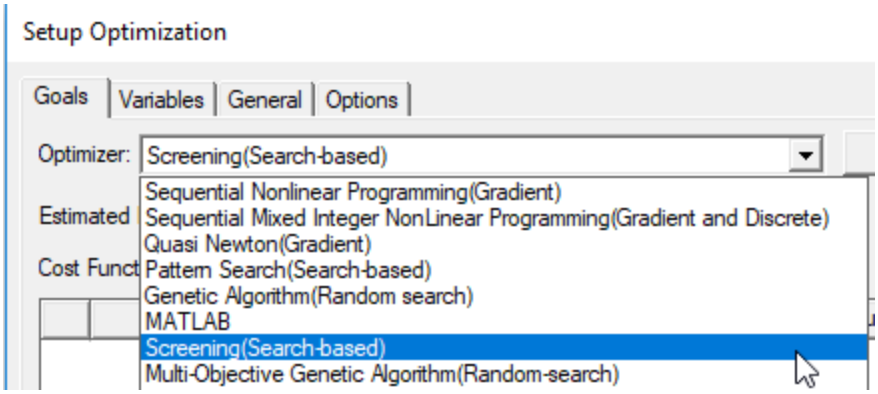
You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Setting Up Screening (Search Based) Optimizer

Following is the procedure for setting up an optimization analysis using the Screening Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

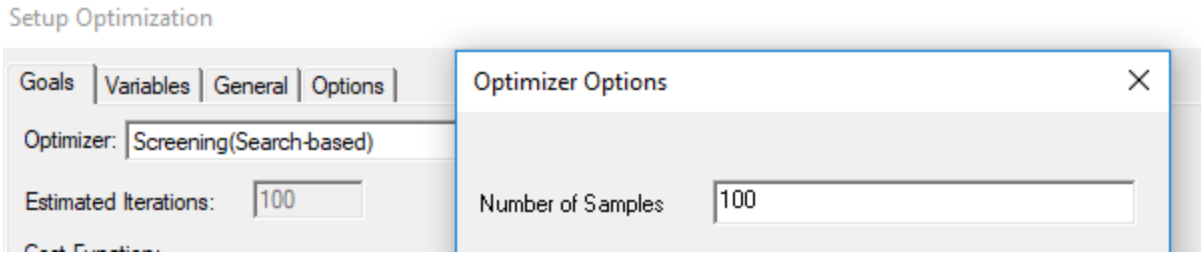
This is a non-iterative direct sampling method that uses a quasi-random number generator based on the Hammersley algorithm. You can start with Screening to locate the multiple tentative optima and then refine with NLPQL or MISQP to zoom in on the individual local maximum or minimum value. Usually Screening is used for preliminary design. Then, if you want to refine, these candidate points are used as starting points for gradient methods.

1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **HFSS** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* window appears.
3. Under the **Goals** tab, select the optimizer by selecting **Screening (Search Based)** from the **Optimizer** drop-down menu.



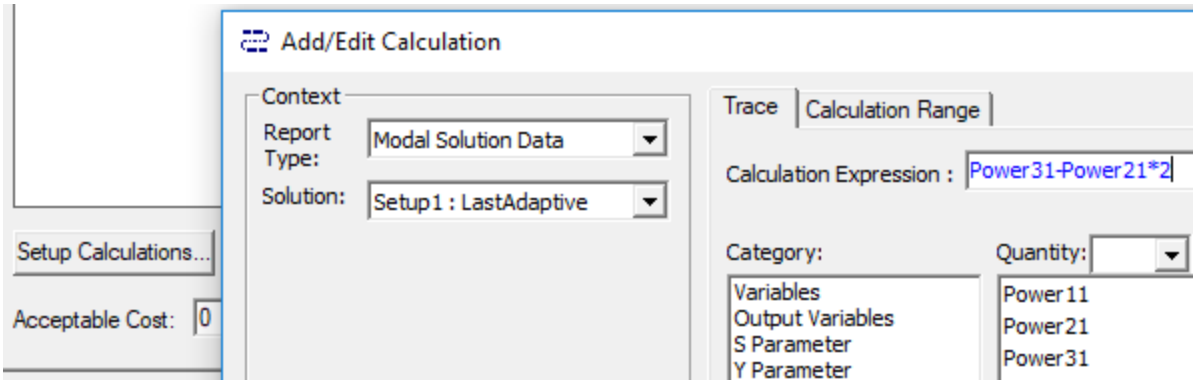
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Optionally, press the **Setup** button to open the **Optimizer Options window** to change the default number of samples from 100.



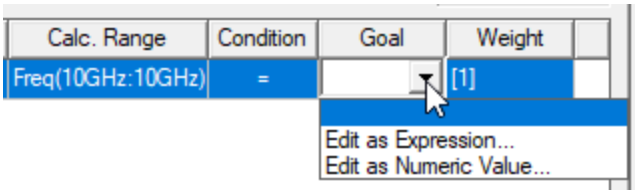
The number of samples must be greater than the number of enabled input parameters. The number of enabled input parameters is also the minimum number of samples required to generate the Sensitivities chart. You can enter a minimum of 2 and a maximum of 10,000. The default is 100 for a Direct Optimization system.

5. Add a cost function by selecting the **Setup Calculations** button to open the **Add/Edit Calculation** window.



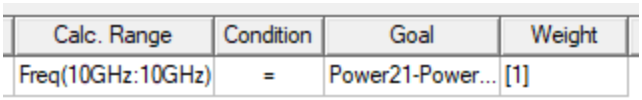
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the **Add/EditCalculation** window.

6. In the Optimization setup, in the drop-down menu for the Goal column, select either Edit as Expression or Edit as Numeric Value... This reopens the **Add/Edit Calculation** window.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. If you are satisfied with the expression or value displayed, click **Done** to close the window. This enters the expression/value to the **Goal** column.



Calc. Range	Condition	Goal	Weight
Freq(10GHz:10GHz)	=	Power21-Power...	[1]

8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** drop-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

     A norm is a function that assigns a positive value to the cost function.

     For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

     The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.
12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.
13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Setting Up Multi-Objective Genetic Algorithm(Random Search) Optimizer

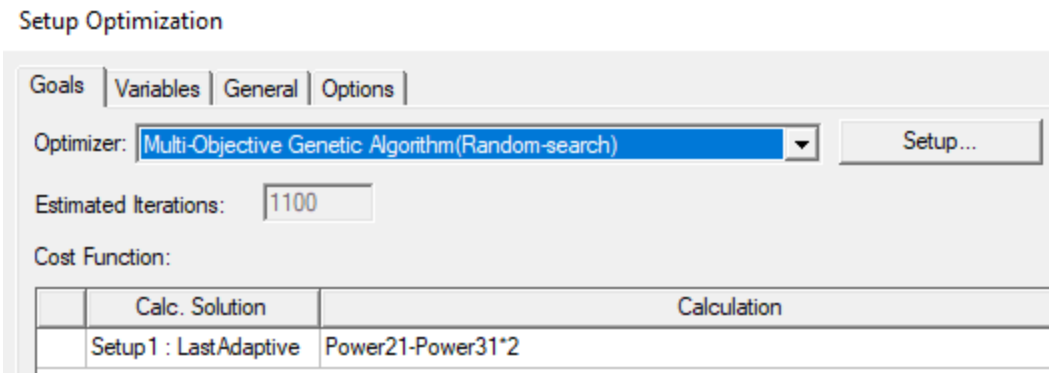
Following is the procedure for setting up an optimization analysis using the Multi-Objective Genetic Algorithm (Random Search) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

The Multi-Objective Genetic Algorithm (MOGA) is a hybrid variant of the popular NSGA-II (Non-dominated Sorted Genetic Algorithm-II) based on controlled elitism concepts. It supports all types of input parameters. The Pareto ranking scheme is done by a fast, non-dominated sorting method that is an order of magnitude faster than traditional Pareto ranking methods. The constraint handling uses the same non-dominance principle as the objectives. Therefore, penalty functions and Lagrange multipliers are not needed. This also ensures that the feasible solutions are always ranked higher than the infeasible solutions.

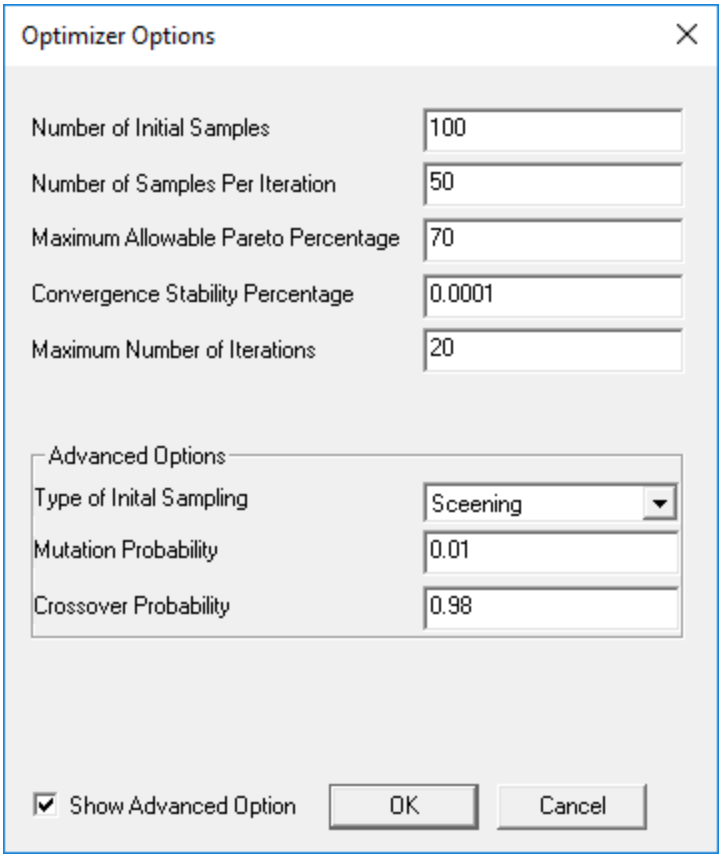
The first Pareto front solutions are archived in a separate sample set internally and are distinct from the evolving sample set. This ensures minimal disruption of Pareto front patterns already available from earlier iterations. You can control the selection pressure (and, consequently, the elitism of the process) to avoid premature convergence by altering the Maximum Allowable Pareto Percentage property.

1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **HFSS** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Multi-Objective Genetic Algorithm (Random Search)** from the **Optimizer** drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Optionally, press the **Setup** button to open the **Optimizer Options window**.



- **Number of Initial Samples**: Initial number of samples to use. This number must be greater than the number of enabled input parameters. The minimum recommended number of initial samples is 10 times the number of enabled input parameters. The larger the initial sample set, the better your chances of finding the input parameter space that contains the best solutions.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The number of enabled input parameters is also the minimum number of samples required to generate the Sensitivities chart. You can enter a minimum of 2 and a maximum of 10000. The default is 100.

If you switch from the Screening method to the MOGA method, MOGA generates a new sample set. For the sake of consistency, enter the same number of initial samples as you used for the Screening method.

- **Number of Samples Per Iteration**: Number of samples to iterate and update with each iteration. This number must be greater than the number of enabled input parameters but less than or equal to the number of initial samples. The default is for a Direct Optimization system.

  You can enter a minimum of 2 and a maximum of 10000.

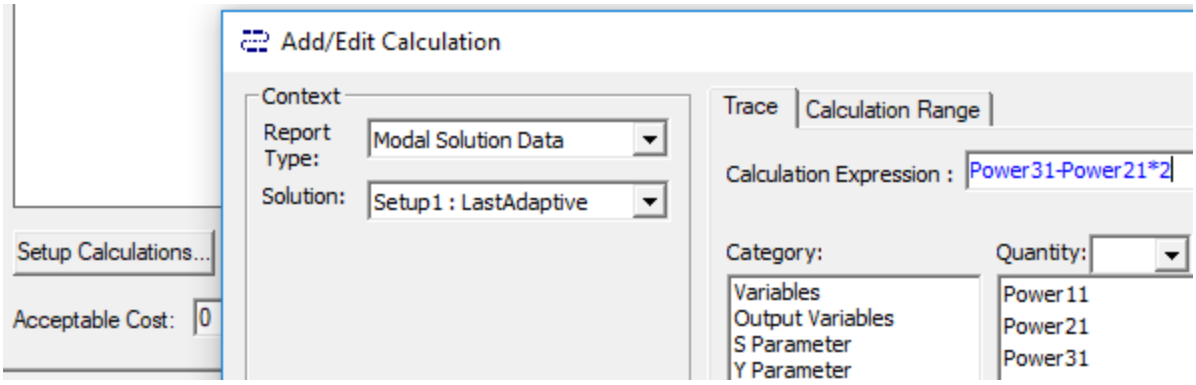
- **Maximum Allowable Pareto Percentage**: Convergence criterion. Percentage value that represents the ratio of the number of desired Pareto points to the number of samples per iteration. When this percentage is reached, the optimization is converged. For example, a value of 70 with Number of Samples Per Iteration set to 200 would mean that the optimization should stop once the resulting front of the MOGA optimization contains at least 140 points. Of course, the optimization stops before that if the maximum number of iterations is reached.
- If the **Maximum Allowable Pareto Percentage** is too low (below 30), the process can converge prematurely. If the value is too high (above 80), the process can converge slowly. The value of this property depends on the number of parameters and the nature of the design space itself. The default is 70. Using a value between 55 and 75 works best for most problems. For more information, see Convergence Criteria in MOGA-Based Multi-Objective Optimization.
- **Convergence Stability Percentage**: Convergence criterion. Percentage value that represents the stability of the population based on its mean and standard deviation. This criterion allows you to minimize the number of iterations performed while still reaching the desired level of stability. When the specified percentage is reached, the optimization is converged. The default percentage is 0.0001. To not take the convergence stability into account, set to 0. For more information, see Convergence Criteria in MOGA-Based Multi-Objective Optimization.
- **Maximum Number of Iterations**: Stop criterion. Maximum number of iterations that the algorithm is to execute. If this number is reached without the optimization having reached convergence, iterations stop. This also provides an idea of the maximum possible number of function evaluations that are needed for the full cycle, as well as the maximum possible time it can take to run the optimization. For example, the absolute maximum number of evaluations is given by:

  Number of Initial Samples + Number of Samples Per Iteration * (Maximum Number of Iterations - 1)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Type of Initial Sampling**: Advanced option for generating different kinds of sampling. If you do not have any parameter relationships defined, set to Screening (default) or Optimal Space-Filling. If you have parameter relationships defined, the initial sampling must be performed by the constrained sampling algorithms because parameter relationships constrain the sampling. For such cases, this property is automatically set to Constrained Sampling.
- **Mutation Probability**: Advanced option for specifying the probability of applying a mutation on a design configuration. The value must be between 0 and 1. A larger value indicates a more random algorithm. If the value is 1, the algorithm becomes a pure random search. A low probability of mutation (<0.2) is recommended. The default is 0.01. For more information on mutation, see MOGA Steps to Generate a New Population.
- **Crossover Probability**: Advanced option for specifying the probability with which parent solutions are recombined to generate offspring solutions. The value must be between 0 and 1. A smaller value indicates a more stable population and a faster (but less accurate) solution. If the value is 0, the parents are copied directly to the new population. A high probability of crossover (>0.9) is recommended. The default is 0.98.

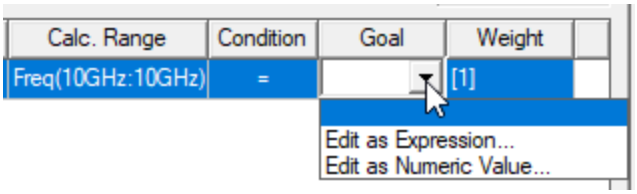
5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.



When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the **Add/EditCalculation** dialog.

6. In the Optimization setup, in the dropdown for the Goal column, select either **Edit as Expression...** or **Edit as Numeric Value...**.

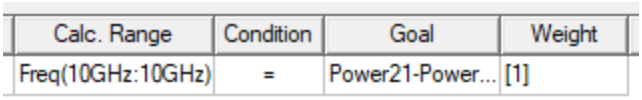
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This reopens the *Add/Edit Calculation* dialog box.

7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box.

   This enters the expression/value into the **Goal** column.



8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.

9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Optionally, set Linear constraints.
- Select the **View all columns** check box to see all columns, including hidden columns.

12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Setting Up Nonlinear Programming by Quadratic Lagrangian (Gradient) Optimizer

Following is the procedure for setting up an optimization analysis using the Nonlinear Programming by Quadratic Lagrangian (Gradient) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

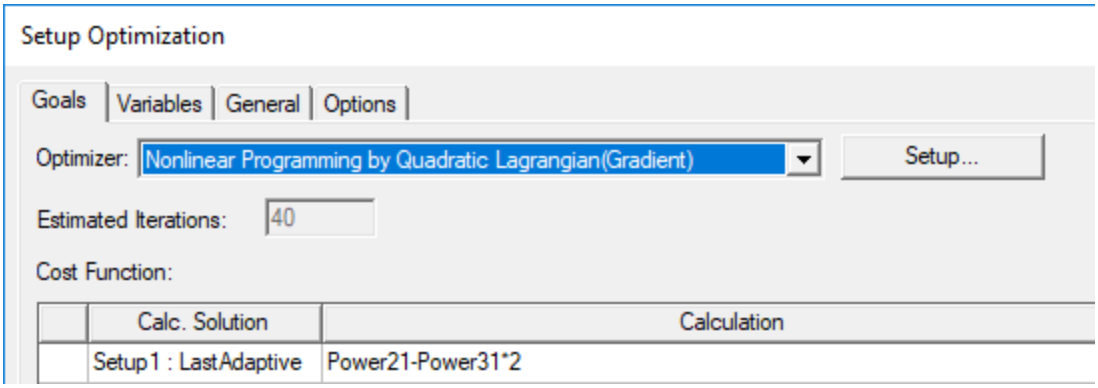
The NLPQL (Nonlinear Programming by Quadratic Lagrangian) method can be used for Direct Optimization systems. It allows you to generate a new sample set to provide a more refined approach than the Screening method. Available for continuous input parameters only, NLPQL can handle only one output parameter goal. Other output parameters can be defined as constraints. For more information, see Convergence Rate % and Initial Finite Difference Delta % in NLPQL and MISQP and Nonlinear Programming by Quadratic Lagrangian (NLPQL).

To generate samples and perform an NLPQL optimization:

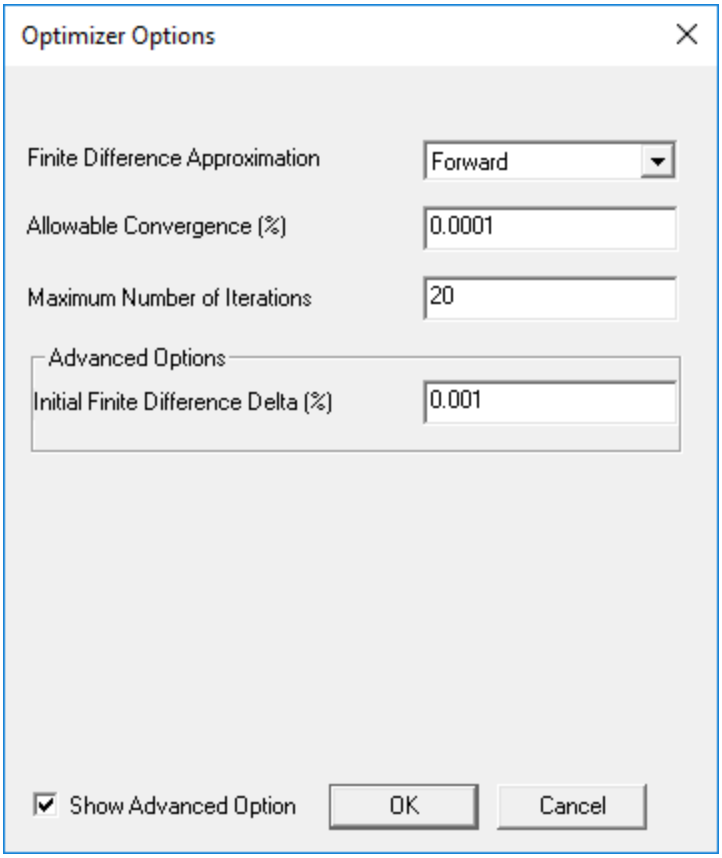
1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **HFSS** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Nonlinear Programming by Quadratic Lagrangian (Gradient)** from the **Optimizer** drop-down menu.



4. Optionally, press the **Setup** button to open the *Optimizer Options window*.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Finite Difference Approximation**: When analytical gradients are not available, NLPQL approximates them numerically. This property allows you to specify the method of approximating the gradient of the objective function. Choices are:
  - **Central**: Increases the accuracy of the gradient calculations by sampling from both sides of the sample point but increases the number of design point evaluations by 50%. This method makes use of the initial point, as well as the forward point and rear point.
  - **Forward**: Uses fewer design point evaluations but decreases the accuracy of the gradient calculations. This method makes use of only two design points, the initial point and forward point, to calculate the slope forward. This is the default method for new Direct Optimization systems.
- **Maximum Number of Iterations**: Stop criterion. Maximum number of iterations that the algorithm is to execute. If convergence happens before this number is reached, the iterations stop. This also provides an idea of the maximum possible number of function evaluations that are needed for the full cycle. For NLPQL, the number of evaluations can be approximated according to the Finite Difference Approximation gradient calculation method, as follows:

  For **Central**: number of iterations * (2*number of inputs +1)

  For **Forward**: number of iterations * (number of inputs+1)

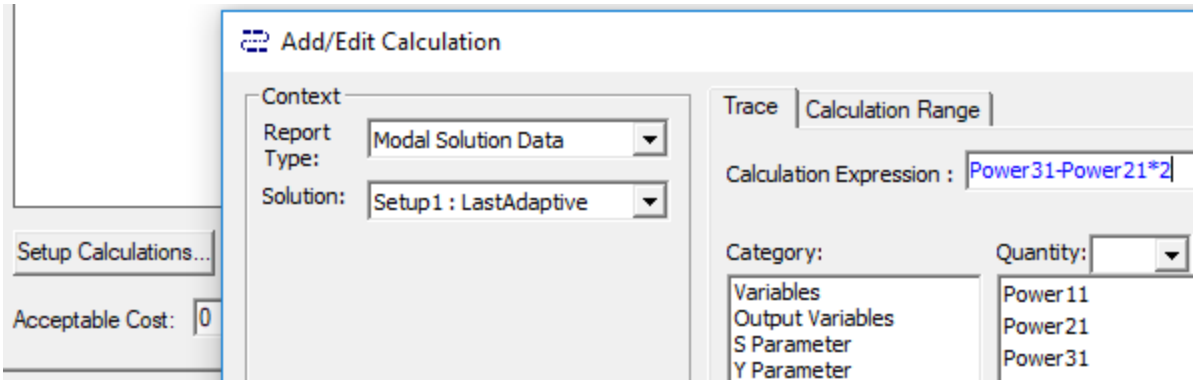
- **Allowable Convergence (%)**: Stop criterion. Tolerance to which the Karush-Kuhn-Tucker (KKT) optimality criterion is generated during the NLPQL process. A smaller value indicates more convergence iterations and a more accurate (but slower) solution. A larger value indicates fewer convergence iterations and a less accurate (but faster) solution. For a Direct Optimization system, the default percentage value is 0.1. The maximum percentage value is 100. These values are consistent across all problem types because the inputs, outputs, and gradients are scaled during the NLPQL solution.
- **Initial Finite Difference Delta (%)**: Advanced option for specifying the relative variation used to perturb the current point to compute gradients. Used in conjunction with Allowable Convergence (%) to ensure that the delta in NLPQL's calculation of finite differences is large enough to be seen above the noise in the simulation problem. This wider sampling produces results that are more clearly differentiated so that the difference is less affected by solution noise and the gradient direction is clearer. The value should be larger than both the value for Initial Finite Difference Delta (%) and the noise magnitude of the model. However, smaller values produce more accurate results, so set Initial Finite Difference Delta (%) only as high as necessary to be seen above simulation noise.

The default percentage value is 0.001. The minimum is 0.0001, and the maximum is 1. For parameters with Allowed Values set to **Manufacturable Values** or Snap to Grid, the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

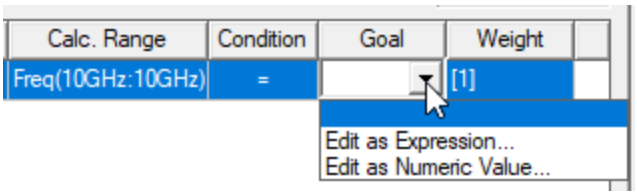
value for Initial Finite Difference Delta (%) is ignored. In such cases, the closest allowed value is used to determine the finite difference delta.

5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.



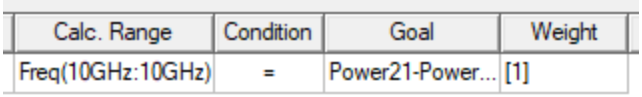
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the *Add/EditCalculation* dialog box.

6. In the Optimization setup, in the drop-down menu for the Goal column, select either Edit as Expression or Edit as Numeric Value...



This reopens the *Add/Edit Calculation* dialog box.

7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box. This enters the expression/value to the **Goal** column.



8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A norm is a function that assigns a positive value to the cost function.

For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.

9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.
12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Up Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete) Optimizer

Mixed-Integer Sequential Quadratic Programming (MISQP) is a mathematical optimization algorithm as developed by Oliver Exler, Thomas Lehmann and Klaus Schittkowski (NLPQL). This method solves Mixed-Integer Nonlinear Programming (MINLP) of the form:

Minimize:

$$f(x,y)$$

Subject to

$$g_j(x,y)=0,\ j=1,\ldots,m_e,$$
$$g_j(x,y)\geq 0,\ j=m_e+1,\ldots,m$$

where

$$x\in\mathbb{R}^{n_c}, y\in\mathbb{N}^{n_i}$$
$$x_l\leq x\leq x_u$$
$$y_l\leq y\leq y_u$$

The symbols and denote the vectors of the continuous and integer variables, respectively. It is assumed that problem functions and are continuously differentiable subject to all . It is not assumed that integer variables can be relaxed. In other words, problem functions are evaluated only at integer points and never at any fractional values in between.

MISQP solves MINLP by a modified sequential quadratic programming (SQP) method. After linearizing constraints and constructing a quadratic approximation of the Lagrangian function, mixed-integer quadratic programs are successively generated and solved by an efficient branch-and-cut method. The algorithm is stabilized by a trust region method as originally proposed by Yuan for continuous programs. Second-order corrections are retained. The Hessian of the Lagrangian function is approximated by BFGS updates subject to the continuous and integer variables. MISQP is able to solve also non-convex nonlinear mixed-integer programs.

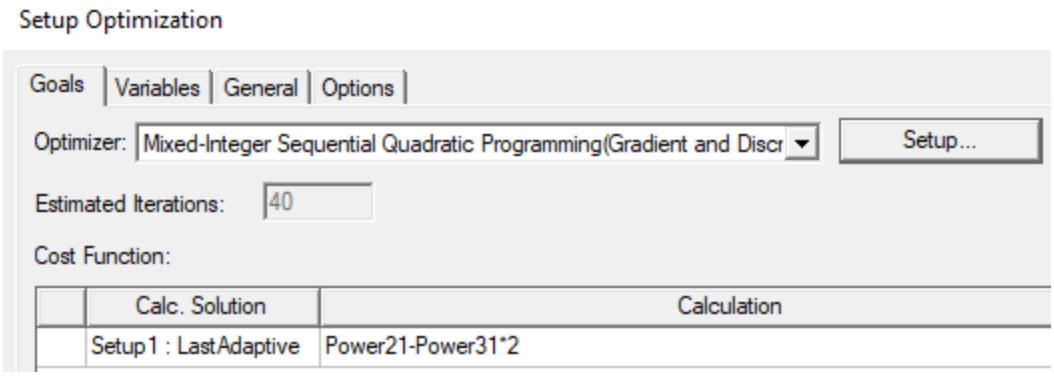
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Following is the procedure for setting up an optimization analysis using the Mixed-Integer Sequential Quadratic Programming optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

The Mixed-Integer Sequential Quadratic Programming (Nonlinear Programming by Quadratic Lagrangian) method (MISQP)can be used for Direct Optimization systems. It allows you to generate a new sample set to provide a more refined approach than the Screening method. MISQP is available for both continuous and discrete input parameters, which is why mixed is in its name. MISQP can handle only one output parameter goal.. Other output parameters can be defined as constraints. For more information, see Convergence Rate % and Initial Finite Difference Delta % in NLPQL and MISQP and Nonlinear Programming by Quadratic Lagrangian (NLPQL).

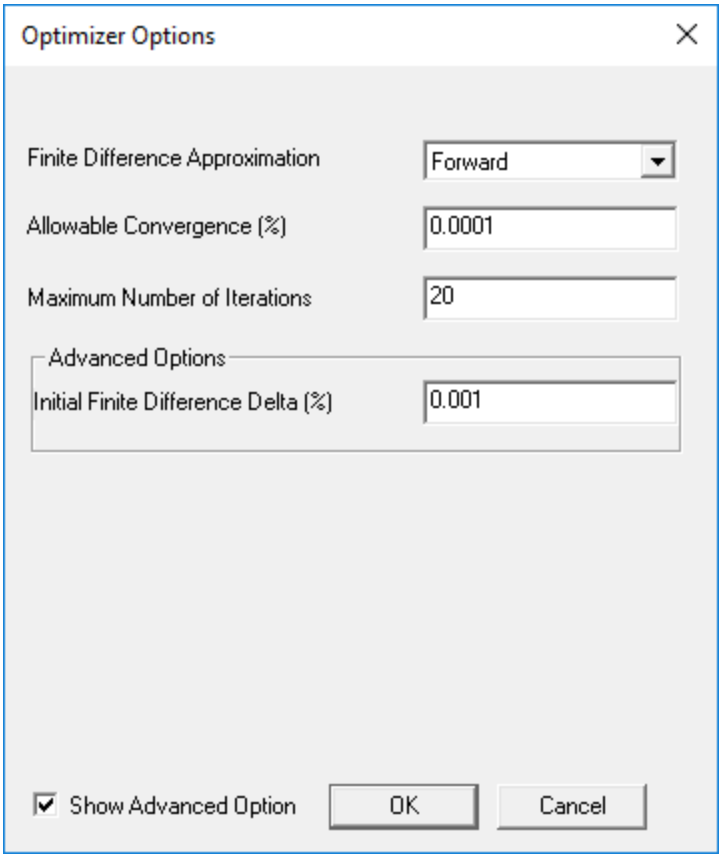
To generate samples and perform an NLPQL optimization:

1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **HFSS** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Mixed-Integer Sequential Quadratic Programming (Gradient and Discrete)** from the **Optimizer** drop-down menu.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Optionally, press the **Setup** button to open the **Optimizer Options window**.



- **Finite Difference Approximation**: When analytical gradients are not available, MISQP approximates them numerically. This property allows you to specify the method of approximating the gradient of the objective function. Choices are:
  - **Central**: Increases the accuracy of the gradient calculations by sampling from both sides of the sample point but increases the number of design point evaluations by 50%. This method makes use of the initial point, as well as the forward point and rear point.
  - **Forward**: Uses fewer design point evaluations but decreases the accuracy of the gradient calculations. This method makes use of only two design points, the initial point and forward point, to calculate the slope forward. This is the default method for new Direct Optimization systems.
- **Maximum Number of Iterations**: Stop criterion. Maximum number of iterations that the algorithm is to execute. If convergence happens before this number is reached, the iterations stop. This also provides an idea of the maximum possible number of function evaluations that are needed for the full cycle. For MISQP, the number of evaluations can be approximated according to the Finite Difference

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Approximation gradient calculation method, as follows:

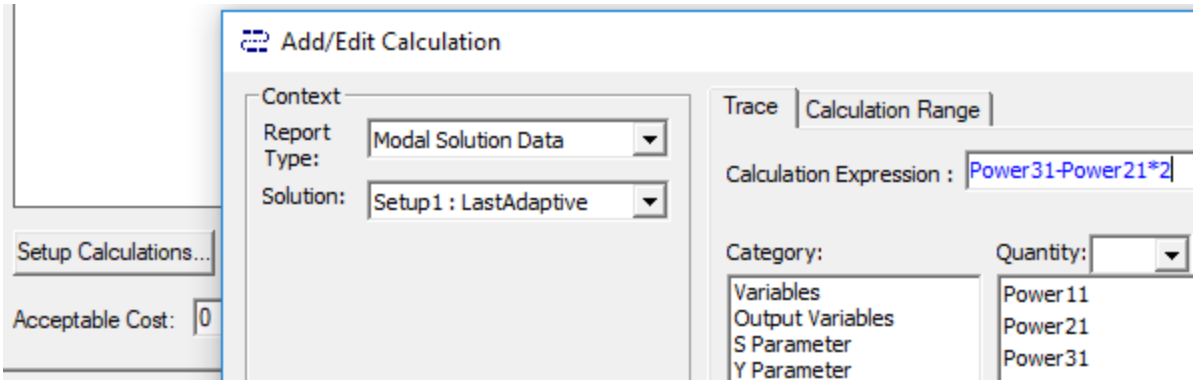
For **Central**: number of iterations * (2*number of inputs +1)

For **Forward**: number of iterations * (number of inputs+1)

- **Allowable Convergence (%)**: Stop criterion. Tolerance to which the Karush-Kuhn-Tucker (KKT) optimality criterion is generated during the MISQP process. A smaller value indicates more convergence iterations and a more accurate (but slower) solution. A larger value indicates fewer convergence iterations and a less accurate (but faster) solution. For a Direct Optimization system, the default percentage value is 0.0001. The maximum percentage value is 100. These values are consistent across all problem types because the inputs, outputs, and gradients are scaled during the MISQP solution.
- **Initial Finite Difference Delta (%)**: Advanced option for specifying the relative variation used to perturb the current point to compute gradients. Used in conjunction with Allowable Convergence (%) to ensure that the delta in MISQP's calculation of finite differences is large enough to be seen above the noise in the simulation problem. This wider sampling produces results that are more clearly differentiated so that the difference is less affected by solution noise and the gradient direction is clearer. The value should be larger than both the value for Initial Finite Difference Delta (%) and the noise magnitude of the model. However, smaller values produce more accurate results, so set Initial Finite Difference Delta (%) only as high as necessary to be seen above simulation noise.

The default percentage value is 0.001. The minimum is 0.0001, and the maximum is 1. For parameters with Allowed Values set to **Manufacturable Values** or Snap to Grid, the value for Initial Finite Difference Delta (%) is ignored. In such cases, the closest allowed value is used to determine the finite difference delta.

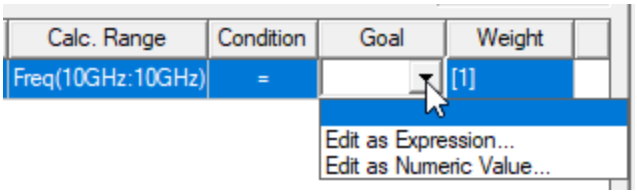
5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

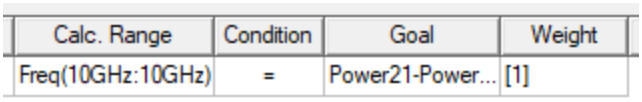
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the *Add/EditCalculation* dialog box.

6. In the Optimization setup, in the drop-down menu, for the Goal column, select either Edit as Expression or Edit as Numeric Value...



This reopens the *Add/Edit Calculation* dialog box.

7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box. This enters the expression/value to the **Goal** column.



8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.

9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
- Optionally, modify the values of fixed variables that are not being optimized.
- Optionally, set Linear constraints.
- Select the **View all columns** check box to see all columns, including hidden columns.

12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

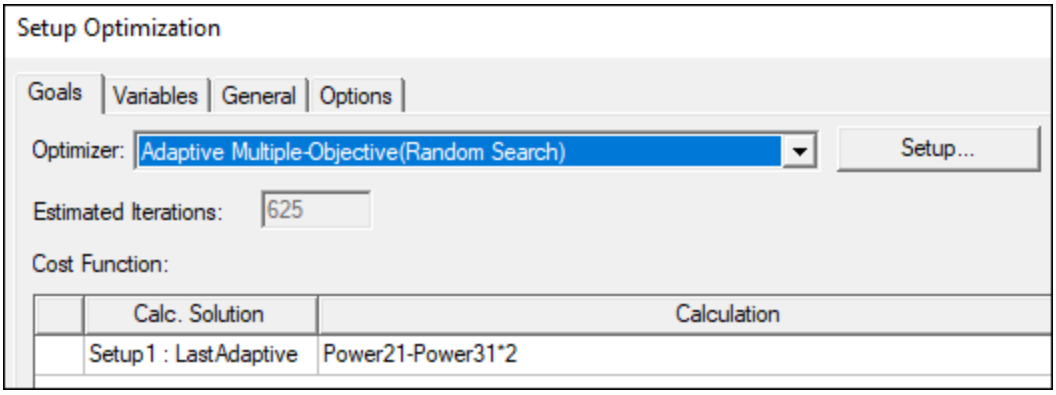
Setting Up Adaptive Multiple-Objective (Random Search) Optimizer

Following is the procedure for setting up an optimization analysis using the Adaptive Multiple Objective (Random Search) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

The Adaptive Multiple Objective (Kriging + MOGA) is an iterative algorithm that allows you to either generate a new sample set or use an existing set, providing a more refined approach than the Screening method. It uses the same general approach as MOGA, but applies the Kriging error predictor to reduce the number of evaluations needed to find the global optimum. The Adaptive Multiple-Objective method is available only for continuous input parameters, including those with manufacturable values. It can handle multiple objectives and multiple constraints. For more information, see Adaptive Multiple-Objective Optimization.

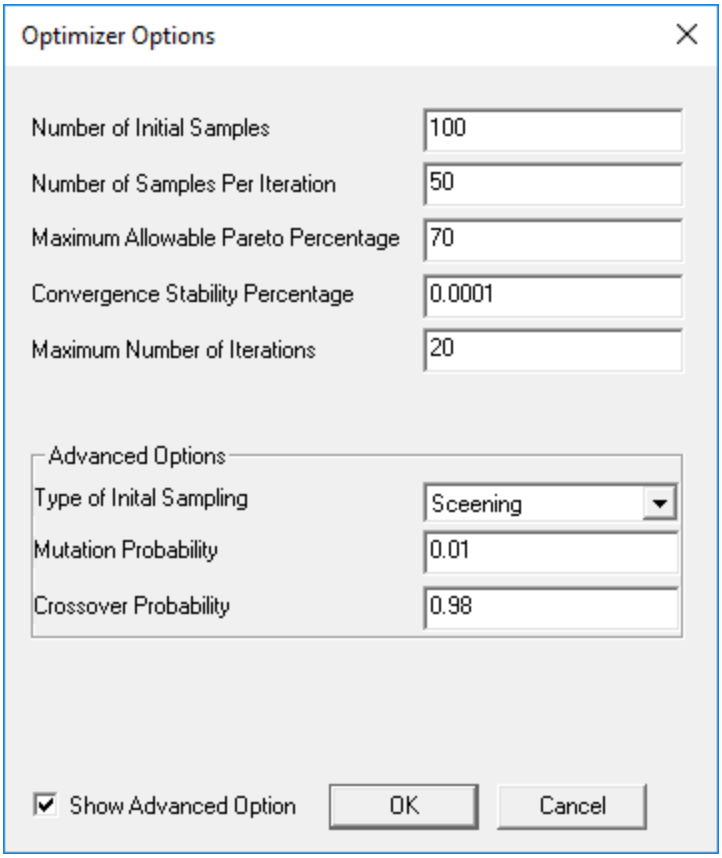
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **HFSS** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, select the optimizer by selecting **Adaptive Multiple Objective (Random-search)** from the **Optimizer** drop-down menu.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Optionally, press the **Setup** button to open the *Optimizer Options* window.



- **Number of Initial Samples**: Initial number of samples to use. This number must be greater than the number of enabled input parameters. The minimum recommended number of initial samples is 10 times the number of enabled input parameters. The larger the initial sample set, the better your chances of finding the input parameter space that contains the best solutions.

  The number of enabled input parameters is also the minimum number of samples required to generate the Sensitivities chart. You can enter a minimum of 2 and a maximum of 10000. The default is 100.

  If you switch from the Screening method to the MOGA method, MOGA generates a new sample set. For the sake of consistency, enter the same number of initial samples as you used for the Screening method.

- **Number of Samples Per Iteration**: Number of samples to iterate and update with each iteration. This number must be greater than the number of enabled input parameters but less than or equal to the number of initial samples. The default is for a Direct Optimization system.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can enter a minimum of 2 and a maximum of 10000.

- **Maximum Allowable Pareto Percentage**: Convergence criterion. Percentage value that represents the ratio of the number of desired Pareto points to the number of samples per iteration. When this percentage is reached, the optimization is converged. For example, a value of 70 with Number of Samples Per Iteration set to 200 would mean that the optimization should stop once the resulting front of the MOGA optimization contains at least 140 points. Of course, the optimization stops before that if the maximum number of iterations is reached.
- If the **Maximum Allowable Pareto Percentage** is too low (below 30), the process can converge prematurely. If the value is too high (above 80), the process can converge slowly. The value of this property depends on the number of parameters and the nature of the design space itself. The default is 70. Using a value between 55 and 75 works best for most problems. For more information, see Convergence Criteria in MOGA-Based Multi-Objective Optimization.
- **Convergence Stability Percentage**: Convergence criterion. Percentage value that represents the stability of the population based on its mean and standard deviation. This criterion allows you to minimize the number of iterations performed while still reaching the desired level of stability. When the specified percentage is reached, the optimization is converged. The default percentage is 0.0001. To not take the convergence stability into account, set to 0. For more information, see Convergence Criteria in MOGA-Based Multi-Objective Optimization.
- **Maximum Number of Iterations**: Stop criterion. Maximum number of iterations that the algorithm is to execute. If this number is reached without the optimization having reached convergence, iterations stop. This also provides an idea of the maximum possible number of function evaluations that are needed for the full cycle, as well as the maximum possible time it can take to run the optimization. For example, the absolute maximum number of evaluations is given by:

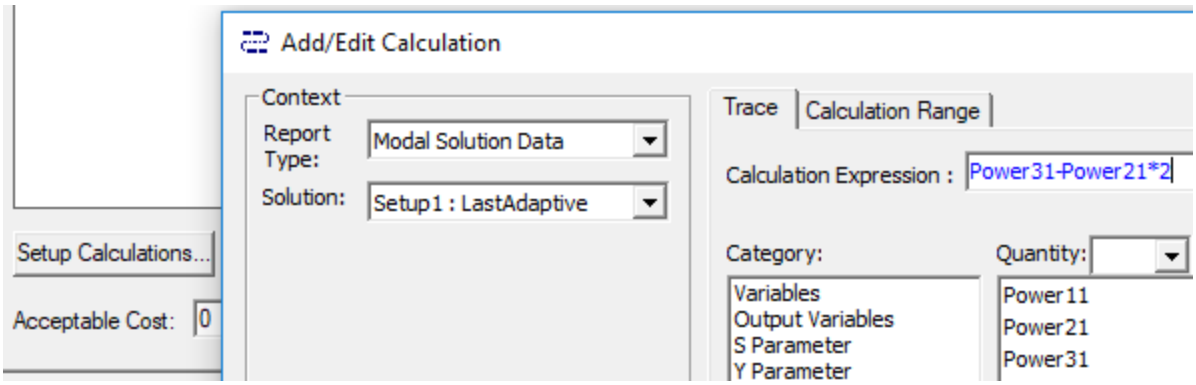
  Number of Initial Samples + Number of Samples Per Iteration * (Maximum Number of Iterations - 1)

- **Type of Initial Sampling**: Advanced option for generating different kinds of sampling. If you do not have any parameter relationships defined, set to Screening (default) or Optimal Space-Filling. If you have parameter relationships defined, the initial sampling must be performed by the constrained sampling algorithms because parameter relationships constrain the sampling. For such cases, this property is automatically set to Constrained Sampling.
- **Mutation Probability**: Advanced option for specifying the probability of applying a mutation on a design configuration. The value must be between 0 and 1. A larger value indicates a more random algorithm. If the value is 1, the algorithm becomes a pure random search. A low probability of mutation (<0.2) is recommended. The default is 0.01. For more information on mutation, see MOGA Steps to Generate a New Population.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

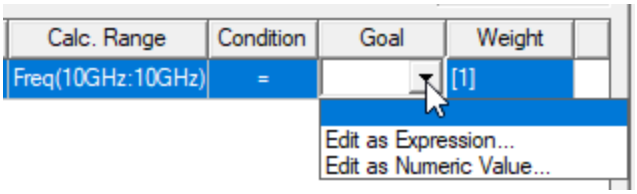
- **Crossover Probability**: Advanced option for specifying the probability with which parent solutions are recombined to generate offspring solutions. The value must be between 0 and 1. A smaller value indicates a more stable population and a faster (but less accurate) solution. If the value is 0, the parents are copied directly to the new population. A high probability of crossover (>0.9) is recommended. The default is 0.98.

5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.

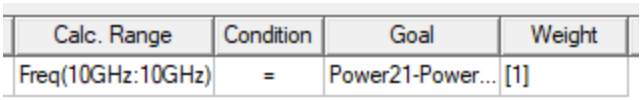


When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the *Add/EditCalculation* dialog box.

6. In the Optimization setup, in the Goal column drop-down menu, select either **Edit as Expression...** or **Edit as Numeric Value...**. This reopens the *Add/Edit Calculation* dialog box.



7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box. This enters the expression/value into the **Goal** column.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The *Cost Function Norm Type* pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.
12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.
13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Setting Up Adaptive Single-Objective (Gradient) Optimizer

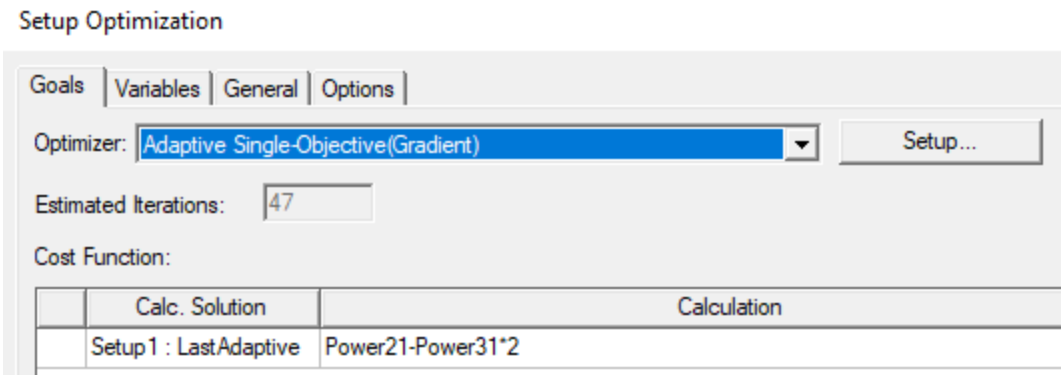
Following is the procedure for setting up an optimization analysis using the Adaptive Single-Objective (OSF + Kriging + MISQP) Optimizer. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes.

This gradient-based method employs automatic intelligent refinement to provide the global optima. It requires a minimum number of design points to build the Kriging response surface, but, in general, this method reduces the number of design points necessary for the optimization. Failed design points are treated as inequality constraints, making it fault-tolerant.

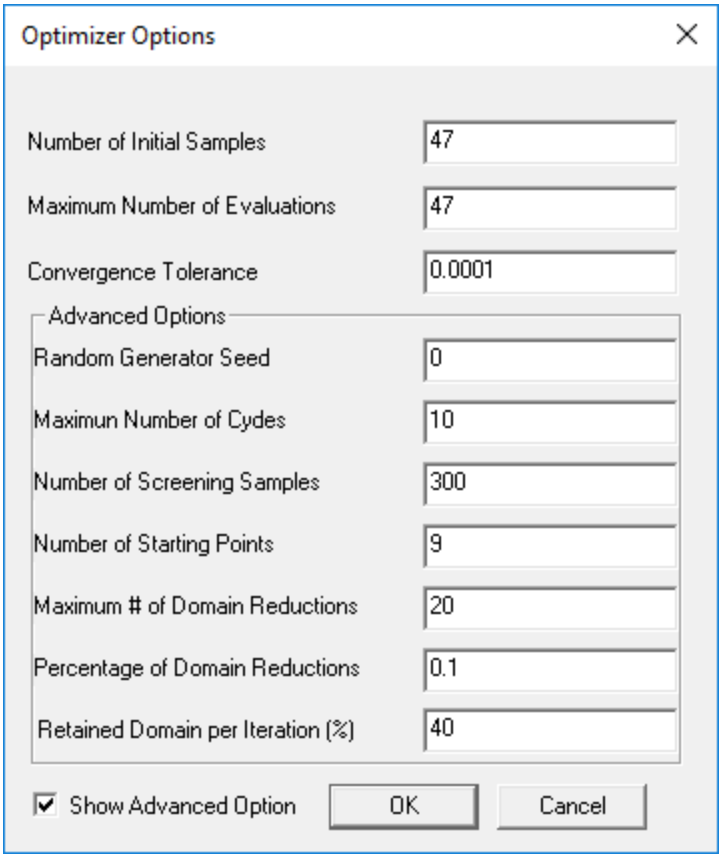
The Adaptive Single-Objective method is available for input parameters that are continuous, including those with manufacturable values. It can handle only one output parameter goal, although other output parameters can be defined as constraints. It does not support the use of parameter relationships in the optimization domain. For more information, see Adaptive Single-Objective Optimization (ASO). It requires advanced options. Ensure that the Show Advanced Options check box is selected.

1. Set up the variables you want to optimize in the *Design Properties* dialog box. The variables must be swept in a Parametric setup.
2. Click **HFSS** > **Optimetrics Analysis** > **Add Screening & Optimization**. The *Setup Optimization* dialog box appears.
3. Under the **Goals** tab, choose the optimizer by selecting **Adaptive Single Objective (Gradient)** from the **Optimizer** drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Optionally, press the **Setup** button to open the *Optimizer Options* window and check **Advanced Options**.



- **Number of Initial Samples**: Number of samples generated for the initial Kriging and after all domain reductions for the construction of the next Kriging. You can enter a minimum of (NbInp+1)*(NbInp+2)/2 (also the minimum number of OSF samples required for the Kriging construction) or a maximum of 10,000. The default

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is (NbInp+1)*(NbInp+2)/2.

Because of the Adaptive Single-Objective workflow (in which a new OSF sample set is generated after each domain reduction), increasing the number of OSF samples does not necessarily improve the quality of the results and significantly increases the number of evaluations

- **Maximum Number of Evaluations**: Stop criterion. Maximum number of evaluations (design points) that the algorithm is to calculate. If convergence occurs before this number is reached, evaluations stop. This value also provides an idea of the maximum possible time it takes to run the optimization. The default is 20*(NbInp +1).
- **Convergence Tolerance**: Stop criterion. Minimum allowable gap between the values of two successive candidates. If the difference between two successive candidates is smaller than the value for Convergence Tolerance multiplied by the maximum variation of the parameter, the algorithm is stopped. A smaller value indicates more convergence iterations and a more accurate (but slower) solution. A larger value indicates fewer convergence iterations and a less accurate (but faster) solution. The default is 1E-06.
- **Random Generator Seed**: The value for initializing the random number generator invoked internally by OSF. The value must be a positive integer. This property allows you to generate different samplings by changing the value or to regenerate the same sampling by keeping the same value. The default is 0.
- **Maximum Number of Cycles**: Number of optimization loops that the algorithm needs, which in turns determines the discrepancy of the OSF. The optimization is essentially combinatorial, so a large number of cycles slows down the process. However, this makes the discrepancy of the OSF smaller. The value must be greater than 0. For practical purposes, 10 cycles is usually good for up to 20 variables. The default is 10.
- **Number of Screening Samples**: Number of samples for the screening generation on the current Kriging. This value is used to create the next Kriging (based on error prediction) and verified candidates.

  You can enter a minimum of (NbInp+1)*(NbInp+2)/2 (also the minimum number of OSF samples required for the Kriging construction) or a maximum of 10,000. The default is 100*NbInp for a Direct Optimization system. There is no default for a Response Surface Optimization system.

  The larger the screening sample set, the better the chances of finding good verified points. However, too many points can result in a divergence of the Kriging.

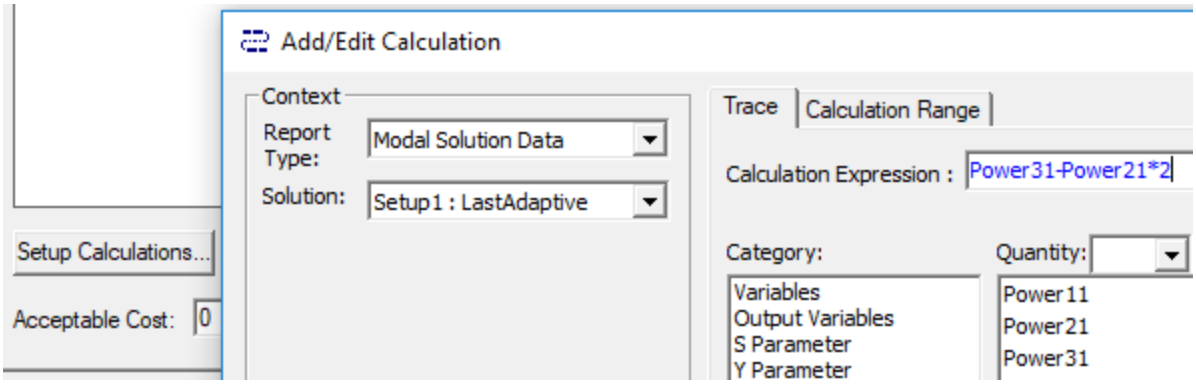
- **Number of Starting Points**: Determines the number of local optima to explore. The larger the number of starting points, the more local optima explored. In the case of a linear surface, for example, it is not necessary to use many points. This value must be less than the value for **Number of Screening Samples** because these

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

samples are selected in this sample. The default is the value for **Number of Initial Samples**.

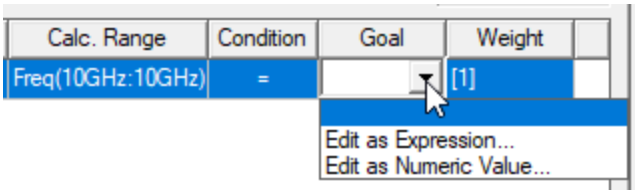
- **Maximum Number of Domain Reductions**: Stop criterion. Maximum number of domain reductions for input variation. (No information is known about the size of the reduction beforehand.) The default is 20.
- **Percentage of Domain Reductions**: Stop criterion. Minimum size of the current domain according to the initial domain. For example, with one input ranging between 0 and 100, the domain size is equal to 100. The percentage of domain reduction is 1%, so the current working domain size cannot be less than 1 (such as an input ranging between 5 and 6). The default is 0.1.
- **Retained Domain per Iteration (%)**: Advanced option that allows you to specify the minimum percentage of the domain you want to keep after a domain reduction. The percentage value must be between 10 and 90. A larger value indicates less domain reduction, which implies better exploration but a slower solution. A smaller value indicates a faster and more accurate solution, with the risk of it being a local one. The default percentage value is 40.

5. Add a cost function by selecting the **Setup Calculations** button to open the *Add/Edit Calculation* dialog box.



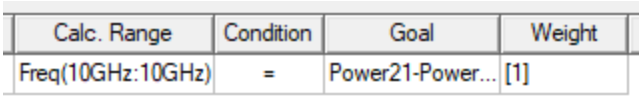
When you have created the calculation, click **Add Calculation** to add it to the **Optimization** setup, and **Done** to close the **Add/EditCalculation** dialog box.

6. In the Optimization setup, in the Goal column drop-down menu, select either **Edit as Expression...** or **Edit as Numeric Value...**. This reopens the *Add/Edit Calculation* dialog box.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. If you are satisfied with the expression or value displayed, click **Done** to close the dialog box. This enters the expression/value into the **Goal** column.



8. In the **Optimization** setup, if you want to select a **Cost Function Norm Type**:
   - Check the **Show Advanced Option** check box.

     The **Cost Function Norm Type** pull-down list appears.
   - Select **L1**, **L2**, or **Maximum**.

   A norm is a function that assigns a positive value to the cost function.

   For **L1** norm the actual cost function uses the sum of absolute weighted values of the individual goal errors. For **L2** norm (the default) the actual cost function uses the weighted sum of squared values of the individual goal error. For the Maximum norm the cost function uses the maximum among all the weighted goal errors, which means that it is always less than zero. (For further details, see Explanation of the L1, L2, and Max Norms in Optimization.)

   The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios.
9. Optionally, set the Acceptable Cost and Cost Function Noise.
10. Optionally, click the button for setting HPC and Analysis Options, which allows you to select or create an analysis configuration.
11. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization.
    - You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.
    - Optionally, modify the values of fixed variables that are not being optimized.
    - Optionally, set Linear constraints.
    - Select the **View all columns** check box to see all columns, including hidden columns.
12. In the **General** tab, specify whether Optimetrics should use the results of a previous Parametric analysis or perform one as part of the optimization process.

    Enabling the **Update design parameters' value after optimization** check box will cause Optimetrics to modify the variable values in the nominal design to match the final values from the optimization analysis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

    > **Note:**
    >
    > Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

    You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

Setting the Maximum Iterations for an Optimization Analysis

The **Max. No. of Iterations** value is the maximum number of design variations that you want Optimetrics to solve during an optimization when using the **Sequential Nonlinear Programming (Gradient)**, **Sequential Mixed Integer NonLinear Programming**, **Quasi Newton (Gradient)**, or **Pattern Search (Search-based) Optimizer**. This value is a stopping criterion; if the maximum number of iterations has been completed, the optimization analysis stops. If the maximum number of iterations has not been completed, the optimization continues by performing another iteration, that is, by solving another design variation.

If the maximum number of iterations has not been reached, the optimizer performs iterations until the acceptable cost function is reached or until the optimizer cannot proceed as a result of other optimization setup constraints, such as when it searches for a variable value with a step size smaller than the minimum step size.

> **Note:**
>
> The **Genetic Algorithm** optimizer does not use the **Max. No. of Iterations** criteria.

To set the maximum number of iterations for an optimization analysis:

- Under the **Goals** tab of the *Setup Optimization* dialog box, type a value in the **Max. No. of Iterations** text box.

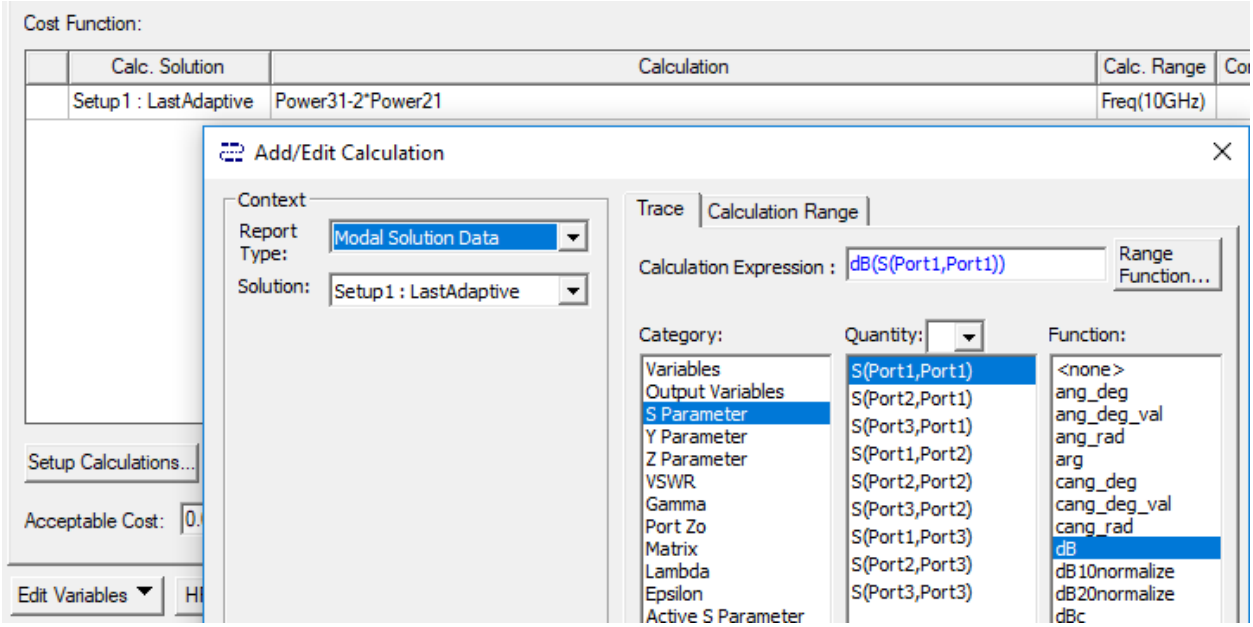
Cost Function

Optimetrics manipulates the model's design variable values to find the minimum location of the cost function; therefore, you should define the cost function so that a minimum location is also the optimum location. For example, if you vary a design to find the maximum transmission from *Wave Port 1* to *Wave Port 2* (S_{21}=>1), define the cost function to be -mag(S (WavePort2,WavePort1)).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When using the Quasi Newton optimizer, which is appropriate for designs that are not sensitive to noise, the best cost function is a smooth, second-order function that can be approximated well by quadratics in the vicinity of the minimum; the slope of the cost function should decrease as Optimetrics approaches the optimum value. The preferred cost function takes values between 0 and 1. In practice, most functions that are smooth around the minimum are acceptable as cost functions. Most importantly, the cost function should not have a sharp dip or pole at the minimum. A well designed cost function can significantly reduce the optimization process time.

The cost function is defined in the **Setup Optimization** dialog box or the **Design of Experiments** setup when you set up an optimization analysis. If you know the exact syntax of the solution quantity on which you want to base the cost function, you can type it directly in the **Calculation** text box. You can also use **Setup Calculations** to add a solution quantity via the *Add/Edit Calculation* dialog box, or to create an output variable that represents the solution quantity in the Output Variables dialog box.



Acceptable Cost

The acceptable cost is the value of the cost function at which the optimization process should stop; otherwise known as the *stopping criterion*.The cost function value must be equal to or below the acceptable cost value for the optimization analysis to stop. The acceptable cost may be a negative value.

Cost Function Noise

The numerical calculation of the electromagnetic field introduces various sources of noise to the cost function, particularly because of changes in the finite element mesh. You must provide the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

optimizer with an estimate of the noise. The noise indicates whether a change during the solution process is significant enough to support achievement of the cost function.

For example, if the cost function, *c*, is

$$c = 10000 \cdot |S_{11}|^2$$

where $|S_{11}|$ is the magnitude of the reflection coefficient, at the minimum, $|S_{11}|$ is expected to be very small, $|S_{11}| \approx 0$.

From the solution setup, the error in $|S_{11}|$ is expected to be $E_{S11} \approx 0.01$. The perturbed cost function is therefore

$$c_{perturbed} = 10000 \cdot (|S_{11}|_{min} + E_{S11})^2$$

Near the minimum, the error in the cost function E_c is given by

$$E_c = c_{perturbed} - c_{min} = 10000 \cdot (0.0 + 0.01)^2 - (10000 \cdot 0.0) = 1.0$$

Therefore, the cost function noise would be 1.0.

Adding a Cost Function

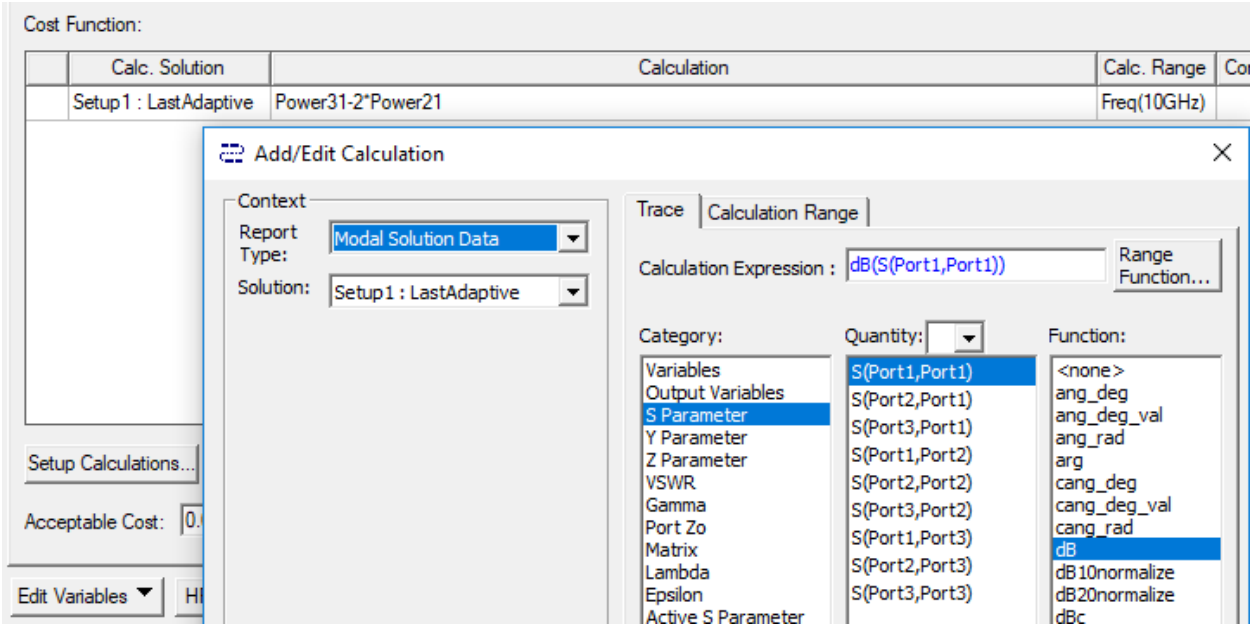
A cost function can include one or more goals for an optimization analysis. Optimetrics manipulates the model's design variable values to fulfill the cost function. The optimization will stop when the solution quantity meets the acceptable cost criterion.

Following is the general procedure for adding a cost function with a single goal:

1. Under the **Goals** tab of the *Setup Optimization* or *Design of Experiments* dialog box, click **Setup Calculations...**

   The Add/Edit Calculation dialog box is displayed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. In the *Add/Edit Calculation* dialog box, follow these general steps to set up a cost function.
   a. Set the **Context** for the calculation.
   b. Choose the **Category** of available data type depending upon the Solution type of the design being optimized.
   c. Select the **Quantity** to add to the **Calculated Expression** field. Available quantities depend upon the **Category** selection.
   d. You may optionally make a selection from the function list to apply to the calculated expression.

      Note:

      Because Optimetrics works in SI values, you should use ang_deg_val or cang_deg_val functions which return unitless numbers in degrees rather than ang_deg or cang_deg, which return angular values in degree units but evaluate in radians in Optimetrics expressions. For example, you could Optimize successfully by changing the expression from abs(**cang_deg**(S(Port2,Port1))) to abs(**cang_deg_val**(S(Port2,Port1))). See the table of available functions and definitions.

   e. When the **Calculation Expression** has the desired equation, click **Add Calculation** to add the expression to the cost function table.
   f. Repeat to add additional calculations to the cost function or click **Done** to exit the **Add/Edit Calculation** dialog box and return to **Setup Optimization**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. To modify the **Solution** on which the calculation is based, click in the **Solution** column and select the solution from which the cost function is to be extracted from the drop-down menu.
4. To edit the calculation on which to base the cost function goal, select **Edit** from the drop-down menu.
5. In the **Condition** text box, click one of the following conditions from the drop-down menu:

<=	Less than or equal to
=	Equal to
>=	Greater than or equal to
Minimize	Reduce the cost function to a minimum value
Maximize	Identify a maximized condition

6. In the **Goal** text box, type the value of the solution quantity that you want to be achieved during the optimization analysis. If the solution quantity is a complex calculation, the goal value must be complex; two goal values must be specified. The **Minimize** and **Maximize** options do not require you to specify a **Goal** value.

   When Minimize is used as an optimization condition, the value of calculation is used as the cost (there is no target value to compare to). For maximize, the negative of calculation value is used as cost.

7. Optionally, if you have multiple goals and want to assign higher or lower priority to a goal, type a different value for the goal's weight in the **Weight** text box. The goal with the greater weight is given more importance. If the goal is a complex value, the weight value must be complex; two weight values must be specified. The weight value cannot be variable dependent.

   > **Note:**
   >
   > Click the **Edit Goal/Weight** button to open the *Edit Goal Value/Weight* dialog box, where you can modify weights for all goals simultaneously and set the **Goal Values** to expressions.

8. Specify other options (such as acceptable cost, noise, and number of passes), and then click **OK**.

The optimization stops when the solution quantity meets the acceptable cost criterion.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specifying a Solution Quantity for a Cost Function Goal

When setting up a cost function, you must identify the solution quantity on which to base each goal. Solution quantities are specified by mathematical expressions that are composed of basic quantities, such as matrix parameters, and output variables.

1. Add a row (a goal) to the cost function table:
    a. Under the **Goals** tab of the *Setup Optimization* dialog box, click **Add**.

       A new row is added to the *Cost Function* table.

    b. In the **Solution** column, click the solution from which the cost function is to be extracted.
2. In the **Solution** text box, click the solution from which the solution quantity is to be extracted.
3. In the **Calculation** text box, specify the solution quantity in one of the following ways:
    - If you know the syntax of the mathematical expression or the output variable's name, type it in the **Calculation** text box.
    - If you want to create an output variable that represents the solution quantity, do the following:
        a. Click **Edit Calculation**.

           The *Output Variables* dialog box appears.

        b. Add the expression you want to evaluate, click **Done**.
        c. Click **Done** to close the *Output Variables* dialog box.

           In the *Setup Optimization* dialog box, the most recently created output variable appears in the *Calculation* text box.

        d. To specify a different defined output variable, click the **Calculation** text box. It becomes a drop-down menu that displays all of the defined output variables. Click an output variable from the drop-down menu.

Setting the Calculation Range of a Cost Function Goal

The calculation range is the range within which you want a cost function goal to be calculated. It can be a single value or a range of values, depending on the solution or solution quantity selected for the goal.

1. Under the **Goals** tab in the *Setup Optimization* dialog box, click **Edit Cal. Range**.
2. In the **Variable** drop-down menu, click a variable.

   If you chose to solve a parametric setup during the optimization analysis, the variables swept in that parametric setup are available in the **Variable** drop-down menu. If you

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

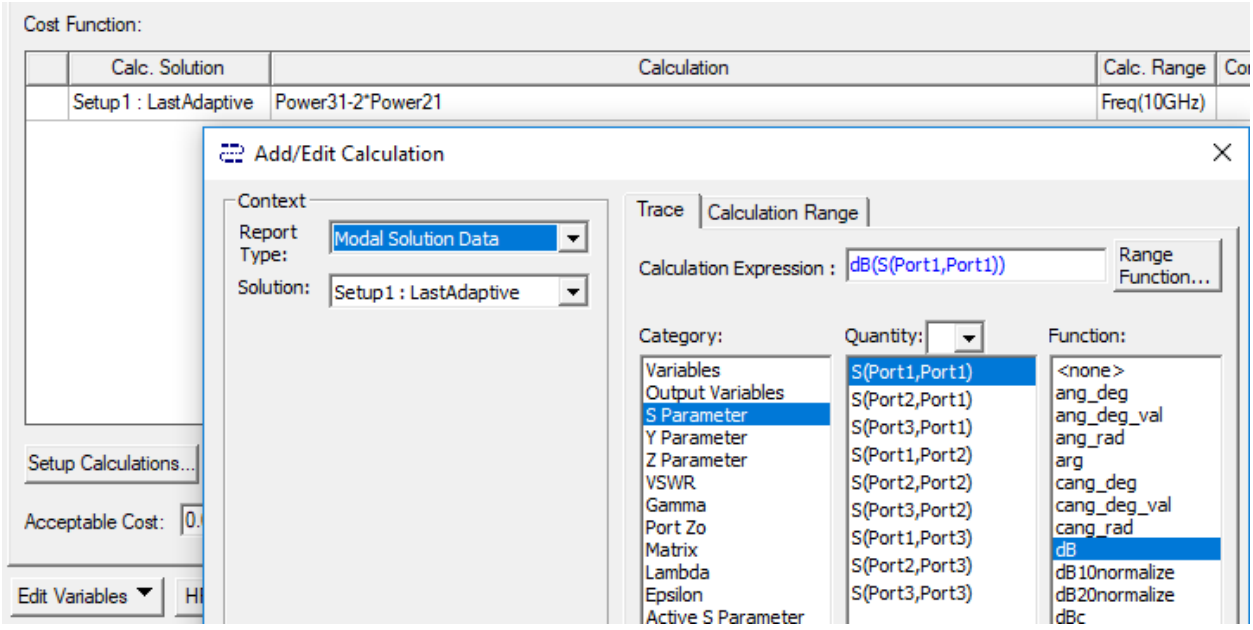
sweep a variable in the parametric setup that is also being optimized, that variable is excluded from the optimization.

Other examples of available variables include frequency, if the solution quantity is an S-parameter quantity, and phi or theta, if the solution quantity is a radiated field quantity.

3. After you select a variable from the **Variable** drop-down menu, you can select a range of values for the calculation range as follows:
    a. Select **Range**.
    b. In the **Start** text box, type the starting value of the range.
    c. In the **Stop** text box, type the final value of the range.
4. To select a single value for the calculation range:
    a. Select **Single Value**.
    b. In the **Value** text box, type the value of the variable at which the cost function goal is to be extracted.
5. Click **Update** and then click **OK**.

Adding or Editing a Cost Function Calculation

The **Add/Edit Calculation** dialog box allows you to define the mathematical equation for one or multiple cost functions. It represents the calculation to be performed on the optimization variables to compare to the goal values.



To set up a calculation for a cost function:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Context** section of the dialog box:
   - Select the **Report Type** with a drop-down selection list containing the available types for this design.
   - Select the **Solution** from the drop-down selection list. This lists the available setups and sweeps. As a minimum, the **LastAdaptive** solution is available.
   - Select the Geometry from the drop-down selection list or select none (the default). This modifies the list of quantities available to the ones that apply to the specific geometry.
2. The **Output Variables** button opens the Output Variables dialog box allowing you to create special output variables to be used in the cost function.
3. The **Calculated Expression** field in the **Trace** tab is used to enter the equation to be used for the cost function. To enter an expression, you may type it directly into the field or use the **Category**, **Quantity**, and **Function** lists as follows:
   - Select the **Category**, these depend on the Solution type and the design. This lets you specify the category of information to be used in the cost function.
   - Select a **Quantity** from the list. Available quantities depend upon the Solution type, as well as the Geometry and Category selection. Selecting a Quantity automatically enters it into the Calculated Expression field.
   - Select a **Function** to apply to the value in the calculated expression.

     > **Note:**
     >
     > Because Optimetrics works in SI values, you should use ang_deg_val or cang_deg_val functions which return unitless numbers in degrees rather than ang_deg or cang_deg, which return angular values in degree units but evaluate in radians in Optimetrics expressions. For example, you could Optimize successfully by changing the expression from abs(**cang_deg**(S(Port2,Port1))) to abs(**cang_deg_val**(S(Port2,Port1))). See the table of available functions and definitions.

   - For swept variables, the Range Function button opens the **Set Range Function** dialog to apply functions to the expression that apply over the sweep range.
4. The **Calculation Range** tab applies to swept variables and allows you to specify the range of the sweep over which to apply the calculation.
5. When the desired **Calculated Expression** has been obtained, click **Add Calculation** to add the entry to the cost function table. You may add multiple entries to the table simply by changing the **Calculated Expression** and using the **Add Calculation** button.
6. To update or edit a selected cost function, enter the desired Calculated Expression and click the **Update Calculation** button.
7. Click **Done** to return to the **Setup Calculations** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting a Goal Value

A goal is the value you want a solution quantity to reach during an optimization analysis. It can be a real value or a complex value. If the solution quantity is a complex calculation, the goal value must be complex. You can type the goal value in the **Goal** text box. Alternatively, you can use the *Edit Goal/Value Weight* dialog box to specify the goal value as a single value, a mathematical expression, or a value dependent on a variable such as frequency.

Specifying a Single Goal Value

1. Under the **Goals** tab in the *Setup Optimization* dialog box, click **Edit Goal/Weight**.

   The *Edit Goal/Weight* dialog box appears.

2. Under the **Goal Value** tab, click **Simple Numeric Value** from the **Type** list.
3. If the goal value is complex, click **real/imag** in the drop-down menu to the right if you want to specify the real and imaginary parts of the goal value.

   Alternatively, click **mag/ang** if you want to specify the magnitude and angle of the goal value.

4. Type the goal value in the **Goal Value** table.

   If the goal value is complex, type both parts of the goal value in the text box below the **Goal Value** heading. For example, type **1, 1** to specify the real part of the goal value as 1 and the imaginary part as 1.

   If the goal value is real, type a real goal value in the text box below the **Goal Value** heading.

5. Click **OK**.

   The goal value you specified appears in the **Goal** text box.

Specifying an Expression as a Goal Value

1. Under the **Goals** tab in the *Setup Optimization* dialog box, click **Edit Goal/Weight**.

   The *Edit Goal/Weight* dialog box appears.

2. Under the **Goal Value** tab, click **Expression** from the **Type** list.
3. If you know the syntax of the mathematical expression or the existing output variable's name, type it in the text box below the **Goal Value** heading.

   Alternatively, if you want to create an output variable that represents the goal value, do the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Click **Edit Expression**.

   The *Output Variables* dialog box appears.

b. Add the expression you want to be the goal value, and then click **Done**.

   The most recently created output variable is entered in the text box below the **Goal Value** heading.

4. Click **OK**.

   The goal value you specified appears in the **Goal** text box.

Specifying a Variable-Dependent Goal Value

1. Under the **Goals** tab in the *Setup Optimization* dialog box, click **Edit Goal/Weight**.

   The *Edit Goal/Weight* dialog box appears.

2. Under the **Goal Value** tab, click **Variable Dependent** from the **Type** list.
3. Click a variable from the pull-down list to the left of the table.
4. Type the value of that variable in the first column of the table.

> **Warning:**
>
> Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

5. Type a corresponding goal value for that variable value in the text box below the **Goal Value** heading.
6. Click **Add** to add another row to the reference curve.
7. Repeat steps 4, 5, and 6 until you have specified the reference curve.
8. Click **OK**.

   The goal value is listed as being variable dependent in the **Goal** text box.

Goal Weight

If an optimization setup has a cost function made up of multiple goals, you can assign a different weight to each goal. The goal with the greater weight is given more importance during the cost calculation.

The error function value is a weighted sum of the sub-goal errors. Each sub-goal, at each frequency at which it is evaluated, gives rise to a (positive) error value that represents the discrepancy between the simulated response and the goal value limit. If the response satisfies

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the goal value limit, then the error value is 0. Otherwise, the error value depends on the differences between the simulated response and the respective goal limit. The error function may be defined as follows:

$$\sum_{j}^{G} \frac{W_j}{N_j} \cdot \sum_{i}^{N_j} e_i$$

where

- G is the number of sub-goals.
- W_j is the weight factor associated with the j^{th} sub-goal.
- N_j is the number of frequencies for the j^{th} sub-goal.
- e_i is the error contribution from the j^{th} sub-goal at the i^{th} frequency.

The value of e_i is determined by the band characteristics, target value, and the simulated response value. The choices for band characteristics are <=, =, and >=.

Band Characteristics (Condition)	**e_i evaluation where s_i is the simulated response and g_i is the desired limit.**
<=	$e_i = \begin{cases} 0 & s_i \le g_i \\ s_i - g_i & s_i > g_i \end{cases}$
=	$e_i = \lvert s_i - g_i \rvert$
>=	$e_i = \begin{cases} 0 & s_i \ge g_i \\ g_i - s_i & s_i < g_i \end{cases}$

If the total error value is within the acceptable cost, the optimization stops.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modifying the Starting Variable Value for Optimization

A variable's starting value is the first value to be solved during the optimization analysis. Optimetrics automatically sets the starting value of a variable to be the current value set for the nominal design. You can modify this value for each optimization setup.

> **Note:**
>
> If you choose to solve a parametric setup before an optimization analysis, a variable's starting value is ignored if a more appropriate starting value is calculated for it during the parametric analysis.

1. In the *Setup Optimization* dialog box, click the **Variables** tab.

   All of the variables that were selected for the optimization analysis are listed.

2. Type a new value in the **Starting Value** text box for the value you want to override, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered is used for this optimization analysis, and the current value set for the nominal model is ignored.

   - Alternatively, you can select the **Override** option first, and then type a new variable value in the **Starting Value** text box.

3. Optionally, click a new unit system in one of the **Units** text boxes.

   > **Note:**
   >
   > To revert to the default starting value, clear the **Override** check box.

Setting the Min. and Max. Variable Values for Optimization

For every optimization setup, Optimetrics automatically sets the minimum and maximum values it will consider for a variable being optimized. Optimetrics sets a variable's minimum value equal to approximately 50% of its starting value. (The starting value is the variable's current value set for the nominal design.) Optimetrics sets the variable's maximum value equal to approximately 150% of the starting value. During the optimization analysis, variable values that lie outside of this range are not considered.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Warning:

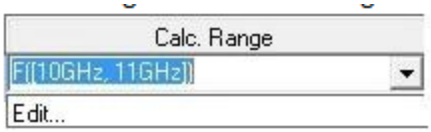
Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any Optimetrics analysis.

Edit Calculation Range

Note:

Functionality featured in the example(s) in this section applies to multiple design types.

In the *Setup Optimization* dialog box, you can enter the Calc. Range Sweep Min/Max by directly editing the Calc. Range field or by accessing an *Edit Calculation Range* dialog box.



The edit field accepts the following forms of text:

- sweep that allows your to select different discrete values:

  discrete values, for example, F(10GHz, 11GHz)

  min/max range, for example, F([10GHz, 11GHz])

- editable sweep, which allows you to customize values (that is a sweep that has an enabled "edited" radio button in sweep selection dialog):

  The min/max is used on top of selected values. For example, if you use the sweep dialog and choose "0 deg, 60 deg, 180 deg, 240 deg", then [60deg, 240deg] will select values "60 deg, 180 deg, 240 deg".

- sweep that uses a full range:

  all values, for example, Time(All)

  min/max range, for example, Time([1ms, 2ms]) HFSS/MAXWELL/TWINBUILDER

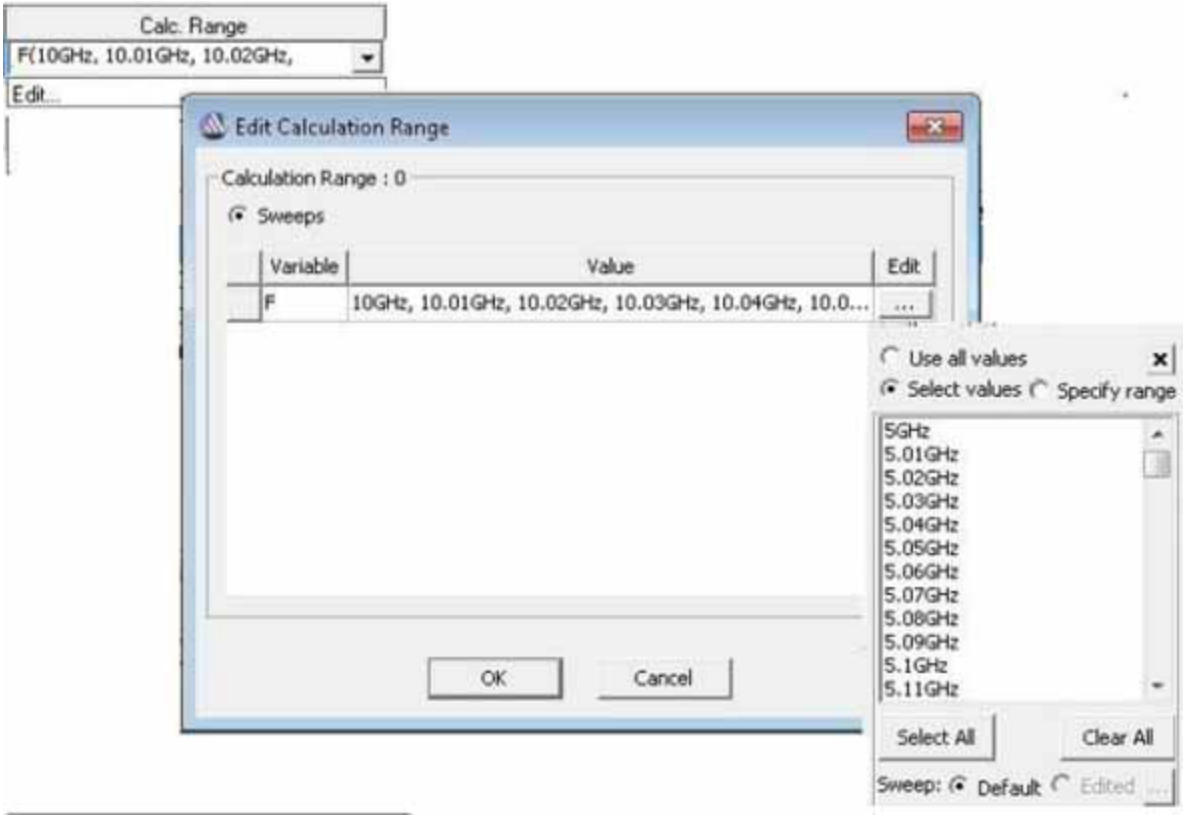
- You solve 1 to 20 GHz step .1 and specify F[10.381GHz, 11.381GHz], it is equivalent to selecting values between 10.4GHz and 11.3GHz.
- You can specify multiple sweep values by separating those with comma ","

  For example, F(1GHz), cap(1pf, 1.2pf)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

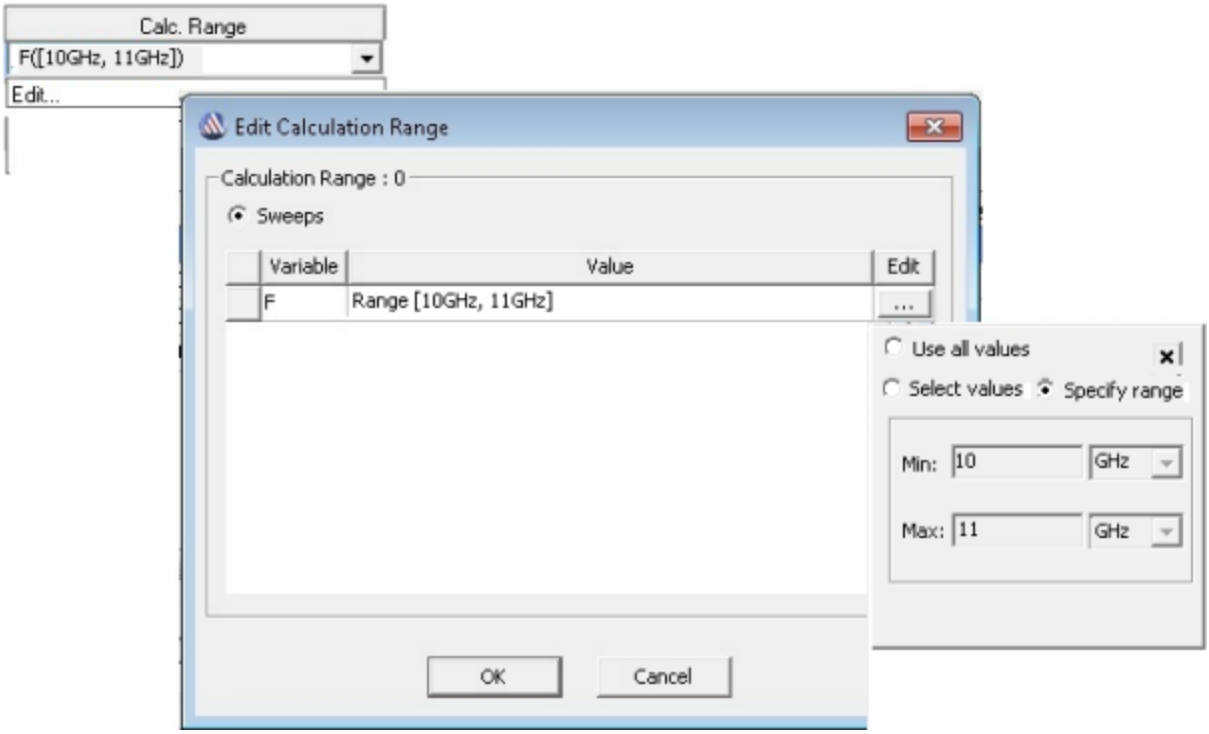
For example, Distance(All), Freq([1ghz,2ghz]), Phase(0 deg)

If you click **Edit..** on the menu, you see the **Edit Calculation Range** dialog box. Click on the ellipsis [...] button to select radio buttons for Use all values, Select Values or Specify range.



This example shows that when you specify a range, how the range appears in the Calc. Range field.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You could also enter the range directly in the Cal. Range field.

Overriding the Min. and Max. Variable Values for a Single Optimization Setup

1. In the *Setup Optimization* dialog box, click the **Variables** tab.

   All of the variables that were selected for optimization analysis are listed.

2. Type a new value in the **Min** or **Max** text box for the value you want to override, and then press **Enter**.

   The *Override* option is now selected, indicating that the value you entered is used for this optimization analysis; the variable's current **Min** or **Max** value in the nominal design is ignored.

   - Alternatively, you can select the **Override** option first, and then type a new value in the **Min** or **Max** text box.

3. Optionally, click a new unit system in one of the **Units** text boxes.

To revert to the default minimum and maximum values, clear the **Override** option.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Changing the Min. and Max. Variable Values for Every Optimization Setup

1. Make sure that the variable's minimum and maximum values are not being overridden in any single optimization setup.
2. If the variable is a design variable, do the following: Click **HFSS**> **Design Properties**.

   If the variable is a project variable, do the following: Click **Project**>**Project Variables**.

   The *Properties* dialog box appears.
3. Select **Optimization**.
4. Type a new value in the **Min** or **Max** text box for the value you want to override, and then press **Enter**.
5. Click **OK**.

   When Optimetrics solves an optimization setup, it does not consider variable values that lie outside of this range.

Step Size

To make the search for the minimum cost value reasonable, the search algorithm is limited in two ways. First, you do not want the optimizer to continue the search if the step size becomes irrelevant or small. This limitation impacts the accuracy of the final optimum. Second, in some cases you do not want the optimizer to take large steps either. In case the cost function is suspected to possess large variations in a relatively small vicinity of the design space, large steps may result in too many trial steps, which do not improve the cost value. In these cases, it is safer to proceed with limited size steps and have more frequent improvements.

For these two limitations, the optimizer uses two independent distance measures. Both are based on user-defined quantities: the minimum and maximum step limits for individual optimization variables. Since the particular step is in a general direction, these measures are combined together in order to derive the limitation for that particular direction.

The step vector between the i^{th} and $(i+1)^{th}$ iterate is as follows:

$$s_i = x_{i+1} - x_i$$

The natural distance measure is

$$\|s_i\| = \sqrt{s_i^T s_i}$$

which is the Euclidean norm.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A more general distance measure incorporates some "stretching" of the design space:

$$\|s_i\|_D = \sqrt{s_i^T D^T D s_i}$$

where the matrix *D* incorporates the linear operation of the stretching of design space. The simplest case is when the *D* matrix is diagonal, meaning that the design space is stretched along the orthogonal direction of the base vectors.

The optimizer stops the search if

$$\|s_i\|_{D_{min}} < 1$$

where D_{min} consists of diagonal elements equal to the inverse of the **Min. Step** value assigned to the corresponding optimization variable. Similarly, the optimizer truncates steps for which

$$\|s_i\|_{D_{max}} > 1$$

where D_{max} has diagonal elements equal to the inverse of **Max. Step** values of the corresponding optimization variables.

Setting the Min. and Max. Step Sizes

For the Quasi Newton and Pattern Search optimizers, the step size is the difference in a variable's value between one solved design variation and the next. The step size is determined when Optimetrics locates the next design variation that should be solved in an effort to meet the cost function.

1. In the *Setup Optimization* dialog box, click the **Variables** tab.
2. Optimetrics displays **Min Step** and **Max Step** columns, with default values for each variable to be optimized.
3. In the **Min Step** text box, type the minimum step size value. Optionally, modify the unit system in the **Units** text box.
4. In the **Max Step** text box, type the maximum step size value. Optionally, modify the unit system in the **Units** text box.

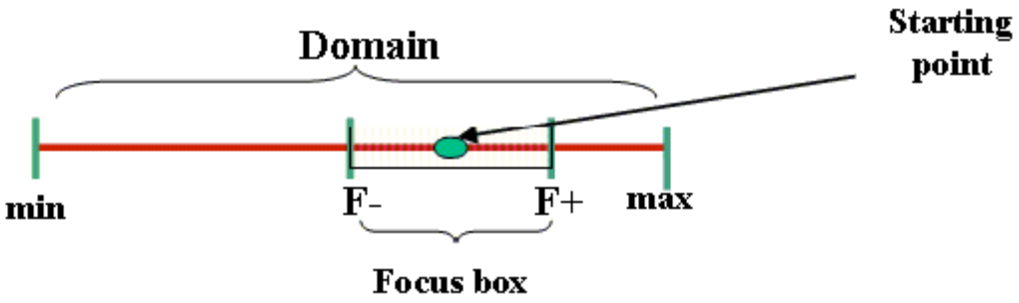
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **OK**.

> **Tip:**
>
> A value of zero is recommended for the minimum step size.

Setting the Min and Max Focus

For the SNLP, SMINLP and Genetic Algorithm optimizers, the min focus and max focus criteria allow you to specify a sub-range of parameter values where the optimizer should look when performing the optimization. This focus box is where you suspect the optimal solution will be, so it is a hint for the optimizer.



- The domain limits the search. The domain = physical limits.
- The focus box does not limit the search. Rather, the Focus box = an initial guess of optimum search domain. The starting point is the center of the focus box, but the search does extend beyond the box.
- This focus must be inside the domain limits. Consequently, it has to be equal or smaller size. An error message is generated if you specify a focus outside the domain.
- The focus box must be at least one hundredth of the domain size. Otherwise, an error message is sent.

Equalizing the Influence of Different Optimization Variables

The optimizer seeks optimal values for the optimization variables. These variables are usually quantities with specified units. The change in one variable could be measured in [mm] and the change in other variable could be measured in [mA]. Instead of those units, the optimizer uses internal abstract units, so that a change in one variable changes the design behavior about as much as the same change in another variable, where changes are measured in the respective internal abstract units. When you define the focus box, the unit of the abstract internal unit is defined as the difference of the upper and lower focus limits. This way you can use the focus box to equalize the influence of different optimization variables on the design behavior.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Min and Max Focus Values

1. In the *Setup Optimization* dialog box, click the **Variables** tab.
2. Optimetrics displays **Min. Focus** and **Max. Focus** columns, with default values for each variable to be optimized.

   If you do not have an initial guess based on your knowledge of the problem, make the focus box equal to the domain; that is, the physical limits. This tells SNLP to search the entire decision space.

   - In the **Min. Focus** text box, type the minimum value of the focus range. Optionally, modify the unit system in the **Units** text box.
   - In the **Max. Focus** text box, type the maximum value of the focus range. Optionally, modify the unit system in the **Units** text box.
3. Click **OK**.

Solving a Parametric Setup Before an Optimization

Solving a parametric setup before an optimization setup is useful for guiding Optimetrics during an optimization.

To solve a parametric setup before an optimization setup:

1. In the *Setup Optimization* dialog box, click the **General** tab.
2. In the **Parametric Analysis** drop-down menu, click the parametric setup you want Optimetrics to solve before optimization.

   > **Note:**
   >
   > The parametric setup must include sweep definitions for the variables you are optimizing.

3. Select **Solve the parametric sweep before optimization**.

If the parametric setup has not yet been solved, Optimetrics solves it. Optimetrics uses the cost value evaluated at each parametric design variation to determine the next step in the optimization analysis. This enables you to guide the direction in which the optimizer searches for the optimal design variation.

Solving a Parametric Setup During an Optimization

Solving a parametric setup during an optimization analysis is useful when you want Optimetrics to solve every design variation specified in the parametric setup at each optimization iteration. A cost function goal could then depend on the value of the variable swept in the parametric setup.

To solve a parametric setup during an optimization analysis:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the *Setup Optimization* dialog box, click the **General** tab.
2. In the **Parametric Analysis** drop-down menu, click the parametric setup you want Optimetrics to solve during an optimization.
3. Select **Solve the parametric sweep during optimization**.
4. Optionally, you can adjust the sweep values to be used during the optimization.
    a. Click on the **Goal** tab, click **Setup Calculations** to specify a calculation.

       The *Add/Edit Calculation* dialog box is displayed.

    b. Click the **Calculation Range** tab.
    c. Click **Edit** for the sweep to be modified.
    d. In the pop-up dialog box, select the sweep values to use.
    e. Close the pup-up dialog box. Click **Done** to close the *Add/Edit Calculation*dialog box.

Automatically Updating a Variable's Value after Optimization

When Optimetrics finds an optimal variable value by solving an optimization setup, it can automatically update that variable's current value set for the nominal model to the optimal value.

1. In the *Setup Optimization* dialog box, click the **General** tab.
2. Select **Update design parameters' values after optimization**.

   When optimization is complete, the current variable value for each optimized variable is changed to the optimal value.

Changing the Cost Function Norm

You can select the norm to be used in the calculation of the cost goal.

1. In the *Setup Optimization* dialog box, click the **Goals** tab.
2. Select **Show Advanced Options**.
3. Select a norm from the drop-down menu in the **Cost Function Norm Type** field. The options are **L1**, **L2**, and **Maximum**. **L2** is the default.

Explanation of L1, L2, and Max Norms in Optimization

When you set multiple goals for an optimization, the question arises as to what is actually going to drive the optimizer which is not a multi-objective one. The cost function will have a lot to do with it. The following discussion explains how the cost function is put together when there are multiple goals.

The general goal setting structure in Optimetrics is a logical sentence with the format:

$Calculation_{(i)}\ Condition_{(i)}\ Goal_{(i)}\ Weight_{(i)}$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The cost function that the optimizer uses is built based on the norm setting as long as there are multiple goals and none of those use the "minimize" or "maximize" conditions. Thus, in this case the error associated with each individual goal (weighted) is combined in a way that is specific for each norm type chosen.

For **L1** norm, the actual cost function uses the sum of absolute weighted values of the individual goal errors:

$$Cost = \sum_{1}^{N} \left| w_i \varepsilon_i \right|$$

For **L2** norm, the cost function computes the weighted sum of the squares of individual goal errors.

$$Cost = \sum_{1}^{N} w_i \varepsilon_i^2$$

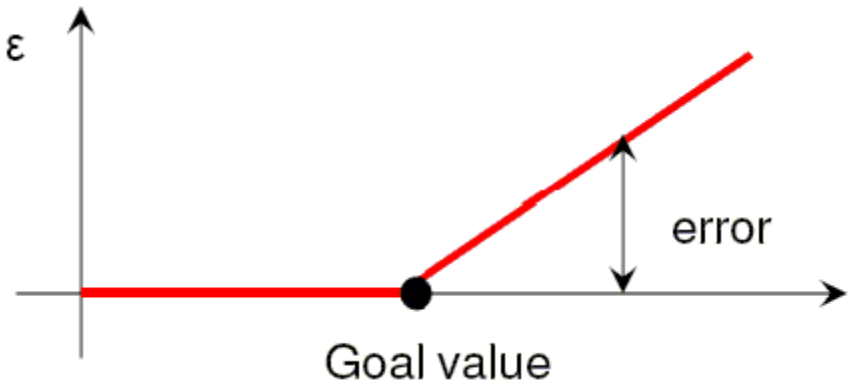
For the **Maximum** norm, the cost function uses the maximum among all the weighted goal errors, which means that cost is always less than zero:

$$Cost = \underset{1}{\overset{N}{Max}} W_i \cdot \varepsilon_i$$

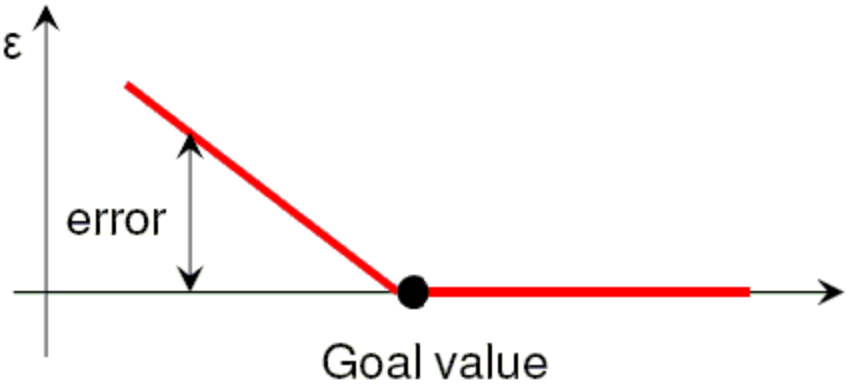
For all the above situations, N is the number of individual goals $w_i \varepsilon_i$ are individual weighting factors and residual error respectively. A minimization of the cost function is performed during optimization since it makes sense to minimize the error in the sense of the chosen norm type.

The graphical representation of the error is possible and depends upon the actual condition being used. If a "<" condition is used, the error can be represented as below:

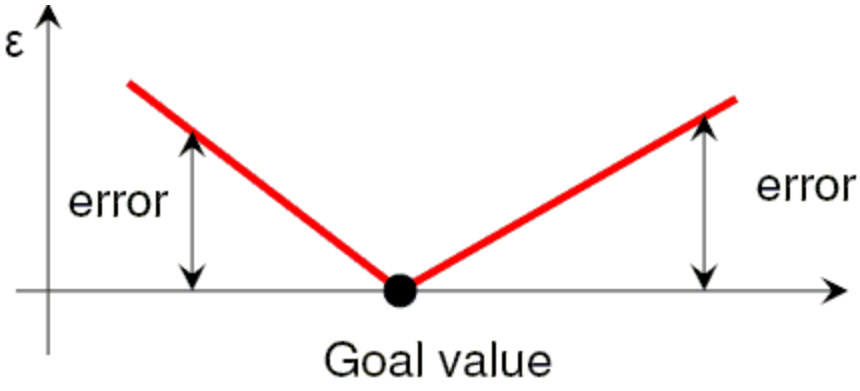
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If a ">" condition is used, the error can be represented as below:



If a "=" condition is used, the error is double-sided and can be represented as below:



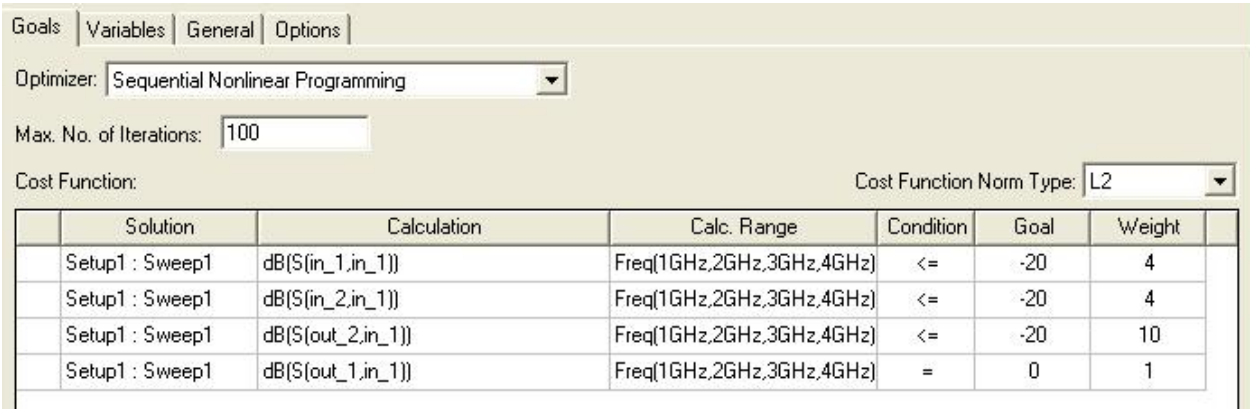
The norm type doesn't impact goal setting that use as condition the "minimize" or "maximize" scenarios. Note that when using "minimize" or "maximize" settings for the condition there should be a single goal setting which in this case coincides with the cost function.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example of a More Complex Cost Function

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

As an example of a more sophisticated cost function, consider the figure. It belongs to a connector simulated in HFSS with more than four ports.



	Solution	Calculation	Calc. Range	Condition	Goal	Weight
	Setup1 : Sweep1	dB(S(in_1,in_1))	Freq(1GHz,2GHz,3GHz,4GHz)	<=	-20	4
	Setup1 : Sweep1	dB(S(in_2,in_1))	Freq(1GHz,2GHz,3GHz,4GHz)	<=	-20	4
	Setup1 : Sweep1	dB(S(out_2,in_1))	Freq(1GHz,2GHz,3GHz,4GHz)	<=	-20	10
	Setup1 : Sweep1	dB(S(out_1,in_1))	Freq(1GHz,2GHz,3GHz,4GHz)	=	0	1

The cost function given here concentrates only on a signal sent into port in_1. Suppose the specifications to be met are: reflection, backward cross talk and forward cross talk all smaller than or equal to -20 dB, of which the forward cross talk is the most important.

The first three entries in the cost function enforce those specifications, with the weight for the forward cross talk being a larger number than the other weights. The actual values for the weights are somewhat arbitrary and serve as examples only. For this cost function, as long as specifications are not met, the optimizer puts the most effort in getting the forward cross talk close to its specification. Once the three specifications have been satisfied, their contributions to the cost function become zero, and only the fourth entry remains. Remember that the connector has more than four ports, so satisfying the given specs does not guarantee maximum transmission.

The fourth line tries to maximize the transmission by asking for S(out_1, in_1) to be 0 dB. That will never be reached, but its presence forces the optimizer to improve the connector a bit beyond the specifications.

The cost function norm type specifies how the four lines are combined into one cost function with one value. With L1 and L2, all four contribute simultaneously, rather than only the largest of the four at any one time.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Advanced Genetic Algorithm Optimizer Options

The Genetic Algorithm (GA) search for Optimization analysis is an iterative process that goes through a number of generations. In each generation some new individuals (Children / Number of Individuals) are created and the so grown population participates in a selection (natural-selection) process that in turn reduces the size of the population to a desired level (Next Generation / Number of Individuals).

If you select the Genetic Algorithm for an Optimization analysis, a **Setup** button is enabled on the **Setup Optimization** page.

1. Click **Setup** to open the *Advanced Genetic Algorithm Optimizer Options* dialog box.
2. Select the Stopping Criteria. Any of the three following, or any combination of these can be selected.
    - **Maximum number of generations**. If checked, this enables a value field.
    - **Elapsed time**. If checked, this enables a drop-down menu with times ranging from five minutes to two weeks.
    - **Slow convergence**.
3. Specify the Parents.

    The first step toward mating is a selection process that determines the participating individuals. Potential parents are selected from the Current Generation. This is a set of individuals that is always a subset of the current generation.

    - **Number of individuals** value field -- specify the number of parents for the optimizer to use. You can set the Number of Individuals to less than or equal to the size of the "Current Generation". One reason to consider fewer parents than the possible maximum is to steer the GA toward improvement by selecting the better portion of the current generation to be able to mate.
    - **Roulette selection** check box -- if checked, this enables the **Selection pressure** value field. This number defines how many times more probable is the selection of the best individual over the worst individual in an elementary spin of the roulette wheel.
4. Specify the Mating pool.

    The Mating pool is created by selecting randomly from the parents, but with each selection, the parent gets "cloned" so it can be selected again and again.

    - **Number of individuals** field -- specify the number individuals to include in the mating pool.
    - **Reproduction setup**-- this button opens the *Genetic Algorithm Optimizer Reproduction Setup* dialog box.
5. Click **Reproduction setup** for the dialog box to specify the Crossover setup, and the Mutation setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The crossover and mutation operator have different roles: *Crossover* mixes "features" of the parents in a new combination, while *mutation* slightly alters the "features" of the individuals. Both need to be present in a GA. The crossover is a way to discover new combinations while the mutation acts as a local search or fine-tuning step. Mutation also keeps diversity in a population, which is a must for GA.

The crossover operator has two steps. It first alters the variable values of the parents according to a distribution. This tends to produce one child that looks a lot like one parent, and one child that looks a lot like the other parent. Next, some of the variable values of the two children can be exchanged in order to achieve more variation.

For crossover there are four possible parameters.

a. **Individual Crossover Probability** determines, for each pair in the mating pool, the probability that their features will be mixed. Usually, this probability should be close or equal to one. If you set it to less than one, some parents will produce two children which are exact clones of the parents. This means that some children inherit all the features of their parents unchanged.
b. Parents often have multiple variables. If the parent is a candidate for mixing, the **Variable Crossover Probability** determines, for each variable, the probability of mixing. This is usually set high to ensure that most or all variables mix.
c. **Variable Exchange Probability**: After the slight change in the variable values has been made, the crossover operation is also able to exchange the values of the variables between the two children that are being constructed. The Variable Exchange Probability governs the likelihood of exchange of any variable.
d. **Mu** is a general parameter defining the sharpness of the distribution that might be used for the **Variable Crossover Probability**. Mu should be greater than one. There is no theoretical upper limit, but we recommend not exceeding 30.

6. Select one of the four **Crossover types** from the drop-down menu.

   The crossover type selected affects the options available.

Uniform	Individual crossover probability Variable crossover probability
One point	Individual crossover probability
Two point	Individual crossover probability
Simulated binary crossover	Individual crossover probability Variable crossover

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

probability
Variable exchange
probability
Mu

7. Select the **Mutation type** – this can be one of three types, which you select from a drop-down menu:
   - **Uniform Distribution**
   - **Gaussian Distribution**
   - **Polynomial Mutation**
8. For the selected mutation type, set the following parameters:
   - **Uniform Mutation Probability**: If this is more than zero (recommendation is to have still a small probability here), then there will be some children whose features are simply a completely random design (design variables randomly selected over the domain).
   - **Individual Mutation Probability** controls, for each child, the likelihood of a mild mutation.
   - **Variable Mutation Probability**. If the child will be mutated, this probability controls at the variable level the likelihood of a mutation of the variables.
   - **Standard Deviation** is the standard deviation of the selected distribution that is being used for the mutation and it is measured relatively to the optimization-domain.
9. When you have completed the Reproduction setup in the *Genetic Algorithm Optimizer Reproduction Setup* dialog box, click **OK** to close it and return to the *Advanced Genetic Algorithm Optimizer Options* dialog box.
10. In the *Advanced Genetic Algorithm Optimizer Options* dialog box, specify the children as a Number of Individuals.
11. Set the **Pareto Front** value.

    This is the number of the very best individuals (identified relative to the cost function) to keep for future generations.

12. Set the Next Generation parameters. The Next Generation is selected from the Parents, the children, and the Pareto front.
    - **Number of individuals** value field -- specify the number of individuals to survive to form the next generation for the optimizer to use.
    - **Roulette selection** check box -- if checked, this enables the **Selection pressure** value field. This number defines how many times more probable is the selection of the best individual over the worst individual in an elementary spin of the roulette wheel.
13. Click **OK** to accept the settings for the Genetic Algorithm and to close the dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sensitivity Analysis Overview

During a sensitivity analysis, Optimetrics explores the vicinity of the design point to determine the sensitivity of the design to small changes in variables. The variables and their attributes define the design point, the problem around which the sensitivity analysis is performed.

When Optimetrics performs a sensitivity analysis, its goal is to calculate the second-order regression polynomials for all of the design's output parameters. The algorithm first determines an appropriate interval for each variable. The intervals are further sub-divided according to the available number of iterations and variables. If the primary output is not used, the specified initial displacement values define those intervals.

When all of the design calculations are complete, the second-order polynomials are fitted for all the output parameters. Optimetrics then reports the following quantities:

- Regression value at the current variable value
- First derivative of the regression
- Second derivative of the regression

> **Note:** Sensitivity Analysis has been placed in legacy mode because it is no longer under active development. Go to **Tools** > **Options** > **General Options** > **Miscellaneous** and toggle the **Enable Legacy Optimetrics Tools** check box to show or hide these legacy tools.

Selecting a Primary Output

During a sensitivity analysis, the design variations that Optimetrics selects to solve are close to the design point, but not so close that numerical noise (from the finite element mesh) affects the analysis. The algorithm that Optimetrics uses to determine the design variations to solve must be based on only one output parameter and that output parameter's numerical noise. Therefore, if you have defined more than one output parameter, be sure to select **Primary Output** for the output variable on which you want the selection of design variations to be based.

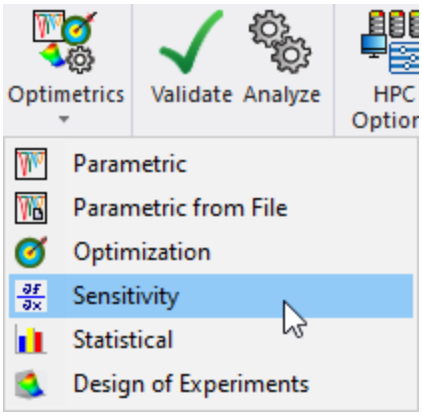
Setting Up a Sensitivity Analysis

Following is the general procedure for setting up a sensitivity analysis. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes. You can create a sensitivity setup before defining variables but all variables must be defined before you start the sensitivity analysis.

1. Before a variable can be included in a sensitivity analysis, you must specify that you intend for it to be used during a sensitivity analysis in the **Design Properties** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **HFSS** > **Optimetrics Analysis** > **Add Sensitivity** . You can also select the **Simulation** tab in the ribbon, and select **Parametric** from the drop-down menu under the Optimetrics icon:



The **Setup Sensitivity Analysis** dialog box appears.

3. Under the **Calculations** tab, type the maximum number of iterations per variable value that you want the software to perform in the **Max. No. of Iterations/Sensitivity Variable** text box.
4. Set up an output parameter calculation, and select a Primary Output.
5. Specify the value of the design point at which the sensitivity analysis should stop in the **Approximate Error in Primary Output** text box.
6. In the **Variables** tab, specify the **Min/Max** values for variables included in the optimization, and the **Initial Displacement (Initial Disp.)** for the analysis.

   You may also override the variable starting values by clicking the **Override** check box and entering the desired value in the **Starting Value** field.

7. In the **General** tab, specify whether Optimetrics should use the results of a previous parametric analysis or perform one as part of the optimization process.

   Checking the Optional **Worst Case Analysis** option does an extreme value analysis that focuses on the upper and lower boundaries of all the analyzed parameters. Some setup is required before performing Worst Case Analysis.

8. Under the **Options tab**, if you want to save the field solution data for every solved design variations in the optimization analysis, select **Save Fields And Mesh**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Do not select this option when requesting a large number of iterations as the data generated will be very large and the system may become slow due to the large I/O requirements.

You may also select **Copy geometrically equivalent meshes** to reuse the mesh when geometry changes are not required, for example when optimizing on a material property or source excitation. This will provide some speed improvement in the overall optimization process.

The following ***optional*** sensitivity analysis setup options can also be used:

- Modify the starting variable value.
- Modify the minimum and maximum values of variables that will be solved.
- Exclude variables from the sensitivity analysis.
- Set the initial displacement.
- Modify the values of fixed variables that are not being modified during the sensitivity analysis.
- Set linear constraints.
- Request that Optimetrics solve a parametric sweep **before** a sensitivity analysis.
- You can also request that Optimetrics solve a parametric sweep **during** a sensitivity analysis.

> **Note:**
>
> Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

Setting the Maximum Iterations Per Variable

The **Max. No. of Iterations/Sensitivity Variable** value is the maximum number of design variations that Optimetrics solves per variable during a sensitivity analysis. This value is a stopping criterion; if the maximum number of iterations has been completed, the sensitivity analysis stops. If the maximum number of iterations has not been completed, the sensitivity analysis continues by performing another iteration, that is, by solving another design variation. It performs iterations until the approximate error in primary output value is reached or until Optimetrics cannot proceed as a result of other sensitivity setup constraints, such as when it searches for a variable value that is larger than the maximum value.

To set the maximum number of iterations for a sensitivity analysis:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Under the **Calculations** tab of the *Setup Sensitivity Analysis* dialog box, type a value in the **Max. No. of Iterations/Sensitivity Variable** text box.

Setting Up an Output Parameter

Following is the general procedure for adding an output parameter to a sensitivity setup:

1. Under the **Calculations** tab of the *Setup Sensitivity Analysis* dialog box, click **Setup Calculations** to open the *Add/Edit Calculations* dialog box.
2. In the *Add/Edit Calculations* dialog box, set up output parameter calculations to be evaluated for sensitivity.
3. To modify the solution from which the output parameter is to be extracted, click in the **Solution** column and select from the options in the pop-up list.
4. You can modify the Calculation specified by clicking on the output parameter in the table and selecting **Edit**.
5. For output parameters based on swept variable, you must choose a single value in the **Calculation Range** at which to evaluate the output parameter.
6. If the output parameter is based on a swept variable, in the **Calculation Range** column, set the value of the variable at which the output parameter is to be computed.
7. If you have more than one output parameter, select **Primary Output** if you want Optimetrics to use the output parameter to base its selection of solved design variations.

> **Note:**
>
> During a sensitivity analysis, the design variations that Optimetrics selects to solve are close to the design point, but not so close that numerical noise (from the finite element mesh) affects the analysis. The algorithm that Optimetrics uses to determine the design variations to solve must be based on only one output parameter and that output parameter's numerical noise. If you have defined more than one output parameter, be sure to select **Primary Output** for the output variable on which you want the selection of design variations to be based.

Specifying a Solution Quantity for an Output Parameter

When setting up an output parameter, you must identify the solution quantity on which to base the output parameter. Solution quantities are specified by mathematical expressions that are composed of basic quantities, such as matrix parameters; and output variables.

The *Add/Edit Calculation* dialog box allows you to define the mathematical equation for one or multiple output parameters. To set up an output parameter:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Context** section of the dialog box:
   - Select the **Report Type** with a pull-down selection list containing the available types for this design.
   - Select the **Solution** from the drop down selection list. This lists the available setups and sweeps. As a minimum, the **LastAdaptive** solution is available.
   - Select the **Geometry** from the drop down selection list or select none (the default). This modifies the list of quantities available to the ones that apply to the specific geometry.
   - When selecting a geometry, you may also be required to specify a point within the geometry where the calculation is to be performed.
2. The **Output Variables** button opens the Output Variables dialog box allowing you to create special output variables to be used in the output parameter.
3. The **Calculation Expression** field in the **Trace** tab is used to enter the equation to be used for the output parameter. To enter an expression, you may type it directly into the field or use the **Category**, **Quantity**, and **Function** lists as follows:
   - Select the **Category**, these depend on the Solution type and the design. This lets you specify the category of information to be used in the output parameter.
   - Select a **Quantity** from the list. Available quantities depend upon the Solution type, as well as the Geometry and Category selection. Selecting a Quantity automatically enters it into the Calculation Expression field.
   - Select a **Function** to apply to the value in the calculated expression.
   - For swept variables, the **Range Function** button opens the **Set Range Function** dialog to apply functions to the expression that apply over the sweep range.
4. The **Calculation Range** tab applies to swept variables and allows you to specify the range of the sweep over which to apply the calculation.
5. When the desired **Calculation Expression** has been obtained, click **Add Calculation** to add the entry to the calculation table in the Setup Sensitivity Analysis dialog box. You may add multiple entries to the table simply by changing the **Calculation Expression** and using the **Add Calculation** button.
6. To update or edit a selected cost function, enter the desired Calculation Expression and click the **Update Calculation** button.
7. Click **Done** to return to the *Setup Sensitivity Analysis* dialog box.

> **Note:**
>
> The solution quantity you specify must be able to be evaluated to a single, real number.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Calculation Range of an Output Parameter

The calculation range of a solution quantity determines the intrinsic variable value at which the solution quantity is to be extracted. For a sensitivity setup, the calculation range must be a single value. If you specified that the solution quantity be extracted from a frequency sweep solution, by default, Optimetrics uses the starting frequency in the sweep. If you specified that the solution be extracted from the last adaptive solution, Optimetrics uses the adaptive frequency defined in the solution setup.

1. Under the **Calculations** tab of the *Setup Sensitivity Analysis* dialog box, click in the **Calculation Range** column of the table for the calculation to be modified.

   The *Edit Calculation Range* dialog box appears.

2. In the **table**, click **Edit** in the row to be modified.

   If you choose to solve a parametric setup during the sensitivity analysis, the variables swept in that parametric setup are available in the pop-up list dialog box. If you sweep a variable in the parametric setup that is also a sensitivity variable, that variable is excluded from the sensitivity analysis.

   Other examples of available variables include frequency, if you selected an S-parameter solution quantity; and phi or theta, if the solution quantity is a radiated field quantity.

3. Click on the value for the calculation range in the list and dismiss the pop-up dialog box.
4. Click **OK** in the *Edit Calculation Range* dialog box to accept the new value for the intrinsic variable, and return to the *Setup Sensitivity Analysis* dialog box.

Modifying the Starting Variable Value for Sensitivity Analysis

The design point of the sensitivity analysis is the starting value of the sensitivity variable and is usually the first variation to be solved. Optimetrics automatically sets the starting value of a variable to be the current value set for the nominal design. You can modify the design point for each sensitivity setup.

> **Warning:**
>
> Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

1. In the *Setup Sensitivity Analysis* dialog box, click the **Variables** tab.

   All of the variables that were selected for the sensitivity analysis are listed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Type a new value in the **Starting Value** text box for the value you want to override, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered is to be used for this sensitivity analysis; the current value set for the nominal model will be ignored.

   - Alternatively, you can select the **Override** option first, and then type a new variable value in the **Starting Value** text box.

3. Optionally, click a new unit system in one of the **Units** text boxes.

To revert to the default starting value, clear the **Override** option.

Setting the Min. and Max. Variable Values

For every sensitivity setup, Optimetrics automatically sets the minimum and maximum values that it will consider for a sensitivity variable. Optimetrics sets a variable's minimum value equal to approximately one-half its starting value. (The starting value is the variable's current value set for the nominal design.) Optimetrics sets the variable's maximum value equal to approximately 1.5 times the starting value. During sensitivity analysis, variable values outside this range are not considered.

> **Warning:**
>
> Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

Overriding the Min. and Max. Variable Values for a Single Sensitivity Setup

1. In the *Setup Sensitivity Analysis* dialog box, click the **Variables** tab.

   All of the variables that were selected for sensitivity analysis are listed.

2. Type a new value in the **Min** or **Max** text box for the value you want to override, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered is to be used for this sensitivity analysis; the variable's current **Min** or **Max** value set in the nominal design is ignored.

   - Alternatively, you can select the **Override** option first, and then type a new value in the **Min** or **Max** text box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Optionally, click a new unit system in one of the **Units** text boxes.

   To revert to the default minimum and maximum values, clear the **Override** option.

Changing the Min. and Max. Variable Values for Every Sensitivity Setup

1. Make sure the variable's minimum and maximum values are not being overridden in any sensitivity setup.
2. If the variable is a design variable, do the following: Click **HFSS**> **Design Properties**.

   If the variable is a project variable, do the following: Click **Project** > **Project Variables**.

   The *Properties* dialog box appears.
3. Select **Sensitivity**.
4. Type a new value in the **Min** or **Max** text box for the value you want to override, and then press **Enter**.

   When Optimetrics solves a sensitivity setup, it does not consider variable values that lie outside of this range.

Setting the Initial Displacement

The initial displacement is the difference in a variable's starting value and the next solved design variation. During the sensitivity analysis, Optimetrics does not consider an initial variable value that is greater than this step size away from the starting variable value.

1. In the *Setup Sensitivity Analysis* dialog box, click the **Variables** tab.
2. Optimetrics displays the **Initial Disp.** column, with default values for each sensitivity variable.
3. In the **Initial Disp.** text box, type the initial displacement value. Optionally, modify the unit system in the **Units** text box.

Solving a Parametric Setup before a Sensitivity Analysis

Solving a parametric setup before a sensitivity setup is useful for guiding Optimetrics in a sensitivity analysis.

To solve a parametric setup before a sensitivity setup:

1. In the *Setup Sensitivity Analysis* dialog box, click the **General** tab.
2. Click the parametric setup you want Optimetrics to solve before the sensitivity setup from the **Parametric Analysis** drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The parametric setup must include sweep definitions for the sensitivity variables.

3. Select **Solve the parametric sweep before analysis**.

If the parametric setup has not yet been solved, Optimetrics solves it. Optimetrics uses the results (of the solution calculation you requested under the **Goals** tab of the *Setup Sensitivity* dialog box) to determine the next design variation to solve for the sensitivity analysis.

Solving a Parametric Setup during a Sensitivity Analysis

Solving a parametric setup during a sensitivity analysis is useful when you want Optimetrics to solve every design variation in the parametric setup at each sensitivity analysis iteration. An output parameter goal could then depend on the value of the variable swept in the parametric setup.

To solve a parametric setup during a sensitivity analysis:

1. In the *Setup Sensitivity Analysis* dialog box, click the **General** tab.
2. Click the parametric setup you want Optimetrics to solve during the sensitivity analysis from the **Parametric Analysis** drop-down menu.
3. Select **Solve the parametric sweep during analysis**.

Performing Worst Case Analysis

Two popular worst case analysis techniques can be implemented: extreme value analysis and Monte Carlo analysis.

Some setup is required before performing worst case analysis. First, identify uncertainties in design and create a local or project variable for each of them. Second, determine the variation range of each variable (Min and Max) – its statistical distribution is optional. Third, determine a measurement of performance, especially for extreme value analysis.

1. **Extreme value analysis:**

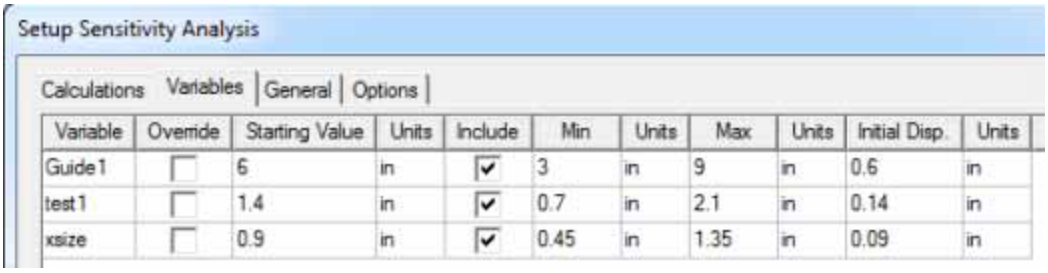
   This is one of the most popular methods to estimate worst-case performance. To start, a sensitivity analysis is performed. The results (sensitivities/first derivative) allow us to pick an extreme value (upper or lower bound) for each variable. The corresponding simulation result is used to predict upper and lower bound of performance. The assumption is that extreme performance is reached at boundary value (note that in certain cases, making such an assumption is not valid).

   To perform an extreme value analysis in Ansys Electronics Desktop, create a new analysis under **Optimetrics** > **Sensitivity**.

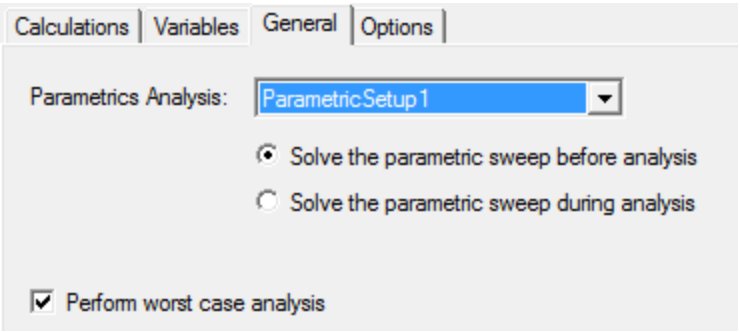
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Setup:

  First, include variables in a sensitivity analysis. To do so, go to your list of variables, under Sensitivity, specify Min and Max values of each variable, and check 'Include'. The following figure shows an example with three variables included.



  Second, follow **Setting Up a Sensitivity Analysis** in the help. During this procedure, set your performance measurement in **Calculations** tab; check all variables in **Variables** tab; check **Perform worst case analysis** in **General** tab.



> **Note:**
>
> Checking **Perform worst case analysis** calculates 1st derivatives for each variable. If we have three variables and for Var1 first derivative is negative, Var2 1st derivative is positive, Var3 1st derivative is positive, then for Worst Case Analysis, we request two more variations:

Var1@minimum, Var2@maximum, Var3@maximun

and Var1@maximum, Var2@minimum, Var3@minimum

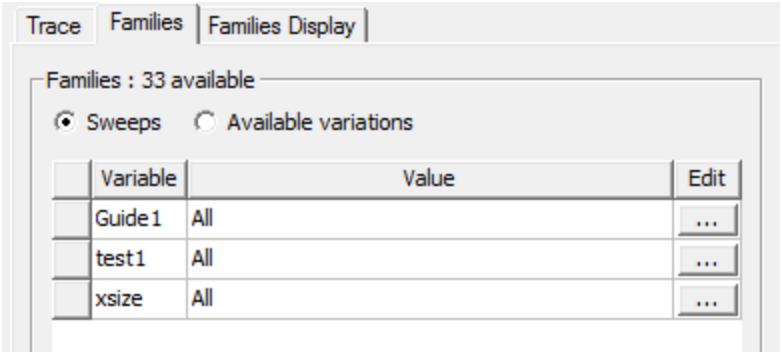
- After setup, analyze your Sensitivity Setup under Optimetrics.
- To view worst-case result (upper bound only):

  Create a Data Table report.

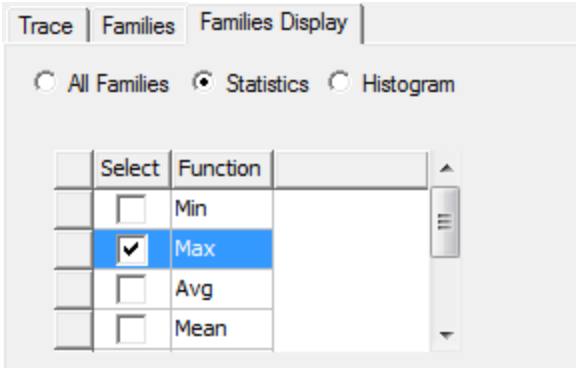
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Under the Context pane in the **Report** dialog box, select matching Solution and Optimetrics setup. Under the **Families** tab, change the setting to 'All' under 'Value' of each variables.



Under the **Families Display** tab, select **Statistics**and then check **Max**.



The associated data table shows the Max values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

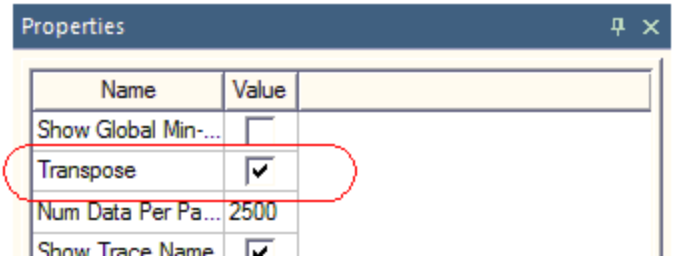
	Freq [GHz]	dB(S(1:1,1:1)) Setup1 : Sweep Max
1	3.000000	-0.000000
2	4.000000	-0.000000
3	5.000000	-0.105323
4	6.000000	-0.248322
5	7.000000	-0.267220
6	8.000000	-0.202039
7	9.000000	-0.298840

To see the corresponding variable values, select 'All Families' under the **Families Display** tab, and locate the max value to see value of variables.

	Freq [GHz]	dB(S(1:1,1:1)) Setup1 : Sweep Guide1='3in' test1='2.1in' xsize='0.45in'
1	3.000000	-0.545922
2	4.000000	-2.300003
3	5.000000	-10.634331
4	6.000000	-4.586505
5	7.000000	-5.950253
6	8.000000	-9.019574
7	9.000000	-3.210097

Tip:

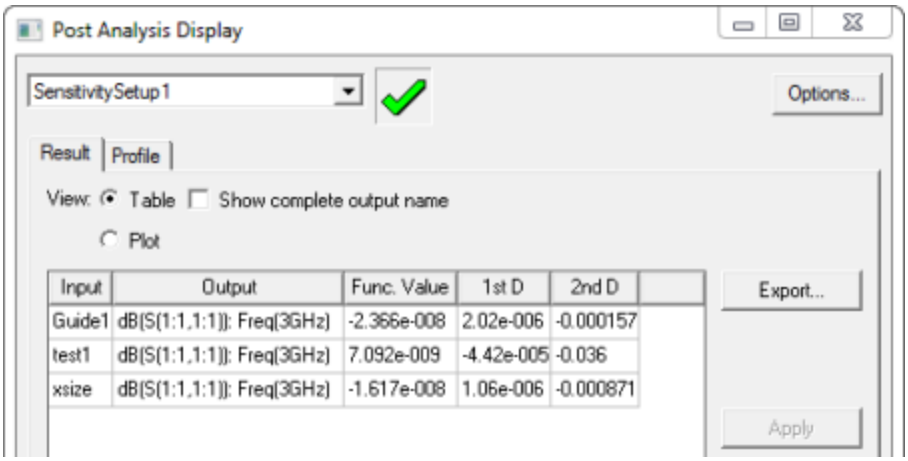
Transpose the table for an alternate view (double-click data table to view the **Properties** window, and on the **Data Table** tab check **Transpose**).



To estimate lower bound of performance:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

First, see the sensitivity of performance to the respect of each variable. (Follow the documentation for Viewing Output Parameter Results for a Sensitivity Analysis). Identify the variables that have major influence over performance.



Second, in your project, manually change these variables to the corresponding bounds: choose Min for positive 1st derivative and Max for negative 1st derivative.

2. **Monte Carlo analysis:**

   This method does not assume a circuit is linear - better accuracy is achieved with more iterations. The cost is computing time and resources.

   To perform Monte Carlo Analysis in Ansys Electronics Desktop, create a new analysis under **Optimetrics> Statistical**. Follow the documentation on Statistical Analysis to set this up. The upper and lower bound of performance can be found on the edge of performance distribution.

   > **Tip:**
   >
   > If distribution of performance is not of interest, set all variables as uniformly distributed.

Statistical Analysis Overview

Statistical analysis allows you to explore the effects of random combinations of values of selected variables on selected global or local available analysis results. Therefore, before a variable can be included in a statistical analysis, you must specify that you intend for it to be used during a statistical analysis. For each variable you must specify the type of distribution (Uniform, Gaussian, Lognormal or User Defined) and the corresponding parameters of the selected distribution.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** A statistical analysis is limited to using a maximum of 30 variables.

In addition to specifying the variables to be used in the statistical analysis and the parameters of the chosen distribution, the output quantities of interest also need to be specified. These quantities can be global ones such as previously defined parameters (Force/torque, inductance / capacitance, etc), other named quantities, quantities defined in the field calculator as global (such a domain integral of a certain field quantity) or local (such as field value at a certain location). The calculations to be performed during the statistical analysis are specified during setup, in a manner similar to other types of analysis in Optimetrics.

Following the analysis the statistical distribution of the output quantities can be visualized in histogram format. To access available reports, after the statistical analysis is complete, right click the respective Statistical analysis setup and select **View Analysis Result**.

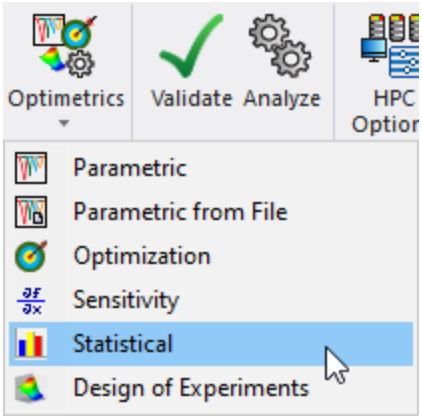
> **Note:**
>
> Statistical Analysis has been placed in legacy mode because it is no longer under active development. Go to **Tools** > **Options** > **General Options** > **Miscellaneous** and toggle the **Enable Legacy Optimetrics Tools** check box to show or hide these legacy tools.

Setting Up a Statistical Analysis

Following is the general procedure for setting up a statistical analysis. Once you have created a setup, you can **Copy** and **Paste** it, and then make changes to the copy, rather than redoing the whole process for minor changes. You can create a statistical setup before defining variables but all variables must be defined before you start the statistical analysis.

1. Before a variable can be included in a statistical analysis, you must specify that you intend for it to be used during a statistical analysis in the **Properties** dialog box.
2. Click **HFSS**> **Optimetrics Analysis** > **Add Statistical** , or select the **Simulation** tab of the ribbon and, under the Optimetrics icon, select **Statistical** from the drop-down menu:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Setup Statistical Analysis** dialog box appears.

3. Under the **Calculations** tab, type the maximum number of iterations you want HFSS to perform in the **Maximum Iterations** text box.
4. If you want to specify an initial seed value, select the **Specify initial seed value** check box and enter a positive value in the text box. Each different seed value creates a new statistical sequence.
5. Specify a solution quantity to evaluate.
6. In the **Calculation** text box, set the value at which the solution quantity is to be computed.
7. Optionally, modify the distribution criteria to be used.
8. The following *optional* statistical analysis setup options can also be used:
   - Modify the starting variable value.
   - Exclude variables from the statistical analysis.
   - Modify the values of fixed variables that are not being modified during the statistical analysis.
   - Request that Optimetrics solve a parametric sweep during a statistical analysis.

   > **Note:**
   >
   > Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

9. If you want to save the field solution data for the design variations solved during analysis, select **Save Fields**.

Setting the Maximum Iterations for a Statistical Analysis

The **Maximum Iterations** value is the maximum number of design variations Optimetrics solves during a statistical analysis. This value is a stopping criterion; if the maximum number of

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

iterations has been completed, the analysis stops. If the maximum number of iterations has not been completed, Optimetrics continues by performing another iteration, that is, by solving another design variation.

To set the maximum number of iterations for a statistical analysis:

- Under the **Calculations** tab of the *Setup Sensitivity Analysis* dialog box, type a value in the **Maximum Iterations** text box.

Specifying the Solution Quantity to Evaluate for Statistical Analysis

When you add a statistical setup, you can identify one or more solution quantities to evaluate. The solution quantities are specified by mathematical expressions that are composed of basic quantities. You can see the distribution of the solution quantities in the results.

1. In the **Calculations** tab of the *Setup Statistical Analysis* dialog box, click **Setup Calculations**.

   The **Add/Edit Calculations** dialog box is displayed, allowing you to define one or more mathematical expressions for statistical evaluation.

2. In the **Context** section of the dialog box:
   - Select the **Report Type** with a drop-down selection list containing the available types for this design.
   - Select the **Solution** from the drop-down selection list. This lists the available setups and sweeps. As a minimum, the **LastAdaptive** solution is available.
   - Select the **Geometry** from the drop-down selection list or select none (the default). This modifies the list of quantities available to the ones that apply to the specific geometry.
   - When selecting a geometry, you may also be required to specify a point within the geometry where the calculation is to be performed.
3. The **Output Variables** button opens the *Output Variables* dialog box allowing you to create special output variables to be used in the output parameter.
4. The **Calculation Expression** field in the **Trace** tab is used to enter the equation to be used for the solution quantities. To enter an expression, you may type it directly into the field or use the **Category**, **Quantity**, and **Function** lists as follows:
   - Select the **Category**, these depend on the Solution type and the design. This lets you specify the category of information to be used in the output parameter.
   - Select a **Quantity** from the list. Available quantities depend upon the Solution type, as well as the Geometry and Category selection. Selecting a Quantity automatically enters it into the Calculation Expression field.
   - Select a **Function** to apply to the value in the calculated expression.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For swept variables, the Range Function button opens the **Set RangebFunction** dialog to apply functions to the expression that apply overbthe sweep range.

5. The **Calculation Range** tab applies to swept variables and allows you to specify the range of the sweep over which to apply the calculation.
6. When the desired **Calculation Expression** has been obtained, click **Add Calculation** to add the entry to the calculation table in the Setup Statistical Analysis dialog box. You may add multiple entries to the table simply by changing the **Calculated Expression** and using the **Add Calculation** button.
7. To update or edit a selected cost function, enter the desired Calculation Expression and click the **Update Calculation** button.
8. Click **Done** to return to the Setup Statistical Analysis dialog box.

> **Note:**
>
> The solution quantity you specify must be able to be evaluated to a single, real number.

Setting the Solution Quantity's Calculation Range

The calculation range of a solution quantity determines the intrinsic variable value at which the solution quantity is extracted. For a statistical setup, the calculation range must be a single value. For a Driven Modal or Driven Terminal design, if you specified that the solution be extracted from the last adaptive solution, Optimetrics uses the adaptive frequency defined in the solution setup. If you specified that the solution quantity be extracted from a frequency sweep solution, Optimetrics will use the starting frequency in the sweep by default. The calculation range should be set during the setup of the solution quantity for statistical evaluation. In order to modify the calculation range, do the following:

1. Under the **Calculations** tab of the *Setup Statistical Analysis* dialog box, click in the **Calculation Range** column of the table for the calculation to be modified.

   The *Edit Calculation Range* dialog box appears.

2. In the table, click **Edit** in the row to be modified.

   If you choose to solve a parametric setup during the statistical analysis, the variables swept in that parametric setup are available in the pop-up list dialog box. If you sweep a variable in the parametric setup that is also a statistics variable, that variable is excluded from the statistics analysis.

   Other examples of available variables include frequency, if you selected an S-parameter solution quantity; and phi or theta, if the solution quantity is a radiated field quantity.

3. Click on the value for the calculation range in the list and dismiss the pop-up dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **OK** in the *Edit Calculation Range* dialog box to accept the new value for the intrinsic variable and return to the *Setup Statistical Analysis* dialog box.

Setting the Distribution Criteria

For every statistical setup, Optimetrics automatically sets the distribution criteria to be uniform within a 10% tolerance of the variable's starting value. You can modify the distribution type and criteria for a single statistical setup or for every statistical setup.

Overriding the Distribution Criteria for a Single Statistical Setup

To override the default distribution criteria for a single statistical setup:

1. In the *Setup Statistical Analysis* dialog box, click the **Variables** tab.

   All of the variables that were selected for statistical analysis are listed.

2. Check or clear the **Include** check box for each variable to define the specific variables to be varied in the statistical analysis setup.
3. For each included variable, select **Uniform**, **Gaussian**, **Lognormal**, or **User Defined** in the **Distribution** column for the variable you want to override.

   If you changed the distribution type, the **Override** option is now selected. This indicates that the distribution type you selected is to be used for this optimization analysis; the current distribution type selected for the variable in the nominal design is ignored in this statistical analysis.

   - Alternatively, you can select the **Override** option first, and then select a different distribution type in the **Distribution**text box.

4. Optionally, if you want to change the distribution criteria, click in **Distribution Criteria** column for the variable you want to override.

   The *Edit Distribution* dialog box appears.

5. If the distribution type is **Gaussian**, do the following:
   a. Type the lower limit of the distribution in the **Cutoff Probability** text box. This is a value =>0 and < 0.1.
   b. Type the mean value of the distribution in the **Mean** text box.
   c. Type the standard deviation of the distribution in the **Std Dev** text box.

   The design variations are solved using a Gaussian distribution within the specified mean and standard deviation values.

6. If the distribution type is **Uniform**, Eeter a tolerance value in the text box.

   The design variations are solved within the tolerance range of the starting value, using an even distribution.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. If the distribution type is **Lognormal**, do the following:
    a. Enter the cutoff probability in the **Cutoff Probability** text box.
    b. Enter the sigma value of the distribution in the **Sigma** text box and select a unit from the pull-down.
    c. Enter the m value of the distribution in the **M** text box.
    d. Enter the theta value in the Theta text box and select a unit from the pull-down.
8. If the distribution type is User Defined, do the following:
    a. Enter the cutoff probability in the **Cutoff Probability** text box.
    b. Click **Edit XY Data** to open the *Edit Datasets* dialog box in which you can select an existing dataset, or create a new one.
9. By default, all variables are set to sample using **Latin Hypercube** sampling. This sampling method provides for greater variability than random sampling by keeping track of chosen samples and guaranteeing that samples cannot be repeated. You may revert to random sampling by clearing the check box in the **Latin Hypercube** column for any desired variable.
10. Click **OK**.

To revert to the default distribution settings, clear the **Override** option.

Changing the Distribution Criteria for Every Statistical Setup

To change the default distribution criteria for every statistical setup:

1. Make sure that the variable's distribution criteria are not being overridden in any statistical setup.
2. If the variable is a design variable, do the following: On **HFSS** > **Design Properties**, select **Statistics**.

    If the variable is a project variable, do the following: Click **Project** > **Project Variables**.

    The *Properties* dialog box appears.
3. Select **Statistics**.
4. Click in the **Distribution** column for the variable you want to change, and then select **Uniform**, **Gaussian**, **Lognormal**, or **User Defined**.
5. Optionally, if you want to change the distribution criteria, click in the **Distribution Criteria** column for the variable you want to change.

    If the distribution type is **Gaussian**, the *Gaussian Distribution* dialog box appears. If the distribution type is **Uniform**, the *Uniform Distribution* dialog box appears.
6. If the distribution type is **Gaussian**, do the following:
    a. Type the lower limit of the distribution in the **Cutoff Probability** text box. This is a value =>0 and < 0.1.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Type the mean value of the distribution in the **Mean** text box.
c. Type the standard deviation of the distribution in the **Std Dev** text box.

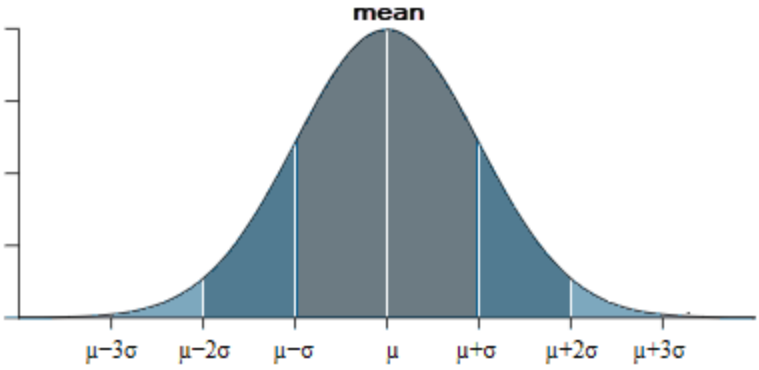
The design variations are solved using a Gaussian distribution within the specified mean and standard deviation values.

7. If the distribution type is **Uniform**, do the following:
   a. Type a cutoff probability value in the **Cutoff Probability** text box.
   b. Type mean and tolerance values in the corresponding text boxes.

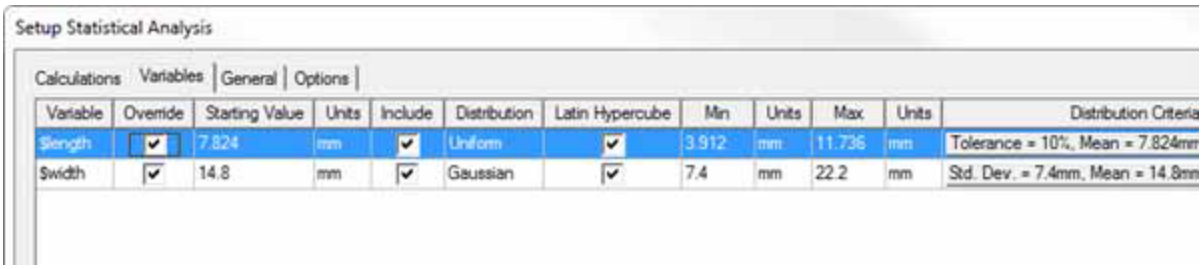
   The design variations are solved within the tolerance range of the starting value, using an even distribution.
8. If the distribution type is **Lognormal**, do the following:
   a. Type a cutoff probability value in the **Cutoff Probability** text box.
   b. Type values for Sigma, M, and Theta in the corresponding text boxes.
9. If the distribution type is **User Defined**, do the following:
   a. Type a cutoff probability value in the **Cutoff Probability** text box.
   b. Click **Edit XY Data** to open the *Edit Dataset* dialog box.
   c. Either type or import the X and Y data values for the distribution in the *Edit Dataset* dialog box.
10. Click **OK**.

Statistical Cutoff Probability

The cutoff probability values affects the Gaussian distribution criteria. This is a value =>0 and < 0.1. The design variations are solved using a Gaussian distribution using a lower limit cutoff probability and specified mean and standard deviation values.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



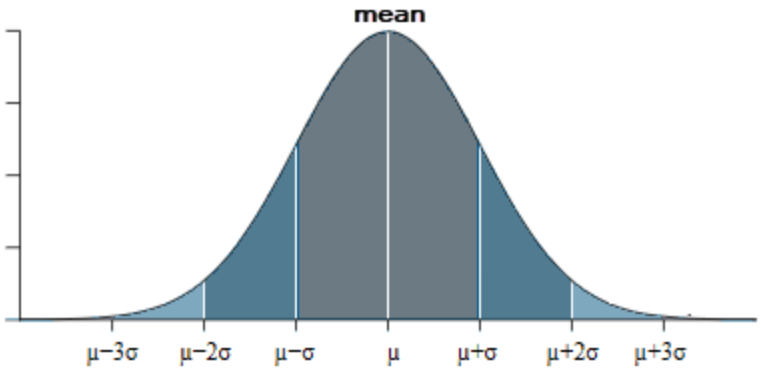
Uniform distributions such as **length** variable above use only the **Tolerance** value and do not have a cutoff probability.

Edit Distribution

When setting the distribution type for a variable, you have the option of changing the distribution parameters from the default values.

1. If the distribution type is **Gaussian**, do the following:
   a. Type the lower limit of the distribution in the **Cutoff Probability** text box. This is a value =>0 and < 0.1.
   b. Type the mean value of the distribution in the **Mean** text box.
   c. Type the standard deviation of the distribution in the **Std Dev** text box.

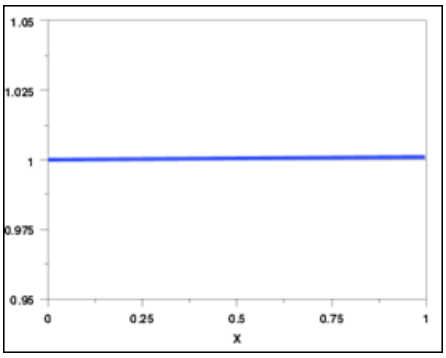
   The design variations are solved using a Gaussian distribution within the specified mean and standard deviation values.



2. If the distribution type is **Uniform**, do the following:
   a. Type the lower limit of the distribution in the **Cutoff Probability** text box.
   b. Type the mean value of the distribution in the **Mean** text box.
   c. Enter the tolerance in the **Tolerance** text box.

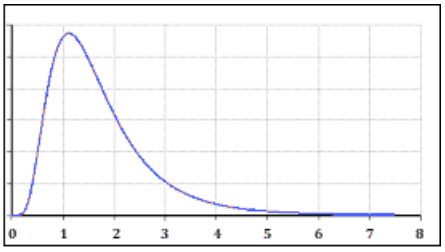
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The design variations are solved within the tolerance range of the starting value, using an even distribution.



3. If the distribution type is **Lognormal**, do the following:
    a. Type the lower limit of the distribution in the **Cutoff Probability** text box.
    b. Enter the shape parameter of the distribution in the **Sigma** text box.
    c. Enter the scale parameter in the **M** text box. The scale parameter should be set to 1 for the standard lognormal distribution.
    d. Enter the location parameter value for **Theta** in the text box. The value for a standard lognormal distribution is 0.

    The design variations are solved with a logarithmic distribution using the shape, scale and location parameters provided.



4. If the distribution type is **User Defined**, do the following:
    a. Type the lower limit of the distribution in the **Cutoff Probability** text box.
    b. Select the **Edit XY Data** button to manually define the data distribution using datasets.

Modifying the Starting Variable Value for Statistical Analysis

A variable's starting value is the first value that is solved during the statistical analysis. Optimetrics automatically sets the starting value of a variable to be the current value set for the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

nominal design. You can modify this value for each statistical setup.

> **Warning:**
>
> Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

1. In the *Setup Statistical Analysis* dialog box, click the **Variables** tab.

   All of the variables selected for the statistical analysis are listed.

2. Type a new value in the **Starting Value** text box for the value you want to override, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered is to be used for this statistical analysis; the current value set for the nominal model will be ignored.

   - Alternatively, you can select the **Override** option first, and then type a new variable value in the **Starting Value** text box.

3. Optionally, click a new unit system in one of the **Units** text boxes.

To revert to the default starting value, clear the **Override** option.

Solving a Parametric Setup During a Statistical Analysis

Solving a parametric setup during a statistical analysis is useful when you want Optimetrics to solve every design variation in the parametric setup at each statistical analysis iteration.

To solve a parametric setup during a statistical analysis:

1. In the *Setup Statistical Analysis* dialog box, click the **General** tab.
2. Click the parametric setup you want Optimetrics to solve during the statistical analysis from the **Parametric Analysis** drop-down menu.
3. Select **Solve the parametric sweep during analysis**.

Using Design of Experiments

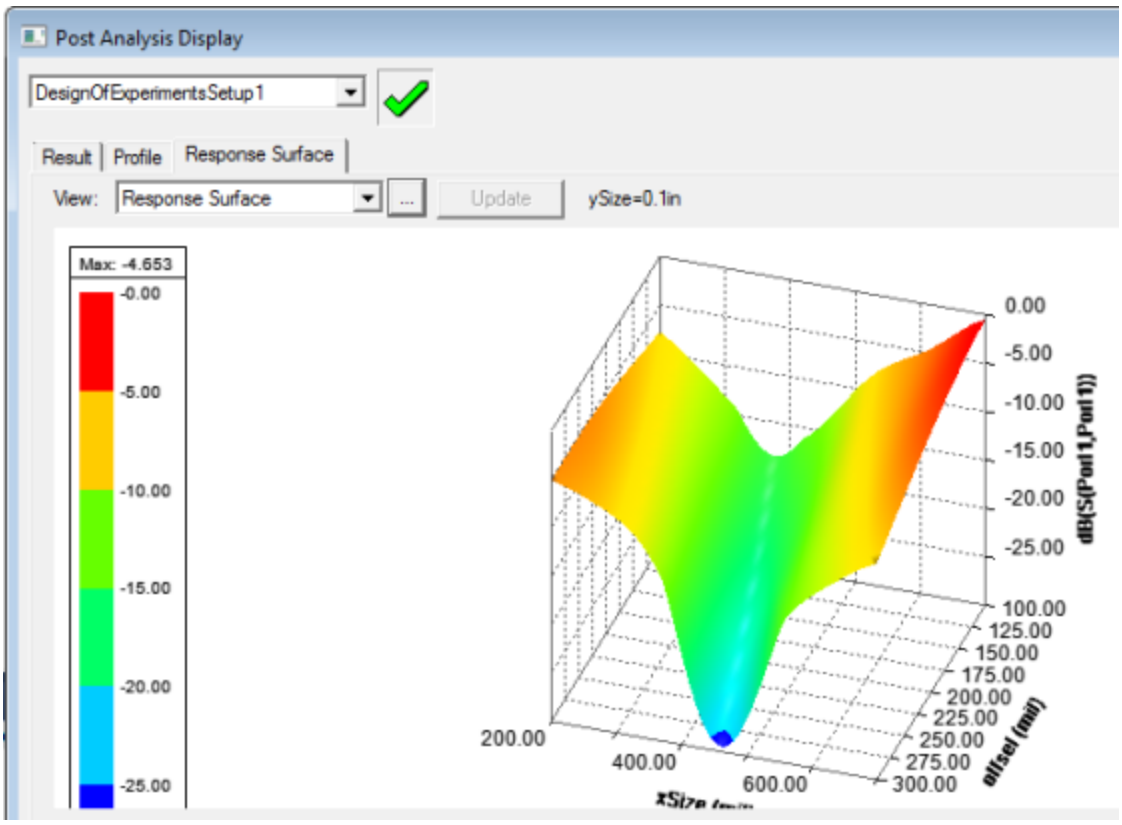
Design of Experiments (DOE) is a technique used to scientifically determine the location of sampling points and is included as part of the Response Surface, Goal Driven Optimization, and Analysis systems. Design of Experiments plus a mathematical approximation of output parameters lets you

- reduce the number of simulations
- interactively explore the design space before running optimization

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

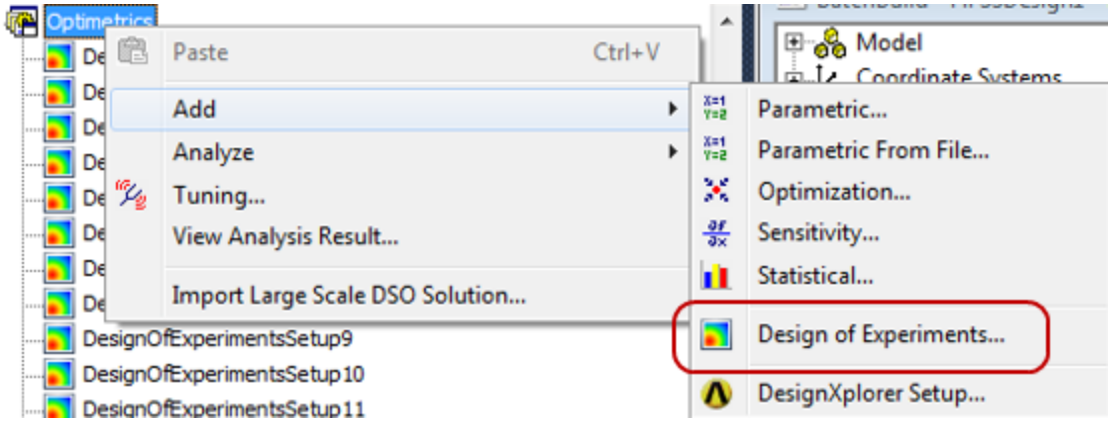
Design of Experiments describes the relationship between the design variables and the performance of the product by using Design of Experiments (DOE), combined with response surfaces. DOE and response surfaces provide all of the information required to achieve Simulation Driven Product Development. Once the variation of the performance with respect to the design variables is known, it becomes easy to understand and identify all changes required to meet the requirements for the product.

The goal is to create a response surface by interpolating through calculated points (a best curve fit). For each design, you can create a response surface for each output parameter. Once the response surfaces are created, you can share the information can in easily understandable terms: curves, surfaces, sensitivities, etc. They can be used at any time during the development of the product without requiring additional simulations to test a new configuration.

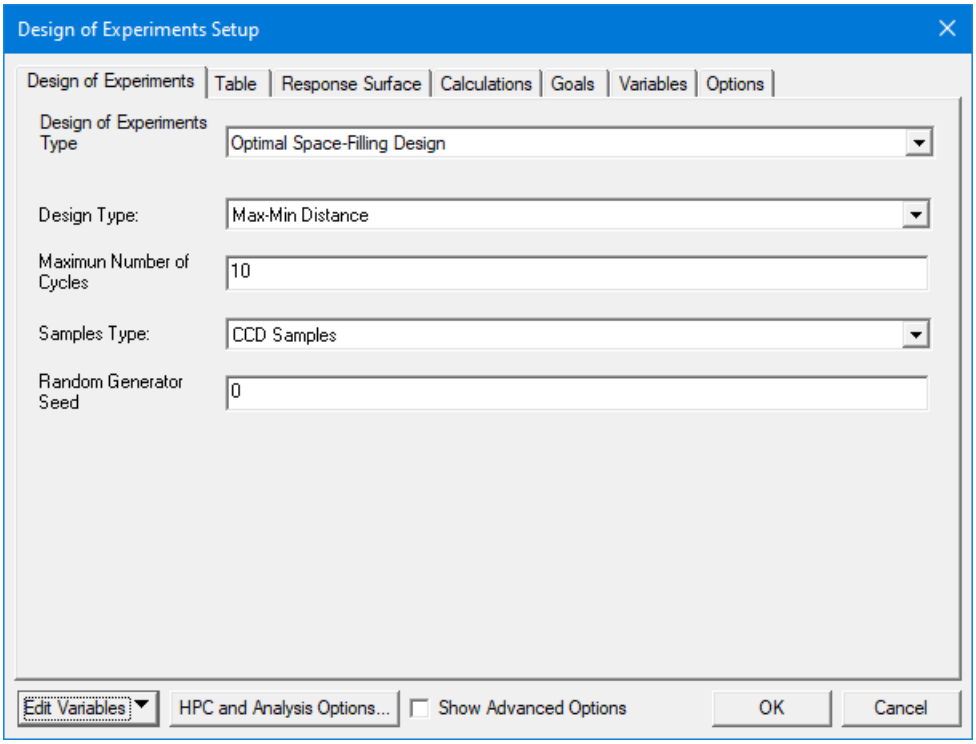


The Design of Experiments feature is integrated inside Electronics Desktop. Combined with Electronics Desktop‘s distributed solve feature, you can build the response surfaces from the DOE variation table much faster.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selecting **Design of Experiments** under **Optimization** opens a window with several tabs:



In the Design of Experiments setup, you select the DOE type, select the Response Surface, specify goals, view and include variables.

There are a wide range of DOE algorithms or methods available in engineering literature. These techniques all have one common characteristic: they try to locate the sampling points such that the space of random input parameters is explored in the most efficient way, or obtain the required information with a minimum of sampling points. Sample points in efficient locations only

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

reduce the required number of sampling points and increases the accuracy of the response surface generated. For more information on the available types of DOE, see Design of Experiments Types.

Once you have set up your input parameters, you can update the DOE, which submits the generated design points to the analysis system for solution. Design points are solved simultaneously if the analysis system is set up to do so; sequentially, if not. After the solution is complete, you can update the Response Surface cell, which generates response surfaces for each output parameter based on the data in the generated design points.

> **Note:**
>
> Requirements and recommendations regarding the number of input parameters vary according to DOE type. For more information, see Number of Input Parameters for DOE Types.

If you change the Design of Experiments type after doing an initial analysis and preview the Design of Experiments Table, any design points generated for the new algorithm that are the same as design points solved for a previous algorithm will appear as up-to-date. Only the design points that are different from any previously submitted design points need to be solved.

You should set up your DOE Properties before generating your DOE Design Point matrix. The following topics describe setting up and solving your Design of Experiments, and viewing the results.

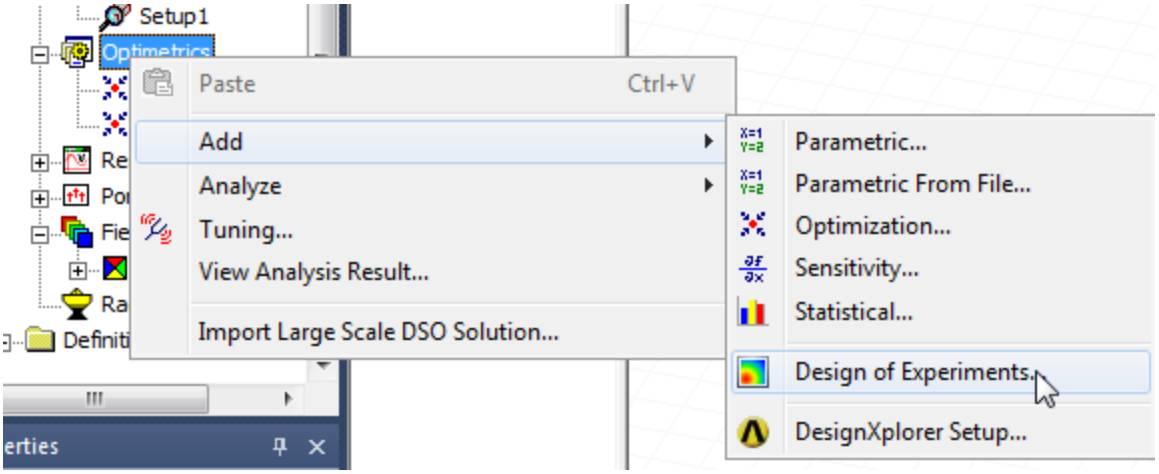
Setting Up Design of Experiments in Ansys Electronics Desktop

The process for setting up a Design of Experiments analysis in Ansys Electronics Desktop follows:

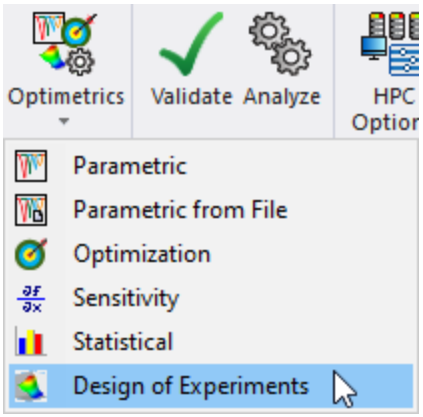
Add the Design of Experiments Setup

Right-click the Optimetrics icon in the Project tree and select **Add** > **Design of Experiments**. You can also use **HFSS** > **Optimetrics Analysis** > **Add Design of Experiments**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

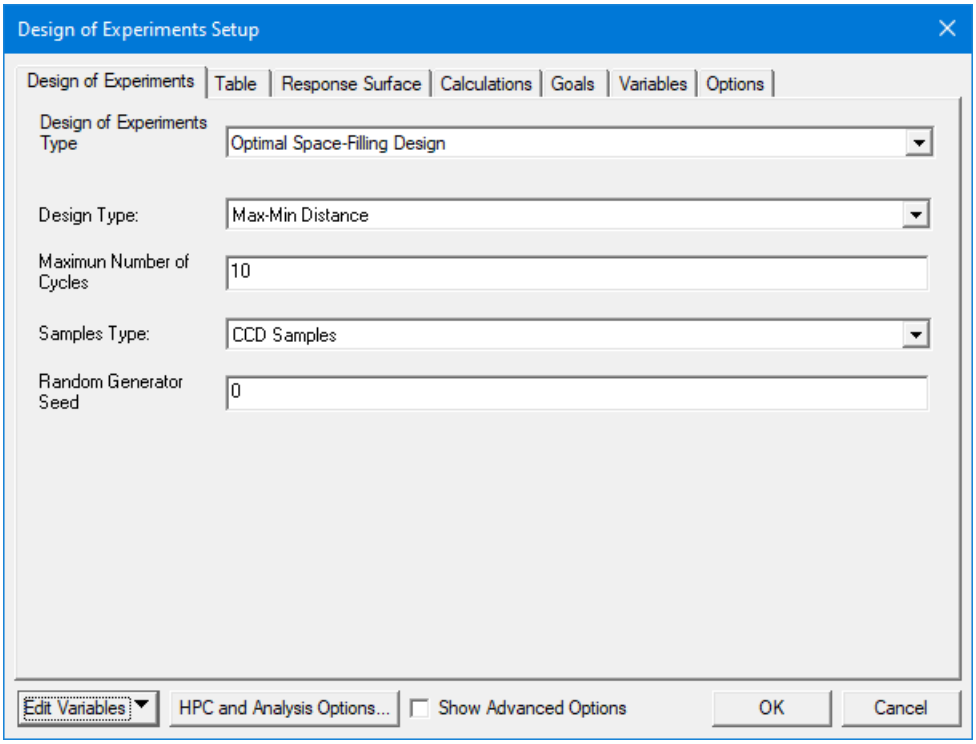


You can also select the **Simulation** tab and under the Optimetrics icon, select Design of Experiments from the drop-down menu:



The **Design of Experiments Setup** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

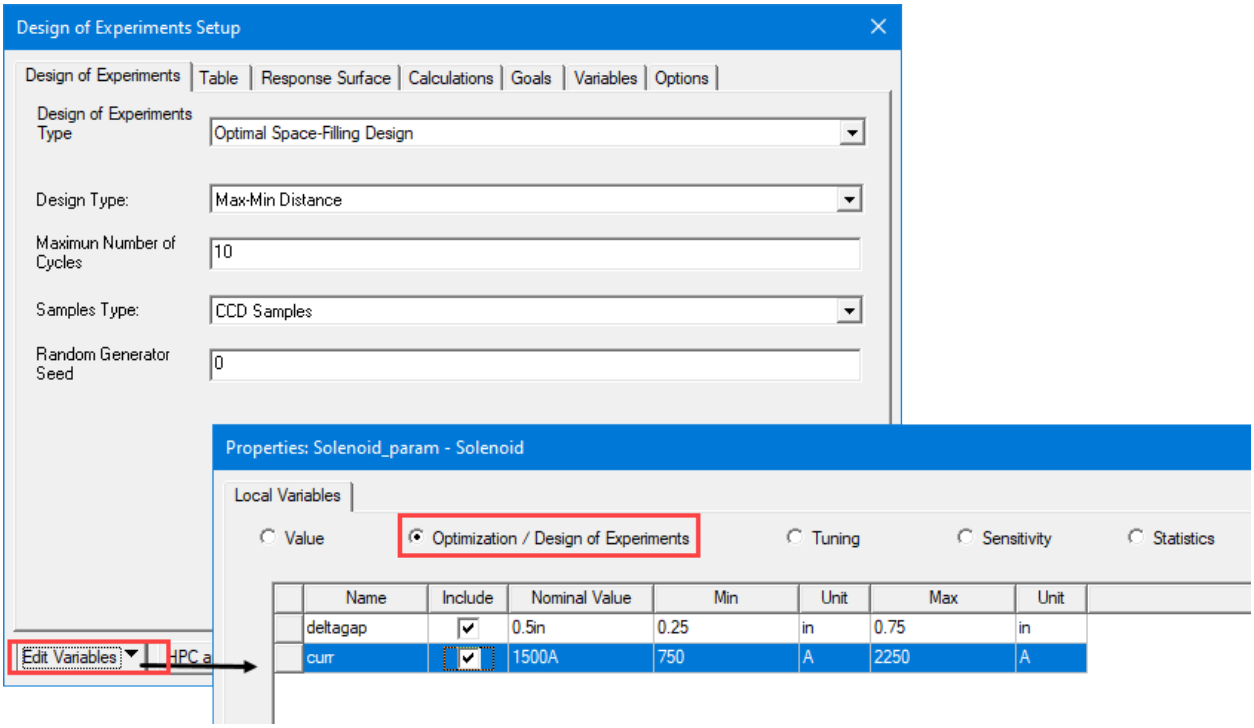


It has tabs for:

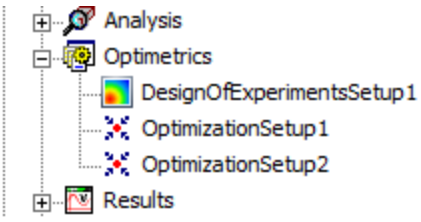
- Design of Experiments, where you specify the sampling points and associated parameters.
- Table, which shows the sampling points defined by the Design of Experiments settings you provide. If you select Custom and the Design of Experiments type, you can import data files as well as add or edit rows. You can also Export files of DOE sampling points you have defined by any method.
- Response Surface, which specifies the Response surface type, and Refinement parameters.
- Calculations, which accesses Optimetrics calculations.
- Goals, including cost functions calculations and norm type.
- Variables, previously defined for Optimization/Design of Experiments as Project or Design variables, and whether to include them, treat as discrete, whether to Use Manufacturable Variables, and the available Levels.
- Options, to Save Fields and Mesh, and whether to copy geometrically equivalent meshes.

You must define local or project variables as Optimization /Design of Experiments variables for the Design of Experiments setup to include the variable.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

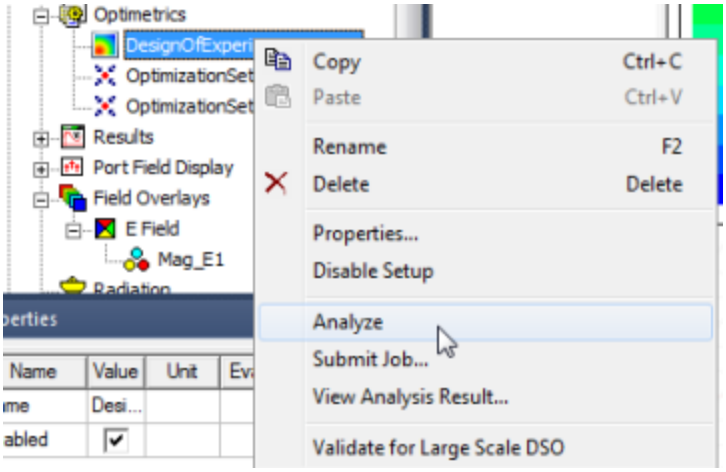


Once you have set parameters and click **OK**, the Design of Experiments setup appears under the Optimetrics icon in the Project tree.



From here you can right-click the DesignOfExperiments setup from the shortcut menu to run **Analyze**, **Submit Job...** or **Validate for Large Scale DSO**.

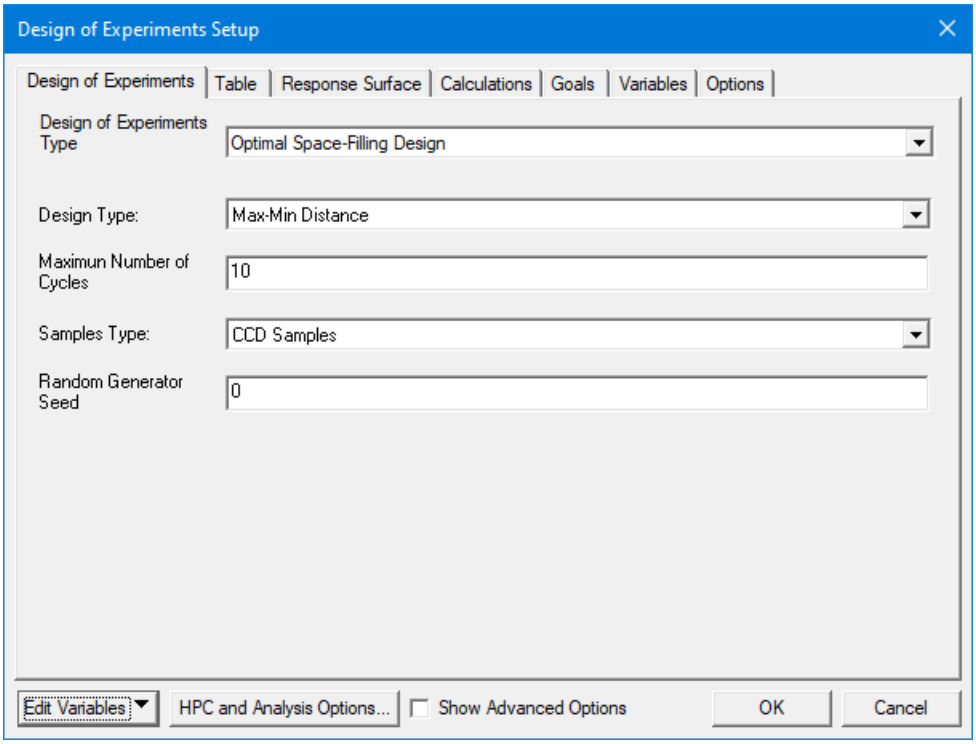
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Stand-alone and distributed simulation will have the same behavior as the parametric setup.

Design of Experiments Tab

The Design of Experiments tab in the DOE Setup includes selections for defining the sampling points that define your experiment. Each selection for Design of Experiments type has a different set of associated parameters so the appearance of the dialog changes to show the parameters for your selection.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The goal in Design of Experiments is to determine the smallest sufficient set of points required to calculate a response surface. Therefore, you choose the type depending on the parametric problem and targeted response surface. The number of points depends on the number of input parameters, or is user defined

The Design of Experiments Types available in the Desktop include the following:

Design of Experiments Types	Brief Description (see links for more details)
Optimal Space Filling (Default)	An optimized Latin Hypercube Sampling maximizing distance between experiments. Several design type criteria are available: • Max-Min Distance • Centered L2 • Maximum Entropy Sampling type determines the number of samples in the design: • CCD samples (Central Composite Designs are five level factorial designs that are suitable for calibrating the quadratic response model) • Linear model samples • Pure quadratic model samples • Full quadratic model samples • User-Defined samples You also specify a Maximum Number of Cycles and a Random Generator Seed.
Central Composite Design	Several design types available: • Face-Centered • Rotatable • VIF-Optimality • G-Optimality • Auto-defined You can also choose a Standard or Enhanced Template.
Box-Behnken	Avoids critical configurations in the corner of the design space. Maximum number of input parameters is 12.
Custom	Lets you customize a DOE matrix, by editing values, adding or removing samples, and/or importing samples from a CSV file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Design of Experiments Types	Brief Description (see links for more details)
	Selecting Custom enables an Import button and the **Table** tab, as well as buttons to **Add** editable rows or **Delete** selected rows. If you previously solved the DOE using one of the other algorithms, those design points are retained and you can add new design points to the table. You can also import and export design points into the custom DOE Table from the Parameter Set.
Latin Hypercube Sampling	Statistical design where no two experiments share input parameters of the same value. The samples Type can be • CCD Samples • Linear Model Samples • Pure Quadratic Model Samples • Full Quadratic Model Samples • User-Defined Samples, for which you also specify the Number of Samples. For each samples type you also specify a Random Generator Seed.

The Table tab provides a preview view of the design points defined by your selections.

Optimal Space Filling Design (OSF)

The goal in Design of Experiments is to determine the smallest sufficient set of points required to calculate a response surface. Therefore, you choose the type depending on the parametric problem and targeted response surface. The number of points depends on the number of input parameters, or is user defined.

Optimal Space-Filling Design (OSF) creates optimal space filling Design of Experiments (DOE) plans according to some specified criteria. Essentially, OSF is a Latin Hypercube Sampling Design (LHS) that is extended with post-processing. It is initialized as an LHS and then optimized several times, remaining a valid LHS (without points sharing rows or columns) while achieving a more uniform space distribution of points (maximizing the distance between points).

To offset the noise associated with physical experimentation, classical DOE types such as CCD focus on parameter settings near the perimeter of the design region. Because computer simulation is not quite as subject to noise, though, the Optimal Space-Filling (OSF) design is able to distribute the design parameters equally throughout the design space with the objective of gaining the maximum insight into the design with the fewest number of points. This advantage makes it appropriate when a more complex meta-modeling technique such as Kriging, Non-Parametric Regression, or Neural Networks is used.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OSF shares some of the same disadvantages as LHS, though to a lesser degree. Possible disadvantages of an OSF design are:

- When the CCD Samples sample type is selected, a maximum of 20 input parameters is supported.
- Extremes, such as the corners of the design space, are not necessarily covered.
- The selection of too few design points can result in a lower quality of response prediction.

The following properties are available for the OSF DOE type.

- **Design Type**: The following choices are available:
  - **Max-Min Distance** (default): Maximizes the minimum distance between any two points. This strategy ensures that no two points are too close to each other. For a small size of sampling (N), the Max-Min Distance design generally lies on the exterior of the design space and fill in the interior as N becomes larger. Generally, this is the faster algorithm.
  - **Centered L2**: Minimizes the centered L2-discrepancy measure. The discrepancy measure corresponds to the difference between the empirical distribution of the sampling points and the uniform distribution. This means that the centered L2 yields a uniform sampling. This design type is computationally faster than the **Maximum Entropy** type.
  - **Maximum Entropy**: Maximizes the determinant of the covariance matrix of the sampling points to minimize uncertainty in unobserved locations. This option often provides better results for highly correlated design spaces. However, its cost increases non-linearly with the number of input parameters and the number of samples to be generated. Thus, it is recommended only for small parametric problems.
- **Maximum Number of Cycles**: Determines the number of optimization loops the algorithm needs, which in turns determines the discrepancy of the DOE. The optimization is essentially combinatorial, so a large number of cycles slows down the process. However, this makes the discrepancy of the DOE smaller. For practical purposes, 10 cycles is generally good for up to 20 variables. The value must be greater than 0. The default is 10.
- **Samples Type**: Determines the number of DOE points the algorithm should generate. This option is suggested if you have some advanced knowledge about the nature of the metamodel. The following choices are available:
  - **CCD Samples** (default): Supports a maximum of 20 inputs. Generates the same number of samples a CCD DOE would generate for the same number of inputs. You can use this to generate a space filling design that has the same cost as a corresponding CCD design.
  - **Linear Model Samples**: Generates the number of samples as needed for a linear metamodel.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - **Pure Quadratic Model Samples**: Generates the number of samples as needed for a pure quadratic metamodel (no cross terms).
    - **Full Quadratic Samples**: Generates the number of samples needed to generate a full quadratic model.
    - **User-Defined Samples**: Specify the desired number of samples.
- **Random Generator Seed**: Set the value used to initialize the random number generator invoked internally by the LHS algorithm. Although the generation of a starting point is random, the seed value consistently results in a specific LHS. This property allows you to generate different samplings by changing the value or regenerate the same sampling by keeping the same value. The default is 0.
- **Number of Samples**: Enabled when **Samples Type** is set to **User-Defined Samples**. Specifies the default number of samples. The default is 10.

Central Composite Design (CCD)

The goal in Design of Experiments is to determine the smallest sufficient set of points required to calculate a response surface. Therefore, you choose the type depending on the parametric problem and targeted response surface. The number of points depends on the number of input parameters, or is user defined.

Central Composite Design (CCD) provides a screening set to determine the overall trends of the metamodel to better guide the choice of options in Optimal Space-Filling Design. The CCD DOE type supports a maximum of 20 input parameters.

The following properties are available for the CCD DOE type.

- **Design Type**: By specifying the **Design Type** for CCD, you can help to improve the response surface fit for DOE studies. For each CCD type, the alpha value is defined as the location of the sampling point that accounts for all quadratic main effects. The following CCD design types are available:
    - **Face-Centered**: A three-level design with no rotatability. The alpha value equals 1.0. A Template Type setting automatically appears, with Standard and Enhanced options. Choose Enhanced for a possible better fit for the response surfaces.
    - **Rotatable**: A five-level design that includes rotatability. The alpha value is calculated based on the number of input variables and a fraction of the factorial part. A design with rotatability has the same variance of the fitted value regardless of the direction from the center point.
    - **VIF-Optimality**: A five-level design in which the alpha value is calculated by minimizing a measure of non-orthogonality known as the Variance Inflation Factor (VIF). The more highly correlated the input variable with one or more terms in a regression model, the higher the VIF.
    - **G-Optimality**: Minimizes a measure of the expected error in a prediction and minimizes the largest expected variance of prediction over the region of interest.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - **Auto-Defined**: Design exploration automatically selects the Design Type based on the number of input variables. Use of this option is recommended for most cases as it automatically switches between the G-Optimality if the number of input variables is 5 or VIF-Optimality otherwise.
      However, you can use the Rotatable design if the default option does not provide good values for the Goodness of Fit from the response surface plots. Additionally, you can use the Enhanced template if the default Standard template does not fit the response surfaces well.
- **Template Type**: Enabled for the Rotatable and Face-Centered design types. The following options are available:
    - **Standard**
    - **Enhanced**: Choose this option for a possible better fit for the response surfaces

Box Behnken Design (CCD)

The goal in Design of Experiments is to determine the smallest sufficient set of points required to calculate a response surface. Therefore, you choose the type depending on the parametric problem and targeted response surface. The number of points depends on the number of input parameters, or is user defined.

A **Box-Behnken Design** is a three-level quadratic design that does not contain fractional factorial design. The sample combinations are treated in such a way that they are located at midpoints of edges formed by any two factors. The design is rotatable (or in cases, nearly rotatable).

One advantage of a Box-Behnken design is that it requires fewer design points than a full factorial CCD and generally requires fewer design points than a fractional factorial CCD. Additionally, a Box-Behnken Design avoids extremes, allowing you to work around extreme factor combinations. Consider using the Box-Behnken Design DOE type if your project has parametric extremes (for example, has extreme parameter values in corners that are difficult to build). Because the Box-Behnken DOE doesn’t have corners and does not combine parametric extremes, it can reduce the risk of update failures.

Possible disadvantages of a Box-Behnken design are

- Prediction at the corners of the design space is poor and that there are only three levels per parameter.
- A maximum of twelve input parameters is supported.

No additional properties are available for the Box-Behnken Design DOE type.

Custom DOE Type

The goal in Design of Experiments is to determine the smallest sufficient set of points required to calculate a response surface. Therefore, you choose the type depending on the parametric

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

problem and targeted response surface. The number of points depends on the number of input parameters, or is user defined.

The **Custom DOE** type allows for definition of a custom DOE Table. You can manually add new design points, entering the input and (optionally) output parameter values directly into the table. If you previously solved the DOE using one of the other algorithms, those design points are retained and you can add new design points to the table. You can also import and export design points into the custom DOE Table from the Parameter Set.

You can change the edition mode of the DOE table to edit the output parameter values. You can also copy and paste data and import data from a CSV file by right-clicking and selecting Import Design Points.

Latin Hypercube Sampling

The goal in Design of Experiments is to determine the smallest sufficient set of points required to calculate a response surface. Therefore, you choose the type depending on the parametric problem and targeted response surface. The number of points depends on the number of input parameters, or is user defined.

In the **Latin Hypercube Sampling** Design DOE type, the DOE is generated by the LHS algorithm, an advanced form of the Monte Carlo sampling method that avoids clustering samples. In a Latin Hypercube Sampling, the points are randomly generated in a square grid across the design space, but no two points share the same value. This means that no point shares a row or a column of the grid with any other point.

Possible disadvantages of an LHS design are

- When the CCD Samples sample type is selected, a maximum of 20 input parameters is supported. For more information, see Number of Input Parameters for DOE Types.
- Extremes, such as the corners of the design space, are not necessarily covered. Additionally, the selection of too few design points can result in a lower quality of response prediction.

> **Note:**
>
> The Optimal Space-Filling Design DOE type is an LHS design that is extended with post-processing.

The following properties are available for the LHS DOE type:

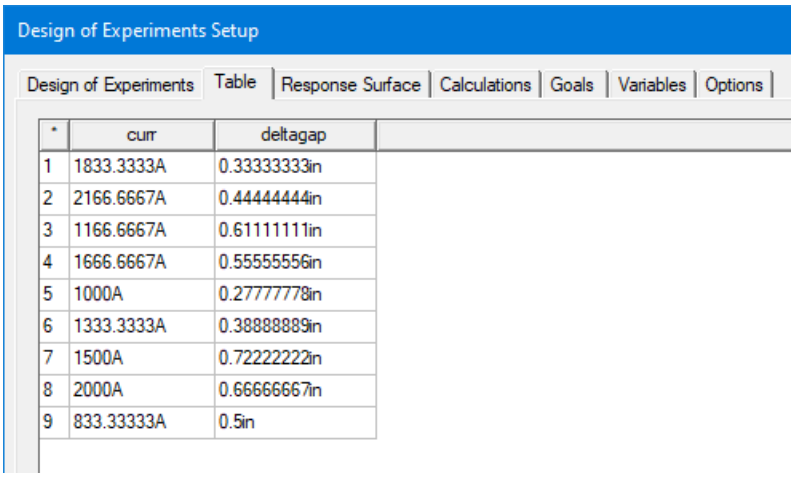
- **Samples Type**: Determines the number of DOE points the algorithm should generate. This option is suggested if you have some advanced knowledge about the nature of the metamodel. The following choices are available:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **CCD Samples** (default): Supports a maximum of 20 inputs. Generates the same number of samples a CCD DOE would generate for the same number of inputs. You can use this to generate a space filling design that has the same cost as a corresponding CCD design.
  - **Linear Model Samples**: Generates the number of samples as needed for a linear metamodel.
  - **Pure Quadratic Model Samples**: Generates the number of samples as needed for a pure quadratic metamodel (no cross terms).
  - **Full Quadratic Samples**: Generates the number of samples needed to generate a full quadratic model.
  - **User-Defined Samples**: Specify the desired number of samples.
- **Random Generator Seed**: Set the value used to initialize the random number generator invoked internally by the LHS algorithm. Although the generation of a starting point is random, the seed value consistently results in a specific LHS. This property allows you to generate different LHS samplings (by changing the value) or to regenerate the same LHS sampling (by keeping the same value). The default is 0.
- **Number of Samples**: Enabled when **Samples Type** is set to **User-Defined Samples**. Specifies the default number of samples. The default is 10.

Table Tab for Design of Experiments

The **Table** tab for the **Design of Experiments Setup** window displays a preview of the design points designed by your selections on the **Design of Experiments** tab. There is one column for each defined variable. The **Export** button lets you create a file of the table in a format you specify.

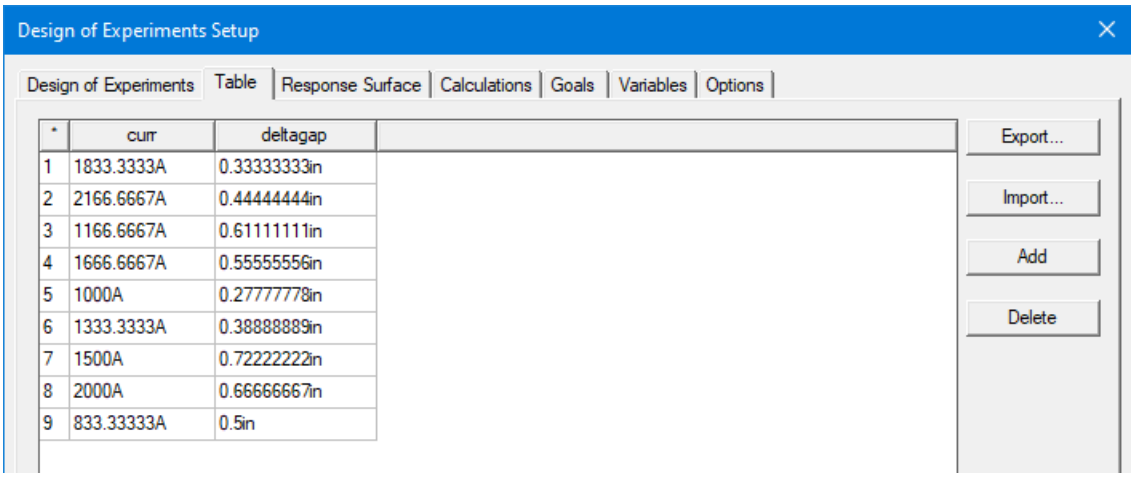


*	curr	deltagap
1	1833.3333A	0.33333333in
2	2166.6667A	0.44444444in
3	1166.6667A	0.61111111in
4	1666.6667A	0.55555556in
5	1000A	0.27777778in
6	1333.3333A	0.38888889in
7	1500A	0.72222222in
8	2000A	0.66666667in
9	833.33333A	0.5in

If you have specified Custom as the Design of Experiments Type, the table is editable, and the **Table** tab includes buttons for **Add** and **Delete** rows. All rows are editable. You can add new rows by entering values in the input parameter

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

columns. Once you have entered a value in one column in the * row, the row is added to the table and the values for the remaining input parameters are set to the initial values of the parameters. You can then edit that row in the table and change any of the other input parameter values if needed. Output parameter values are then calculated when the design is solved updated.



Depending on the context, the tables are read-only and filled automatically, or they are partially or completely editable. The background color of a cell indicates if it is editable or not:

- A gray background indicates a read-only cell
- A white background indicates an editable cell

Output parameter values calculated from a simulation (a design point update) are displayed in black text.

The Custom Table view also includes an Import button. Import and Export files can be

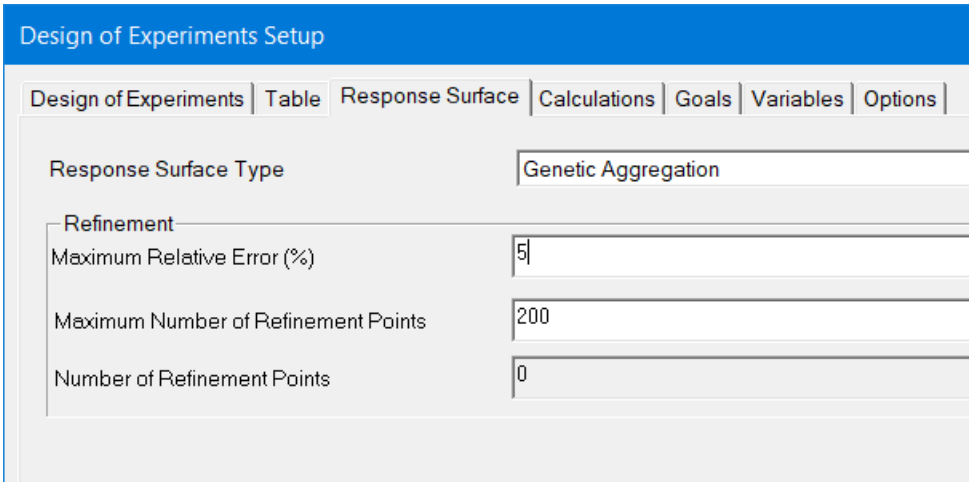
- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansoft Plot data files, (*.dat)
- Post Processor format data files, (*.txt)

The Table updates automatically when you change your selections on the **Design of Experiments** tab.

Response Surface Tab for Design of Experiments

The Response Surface tab for the **Design of Experiments Setup** dialog lets you select the Response Surface type as Genetic Aggregation or Standard Response Surface.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



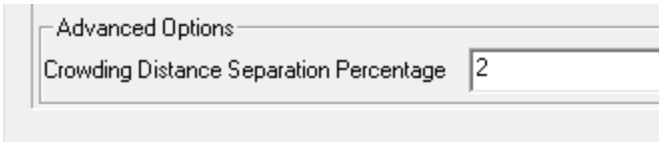
The Response Surface type selection specifies the refinement applied to the initial Design of Experiments. The Genetic Aggregation Response Surface finds the best possible response surface for each output automatically by combining

- Metamodels
- Settings
- Kernel Variation
- Polynomial Regression

For each output, a fitness factor works to minimize error, including cross-validation errors. The automatic refinement adds design points to the DOE until the response surface accuracy meets user requirements. You can specify requirements for:

- **Maximum Relative Error %**: This apply to all output calculations. Empty this field to turn off Auto refinement. (Note: Workbench does not have this field.)
- **Maximum Number or Refinement Points**: Determines the maximum number of refinement points that can be generated for use with the Genetic Aggregation algorithm.
- **Number of Refinement Points**: Read-only property indicating the number of existing refinement points.

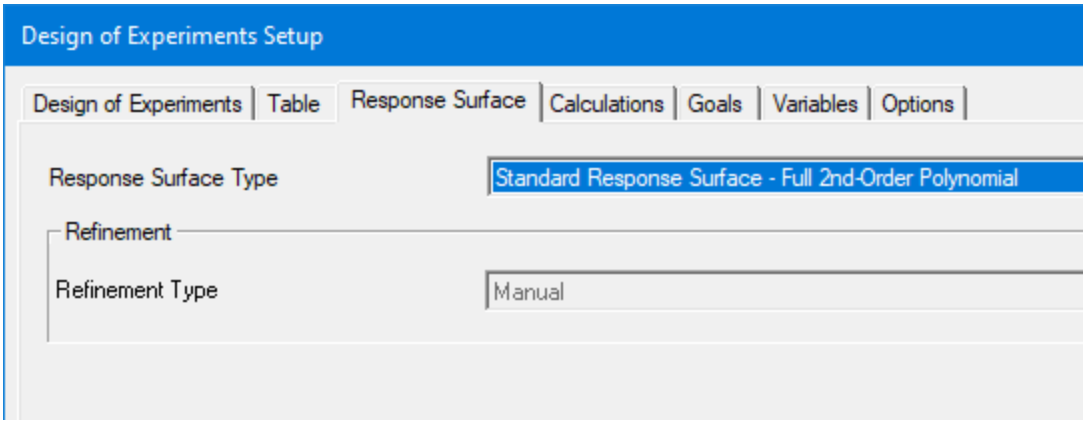
If you check Show Advanced Options at the bottom of the **Setup** window, you can also specify Crowding Distance Separation Percentage:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Crowding Distance Separation Percentage: determining the minimum allowable distance between new refinement points, implemented as a constraint in the search for refinement points.

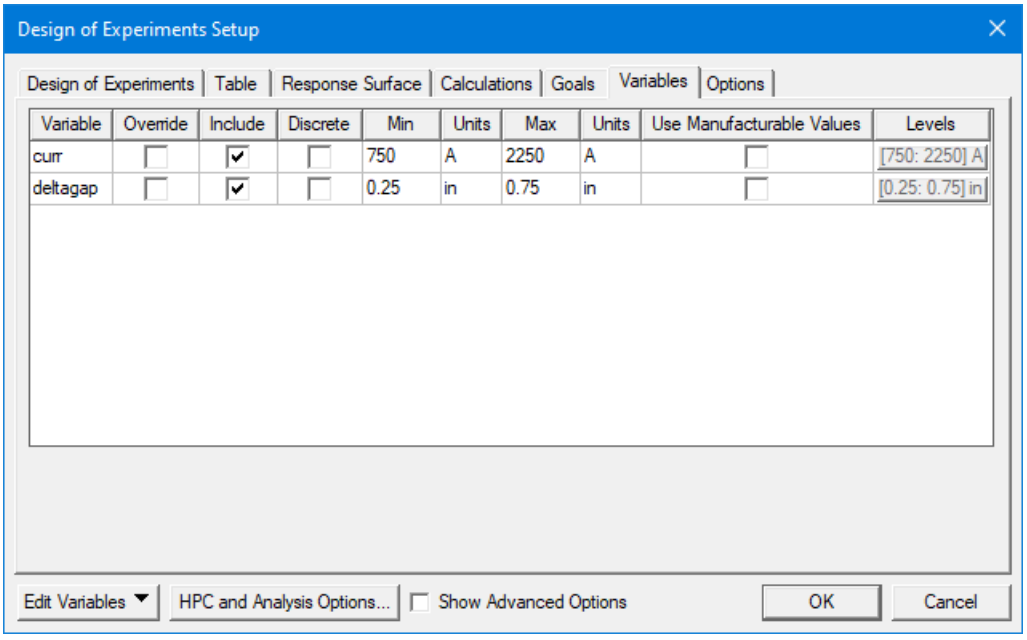
You do have the option of selecting Standard Response Surface- Full 2nd Order Polynomial.



After you have completed an analysis you can view the generated plot.

Variables Tab for Design of Experiments

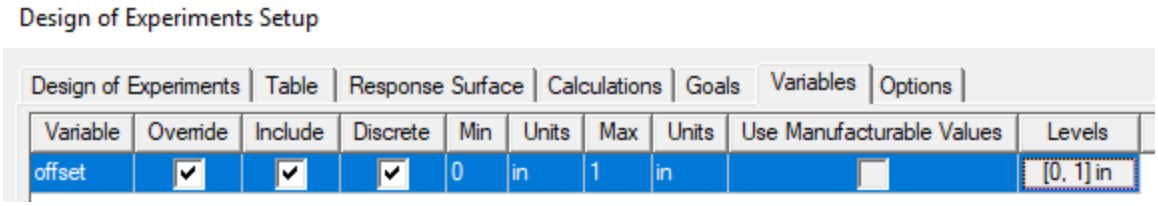
The Variables tab for the **Design of Experiments Setup** window displays list of the variables defined for the Design and Project as Optimetrics/Design of Experiments variables.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The columns list the Variable names, the current Min and Max Values and Units, and provides options for the following:

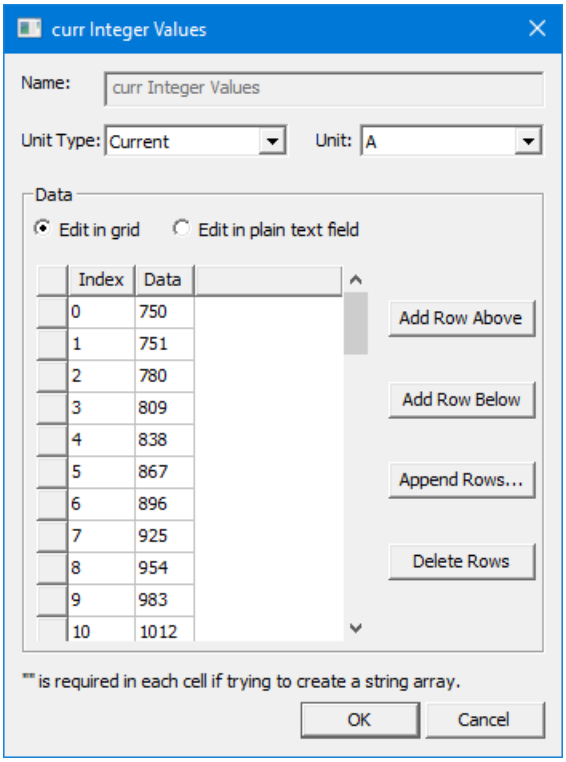
- **Override** – check this to override the current design value. Unchecking this causes a dialog to appear asking you to confirm the return to the design value.
- **Include** – whether to include the variable in an analysis.
- **Discrete** – **Discrete Variables** physically represent different configurations or states of the model. When check box in the Discrete column is checked, the button in the “Levels” column will be enabled and Use Manufacturable Values is disabled.



The Levels show the variable values. For continuous variable, Levels is determined by the min/max columns. For Integer variables Levels can be a subset of the min/max range. The discrete values can be bounded by a min/man range and/or manufacturable values.

Click the enabled **Levels** button for the row to edit the Discrete values. An edit dialog box for the variable appears:

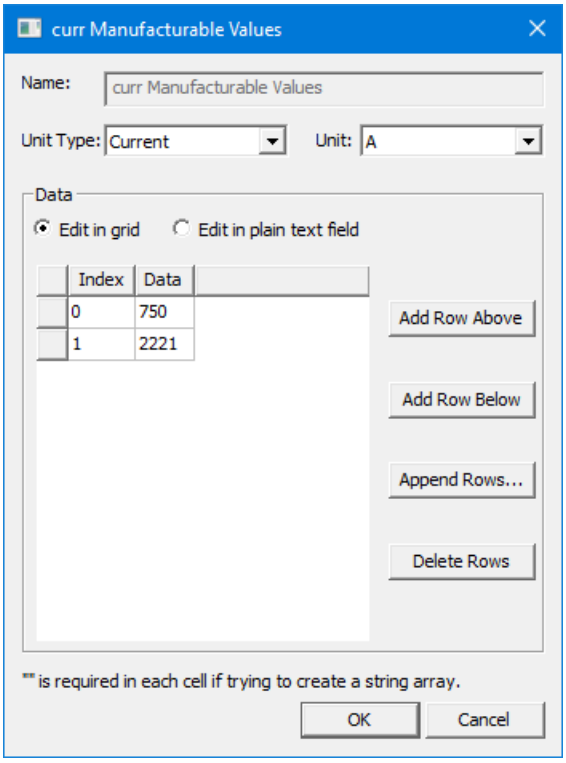
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For continuous variables (with **Discrete** unchecked), the values physically vary in a continuous manner between a lower and an upper bound (min/max) defined by the user. With continuous variables, you can also check **Use Manufacturable Values**.

- When you enable the check box in the "**Use Manufacturable Values**" column the button in the "**Levels**" column is enabled. Manufacturable Values represent real-world manufacturing or production constraints. The min max values for the variable will be upper and lower constraints on the available manufacturable values bounded by the min/max range. If you check **Use Manufacturable values**, you edit the Levels to define the Manufacturable values. Click the Levels button to edit the values. An edit dialog for the variable appears.

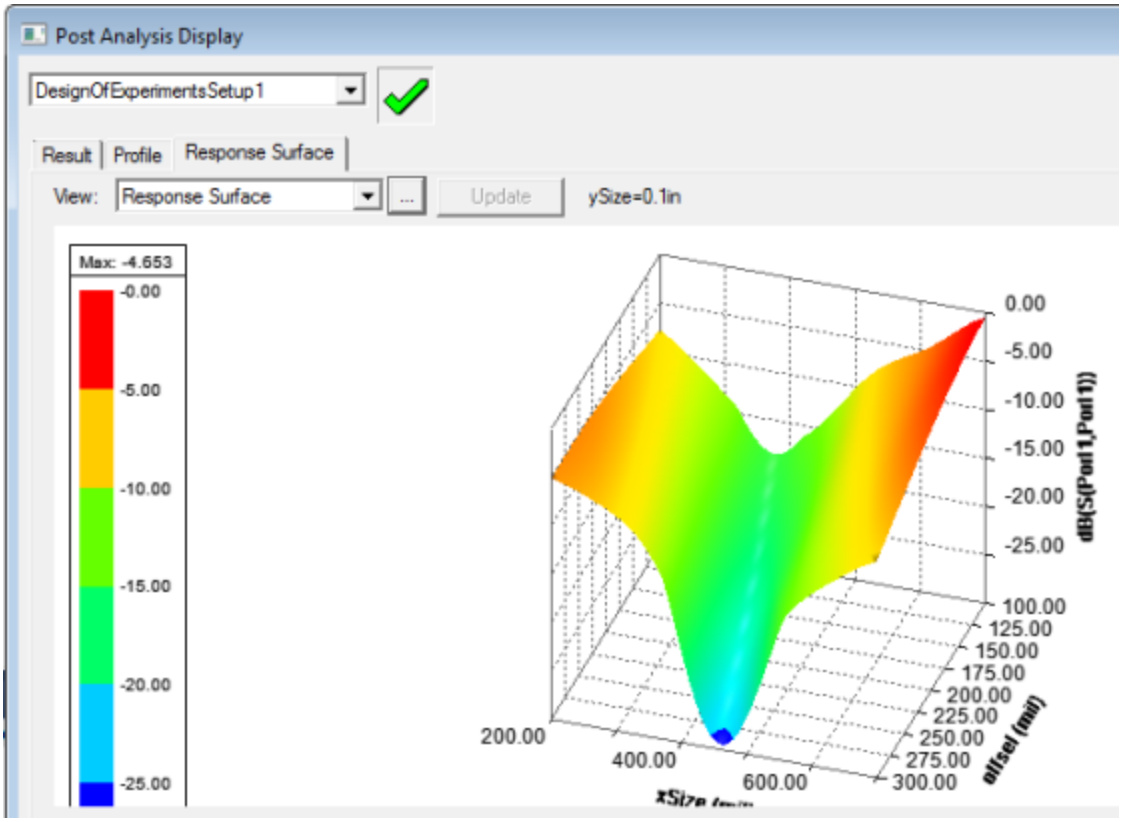
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Viewing Analysis Result for Design of Experiments

After the Analysis has competed, you can right-click the Design of Experiments setup in the **Project tree** and click **View Analysis Result** on the shortcut menu. This opens a dialog that includes a **Result** tab that lists the variations and variable values, a solution **Profile** tab with start, stop, time elapsed, and machines used, as well as a **Response Surface** tab. When you include more than one variable in the setup, response surface view is available. You can choose any two variables as the X, Y axis, and choose an output calculation as the Z axis, by clicking the ellipsis [...] button next to the view list box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



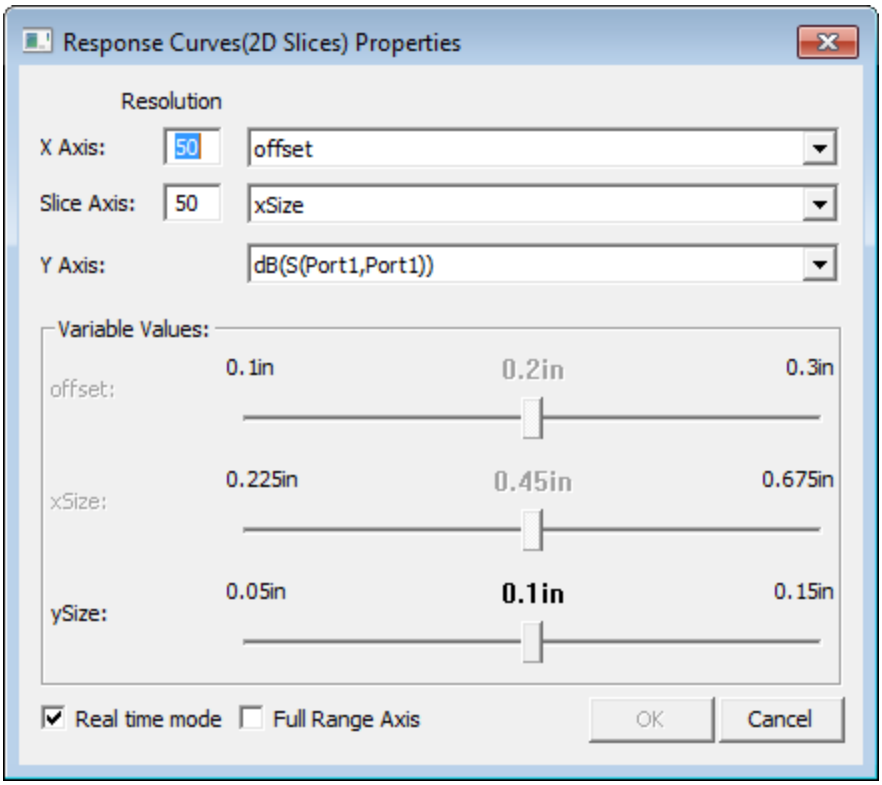
From the **Response Surface** tab, the “View” list box lets you select all available views of the selected response-surface-setup.

- Min Max Search
- Refinement points table
- Response points table
- Verification points table
- Goodness of Fit
- Response Curve
- Response Curve (2D Slices)
- Response Surface
- Local Sensitivity Charts
- Local Sensitivity Curves

The **Update** push button is disabled when the response surface is up to date. After the setup, if you modify a verification point or refinement points, it is enabled. Click this button to regenerate the response-surface with new settings. It may start new simulations if any of the design points in the DOE, refinement points, or verification points has not been solved.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clicking the ellipsis button [...] by the View for Response Surface opens a dialog box that lets you adjust the variables selected and the values applied.



Tuning a Response Curve

For the X Axis and Y Axis, you can specify a Resolution, and the variable to use. For the Z Axis you can select the Cost or calculation. For variables not selected for the X and Y axis, a slider is enabled that lets you adjust the value to see the effect on the response curve plot. You can enable or disable **Real time mode** by using the check box at the lower left.

Full Range Axis check box

When it is checked all Axes are set to their maximum ranges, and the ranges won't be changed while tuning unless you change the axis variable.

When it is unchecked, the Y(2D)/Z(3D) axis range is auto updated to fit the curve/surface.

Exporting Response Curve Data

You can Export the response curve data as a table in the following formats:

- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansys Plot data files, (*.dat)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

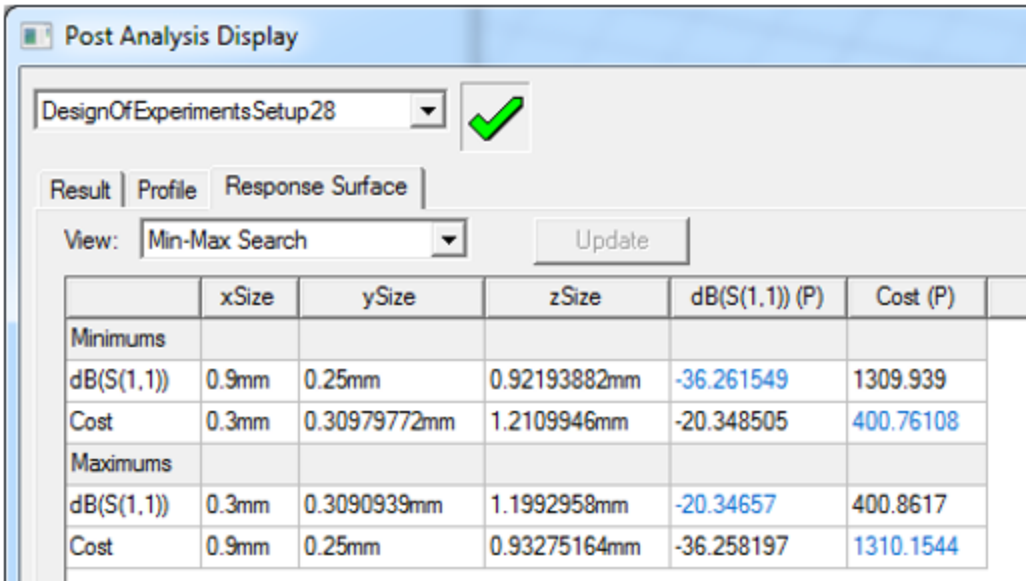
- Post Processor format data files, (*.txt)
- Ansys Report Data *.rdat files

Exported files can be imported into a report.

Min-Max Search View for Design of Experiments Result

From the **Response Surface** tab of the Design of Experiments **Post Analysis Display** dialog box, the **View** list box lets you select all available views of the selected response-surface-setup.

The **Min-Max Search** view examines the entire output parameter space from a response surface to approximate the minimum and maximum values of each output parameter. When you select a Min/Max row **Apply** is enabled, and you can then apply the selected variation variable values to the variables' nominal values.



	xSize	ySize	zSize	dB(S(1,1)) (P)	Cost (P)
Minimums					
dB(S(1,1))	0.9mm	0.25mm	0.92193882mm	-36.261549	1309.939
Cost	0.3mm	0.30979772mm	1.2109946mm	-20.348505	400.76108
Maximums					
dB(S(1,1))	0.3mm	0.3090939mm	1.1992958mm	-20.34657	400.8617
Cost	0.9mm	0.25mm	0.93275164mm	-36.258197	1310.1544

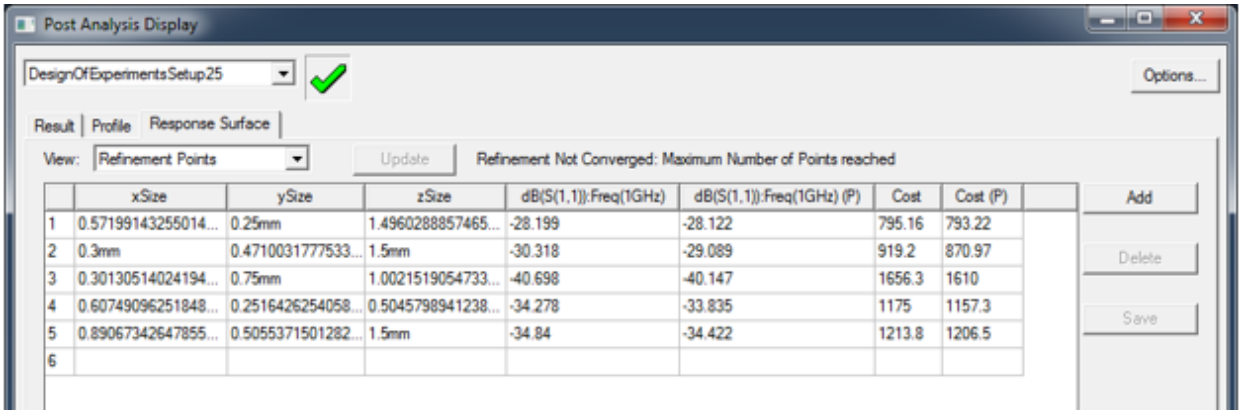
You can Export the Table in the following formats:

- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansoft Plot data files, (*.dat)
- Post Processor format data files (*.txt)

Refinement Points Table

From the Response Surface tab, the "View" list box lets you select all available views of the selected response-surface-setup. All refinement points are shown in the Refinement Points table:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



	xSize	ySize	zSize	dB(S(1,1)):Freq(1GHz)	dB(S(1,1)):Freq(1GHz) (P)	Cost	Cost (P)
1	0.57199143255014...	0.25mm	1.4960288857465...	-28.199	-28.122	795.16	793.22
2	0.3mm	0.4710031777533...	1.5mm	-30.318	-29.089	919.2	870.97
3	0.30130514024194...	0.75mm	1.0021519054733...	-40.698	-40.147	1656.3	1610
4	0.60749096251848...	0.2516426254058...	0.5045798941238...	-34.278	-33.835	1175	1157.3
5	0.89067342647855...	0.5055371501282...	1.5mm	-34.84	-34.422	1213.8	1206.5
6							

Refinement points are points added to your model to enrich and improve your response surface. They can either be generated automatically with the response surface update or added manually, as described in **Performing a Manual Refinement**. As with design points, DesignXplorer must perform a design point update (a "real solve") in order to obtain the output parameters for the refinement points.

Upon update, the refinement points are used to build the response surface and are taken into account for the generation of verification points. Along with DOE points, refinement points are also used as "learning points" for Goodness of Fit calculations.

Performing a Manual Refinement

Manual refinement is a way to force the response surface to take into account points of your choice, in addition to the points already in the Design of Experiments. You can insert a refinement point in the Refinement Points table, and you do not need to solve for an initial response surface (without the refinement point) before updating it with your manual refinement. Manual refinement is available for all response surface types except for Sparse Grid.

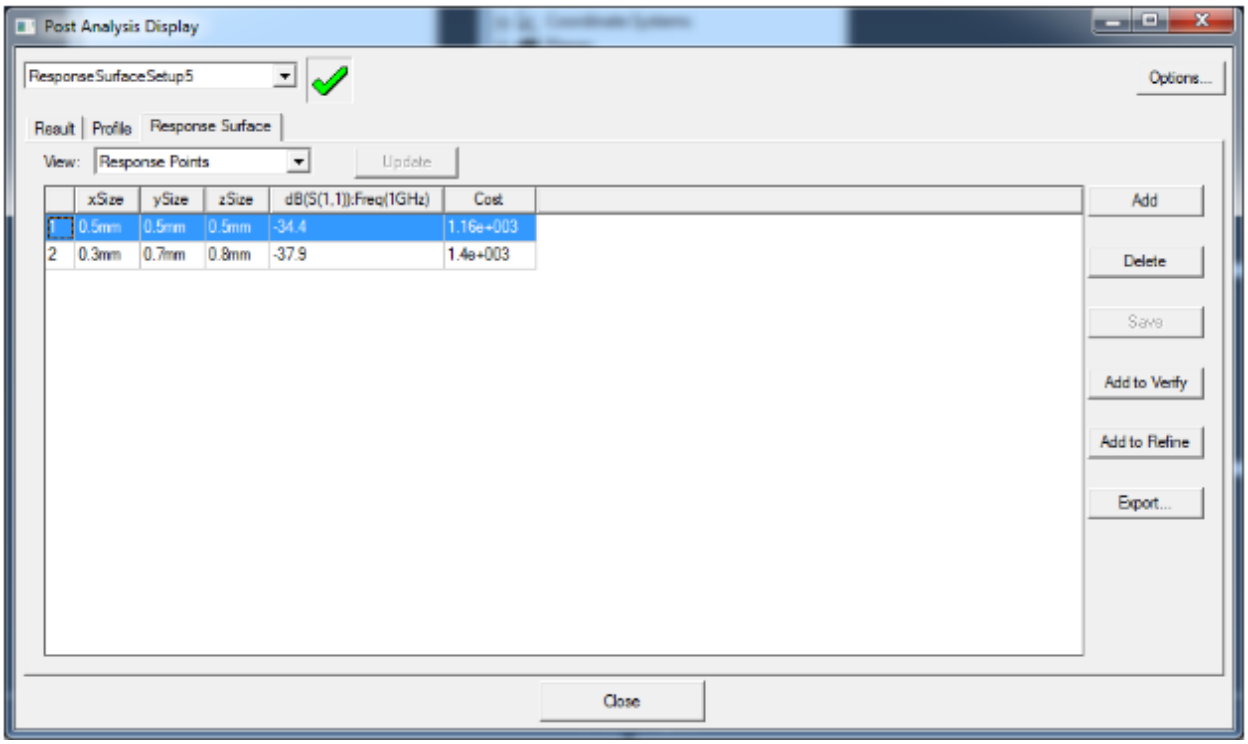
You can add, delete, or modify refinement points by clicking the **Add** or **Delete** buttons, or by modifying point values directly in the grid.

Manual Refinement Point can be inserted from the Response point table and Verification Points Table.

Response Points Table

From the Response Surface tab, the **View** list box lets you select all available views of the selected response-surface-setup. All Response points are shown in the Response Points table:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A response point is defined by a snapshot of variable values where output calculation values were calculated in Ansys DesignXplorer from a response surface. As such, the output calculation(or cost) values are approximate and calculated from response surfaces.

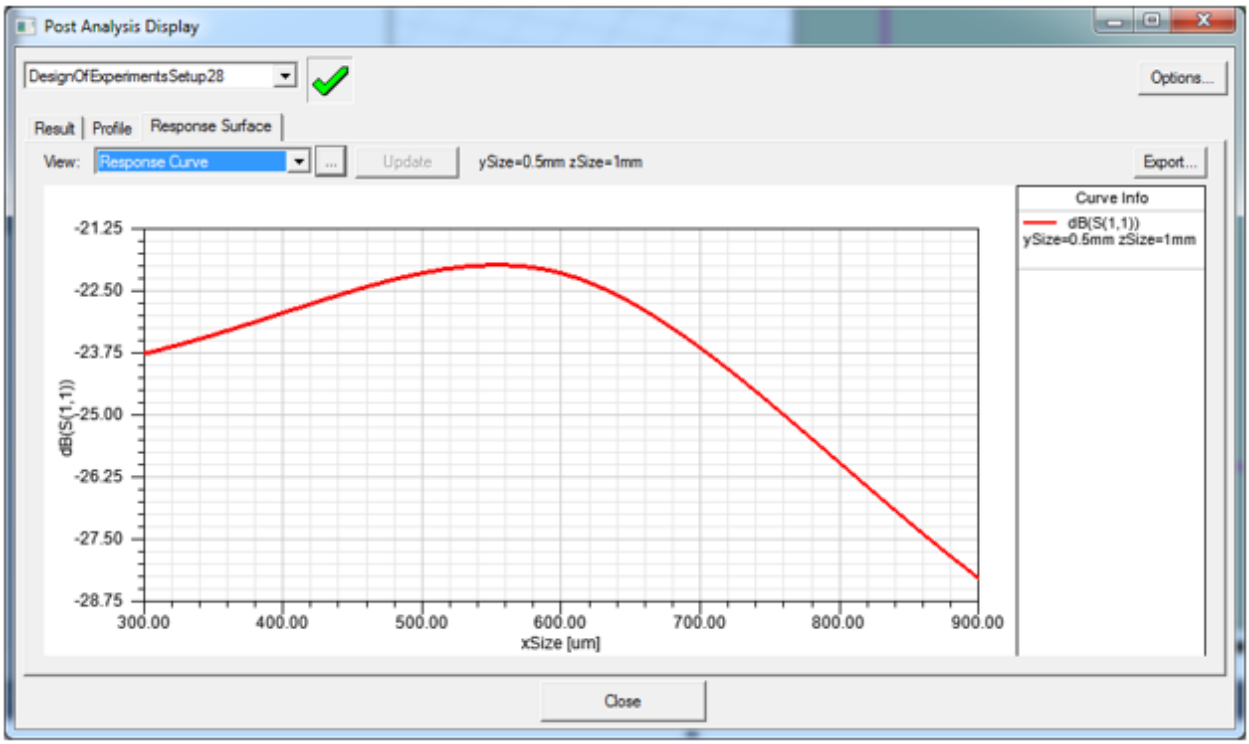
You can add, delete, save, or export response points by using the command buttons in the dialog or you can modify response points manually by modifying point values directly in the grid.

Click **Add to Verify** or **Add to Refine** buttons to insert the selected response point to the verification table or refinement table.

Response Curve

When Continuous Variable is chosen as the X-axis, and Continuous XY plot will be shown.

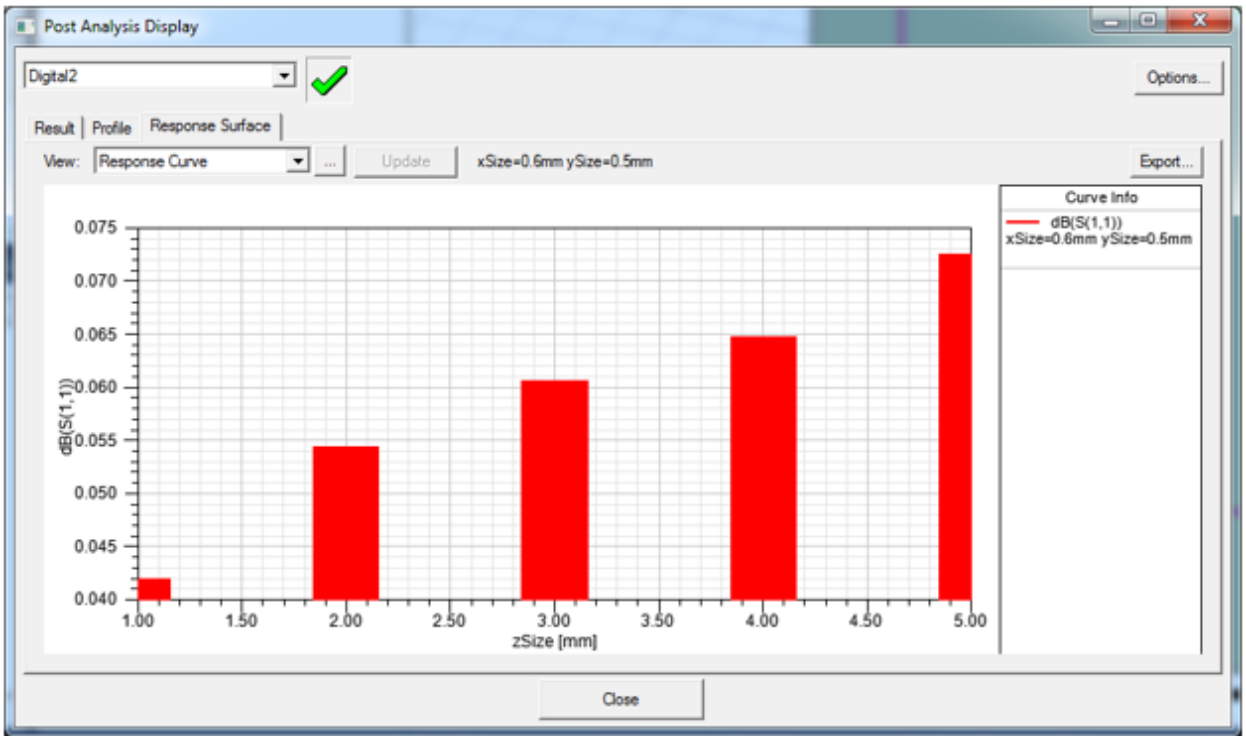
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Discrete Variable

If you choose Discrete Variable as the X-axis, a bar chart plot is shown:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tuning a Response Curve

For the X Axis and Y Axis, you can specify a Resolution, and the variable to use. For the Z Axis you can select the Cost or calculation. For variables not selected for the X and Y axis, a slider is enabled that lets you adjust the value to see the effect on the response curve plot. You can enable or disable **Real time mode** by using the check box at the lower left.

Full Range Axis check box

When it is checked all Axes are set to their maximum ranges, and the ranges won't be changed while tuning unless you change the axis variable.

When it is unchecked, the Y(2D)/Z(3D) axis range is auto updated to fit the curve/surface.

Accumulate Response Curve

It always be unchecked when the tuning dialog started, and won't restore it last check/uncheck state. When it is checked, it will retaining the existing curves, and add new curve to the plot. Uncheck the check box won't clear the accumulated curves, just stop to accumulate new curve. When the Axis variable is changed, all accumulated curves will be cleared.

Exporting Response Curve Data

You can Export the response curve data as a table in the following formats:

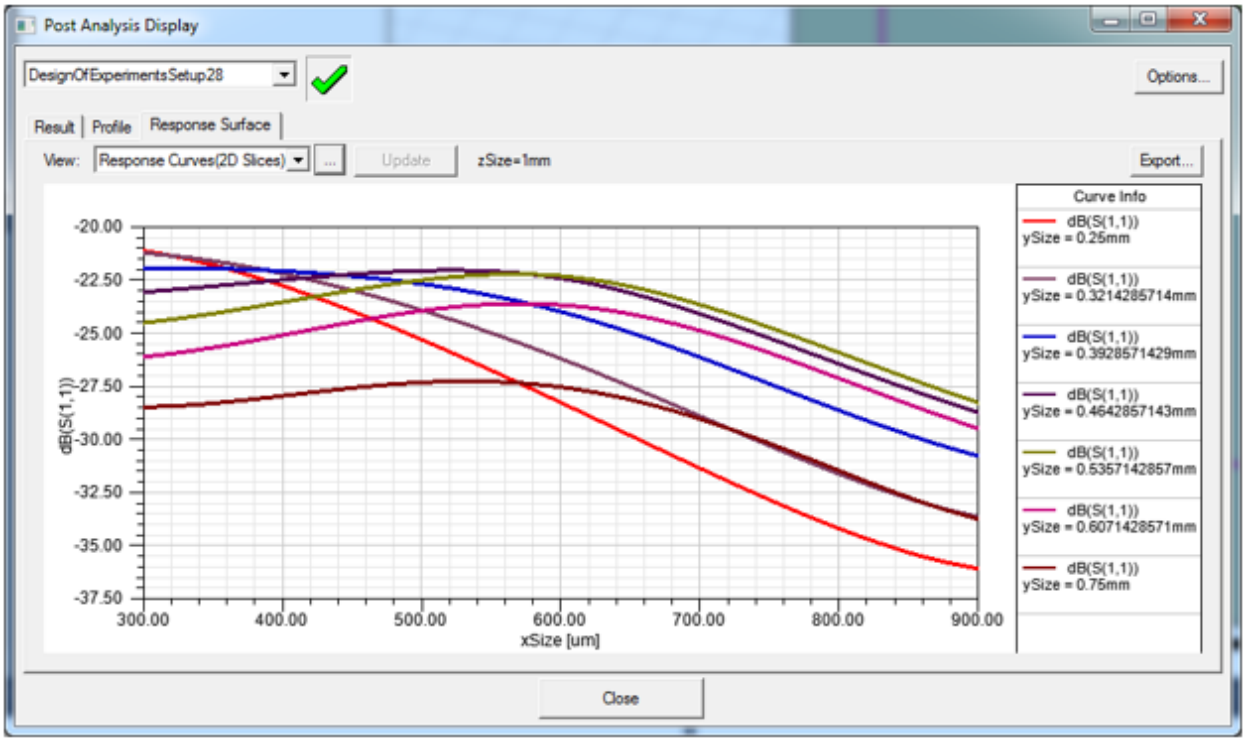
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansys Plot data files, (*.dat)
- Post Processor format data files, (*.txt)
- Ansys Report Data *.rdat files

Exported files can be imported into a report.

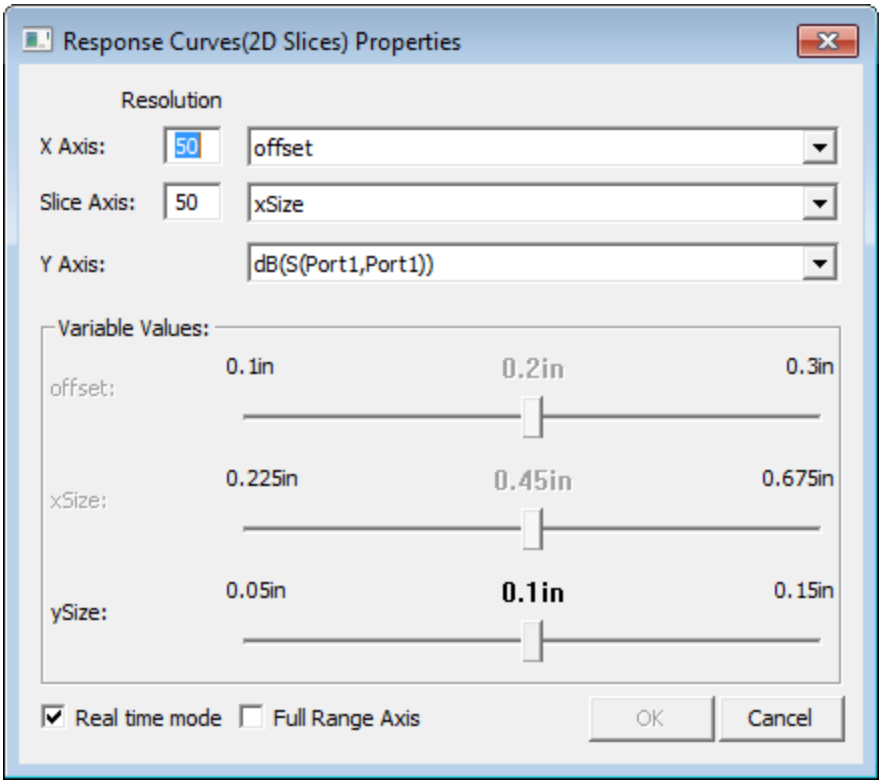
Response Curve (2D Slices)

If you have included more than one variable in the setup, the Response Curves (2D) view is available:



Clicking the ellipsis button [...] by the View for Response Surface opens a dialog that lets you adjust the variables selected and the values applied.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tuning a Response Curve

For the X Axis and Y Axis, you can specify a Resolution, and the variable to use. For the Z Axis you can select the Cost or calculation. For variables not selected for the X and Y axis, a slider is enabled that lets you adjust the value to see the effect on the response curve plot. You can enable or disable **Real time mode** by using the check box at the lower left.

Full Range Axis check box

When it is checked all Axes are set to their maximum ranges, and the ranges won't be changed while tuning unless you change the axis variable.

When it is unchecked, the Y(2D)/Z(3D) axis range is auto updated to fit the curve/surface.

Exporting Response Curve Data

You can Export the response curve data as a table in the following formats:

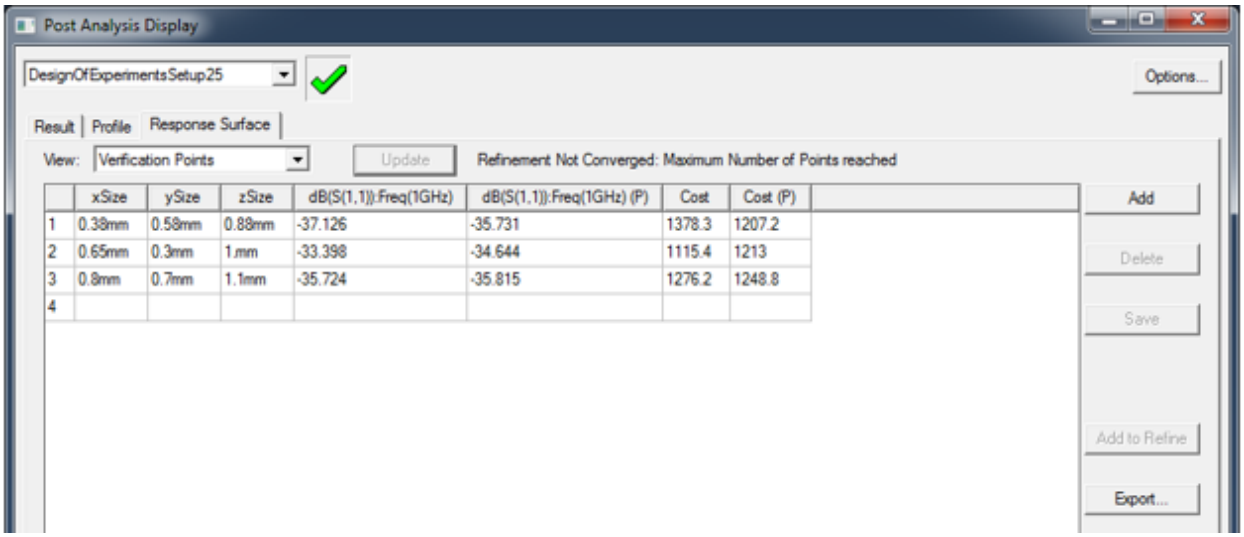
- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansys Plot data files, (*.dat)
- Post Processor format data files, (*.txt)
- Ansys Report Data *.rdat files

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Exported files can be imported into a report.

Verification Points Table

From the Response Surface tab, the **View** list box lets you select all available views of the selected response-surface-setup. All Verification points are shown in the Verification Points table:



Verification points enable you to verify that the response surface accurately approximates the output parameter values; they compare the predicted and observed values of the output parameters.

You can add, delete and modify Verification points manually:

- Same as add/delete/modify refinement points.
- Insert from Response Points Table.

Click **Add to Refine** button to insert the selected response point to the refinement table.

A design point update (that is, a "real solve") calculates each verification point. These verification point results are then compared with the response surface predictions and the difference is calculated.

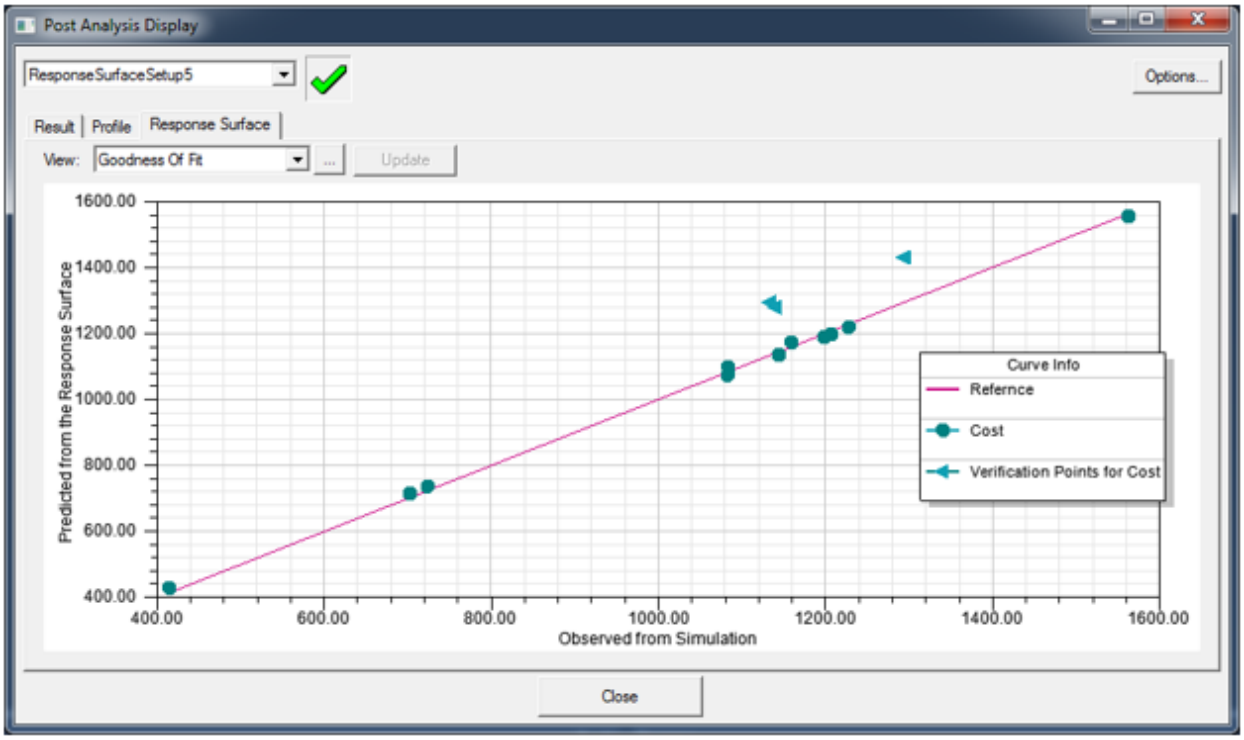
Verification points are useful in validating any type of response surface. In particular, however, you should always use verification points to validate the accuracy of interpolated response surfaces, such as Kriging or Sparse Grid.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Goodness of Fit (Predicted vs Observed Chart)

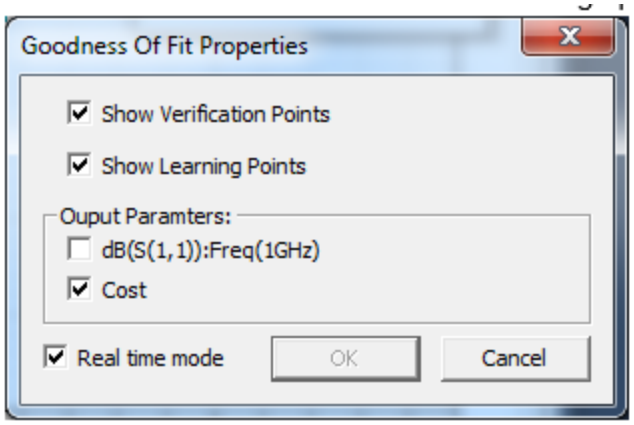
From the Response Surface tab, the **View** list box lets you select all available views of the selected response-surface-setup.Response surfaces are built from design points in the Design of Experiments (DOE) and refinement points (collectively, called "learning points"). The Goodness of Fit calculations compare the response surface outputs with the DOE results used to create them.

The closer the points are to the diagonal line, the better the response surface fits the points.



You can view Goodness of Fit information for any of the output parameters in a response surface. To do so click the ellipsis button [...] button to bring up this dialog box:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

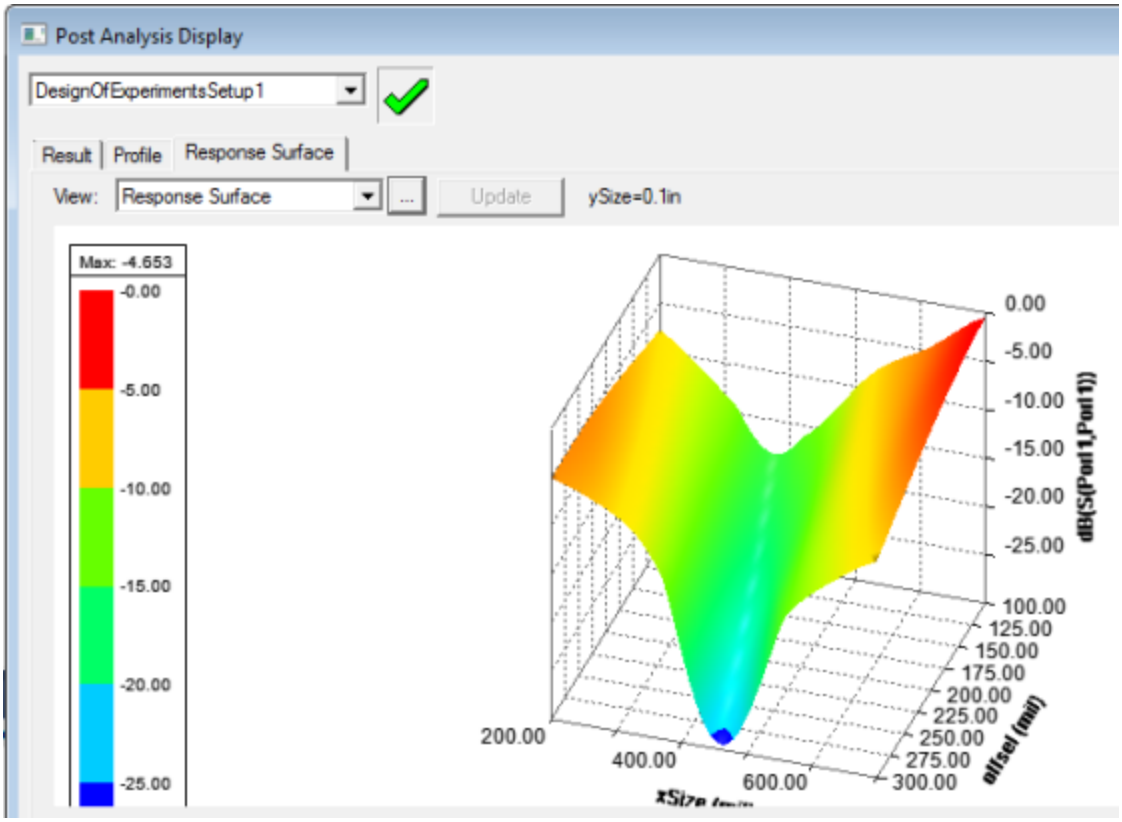


Response Surface Results

When more than one variable is included in the setup, the Response Surface view is available. You can choose any two variables as the X, Y axis, and choose an output calculation as the Z axis, by clicking the "..." button next to the view list box. From the **Response Surface** tab of the Design of Experiments **Post Analysis Display** dialog box, the "View" list box lets you select all available views of the selected response-surface-setup.

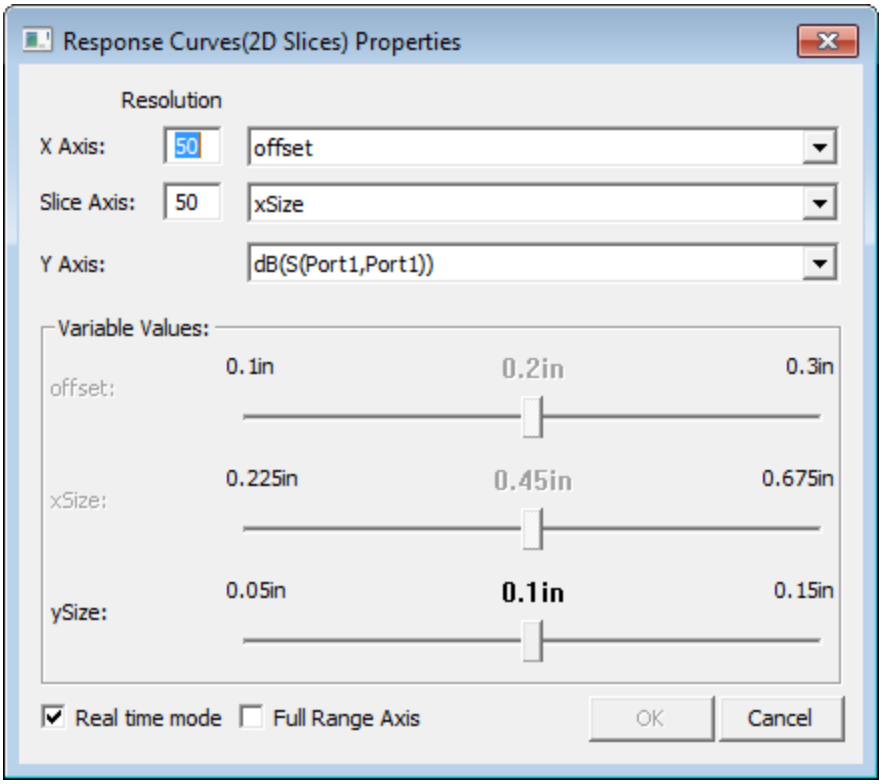
Response surfaces are functions of varying natures in which the output parameters are described in terms of the input parameters. Built from the Design of Experiments (DOE), they quickly provide the approximated values of the output parameters throughout the design space without having to perform a complete solution. The accuracy of a response surface depends on several factors: the complexity of the variations of the solution, the number of points in the original DOE, and the response surface type. Once a response surface has been generated, you can create and manage response points and charts. These postprocessing tools help you to understand how each output parameter is driven by input parameters and how you can modify your design to improve its performance.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Clicking the ellipsis button [...] by the View for Response Surface opens a dialog that lets you adjust the variables selected and the values applied.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tuning a Response Surface

For the X Axis and Y Axis, you can specify a Resolution, and the variable to use. For the Z Axis you can select the Cost or calculation. For variables not selected for the X and Y axis, a slider is enabled that lets you adjust the value to see the effect on the response surface plot. You can enable or disable **Real time mode** by using the check box at the lower left.

Full Range Axis check box

When it is checked, all Axes are set to their maximum ranges, and the ranges won't be changed while tuning unless you change the axis variable.

When it is unchecked, the Y(2D)/Z(3D) axis range is auto updated to fit the curve/surface.

Exporting Response Surface Data

You can Export the Response surface data as a table in the following formats:

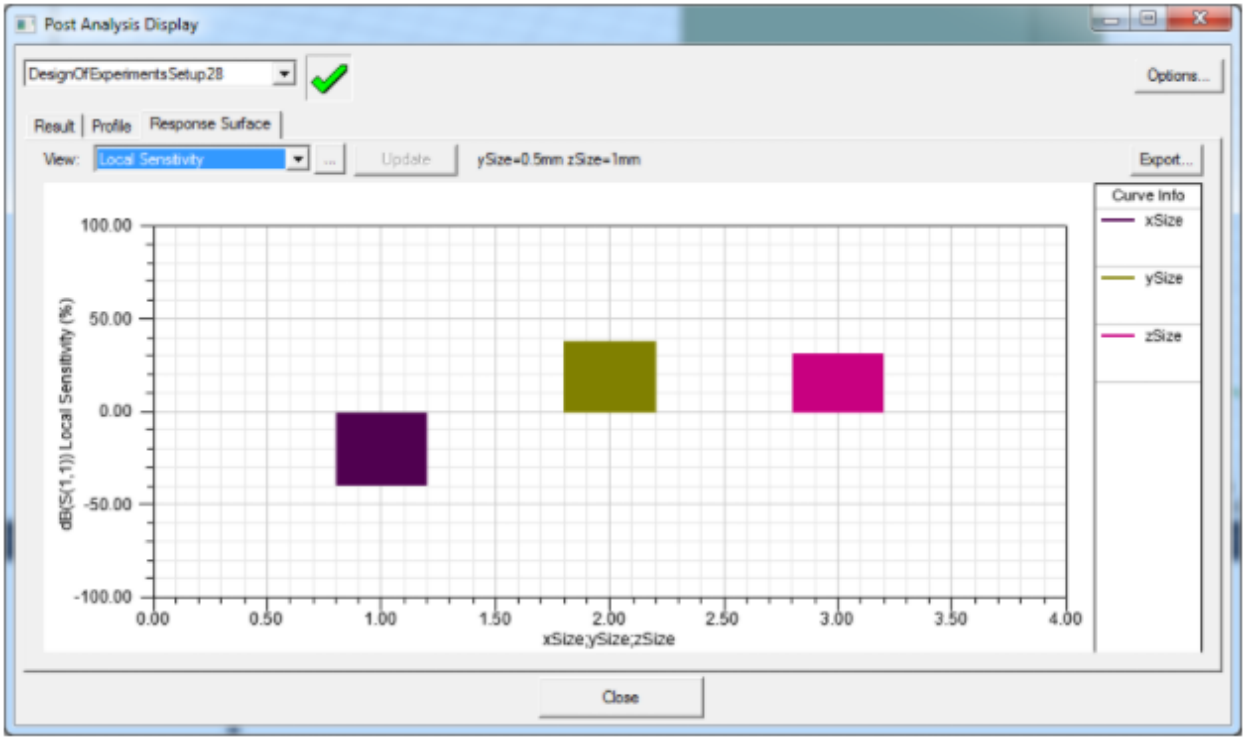
- Comma delimited data files, (*.csv)
- Tab delimited data files, (*.tab)
- Ansoft Plot data files, (*.dat)
- Post Processor format data files (*.txt)

Exported files can be imported into a report.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

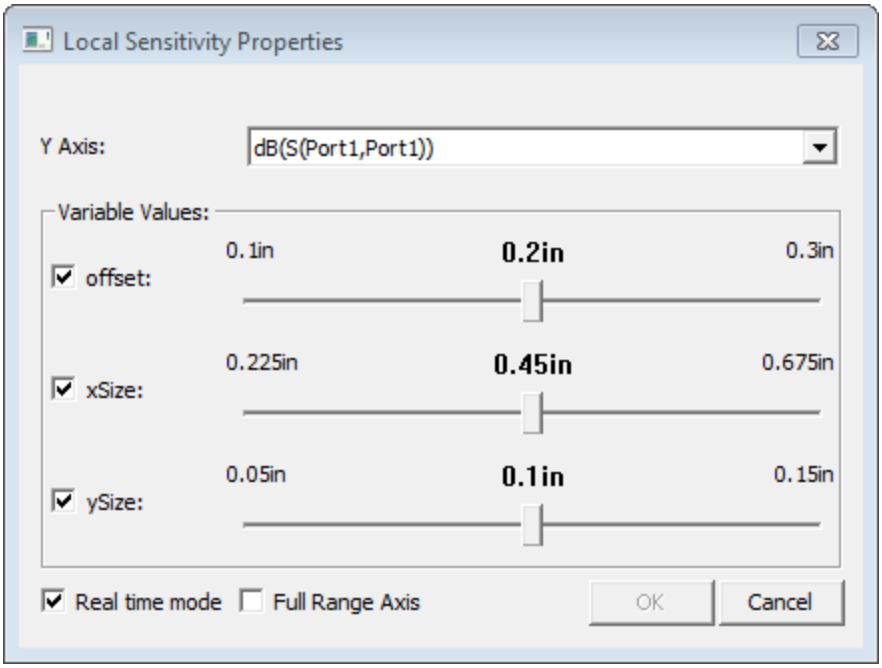
Local Sensitivity Charts

From the **Response Surface** tab, the **View** list box lets you select all available views of the selected response-surface-setup. Local Sensitivity Charts allow you to graphically view the impact that changing each input parameter has on the output parameters. The Local Sensitivity chart can be a powerful exploration tool. For each output, it allows you to see the weight of the different input. This chart calculates the change of the output based on the change of each input independently, at the current value of each input parameter in the project.



Clicking the ellipsis button [...] opens the **Local Sensitivity Properties** dialog that lets you adjust the variables selected and the values applied.

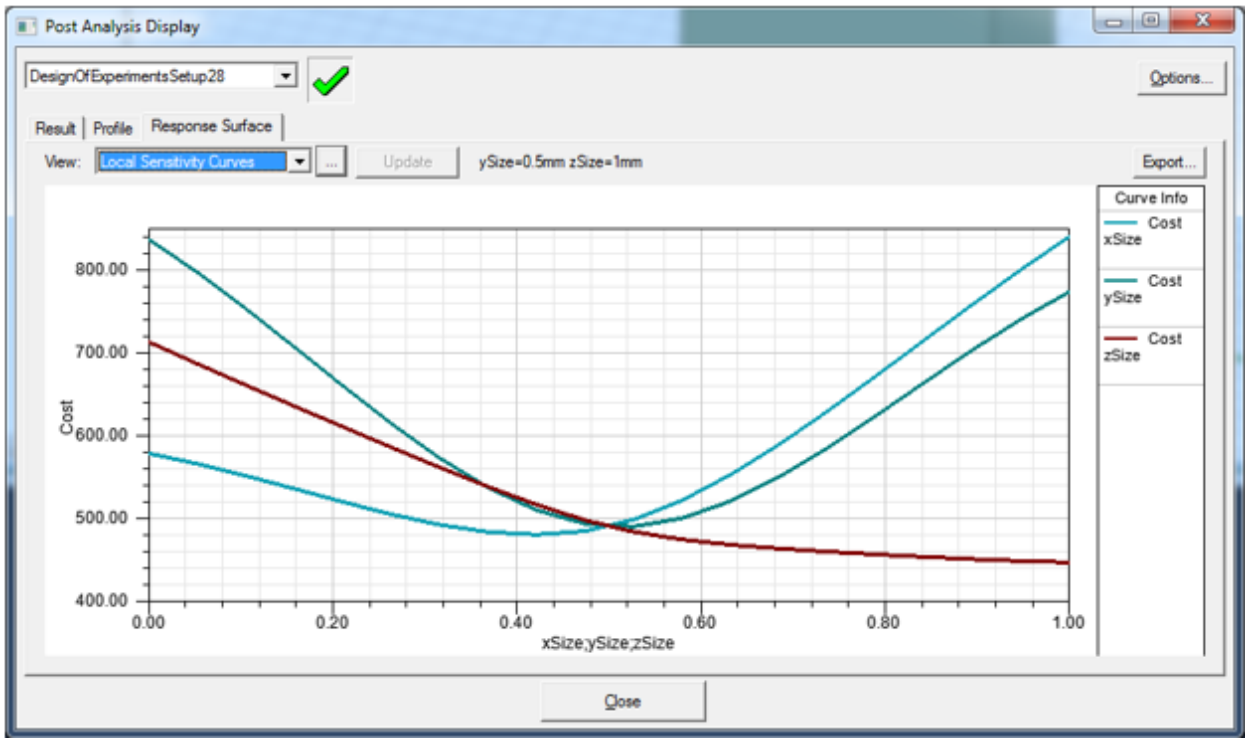
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Local Sensitivity Curves

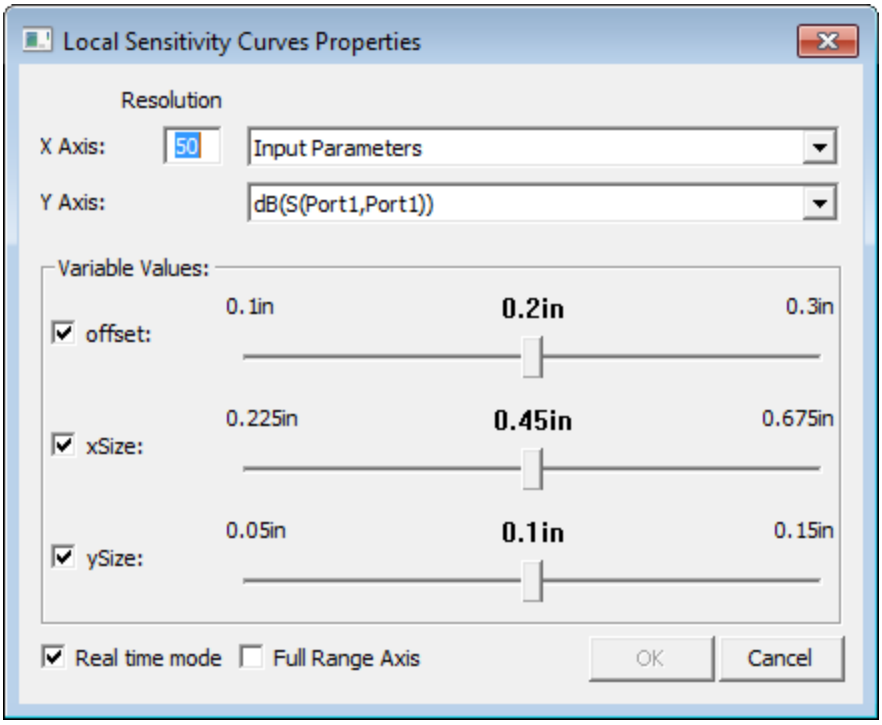
From the **Response Surface** tab, the **View** list box lets you select all available views of the selected response-surface-setup. Local Sensitivity Curves allow you to graphically view the impact that changing each input parameter has on the output parameters. The Local Sensitivity Curves chart helps you to focus your analysis by allowing you to view independent parameter variations within the standard Local Sensitivity chart. This multi-curve chart provides a means of viewing the impact of each input parameter on specific outputs, given the current values of the other parameters. The Local Sensitivities Curves chart shows individual local sensitivities, with a separate curve to represent the impact of each input on one or two outputs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Clicking the ellipsis button [...] opens the **Local Sensitivity Properties** dialog box that lets you adjust the variables selected and the values applied.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tuning Overview

Tuning a variable is useful when you want to manually modify its value and immediately perform an analysis of the design. For example, it is useful after performing an optimization analysis, in which Optimetrics has determined an optimal variable value, and you want to fine tune the value to see how the design results (for example, traces in a report) are affected.

A design can be updated after a tuning analysis to reflect a design variation solved during a tuning analysis and the results, including field solutions if you select **Save Fields and Mesh** on the **Options** tab of the associated setup dialog box.

Tuning a Variable

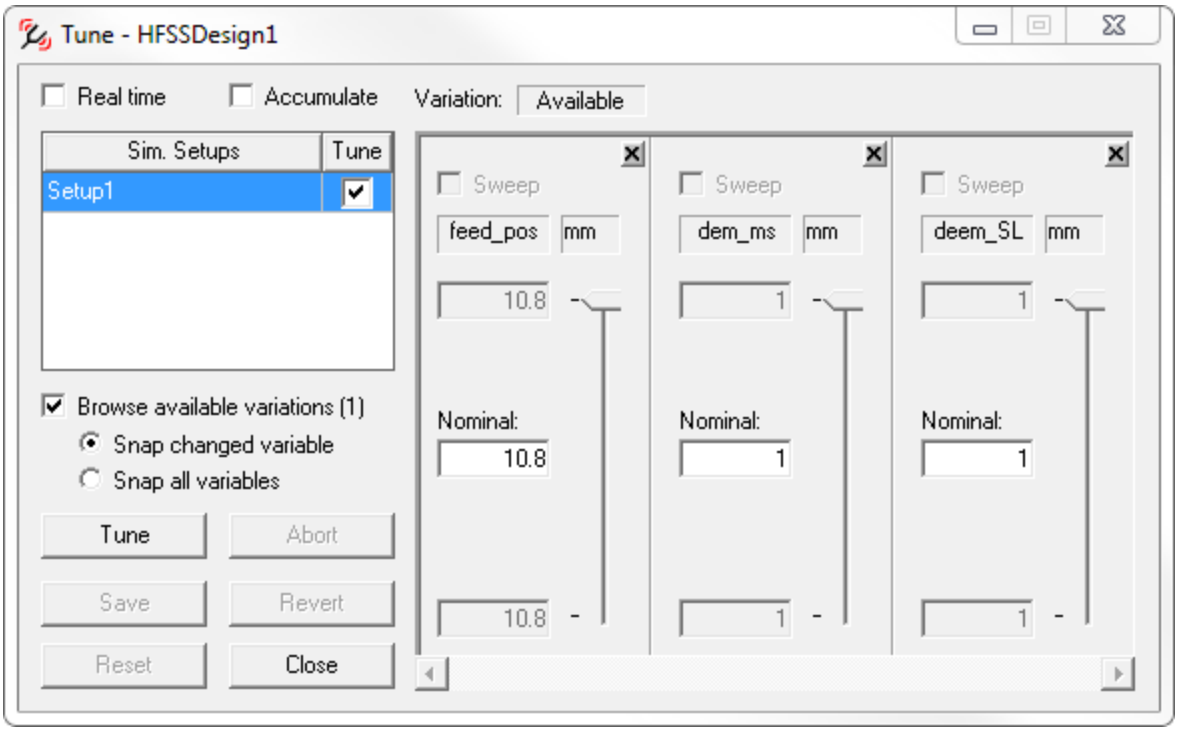
If you want to ensure that tuning does not resolve variations already solved by an optimization setup, you must check **Save Fields and Mesh** in the **Options** tab of that setup.

1. Before a variable can be tuned, you must specify that you intend for it to be used during a tuning analysis in a Project or Design **Properties** dialog box.
2. After running the simulation, click the product in the menu bar and then select **Tune**.

   Alternatively, right-click **Optimetrics** in the Project Manager and choose **Tuning** from the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Tune** dialog box appears, listing the variables which have been included for tuning.



3. Clear the **Real Time** option.

   Clearing the **Real Time** option enables the **Tune** button. If this option is selected, a simulation begins immediately after you move the slider. Otherwise, you use the **Tune** button to apply the current values to a simulation.

4. If you want to see updates to an open Report plot while tuning a post processing variable, you must select the **Browse available variations** check box. Selecting **Browse available variations** disables the sweep check box, and the fields for minimum and maximum variable values. This feature lets you see the effect of changes to the post processing variables on plotted results.

   Clearing **Browse available variations** enables the Sweep check box, the minimum and maximum fields, and changes the **Nominal** field to **Step**. (See step 6.)

5. In the **Sim. Setups** column, select the solution setup you want to use when it solves the specified design variation.

   The analysis is solved using the solution setup you select. If you select more than one, results are generated for all selected solution setups.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Checking the Tune box for a Sim Setup enables the Real Time check box, the Browse available variations check box, and the Snap radio buttons. Clearing the Tune box disables those selections.

6. In the **Nominal** text box for the variable you want to tune, type the value of the variable you want to solve, or drag the slider to increase or decrease its value.

   > **Warning:**
   >
   > Variable values must be single real numbers, or expressions that evaluate to single real numbers. Complex numbers cannot be used as the values of variables in any optimetric analysis.

   Alternatively, if you want to solve a range of values, specify a linear range of values with a constant step size:

   a. Select the **Sweep** check box. (You must have cleared the **Browse available variations** check box).
   b. In the text box below the **Step** value, type the starting value in the variable range.
   c. Type the step size, or difference between variable values in the sweep definition, in the **Step** text box. The step size determines the number of design variations between the start and stop values. The model is solved at each step in the specified range, including the start and stop values.
   d. In the text box just below the variable name, type a stopping value in the variable range.

7. If you have cleared the Real Time check box, click **Tune** to apply the changes you have made to the variable values.

   > **Note:**
   >
   > Sweeping or using a complex variable is not allowed in any optimetrics setup, including optimization, statistical, sensitivity, and tuning setups.

8. Changing a variable value with the sliders of by typing in the text field enables the **Save** and **Reset** buttons.

   Clicking **Save** opens a **Save As** dialog box with a name field and an **Apply tuned values to design** check box.

   Clicking **Reset** changes the variable values back to what they were originally.

9. If you have changed one or more included variables, clicking **Close** on the **Tuning** dialog box opens the **Apply Tuned Variation** dialog box. This lists the included variables and the values for each tuning. If you have tried multiple values, they are listed, and the current

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

value is highlighted. Select another value to change the highlight. Click **OK** to apply the highlighted values to the design, or **Don't Apply** to ignore the changes from the original variable values.

If you have applied variant values, you should see the new values listed in the relevant Design or Project Properties lists of variables and values, and if the changes affect plots or physical features of a model, those changes should also appear.

Click **Cancel** to close the dialog box and go back to the **Tune** dialog box.

Applying a Tuned State to a Design

You can apply the variable values solved during a tuning analysis to the nominal design in one of the following three ways:

- When closing the **Tune** dialog box
- When saving a tuned state
- When reverting to a tuned state

When closing the **Tune** dialog box:

1. Click **Close** to exit the **Tune** dialog box.

   The **Apply Tuned Variation** dialog box appears.

2. Click the design variation you want to apply, and then click **OK**.

   The variable values from the solved design variation become the current variable values for the nominal design. If you have applied variant values, you should see the new values listed in the relevant Design or Project Properties lists of variables and values, and if the changes affect plots or physical features of a model, those changes should also be apparent.

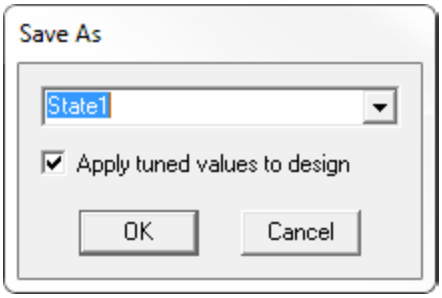
Saving a Tuned State

You can save the settings in the **Tune** dialog box, including the variable values you specified for a tuning analysis. Saved states are only available during the current session of the **Tune** dialog box; they are not stored for the next session.

1. After tuning a variable, click **Save** in the **Tune** dialog box.

   A **Save As** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Type a name for the tuned state in the text box.
3. Select **Apply tuned values to design** if you want to update the model to the new variable values.
4. Click **OK** to return to the **Tune** dialog box.

Reverting to a Saved Tuned State

You can revert to a group of saved settings in the **Tune** dialog box, including the variable values you specified for a specific tuning analysis. Saved states are only available during the current session of the **Tune** dialog box; they are not stored for the next session.

1. In the **Tune** dialog box, click **Revert**.

   The **Revert** dialog box appears.

2. Type the name of the tuned state you want to apply or click a name in the drop-down menu.
3. Select **Apply tuned values to design** if you want to update the model to the selected tuned state's variable values.
4. Click **OK** to return to the **Tune** dialog box.

Resetting Variable Values after Tuning

If you want to reset variable values to the values they were set to when you started the current session of the **Tune** dialog box:

- After tuning a variable, click **Reset** in the **Tune** dialog box.

  Solutions for the design variations solved during tuning analyses remain available for post processing.

Saving Field Solutions for Optimetrics Analyses

In order to preserve disk space, by default Ansys Electronics Desktop does not save field solution data for every solved design variation in an optimization analysis. It only saves the field

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solutions for the nominal design when an adaptive analysis is specified in the solution setup or when you request that fields be saved for each solved point in a sweep. If the nominal design is not included in the optimization analysis, all field solutions are deleted.

You can change the general options for a particular design type (HFSS, Q3D, and so on) so that any new Optimetrics setup you define will default to saving the fields. The instructions to do so are included on this page.

Alternatively, you can choose to save the fields on an individual setup basis, without enabling the option to save fields by default. See the subtopics listed in the Related Topics section at the bottom of this page for instructions to do so for each type of optimetrics setup.

To save the fields for all design variations, change the default setting for all projects:

1. Using the menu bar, select **Tools** > **Options** > **General Options**, or from the **Desktop** ribbon tab, click **General Options**.

   The **Options** dialog box appears.

2. In the tree on the left side of the dialog box, expand the branch for the product/design type of interest (HFSS, Q3D, Mechanical, and so on).
3. Select **General**.

   The appropriate group of options appears on the right side of the dialog box.

4. Select **Save Optimetrics field solutions**.
5. Repeat steps 2 through 4 for all products in which you want to save field results by default for all new Optimetrics setups.

   The **Save Fields** option will now be selected by default when you create a new Optimetrics setup in each product for which you selected the **Save Optimetrics field solutions** option.

Saving Field Solutions for a Parametric Setup

In order to preserve disk space, by default Ansys Electronics Desktop does not save field solution data for every solved design variation in a parametric setup. It only saves the field solutions for the nominal design. If the nominal design is not included in the parametric setup, by default field solutions will not be available.

To save the fields for all design variations solved during a parametric analysis:

1. Right-click **Optimetrics** in the Project Manager and choose **Add** > **Parametric** from the shortcut menu to create a new parametric setup.

   Alternatively, to change the settings of an existing setup, right-click the desired Parametric setup under **Optimetrics** in the Project Manager and choose **Properties**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Setup Sweep Analysis** dialog box appears.

2. Select the **Options** tab.
3. Select **Save Fields And Mesh**. Optionally, select **Copy geometrically equivalent meshes**, and choose whether to **Solve with copied meshes only**, or to **Solve with copied meshes and continue adaptive passes**.

   Ansys Electronics Desktop will save the field solution data for every solved design variation in the parametric setup.

> **Note:**
>
> Alternatively, you can modify the Ansys Electronics Desktop General Options to save fields by default for any new Optimetrics setup. See: Saving Field Solutions for Optimetrics Analyses for instructions.
>
> Changing the default setting in the General Options has no effect on previously defined Optimetrics setups.

Saving Field Solutions for an Optimization Setup

In order to preserve disk space, by default Ansys Electronics Desktop does not save field solution data for every solved design variation in an optimization analysis. It only saves the field solutions for the nominal design when an adaptive analysis is specified in the solution setup or when you request that fields be saved for each solved point in a frequency sweep. If the nominal design is not included in the optimization analysis, all field solutions are deleted.

To save the fields for all design variations solved during an optimization analysis:

1. Right-click **Optimetrics** in the Project Manager and choose **Add** > **Optimization** from the shortcut menu to create a new optimization setup.

   Alternatively, to change the settings of an existing setup, right-click the desired Optimization setup under **Optimetrics** in the Project Manager and choose **Properties**.

   The **Setup Optimization** dialog box appears.

2. Select the **Options** tab.
3. Select **Save Fields And Mesh**. Optionally, select **Copy geometrically equivalent meshes**, and choose whether to **Solve with copied meshes only**, or to **Solve with copied meshes and continue adaptive passes**.

   Ansys Electronics Desktop will save the field solution data for every solved design variation in the optimization setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Alternatively, you can modify the Ansys Electronics Desktop General Options to save fields by default for any new Optimetrics setup. See: Saving Field Solutions for Optimetrics Analyses for instructions.

Changing the default setting in the General Options has no effect on previously defined Optimetrics setups.

Saving Field Solutions for a Sensitivity Setup

In order to preserve disk space, by default Ansys Electronics Desktop does not save field solution data for every solved design variation in a sensitivity analysis. It only saves the field solutions for the nominal design when an adaptive analysis is specified in the solution setup or when you request that fields be saved for each solved point in a frequency sweep. If the nominal design is not included in the sensitivity analysis, all field solutions are deleted.

To save the fields for all design variations solved during a sensitivity analysis:

1. Right-click **Optimetrics** in the Project Manager and choose **Add** > **Sensitivity** from the shortcut menu to create a new sensitivity analysis setup.

   Alternatively, to change the settings of an existing setup, right-click the desired Sensitivity analysis setup under **Optimetrics** in the Project Manager and choose **Properties**.

   The **Setup Sensitivity Analysis** dialog box appears.

2. Select the **Options** tab.
3. Select **Save Fields And Mesh**. Optionally, select **Copy geometrically equivalent meshes**, and choose whether to **Solve with copied meshes only**, or to **Solve with copied meshes and continue adaptive passes**.

   Ansys Electronics Desktop will save the field solution data for every solved design variation in the sensitivity analysis.

Note:

Alternatively, you can modify the Ansys Electronics Desktop General Options to save fields by default for any new Optimetrics setup. See: Saving Field Solutions for Optimetrics Analyses for instructions.

Changing the default setting in the General Options has no effect on previously defined Optimetrics setups.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Saving Field Solutions for a Statistical Setup

In order to preserve disk space, by default Ansys Electronics Desktop does not save field solution data for every design variation solved in a statistical analysis. It only saves the field solutions for the nominal design when an adaptive analysis is specified in the solution setup or when you request that fields be saved for each solved point in a frequency sweep. If the nominal design is not included in the statistical analysis, all field solutions are deleted.

To save the fields for all design variations solved during a statistical analysis:

1. Right-click **Optimetrics** in the Project Manager and choose **Add** > **Statistical** from the shortcut menu to create a new statistical analysis setup.

   Alternatively, to change the settings of an existing setup, right-click the desired Statistical analysis setup under **Optimetrics** in the Project Manager and choose **Properties**.

   The **Setup Statistical Analysis** dialog box appears.

2. Select the **Options** tab.
3. Select **Save Fields And Mesh**. Optionally, select **Copy geometrically equivalent meshes** and choose whether to **Solve with copied meshes only**, or to **Solve with copied meshes and continue adaptive passes**.

   Ansys Electronics Desktop will save the field solution data for every solved design variation in the statistical analysis.

> **Note:**
>
> Alternatively, you can modify the Ansys Electronics Desktop General Options to save fields by default for any new Optimetrics setup. See: Saving Field Solutions for Optimetrics Analyses for instructions.
>
> Changing the default setting in the General Options has no effect on previously defined Optimetrics setups.

Saving Field Solutions for a Design of Experiments Setup

In order to preserve disk space, by default Ansys Electronics Desktop does not save field solution data for every solved design variation in a Design of Experiments setup. It only saves the field solutions for the nominal design. If the nominal design is not included in the Design of Experiments setup, by default field solutions will not be available.

To save the fields for all design variations solved during a Design of Experiments analysis:

1. Right-click **Optimetrics** in the Project Manager and choose **Add** > **Design of Experiments** from the shortcut menu to create a new Design of Experiments setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternatively, to change the settings of an existing setup, right-click the desired Design of Experiments setup under **Optimetrics** in the Project Manager and choose **Properties**.

The **Design of Experiments Setup** dialog box appears.

2. Select the **Options** tab.
3. Select **Save Fields And Mesh**. Optionally, select **Copy geometrically equivalent meshes** and choose whether to **Solve with copied meshes only**, or to **Solve with copied meshes and continue adaptive passes**.

   Ansys Electronics Desktop will save the field solution data for every solved design variation in the design of experiments setup.

> **Note:**
>
> Alternatively, you can modify the Ansys Electronics Desktop General Options to save fields by default for any new Optimetrics setup. See: Saving Field Solutions for Optimetrics Analyses for instructions.
>
> Changing the default setting in the General Options has no effect on previously defined Optimetrics setups.

Copying Meshes in Optimetrics Sweeps

An option in the Optimetrics Analysis setups allows you to request Ansys Electronics Desktop to copy a mesh that was calculated for one sweep variation for reuse on a geometrically equivalent sweep variation. For example, with this option selected a sweep on a scan angle would not need to generate meshes for each solution. The option is available on the setups for sweeps on parametrics, optimization, sensitivity, and statistics.

To copy and reuse meshes on geometrically equivalent parametric variations:

1. Define a variable for the kind of Optimetrics sweep you intend to set up.
2. Select Ansys Electronics Desktop and then select the appropriate **Optimetrics** > **Add** command to display the associated **Setup** dialog box.
3. Click the **Options** tab in the setup dialog box.
4. Select **Copy geometrically equivalent meshes**.
   - When this option is enabled, you can additionally select **Solve with copied meshed only** or **Solve with copied meshes and continue adaptive passes**. The **Solve with copied meshes only** option is not available for Maxwell 3D/2D magnetic and electric transient designs. It is available for HFSS Transient but does not apply for a Transient solve setup with a mesh link.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electronics Desktop copies the mesh for a particular parametric sweep for reuse on each geometrically equivalent sweep variation.

> **Note:**
>
> This option is available with all Optimetrics setups, and is applied when these analyses generate geometrically equivalent values. However, it is most relevant to parametric sweep, where such equivalences are more likely to occur.

The **Copy geometrically equivalent mesh** option is not recommended for use when the frequency is varying, since meshing is frequency-dependent. You may wish to turn this option off when the first geometrically equivalent variation requires numerous passes after the initial mesh, but the other geometrically equivalent variations require fewer additional passes, so that it is cheaper to start with the initial mesh each time.

Adding an Expression in the Output Variables Window

When you are in the **Output Variables** window (after clicking **Edit Calculation** from one of the setup analysis windows), do the following to specify an expression:

1. Type a name for the expression in the **Name** text box.
2. Do the following in the **Calculation** section of the window to insert a quantity into the expression:
    a. Select the **Report Type** and **Solution** from the drop-down menus.
    b. Select a **Category**, **Quantity**, and **Function** from the lists, and click **Insert Quantity Into Expression**.
    c. If you want to insert a specific predefined function, select one from the **Function** drop-down menu, and click **Insert Function**.
3. You can also type numbers or expression by hand directly into the **Expression** area.

Excluding a Variable from an Optimetrics Analysis

To exclude a variable from being optimized or included in a sensitivity or statistical analysis:

1. Do one of the following:
    - In the **Setup Optimization** dialog box, click the **Variables** tab.
    - In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
    - In the **Setup Statistical Analysis** dialog box, click the **Variables** tab.

    All of the independent variables that were selected for the optimization analysis are listed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Clear the **Include** option for the variable you want to exclude from the analysis.

   The **Override** option is now selected. This indicates that, for this optimization analysis, the variable is not included.

   > **Note:**
   >
   > Alternatively, you can select the **Override** option first, and then clear the **Include** option for the variable you want to exclude.

3. Click **OK**.

Modifying the Value of a Fixed Variable

If you are not including a variable in an optimization, sensitivity, or statistical analysis, Optimetrics uses that variable's current value during the analysis.

To override the current value of a fixed variable for an Optimetrics setup:

1. Do one of the following:
   - In the **Setup Optimization** dialog box, click the **Variables** tab.
   - In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
   - In the **Setup Statistical Analysis** dialog box, click the **Variables** tab.
2. Click **Set Fixed Variables**.

   The **Setup Fixed Variables** dialog box appears. Under **Fixed Variables**, all of the current independent variable values are listed.

3. Click the **Value** text box of the variable with the value you want to override.
4. Type a new value in the **Value** text box, and then press **Enter**.

   The **Override** option is now selected. This indicates that the value you entered is used for this Optimetrics setup; the current variable value set for the nominal design is ignored.

   > **Note:**
   >
   > Alternatively, you can select the **Override** option first, and then type a new value in the **Value** text box.

5. Optionally, click a new unit system in the **Units** text box.
6. Click **OK**.

To revert to a default variable value, clear the **Override** option.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Linear Constraints

Once the optimization variables are specified, the optimizer handles each of them as an *n*-dimensional vector *x*. Any point in the design space corresponds to a particular *x*-vector and to a design instance. Each design instance may be evaluated via Finite Element Analysis and assigned a cost value; therefore, the cost function is defined over the design space

$$cost(x): R^n \rightarrow R$$

where *n* is the number of optimization variables.

In practice, a solution of the minimization problem is sought only on a bounded subset of the R^n space. This subset is called the feasible domain and is defined via linear constraints.

You may constrain the feasible domain of a design variable by defining linear constraints for the optimization process. The feasible domain is defined as the domain of all design variables that satisfy all upper and lower bounds and constraints. Linear constraints are defined by the following inequalities:

$$\sum_i \alpha_{ij} x_i < c_j \forall j$$

where

- α_{ij} are coefficients.
- c_j is a comparison value for the j^{th} linear constraint.
- x_i is the i^{th} parameter.

Setting a Linear Constraint

A linear constraint defines the linear relationship between variables. Setting linear constraints in Optimetrics is useful for establishing limitations involving linear combinations of variable values.

1. Do one of the following:
   - If you are setting up an optimization analysis: In the **Setup Optimization** dialog box, click the **Variables** tab.
   - If you are setting up a sensitivity analysis: In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
2. Click **Linear Constraint**.

   The **Linear Constraint** dialog box appears.
3. Click **Add**.

   The **Edit Linear Constraint** dialog box appears.
4. Click a **Coeff** text box and type a positive or negative coefficient value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click a condition, **<** (less than) or **>** (greater than), from the drop-down menu.
6. Type the inequality value, which should be a constant value, in the text box to the right of the condition.
7. Click **OK**.

   You return to the **Linear Constraint** dialog box. The left-hand side of the constraint appears in the **LHS** (left-hand side) column. The condition is listed in the **Condition** column, and the inequality value is listed in the **RHS** (right-hand side) column.

Modifying a Linear Constraint

1. Do one of the following:
   - If you are setting up an optimization analysis: In the **Setup Optimization** dialog box, click the **Variables** tab.
   - If you are setting up a sensitivity analysis: In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
2. Click **Linear Constraint**.

   The **Linear Constraint** dialog box appears.

3. Click the row listing the constraint you want to modify, and then click **Edit**.

   The **Edit Linear Constraint** dialog box appears.

4. Optionally, click a **Coeff** text box and type a new coefficient value.
5. Optionally, click a different condition, **<** (less than) or **>** (greater than), in the pull-down list.
6. Optionally, type a different inequality value in the text box to the right of the condition, and then click **OK**.

   You return to the **Linear Constraint** dialog box. The new coefficient value, the condition, and the inequality value appear in the **LHS** (left-hand side), **Condition**, and **RHS** (right-hand side) columns, respectively.

Deleting a Linear Constraint

1. Do one of the following:
   - If you are setting up an optimization analysis: In the **Setup Optimization** dialog box, click the **Variables** tab.
   - If you are setting up a sensitivity analysis: In the **Setup Sensitivity Analysis** dialog box, click the **Variables** tab.
2. Click **Linear Constraint**.

   The **Linear Constraint** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click the row listing the constraint you want to delete, and then click **Delete**.

   The constraint is deleted.

Running an Optimetrics Analysis

Once you have created all necessary Optimetrics based analyses, you have several options for running the simulations.

- To use the **Analyze All** command at the Project or design level to simulate the nominal problem and subsequently run all Optimetrics setups, do the following:
  1. In the Project Manager window, right-click on the **project** or **design** name.
  2. Click **Analyze All** from the shortcut menu.
- To use the **Analyze All** command from the Optimetrics menu to simulate only the Optimetrics based setups, do the following:
  1. In the Project Manager window, right-click on **Optimetrics**.
  2. Click **Analyze** > **All** from the shortcut menu.
- You can choose to analyze only the setups related to a specific Optimetrics type of analysis. In order to simulate setups of a specific type, do the following:
  1. In the Project Manager window, right-click on **Optimetrics**.
  2. Click **Analyze** > **All{*TYPE*}** from the shortcut menu, where *TYPE* is the specific analysis type of interest, Parametric, Optimization, Sensitivity, or Statistical.

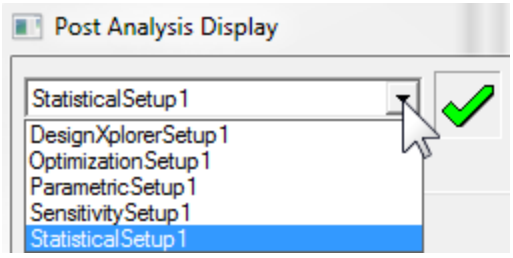
Viewing Analysis Results for Optimetrics Solutions

To view data specific to an Optimetrics solution:

1. In the project tree, right-click the Optimetrics setup for which you want to view the results, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Use the drop-down menu to select from available setups.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click the **Results** tab to view results in plot or table form.

   When you view results in Table format, you can sort the results based on each column. Click a column's header to sort. Click again to invert the current sort.

4. Click **Options...** to open a dialog box that allows you to specify the Maximum number of significant digits to display when showing the analysis result. The default is 4.
5. Select the **Profile** tab to view start, stop, and elapsed times for each variable, and the analysis machine for each variation. Click a column heading to sort the table by variation number, variable value, start, stop, elapsed time, or machine.

See the help topics in this section for more details about viewing Optimetrics analysis results.

Viewing Solution Data for an Optimetrics Design Variation

To view the convergence information, or computing resources used, or matrices computed for any design variation solved during an optimization analysis, you must first select the design variation in the **Set Design Variation** dialog box. This dialog box is accessible from the **Solutions** window and via the **Results** > **Apply Solved Variation** command.

1. Click the **HFSS** menu and then select **Results** > **Solution Data**.

   The **Solutions** dialog box appears.

2. Click the browsing dots beside the **Design Variation** box (...).

   The **Set Design Variation** dialog box appears.

3. Clear the **Use nominal design** option.
4. Click the design variation for which you want to view the solution data, and then click **OK**.

   The solution data is displayed in the table.

Viewing an Optimetrics Solution's Profile Data

At any time during or after the Optimetrics solution process, you can see an overview of the computing resources or profile data that was used by HFSS as it solved each design variation. Optimetrics writes the variation information to the profile table before the solve. It then updates the entry with end data (end time, elapsed time, etc.) once the solve variation is completed.

1. In the project tree, right-click the Optimetrics solution setup of interest, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Click the **Profile** tab.
3. Select the Optimetrics setup with the results you want to view from the drop-down menu at the top of the dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Optionally, to examine more detailed profile data for a specific design variation, do the following:
   a. Click a design variation in the table.
   b. Click **Solver Profile**.

   The **Solutions** dialog box appears with the profile data for the selected design variation.

   The profile line for the matrix solver is in the following format:

   Solver *123*

   where:

   - *1* is the precision type: M (mixed) or D (double)
   - *2* is the matrix data type: R (real) or C (complex)
   - *3* is the symmetry type: S (symmetric), A (asymmetric), H (hermitian)

Viewing Results for Parametric Solution Quantities

1. In the project tree, right-click the parametric setup for which you want to view the results calculated for the solution quantities, and then click **View Analysis Result** on the shortcut menu.

   The **Post Analysis Display** dialog box appears.

2. Select the parametric setup with the results you want to view from the drop-down menu at the top of the dialog box.
3. If it is not already selected, select **Table** as the view type.

   The results for the selected solution quantities are listed in table format for each solved design variation. The variation column in the table lists the entries in order. Clicking the Vision header inverts the order. Clicking other headers sorts the entries by value, and clicking again inverts the order.

4. Optionally, select **Show complete output name**.

   The complete name of the solution for which the results are being displayed will be listed in the column headings.

5. Optionally, click a design variation in the table, and then click **Apply** (at the far right side of the dialog box).

   The design displayed in the **Modeler** window is changed to represent the selected design variation.

Plotting Solution Quantity Results vs. a Swept Variable

To plot solution quantity results versus a swept variable's values on a rectangular (x - y) plot:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the project tree, right-click the parametric setup for which you want to view the results, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. If it is not already selected, select **Plot** as the view type.
3. Select the variable with the swept values you want to plot on the x-axis from the **X** drop-down menu.
4. Only one sweep variable at a time can be plotted against solution quantity results. Any other variables that were swept during the parametric analysis remain constant.

   Optionally, to modify the constant values of other swept variables, do the following:

   a. Click **Set Other Sweep Variables Value**.

      The **Setup Plot** dialog box appears. All of the other solved variable values are listed.

   b. Click the row with the variable value you want to use as the constant value in the plot, and then click **OK**.
5. Select the solution quantity results you want to plot on the y-axis from the **Y** drop-down menu.

   The xy plot appears in the view window.

6. Right-click in the plot area to get the shortcut menu where you can set modify the plots display properties, print, copy to the clipboard, or export the data to a file.

Viewing Cost Results for an Optimization Analysis

To view cost values versus completed iterations in data table format:

1. In the project tree, right-click the optimization setup for which you want to view the cost results, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Under the **Result** tab, select **Table** as the view type, if it is not already selected.

   The cost value at each solved design variation is listed in table format.

3. Optionally, click a design variation in the table, and then click **Apply**.

   The software now points to the selected design variation as the nominal solution and as a result, the design displayed in the **Modeler** window is changed to represent the selected design variation.

   Click **Revert** to return the design in the view window to the original value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plotting Cost Results for an Optimization Analysis

To view cost values versus completed iterations in rectangular (x-y) plot format:

1. In the project tree, right-click the optimization setup for which you want to view the cost results, and then click **View Analysis Result** on the shortcut menu.

   The **Post Analysis Display** dialog box appears.

2. Under the **Result** tab, select **Plot** as the view type.

   A plot of the cost value at each iteration appears.

Viewing Output Parameter Results for a Sensitivity Analysis

To view actual output parameter values versus design point in data table format:

1. In the project tree, right-click the sensitivity setup for which you want to view the parameter results, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Under the **Result** tab, select **Table** as the view type, if it is not already selected.

   The following values are listed in table format:

   - The regression value of the output parameter at the design point is listed in the **Func. Value** column.
   - The first derivative of the regression is listed in the **1st D** column.
   - The second derivative of the regression is listed in the **2nd D** column.

3. Click **Apply**.

   The software now points to the selected design variation as the nominal solution and as a result, the design displayed in the **Modeler** window is changed to represent the selected design variation.

   Click **Revert** to return the design in the view window to the original value.

Plotting Output Parameter Results for a Sensitivity Analysis

To plot output parameter results versus sensitivity variable values on a rectangular (xy) plot:

1. In the project tree, right-click the sensitivity setup for which you want to view the output parameter results, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Under the **Result** tab, select **Plot** as the view type.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the sensitivity variable with the sweep values you want to plot on the x-axis from the **X** drop-down menu.
4. Select the output parameter results you want to plot on the y-axis from the **Y** drop-down menu.

   The xy plot appears in the **Post Analysis Display** dialog box.

   The plot displays actual output parameter results for each solved design variation. It also displays a parabola that best fits these results. The parabola is a more accurate representation of sensitivity around the design point than any individual solved design variation.

Viewing Distribution Results for a Statistical Analysis

1. In the project tree, right-click the statistical setup for which you want to view the distribution results calculated for the solution quantities, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Select the statistical setup with the results you want to view from the drop-down menu at the top of the dialog box.
3. To view the results in tabular form, select **Table** as the view type.

   The distribution results for the selected solution quantities are listed in table format for each solved design variation.

4. Optionally, click a design variation in the table, and then click **Apply** (at the far right side of the dialog box).

   The design displayed in the **Modeler** window is changed to represent the selected design variation.

5. To view the results in graphic format, select **Plot** as the view type.
6. Type the number of bins you want to plot on the x-axis.
7. Select the solution quantity for which you want to plot distribution results on the y-axis from the **Y** drop-down menu.

   A histogram plot appears in the **Post Analysis Display** dialog box. It displays the distribution of the selected solution quantity.

8. Optionally, click a design variation in the table, and then click **Apply** (at the far right side of the dialog box).

   The software now points to the selected design variation as the nominal solution and as a result, the design displayed in the **Modeler** window changes to represent the selected design variation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click **Revert** to return the design in the view window to the original value.

Plotting Distribution Results for a Statistical Analysis

1. In the project tree, right-click the statistical setup for which you want to view the distribution results calculated for the solution quantities, and select **View Analysis Result**.

   The **Post Analysis Display** dialog box appears.

2. Select the statistical setup with the results you want to view from the drop-down menu at the top of the dialog box.
3. If it is not already selected, select **Plot** as the view type.
4. Type the number of bins you want to plot on the x-axis.
5. Select the solution quantity for which you want to plot distribution results on the y-axis from the **Y** drop-down menu.

   A histogram plot appears in the **Post Analysis Display** dialog box. It displays the distribution of the selected solution quantity.

Link to DesignXplorer

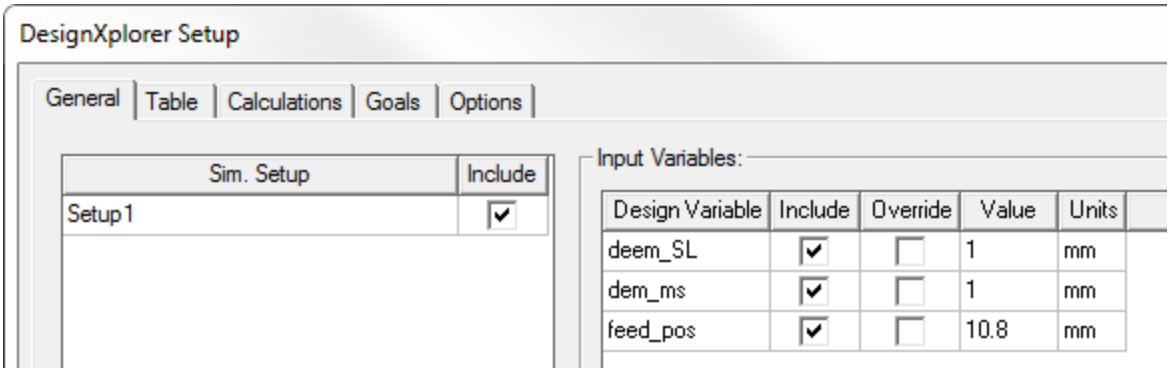
You can export a .xml file containing information on an HFSS setup, optimization variables, and output variables that enables Ansys Design Xplorer to manage the simulations (for example, for design of experiments and optimization). Design Xplorer will launch Ansys Electronics Desktop simulations of design variations and evaluate the outputs.

To do so:

1. Click your product on the menu bar and then **Optimetrics Analysis** > **Add Design Xplorer Setup** or right-click on **Optimetrics** in the **Project Manager** window, and select **Add Design Xplorer Setup** from the short-cut menu.

   This opens the **Design Xplorer** dialog box with the **General** tab selected. it lists the setups available in the current project, and the input variables it contains.

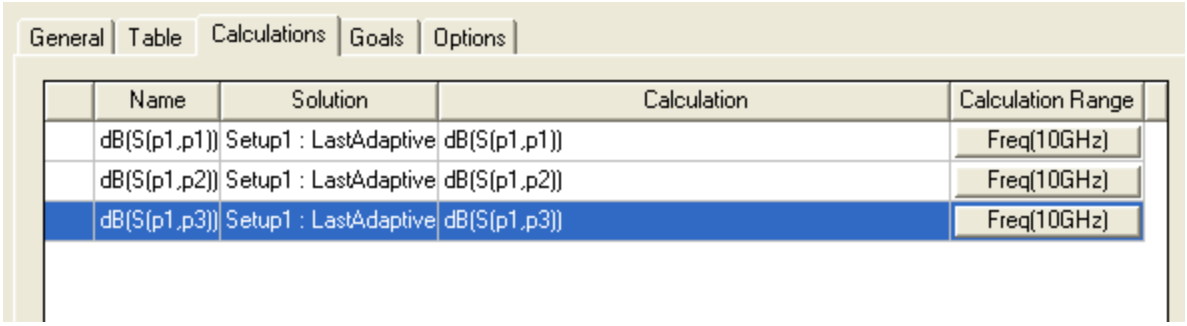
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select **Include** for the simulation setups you want to use.
3. Check the Design variables to use. You can also chose to Override the value of a design variable. You can edit the Value and Units fields. Unchecking **Override** returns the values to their original state.
4. To set up any output calculations, click the **Calculation** tab and click **Setup Calculations**.

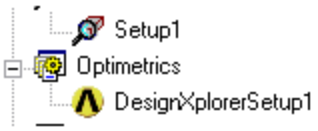
   This opens the **Add/Edit Calculation** dialog box. Here you can define the simulation results of interest. The dialog box contains distinct panes and tabs to set the **Context**, the **Calculation Expression**, and the **Calculation Range**. See: Setup Calculations for Optimetrics for details.

   Use the **Add Calculation** button to add expressions to the Calculations table.



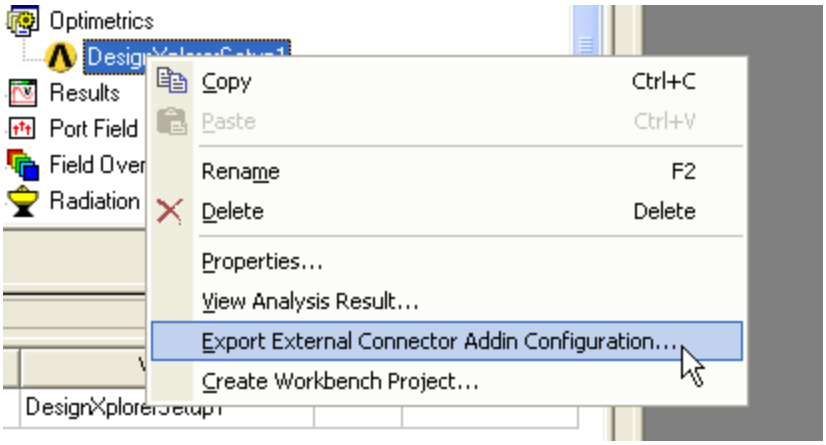
5. When you have added the calculations of interest, click **OK** to save the setup.

   An icon for the Design Xplorer setup appears under Optimization in the Project tree.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. To create a .xml file with the setup information for Design Xplorer, first Save your project.
7. Then right-click the setup and select **Export External Connector Addin Configuration**.



This displays a browser dialog that you can use to navigate your file system and name and saves the .xml file. This file contains information regarding the path along with the setup, variables, and simulation results that you specified.

8. If you have an Ansys Workbench installation, you can perform additional steps. You should have provided a path to the Workbench installation in the **Tools** > **General Options** dialog box Miscellaneous tab, to provide a path.
9. Then click **Create Workbench Project**.

   This lets you name a Workbench project containing the information in the setup. The Ansys Workbench will be launched with the connection to the project established. To this connection, you can add a Design Xplorer Setup. See the documentation of Ansys Workbench for details on Design Xplorer.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9 - High Performance Computing

Ansys Electronics Desktop affords High Performance Computing (HPC) options that allow you to speed up processing. HPC leverages multiple cores through matrix multiprocessing, distributed frequency points (called spectral decomposition method or SDM), domain decomposition (DDM), parallel hybrid FEM/IE solving or the finite antenna array DDM. In addition, hierarchical HPC solving is possible where frequency points can be distributed with each frequency point using multiple cores or machines for large scale DDM analysis at each frequency point, all in parallel. HFSS intelligently determines which jobs are to be performed and how to distribute them for the simulation. It automatically apportions the jobs during the simulation process and makes optimum use of the available resources. This computing technology enables generating accurate solutions for large, complex, higher-fidelity models.

See the following topics for more information and configuration tips:

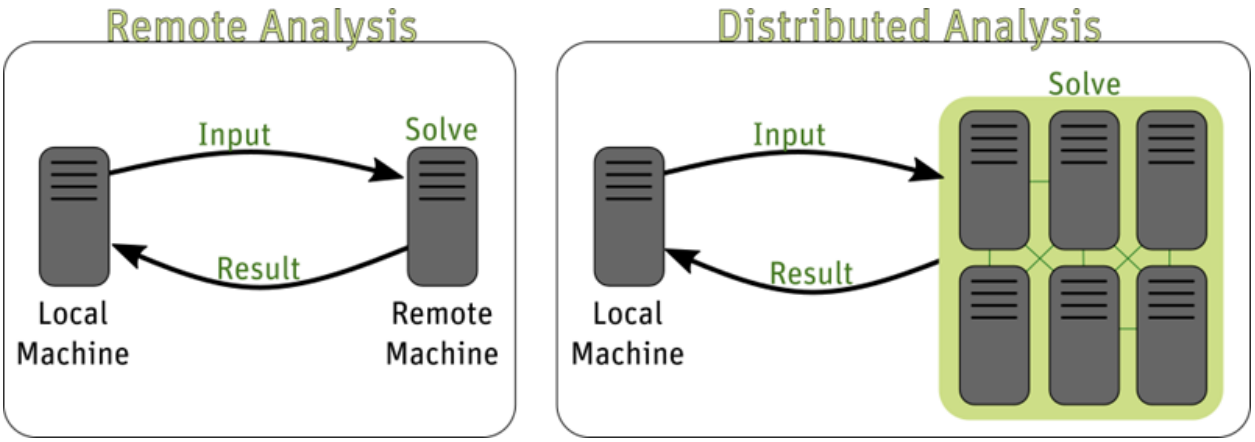
- Remote Analysis
- Distributed Analysis
- Ansys Cloud Direct Support for HPC Job Management
- Large Scale DSO for Parametric Analysis
- Interactive Scheduler Jobs
- HPC Integration
- Multi-Step Job Submission
- Windows to Linux Job Submission
- Distribution Command Line Options
- GPU Acceleration

Remote Analysis

It is possible to solve a project on a different machine from the one on which you are running Ansys Electronics Desktop. This is particularly useful when you want to take advantage of a more powerful machine but it is not convenient to access that machine. This process involves configuring the machine that will perform the solve (the remote machine), as well as the machine from which the simulation is to be launched (the local machine).

This can also be extended into distributed analysis, where a specified analysis, if supported, is concurrently solved on multiple machines.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Important:

In both Remote and Distributed Analysis, communication between machines can drastically affect performance. Use of a high-speed network system, such as Gigabit or Infiniband, is recommended for optimal performance.

The rest of this topic covers the following:

- Prerequisites for Remote and Distributed Analysis
- Configuring the Local Machine to Solve Remotely
- Remote Analysis Options
- Running Remote Analysis

The **Export Options Files** command writes XML files containing options settings at all levels to the specified directory. **Tools** > **Options** > **Export Options** makes it easier for different users to use Ansys EM tools installed on shared directories or network drives. For some example use cases, see: Example Uses for Export Options Features.

Prerequisites for Remote and Distributed Analysis

1. You must have installed Ansys Remote Simulation Manager (RSM) or a supported High Performance Computing (HPC) management software program (See: High Performance Computing (HPC) Integration).

   For a list of currently supported HPC software, select the topic, *Ansys 2025 R1 - Job Schedulers and Queuing Systems Support* from the Platform Support page.

2. Ansys Electronics Desktop must be accessible from all remote machines as well as on the local machine. If the analysis uses MPI, then the path of the Ansys Electronics Desktop installation must be the same on all of the machines used for the analysis (remote and local). This may be a shared network path accessible from all hosts. Alternatively, it may

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

be a local installation on each host; in this case the installation path must be the same for all hosts. If using a shared network path, there should not be a local installation of the same Ansys Electronics version on any of the hosts. See Distributed Analysis for information on whether MPI is needed for the analysis.

3. If you use RSM, it must be accessible from all remote machines. In addition, the Ansys Electronics Desktop engines must be registered with each initialization of RSM.

   To do this on each machine:

   - From Windows, click **Start** > **Programs** > **Ansys EM Suite 2025 R1** > **Register with RSM**. You can also run **RegisterEnginesWithRSM.exe**, located in the product subdirectory (for example, C:\Program Files\ANSYS Inc\v251\AnsysEM\RegisterEnginesWithRSM.exe).

     In each case, you see a dialog box confirming the registration. Click **OK** to confirm.

   - From Linux, run **RegisterEnginesWithRSM.pl**, located in the product installation directory (for example, /opt/ansys_inc/v251/AnsysEM/RegisterEnginesWithRSM.pl).

     If the RSM service cannot run due to permission issues for the configuration file, it issues an error message and exits.

     If your product is not registered with RSM, the analysis will run locally.

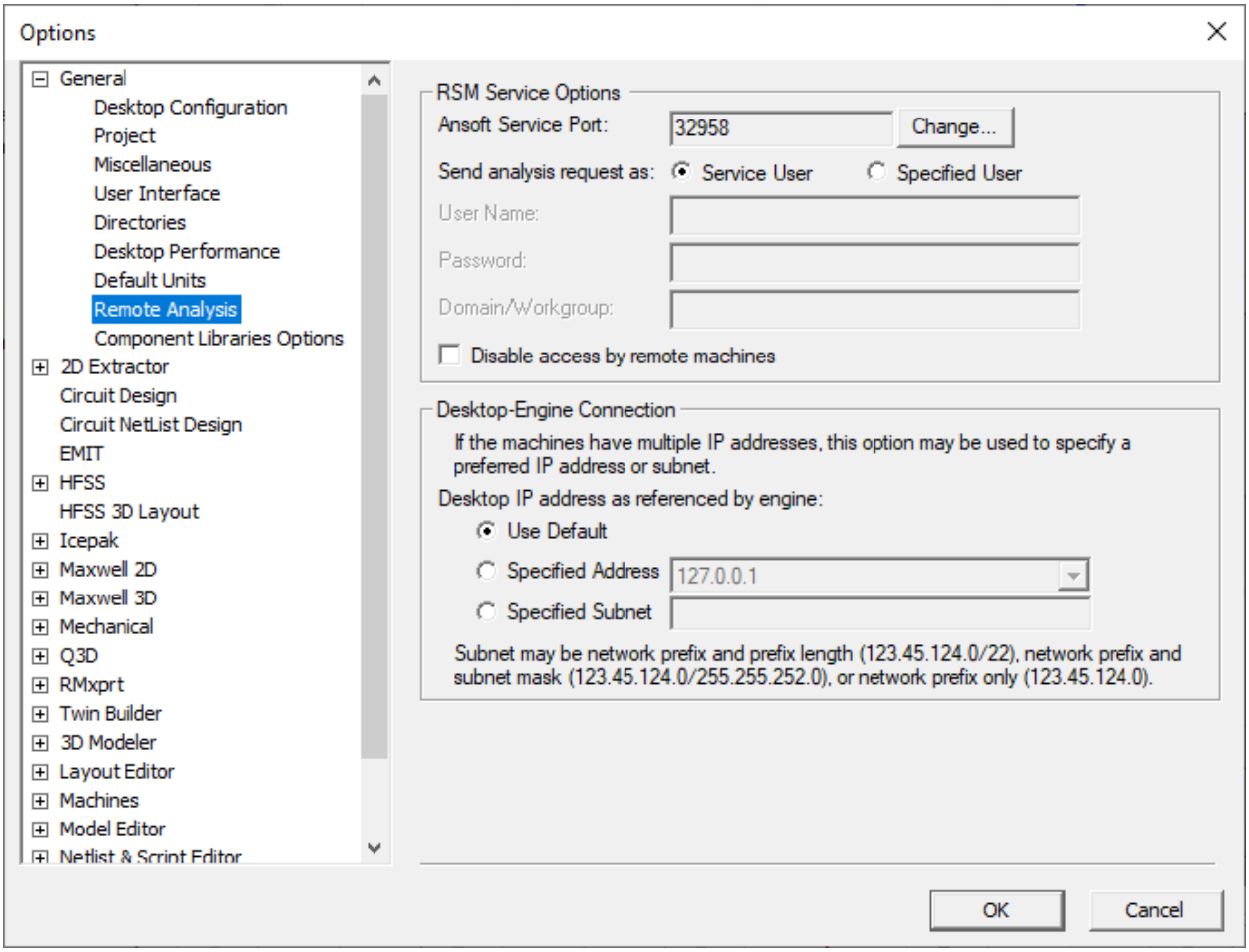
Configuring the Local Machine to Solve Remotely

To set HFSS Analysis options, see: Distributed Analysis.

Remote Analysis Options

You can also set Remote Analysis Options from **Tools** > **Options** > **General Options** > **General** > **Remote Analysis**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Select whether to run simulation processes as the user running RSM (Service User), or sd a Specified User. If you select Specified user, you must provide the User Name, Password, and any Domain/Workgroup on which this user is defined. If the name or password is incorrect, the Messages window issues a warning, and the solver attempts to perform the analysis as the Service User.

Running Remote Analysis

When you run a simulation remotely, you should see a message in the Progress window identifying the design name and the specified remote machine. You will see Progress messages as the simulation continues. When the simulation is complete, you will see a message in the **Messages** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Troubleshooting

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

Problem:

When you try to solve from local to remote machine, a COM engine process starts on the remote machine, but the user interface hangs indefinitely.

This occurs if the remote solve option is enabled after the COM daemon is started, or when the option "Don't allow exceptions" is selected for the Windows firewall.

Resolution:

Remote solve needs either firewall exceptions to be turned on, or firewall to be completely turned off.

Problem:

When you try to solve from a local to a remote machine, you receive the following error message:

[error] Unable to locate or start COM engine on 'nomachine' : Unable to reach AnsoftRSMService. Check if the service is running and if the firewall allows communication. (10:57:13 PM Aug 13, 2019)

Resolution:

This message can happen if the machine is not present, the network connection is down, if there are firewall issues, or if the service is not running.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Problem:

A solve that is distributed to multiple hosts using MPI fails because the AnsysEM installation path is different on different hosts.

Resolution:

Ansys Electronics Desktop must be accessible from all remote machines as well as on the local machine. If the analysis uses MPI, then the path of the Ansys Electronics Desktop installation must be the same on all of the machines used for the analysis (remote and local). This may be a shared network path accessible from all hosts. Alternatively, it may be a local installation on each host; in this case the installation path must be the same for all hosts. If using a shared network path, there should not be a local installation of the same Ansys Electronics version on any of the hosts. See Distributed Analysis for information on whether MPI is needed for the analysis.

Problem:

The command `ansoftrsmservice stop` will not work as root.

Resolution:

Use systemctl command to stop the service:

```
sudo systemctl stop ansoftrsmservice
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Remote Solve Node = Windows

Error:

"Unable to locate or start COM engine on <remote node> : Unable to reach AnsoftRSMService. Check if the service is running and if the firewall allows communication."

Resolution:

1. Try disabling the firewall.
2. Confirm that you have not changed the Service Port in **Tools** > **Options** > **General Options** > **General** > **Remote Analysis**. If you have, change it back to the default (32958), restart Ansys Electromagnetics Desktop, and try to solve again.
3. Make sure that the local machine is able to contact the RSM port on the remote node. Open a command prompt on the local machine and type:
   `telnet <remote node name> 32958`
   If the terminal appears to hang, the connection was successful.
4. Make sure the user listed in the service is an administrator.
5. Make sure the COM engine is registered with the RSM Service. From the Windows menu, choose **Start** > **All Programs**> **Ansys EM Suite 2025 R1** > **Register with RSM** to register the engines.
6. If none of these steps fixes the problem, contact Ansys Support.

Error:

"Unable to locate or start COM engine on <remote node>: Engine is not registered with the Ansoft RSM service which is running on this machine."

Resolution:

1. To register the engine from the Windows menu, select **Start** > **All Programs** > **Ansys Electromagnetics Suite 2025 R1** > **Register with RSM**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Remote Solve Node = Linux

Error:

"Unable to locate or start COM engine on <remote node>: Unable to reach AnsoftRSMService. Check if the service is running and if the firewall allows communication."

Resolution:

1. Try disabling the firewall.
2. Confirm that you have not changed the Service Port in **Tools** > **Options** > **General Options** > **General** > **Remote Analysis**. If you have, change it back to the default (32958), restart Ansys Electromagnetics Desktop, and try to solve again.
3. Make sure that the local machine is able to contact the RSM port on the remote node. Open a command prompt on the local machine and type:
   `telnet <remote node name> 32958`
   If the terminal appears to hang, the connection was successful.
4. Check to make sure Remote Simulation Manager is running:
   a. Go to the rsm subdirectory of the Remote Simulation Manager installation directory, <RSM installdir>/rsm.
   b. Type
      `./ansoftrsmservice status`
   c. If the status query indicates that the service is stopped, type ./ansoftrsmservice start.
5. Make sure the COM engine is registered with RSM. Type:
   `./RegisterEnginesWithRSM.pl status`
   from within the Electronics Desktop installation directory. If the status query indicates "Not registered", type:
   `./RegisterEnginesWithRSM.pl add`
6. If none of these steps fixes the problem, contact Ansys Support.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Error:

"Unable to locate or start COM engine on <remote node>: Engine is not registered with the Ansoft RSM service which is running on this machine."

Resolution:

1. To register the engine, go to the Ansys Electromagnetics product installation directory and type:
   `./RegisterEnginesWithRSM.pl add`

Distributed Analysis

Distributed analysis allows users to split certain types of analyses and solve each portion of an analysis simultaneously on multiple machines. Simulation times can be greatly decreased by using this feature.

HFSS supports the following form(s) of distributed analysis:

- Distributing rows of a parametric table, either as a regular DSO or as Large Scale DSO performed through command line.
- Distributing array solves.
- Distributing domain solves.
- Distributing single or discrete interpolating sweeps.

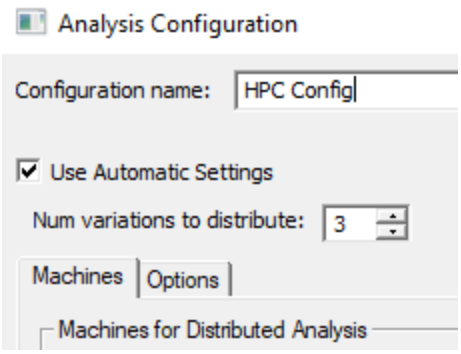
> **Note:**
>
> Communication between machines in remote analysis and distributed analysis can drastically affect performance. Use of a high-speed network system, like Gigabit or Infiniband, is recommended for optimal performance.

All types of distribution use MPI, except when only distributing rows of a parametric table, either as a regular DSO or as Large Scale DSO performed through command line. MPI may also be used if Auto mode is specified, and rows of a parametric table are distributed. MPI must be correctly configured if the distributed analysis uses MPI. See Setting HPC and Analysis Options for setting MPI Licensing and the appropriate Ansys Windows or Linux Installation Guide for details on installing MPI.

Beta Feature: Parallel Component Mesh Adapt for 3D Component Array

If you enable the Beta Feature for Parallel Component Mesh Adapt, you can have parallel mesh adaption for 3D Component arrays. This feature works only with Use Automatic Settings enabled on the active Analysis Configuration.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When running a suitable design with this feature enabled, the Solution Profile will show multiple frequencies being adapted at the same time.

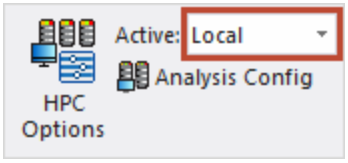
Configure a Distributed Analysis

To configure a distributed analysis, you must select a distributed machine configuration containing a list of machines to use for a simulation, based on memory and CPU considerations (See: Selecting an Optimal Configuration for Distributed Analysis). To create an new distributed machine configuration or to edit an existing one, see Editing Distributed Machine Configurations.

Before you can select a configuration, it must be active. See: Setting HPC and Analysis Options.

To select an existing, active configuration:

- From the **Simulation** tab, use the **Active** drop-down menu to select a configuration.



Distributed Analysis Configuration

After you have selected the active distributed machine configuration, you can edit its configuration in the **Analysis Configuration** window.

To edit the configuration:

1. Access the **Analysis Configuration** window one of three ways:
    - From the **Simulation** tab, click **Analysis Config**.
    - Click **Tools** > **Edit Active Analysis Configuration**.

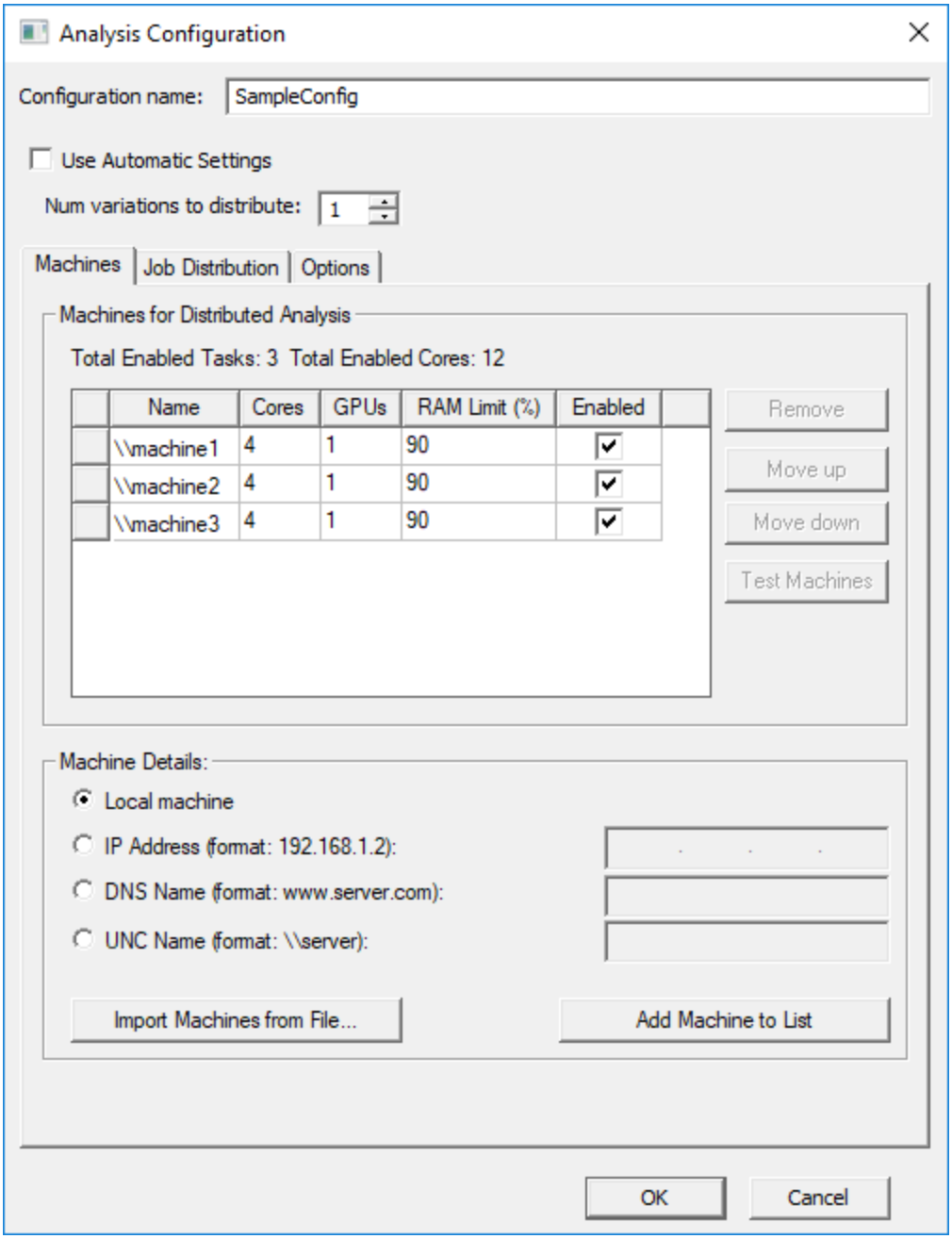
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- From the **HPC and Analysis Options** window (opened from the **HPC Options** icon on the **Simulation** ribbon or **Tools** > **Options** > **HPC and Analysis Options**), select the configuration and click **Edit**.

> **Note:**
>
> You can also access the Analysis Configuration window from the **HPC and Analysis Options** window by clicking **Add** or **Copy**. When using **Add** or **Copy**, the steps are the same as below, but you will need to specify a **Configuration Name**. The name cannot be empty and it cannot be a reserved name.

The **Analysis Configuration** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> Available options may vary slightly depending on the solver and design.

2. If automatic settings are supported, the **Use Automatic Settings** check box allows you to select this option and set parameters based on the best use of available resources for the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

current analysis. Deselecting it enables the **Job Distribution** tab, where you can assign resources manually.

HFSS and HFSS 3D Layout use Automatic Settings by default. For HFSS Auto Solutions, you must select **Use Automatic Settings**.

Selecting **Use Automatic Settings** enables the **Num Variations to Distribute** option. If you specify more than 1, the distribution of variations is in proportion to RAM availability on the machines listed. Depending on the resources available and whether the simulation defines optimetrics variations and other distributed types, the simulation can include up to three levels of distribution. Variations will not be distributed when the number of variations to distribute is set to 1.

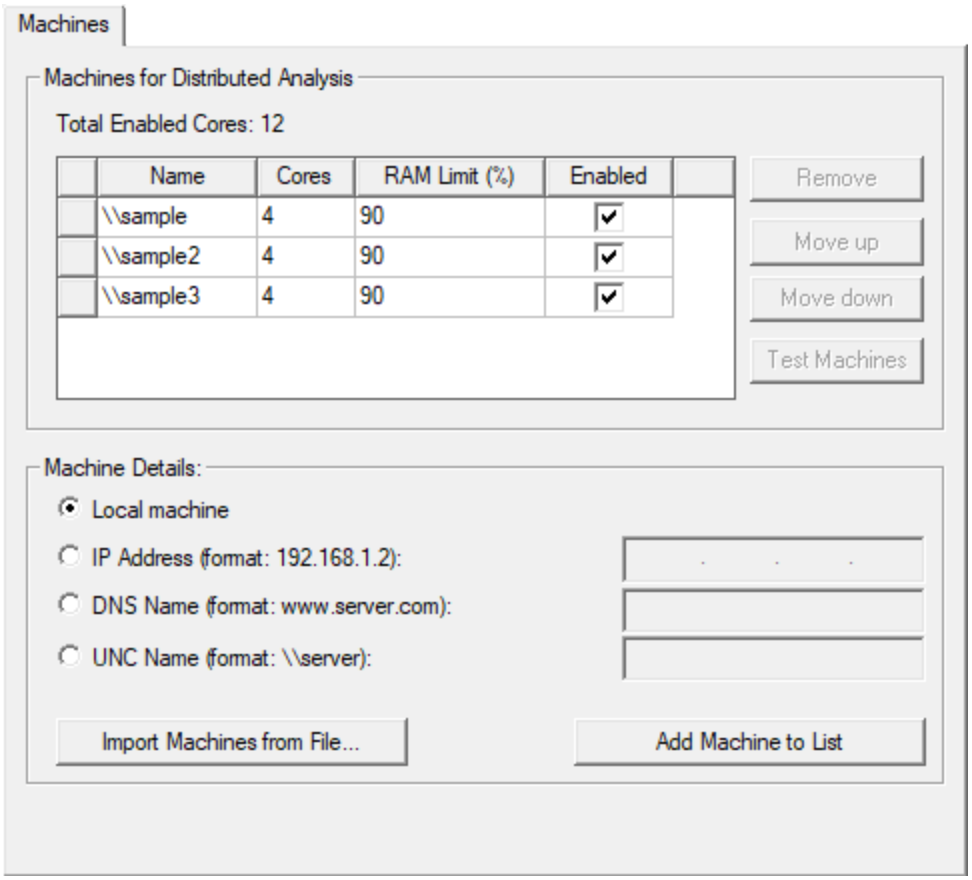
The GPUs column applies only to SBR+ solutions.

3. Use the tabs to navigate through the rest of the settings:
   - **Machines** – contains the machine list. You can add, enable, remove, test, and reorder machines from the list.
   - **Job Distribution** – when **Use Automatic Settings** is deselected, this tab appears and allows you to select job distribution settings manually.
   - **Options** – allows you to specify additional options, depending on the design type.
4. Click **OK** to save the configuration settings.

Distributed Analysis Configuration - Machines Tab

On the **Analysis Configuration** window, the **Machines** tab allows you to provide machine information, either by specifying remote machine details, or by importing a list of machines from a file. You can then add, enable, remove, test, and reorder machines from the list.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Manually Adding a Machine

You can add the **Local machine**, or add a machine by **IP Address**, **DNS Name**, or **UNC Name**. Select the applicable radio button, enter the machine's information in the specified format, then click **Add Machine to List**.

> **Important:**
>
> The remote machines must have the same Ansys Electromagnetics Suite version installed on the same OS version, and have active RSM service.

The added machine(s) will appear under **Machines for Distributed Analysis**.

Importing a Machine List

You can import a machine list from a text file. Each line of the file should contain a machine's information in the following format:

`<MachineName>:<NumTasks>:<NumCores>`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Machine List Details

The **Machines for Distributed Analysis** area lists machines in the order in which you entered them, irrespective of the load on the machines. To control the list order, select one or more machines, and use the **Move up** and **Move down** buttons. To remove one or more machines, select the machine(s) and click **Remove**.

The following columns can appear in the list:

- **Name** – the machine name.
- **Tasks** – this column allows you to specify the number of tasks a given machine will perform simultaneously. *Each separate solver or instance counts as one task.*

  If you have selected **Use Automatic Settings**, this column does not appear because there is no need for it.

- **Cores** – specifies the total number of cores that will be used on the given machine. The total number of Tasks and Cores are described just above the machine list.

  For distributed tasks, the software will allocate the total cores on a given machine to that machine's tasks. If a machine with 8 cores is running 2 distributed tasks, the software will automatically allocate 4 cores to each task. If it is running 4 distributed tasks, each gets 2 cores. And if it is running 3 distributed tasks, the first two tasks get 3 cores and the last task gets 2 cores. *The number of Cores must always be greater than or equal to the number of Tasks.*

  For a given variation (for example, frequency or geometry), you should make assignments so that each task has the same number of cores. This is because the solvers attempt to make each task computationally balanced. For example, with two machines, one with eight cores and another with four, assuming that the memory is proportionally equivalent, you could assign two tasks for machine 1, and one task for machine 2, giving all tasks the same number of cores.

- **GPUs** – supports GPU acceleration. See: Enabling GPU.
- **RAM Limit (%)** – specifies the maximum percentage of each machine's RAM you would like to be in use by the solver.
- **Enabled** – use the check boxes to enable or disable machines.

In general, Ansys Electromagnetics Suite solvers use machines in the distributed analysis machines list in the order in which they appear. If you select a distributed configuration (rather than Local) from the Toolbar menu and you launch multiple analyses from the same UI, Ansys Electromagnetics Suite solvers select the machines that are running the fewest number of engines in the order in which the machines appear in the list. For example, if the list contains 4 machines, and you launch a simulation that requires one machine, the solver chooses the first machine in the list. If another simulation is launched while the previous one is running, and this simulation requires two machines, the solver chooses machines 2 and 3 from the list. If the first

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

simulation then terminates and we launch another simulation requiring three machines, the solver chooses 1, 4, and 2 (in that order).

Testing Machines

When multiple users on a network are using distributed solve or remote solve, they should check the status of their machines before launching a simulation to ensure that no other Ansys EM processes are running on the machine. To do this, select one or more machines and click **Test Machines**.

The **Test Machines** dialog box appears. The test goes through the current machine list and gives a report on the status of each machine. A progress bar shows testing progress. When the test is complete, click **OK** to close the dialog box. If you need to disable or enable machines based on the report, you can do so in the **Analysis Configuration** window.

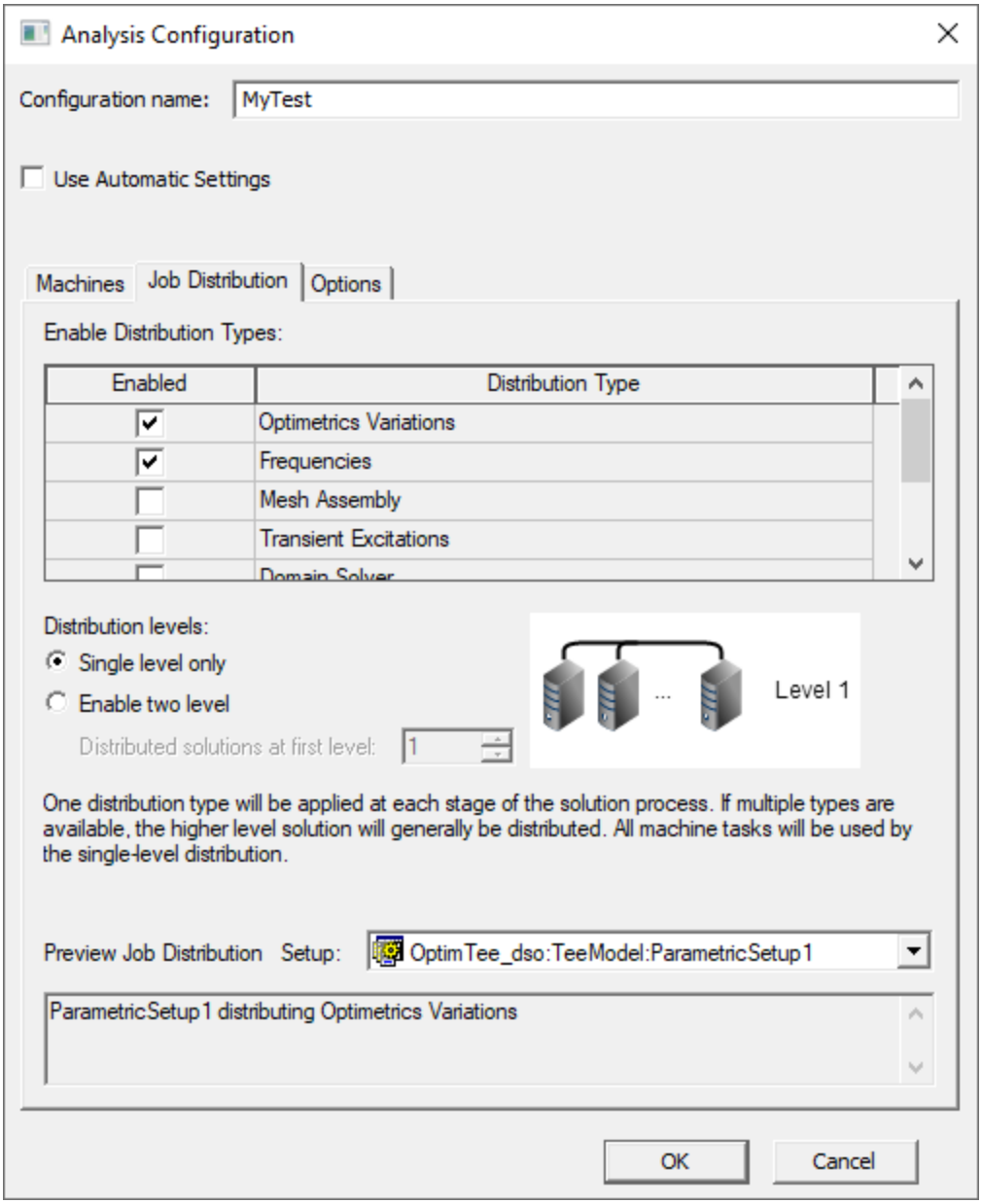
Distributed Analysis Configuration - Job Distribution Tab

On the **Analysis Configuration** window, the **Job Distribution** tab allows you to manually enable specific types of job distribution and enable multi-level solves.

The **Job Distribution** tab is disabled if you selected **Use Automatic Settings**.

> **Note:**
>
> Different design types have different job distribution types.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The job distribution list allows you to specify which job distribution types to allow for the current analysis configuration. Use the check boxes enable and disable distribution types. At solve time, HFSS automatically selects the best distribution type from the enabled types.

Enabling a distribution type does not mean it will be used. It must be also allowed by the solve setup. If you enable a distribution type for a given setup, and distribution is allowed, the preview window updates to describe the distributions. Note that enabled distribution types apply to all setups of the given design type, so it is possible for different setups in a design to be solved using different distribution types.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

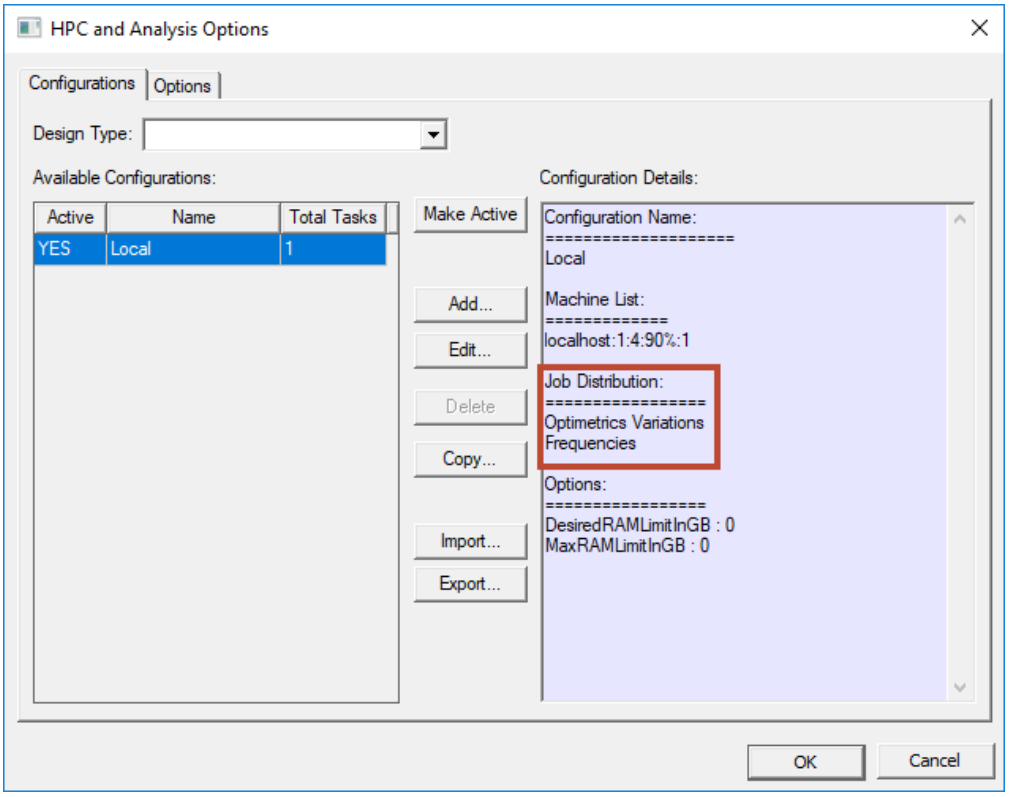
The concurrent initial mesh generation workflow with Distributed Mesh Assembly relies on the MPI based distribution technology inside the MeshAssemblyManager. The decision whether to launch sequential or parallel mesh generation is based on the combination of the number of individual meshes present, the HPC setting, and the number of tasks available.

- If there's only one single mesh or if curvilinear contact exists: then normal initial mesh workflow with single G3dmesher will be launched.
- If there are more than one individual mesh and no curvilinear contact exists:
    - If Use Automatic Setting is not selected, under the **Job Distribution** tab, if "Mesh Assembly" is not enabled, then meshes will be generated sequentially. If "Mesh Assembly" is enabled and user assigns more than 1 tasks, then parallel mesh generation will be launched.
    - If Use Automatic Setting is selected and user assigns more than 1 core, then parallel mesh generation will be launched.
    - If Use Automatic Setting, the number of MPI tasks will be the number of cores user assigned or the number of geometries to be meshed, whichever is smaller. If there are more geometries than cores, the geometries will be dynamically assigned to tasks. If there are more cores than geometries, the remaining available cores will provide some of the tasks with multi-thread capabilities.

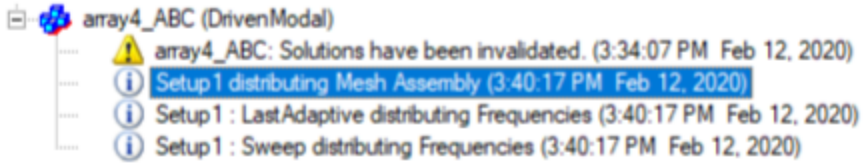
All the detailed progress information from the Mesher is suppressed and the progress will report the number of meshes being finished. All the mesh profiles will be available under profile report and the mesh feedback will also be available under mesh feedback tool.

Enabled distribution types are listed when you select a configuration in the **HPC and Analysis Options** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

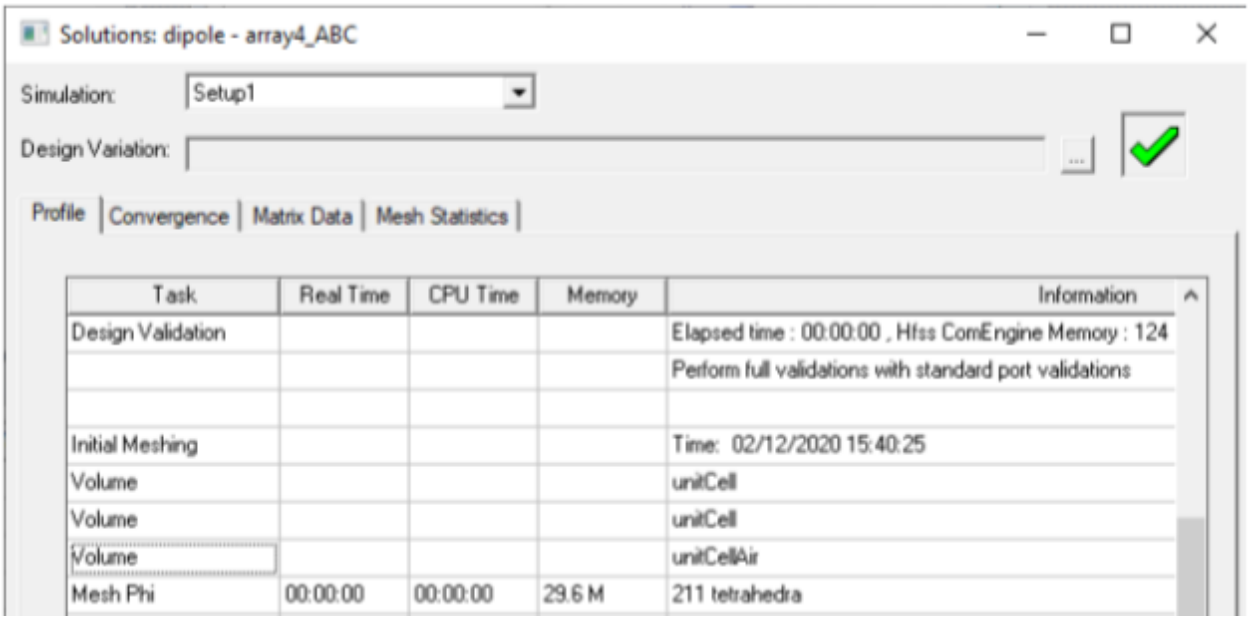


When you run a simulation, the **Messages** window describes distributions.



When you view the Solution Profile, distributions for unitCell show as parallel Volume Tasks.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



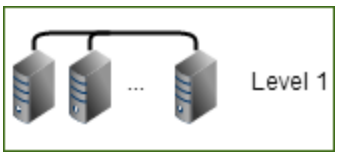
Distribution Levels

For products and designs that support two-level distribution, you can select either single or two-level distribution.

If you select single level, one distribution type will be applied at each stage of the solution process. If multiple types are available, the higher-level solution will generally be distributed. All machine tasks will be used by the single-level distribution.

Single-level Distributions

In a single-level distribution, one distribution type is applied at each stage of the solution process. Common stages include LastAdaptive, Sweep, and Parametric. All machine tasks will be used by the single-level distribution.



Supported distribution types include:

- Optimetrics Variations
- Frequencies
- Mesh Assembly

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Transient Excitations
- Domain Solver(for Technical details, see Domain Decomposition Method)
- Iterative Solver Excitations(for Technical details, see Multiprocessing and the Iterative Solver)
- Direct Solver Memory(for Technical details, see Distributed Memory Solutions with HFSS)

Solver distributions require MPI. See: Setting HPC and Analysis Options.

Parallel distribution types (such as Optimetrics Variations, Frequencies, and Excitations) do not require distribution. If these types are not able to distribute, the simulation can run sequentially.

Memory distribution types (such as Direct Solver Memory and Domain Solver) are set to require distribution. If these types are enabled, the software will assume that distribution is necessary to extend the simulation scale or add fundamental solution capabilities.

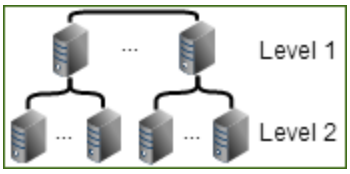
When multiple distribution types are available, the higher-level solution will generally be distributed. For example, when both Optimetrics Variations and Iterative Solver Excitations are enabled, Optimetrics Variations will be distributed. When both Optimetrics and Mesh Assembly, are selected Optimetrics are distributed. Domain Solver and Direct Solver Memory are exceptions because they are required; even though they are lower level, these types are distributed instead of parallel distribution types.

Two-Level Distributions

Selecting **Enable two-level** enables the **Distributed solutions at first level** box.

In a two-level distribution, the first level distributes the specified number of solutions. Each solution will then use a subset of machine tasks to distribute the second level. A solver distribution type must be available for the second level; otherwise, single-level distribution will be applied.

For two-level distribution, the total number of tasks must be greater than or equal to the number of tasks for level 1.



The following are examples of two-level distributions:

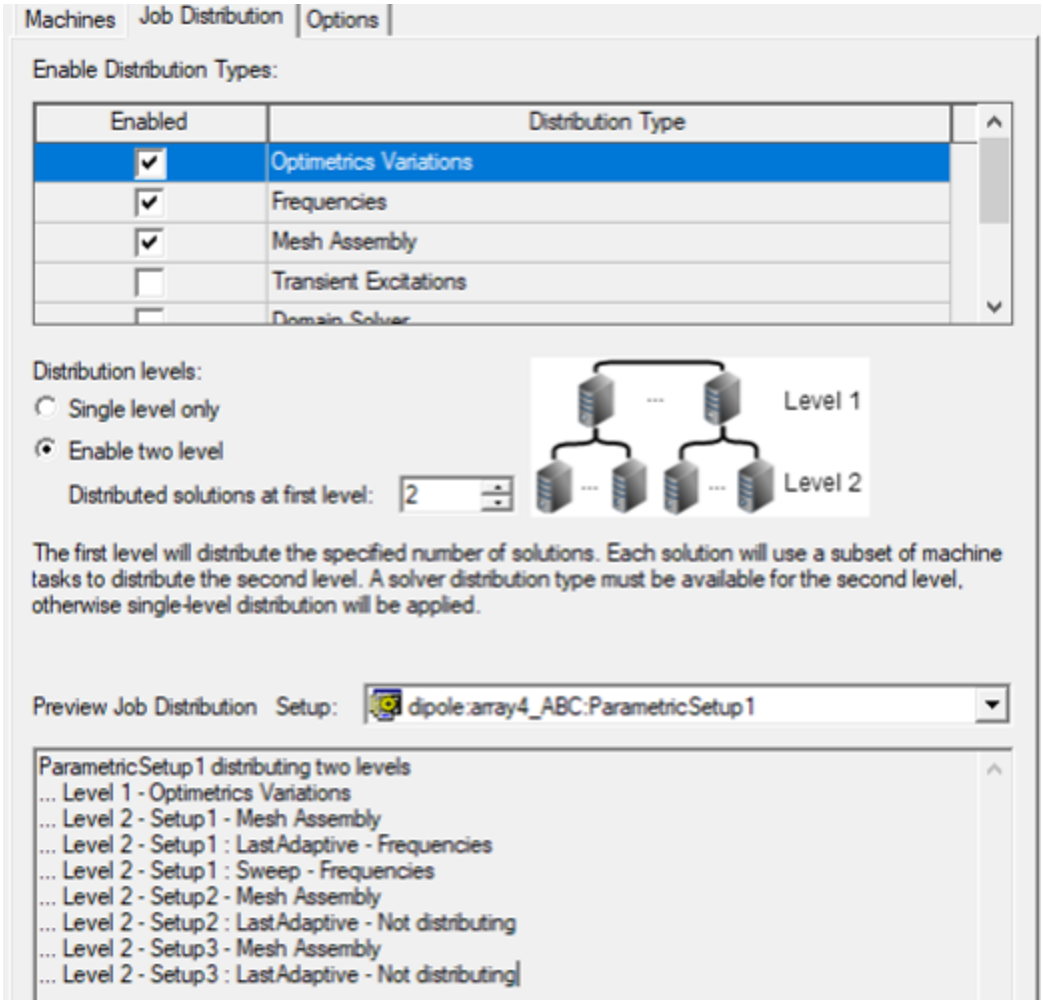
- A parametric setup distributing Optimetrics variations as level 1 and iterative solver excitations as level 2.
- A parametric setup for a non-transient problem distributing optimetrics variations as level one and frequencies as level 2.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A frequency sweep distributing frequencies as level 1 and direct solver memory as level 2.
- A parametric setup for a transient network problem distributing optimetrics variations as level one and transient excitations as level 2.

The **Analysis Configuration** window displays a preview of the job distribution, if allowed for that configuration:

Here is another example:



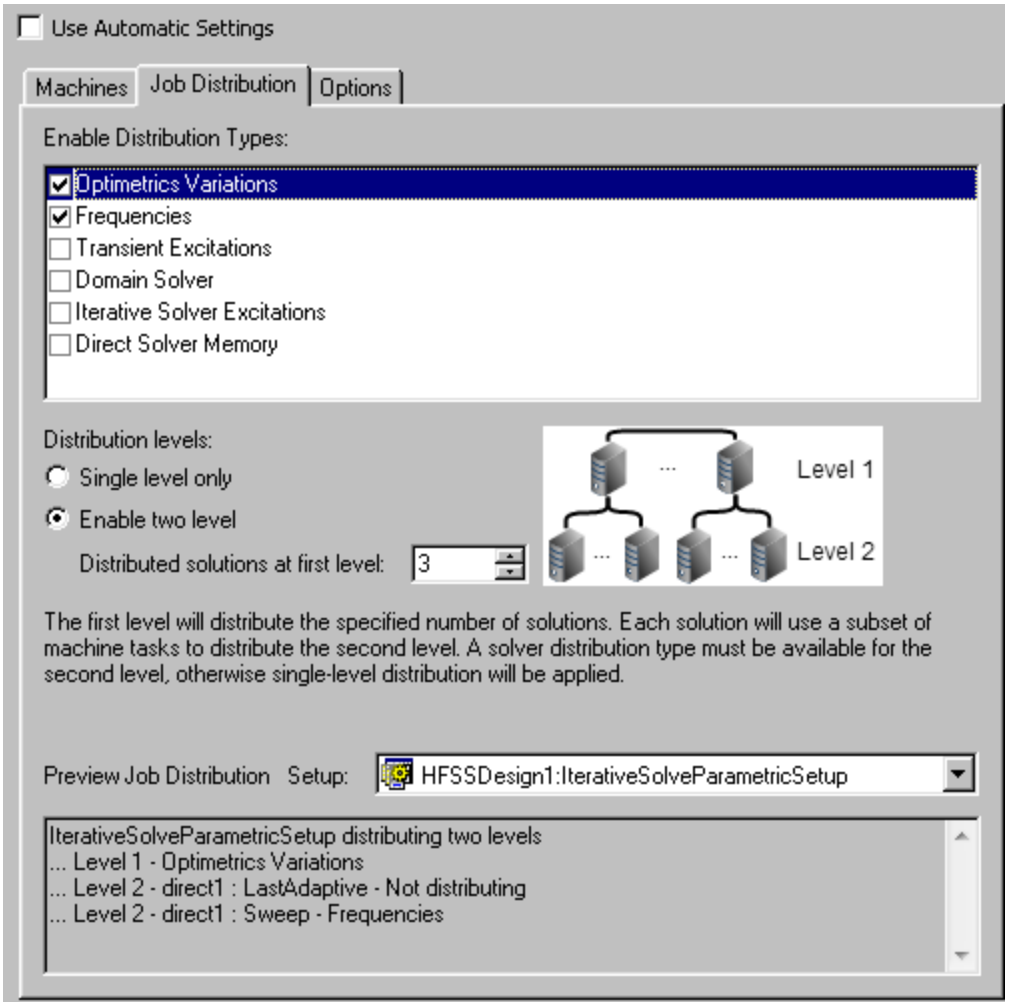
For more information, see: Two-Level Distribution Guidelines.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HFSS Frequency Distribution

HFSS Frequency Distribution can be treated as both of first and second level distribution. The following bullets described the cases for an HFSS design with frequencies sweep setup and parametric solve setup (for non transient solution).

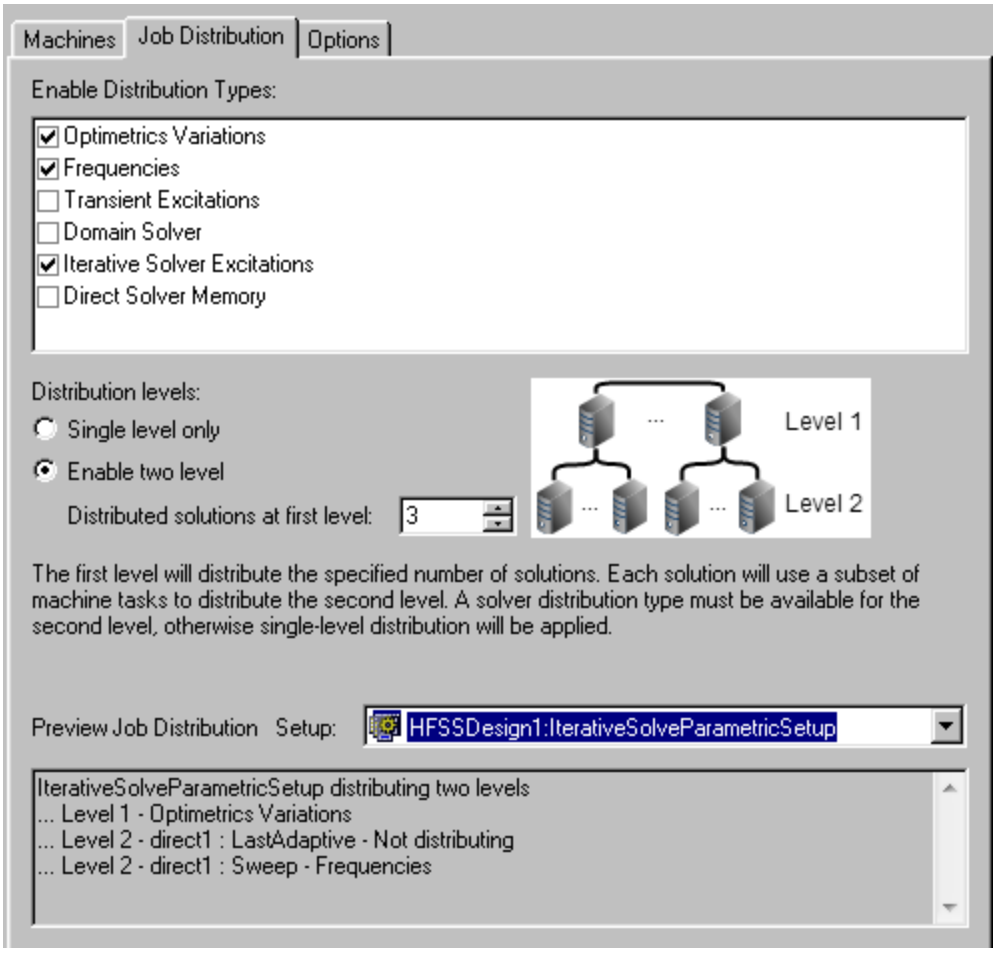
- In the **Analysis Configuration** window, if you select “Optimetrics Variations” and “Frequencies” with two-level distribution enabled, HFSS does two level distribution: first level, “Optimetrics Variations”; second level “Frequencies”. We can see this described in the “Preview Job Distribution” area:



- In the **Analysis Configuration** window, if you select “Optimetrics Variations”, “Frequencies”, and “Iterative Solver Excitations“ with two-level distribution enabled, HFSS does two level distribution: first level, “Optimetrics Variations”; second level “Frequencies”. There is no “Iterative Solver Excitations“ distribution. We can see this in the “Preview Job

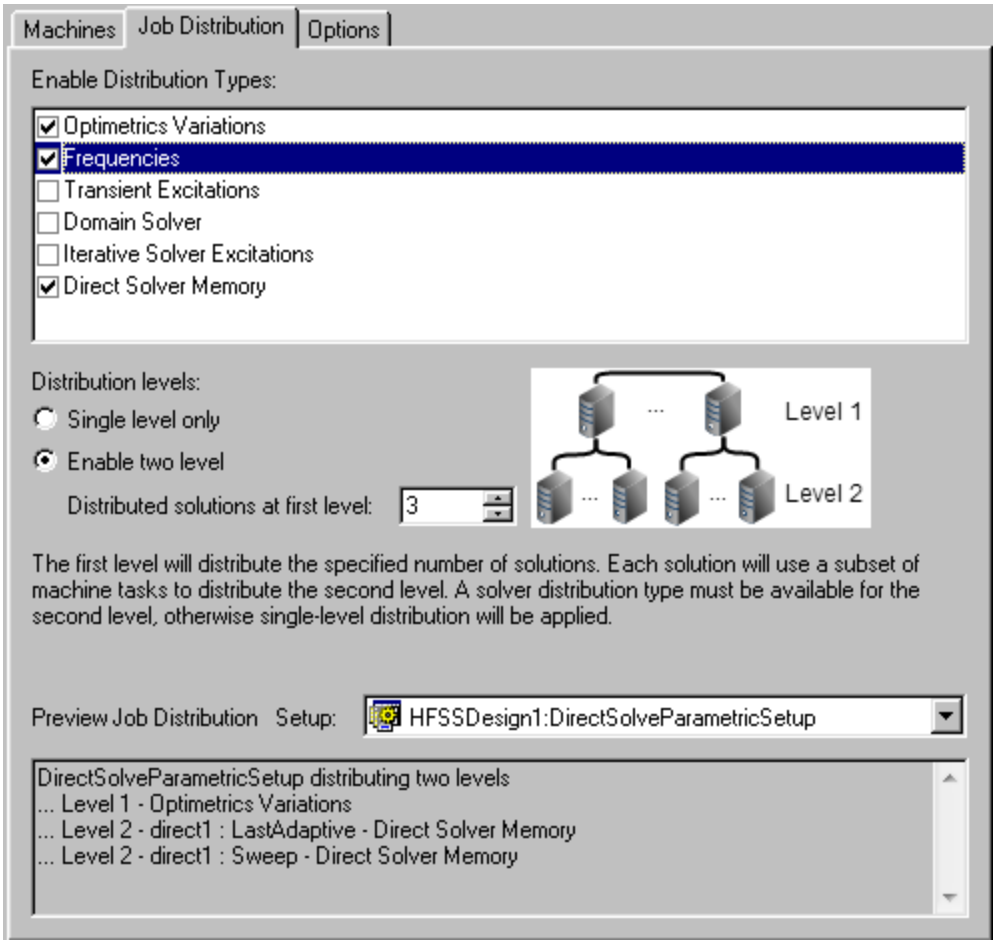
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Distribution” area:



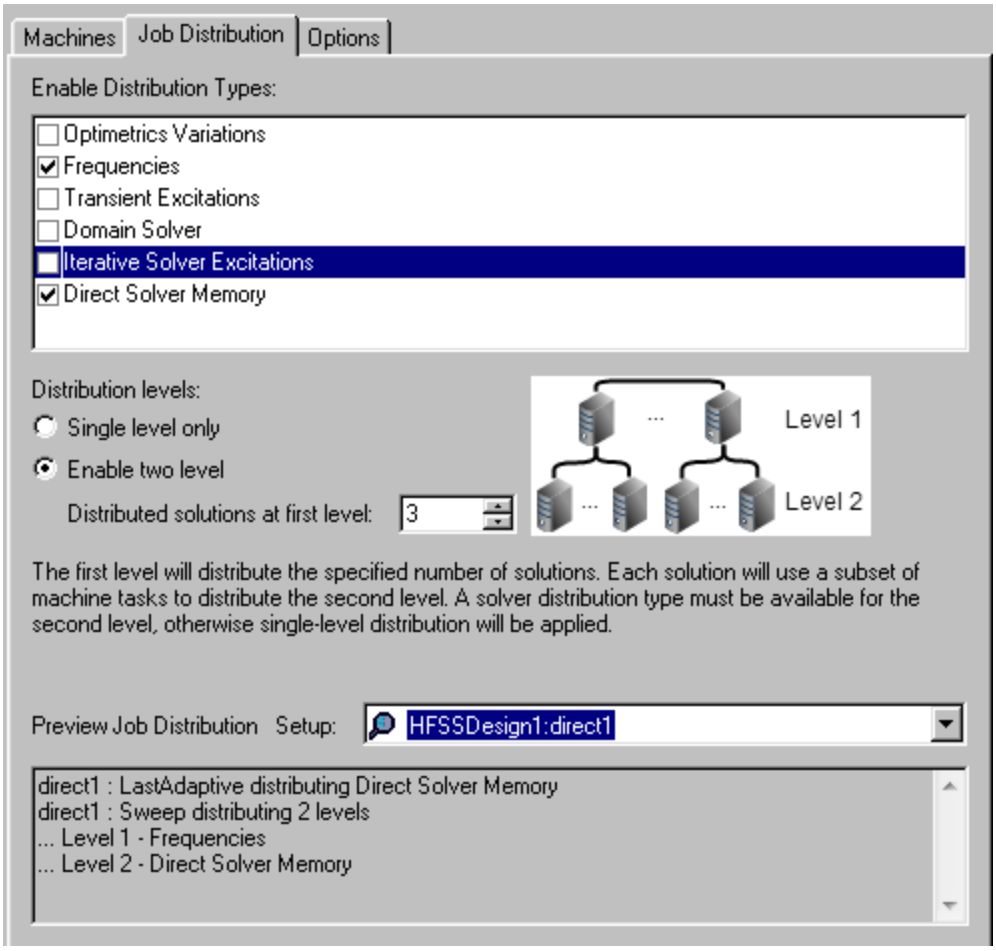
- In the **Analysis Configuration** window, if you select “Optimetrics Variations”, “Frequencies”, and “Direct Solver Memory“ (or “Domain Solver”) with two-level distribution enabled, and if “Direct Solver Memory“ option is selected in the solve setup, HFSS does two level distribution: first level, “Optimetrics Variations”; second level, “Direct Solver Memory“ (or “Domain Solver”). There is no “Frequencies” distribution. We can see this in the “Preview Job Distribution” area:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- In the **Analysis Configuration** dialog box, if you select “Frequencies”, and “Iterative Solver Excitations“(or “Direct Solver Memory“, or “Domain Solver”) with two-level distribution enabled and if you select “Direct Solver Memory“ option (or corresponding solver option) in the solve setup, HFSS only does two level distribution: first level, “Frequencies”; second level, “Iterative Solver Excitations“ (or corresponding solver. We can see this in the “Preview Job Distribution” area. See the following two examples. The first shows Enable Job Distribution Types for “Frequencies” and “Direct Solver Memory”.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

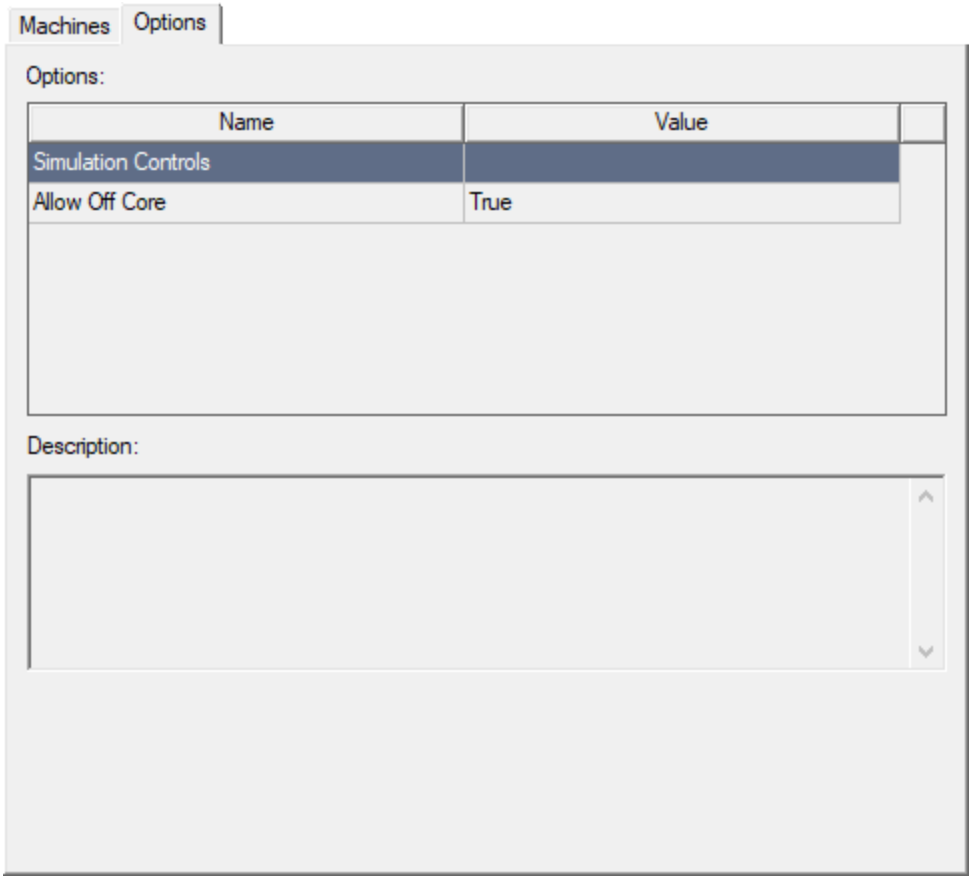


Distributed Analysis Configuration - Options Tab

On the **Analysis Configuration** window, the **Options** tab allows you to specify options for the current analysis configuration.

> **Note:**
>
> Different design types have different options.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



These options settings will be in effect only when all the following are true:

- A design is being solved whose design type matches the analysis configuration design type.
- The analysis configuration is the active configuration for its design type.
- You have not specified corresponding `batchoptions` on the command line.

Relation to Batchoptions

Command line `batchoptions` can be used to override the options specified by the active configuration.

Analysis configuration option settings can be overwritten by specifying the option name and value inside a `-batchoption` string.

To add batchoptions from the UI, click **HFSS** > **Submit Job**. Under **Analysis Options**, click **Add**. This opens the **Add Batchoption** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See: Running Electronics Desktop from the Command Line and Batchoptions Command Line Examples.

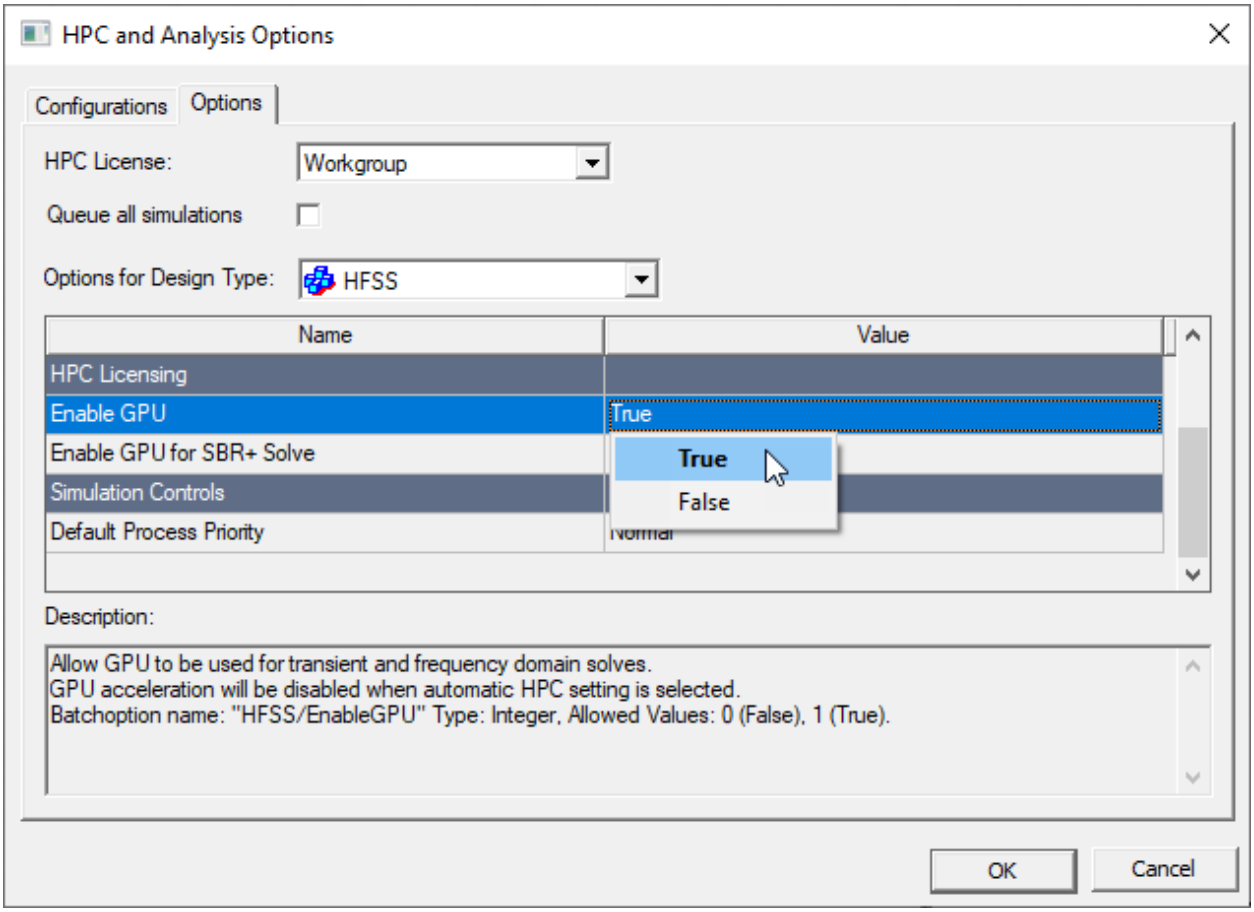
Adding Configurations or Accepting Edits

Click **OK** to accept the changes and close the **Analysis Configuration** window. Only machines marked **Enabled** appear in the machine list.

Regardless of the machine(s) on which the analysis is actually run, the number of processors and RAM Limit (%) settings, and the default process priority settings are now read from the machine from which you launch the analysis. See: Setting HPC and Analysis Options.

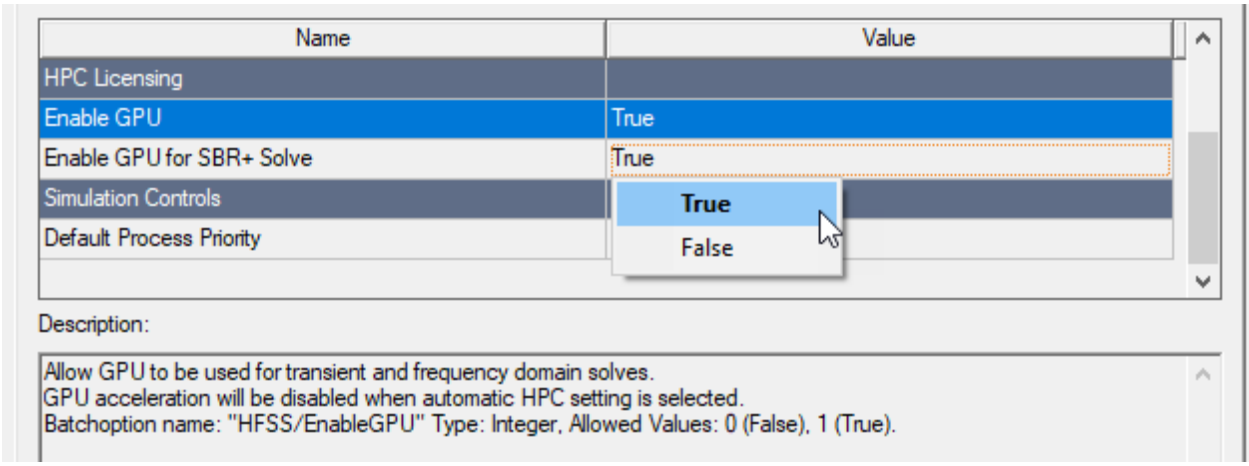
Enabling GPU

Ansys Electronics Desktop supports GPU acceleration for transient, frequency domain, and Maxwell 3D eddy current matrix simulations. For details on the requirements for GPU use, see: GPU Acceleration.

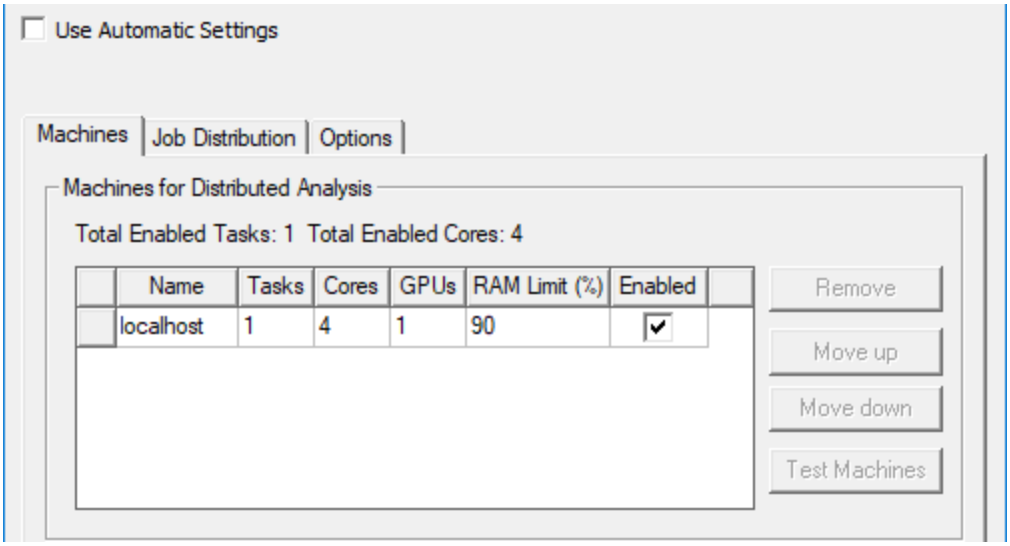


For HFSS SBR+ projects, you can also Enable GPU for SBR+ solves.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To enable use of GPU or MIC acceleration, you can set **Enable GPU** to **True** and set **Use Automatic Settings** to **off** in the **Analysis Configuration** window. The matrix solver automatically determines if all cores should be used, or if one GPU should be used to give the best performance. For example, for a Maxwell Eddy Current Solution Matrix, if you specify 4 cores for the simulation, the 3D Eddy current solver will use 4 cores in parallel during matrix assembly while the matrix solver will use either 4 cores or 1 GPU.



For HFSS, **Use Automatic Settings** supports SBR+ Solution types only. The GPU column is always shown, but only applies when solving the SBR+ solution type. For all other solution types, the GPU count is ignored, and treated as if set to zero.

For HFSS SBR+ in auto mode, each machine has an allocated number GPUs, but no preset number of tasks. In this case, since HFSS SBR+ acceleration is mainly through GPUs, the variation distribution prioritizes use of GPUs. In other words, the distribution divides the GPUs

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

equally among the variations, although it may not be possible for every variation to have the same number of GPUs. Below the level of variations, the allocated GPUs per machine will be passed to the solver's auto algorithm, which then allocate these GPUs to the distributed solver instances.

For solvers that can use GPUs, when you uncheck **Use Automatic Settings**, the analysis configuration lets you specify the number of GPUs to be used by each machine, along with the existing settings for cores and RAM Limit. This makes sense, because GPUs, cores, and RAM are all resources that the user is managing, and GPUs and cores both contribute to HPC count. Each machinehas tasks and GPUs. The GPUs are distributed among the tasks, similarly to how cores are distributed. For example, if a machine has 3 tasks and 2 GPUs, then the first two tasks will get one GPU each, and the third task will get no GPU.

For design types like HFSS that support GPU for different types of solve (frequency, transient, SBR+), the GPU setting when **Use Automatic settings** is unchecked applies to all types. To use GPU for some solves and not for others, you can create multiple configurations.

Determining When a GPU or MIC is Being Used

You can determine if a GPU or MIC is being used for acceleration by viewing the **Solutions** dialog box, **Profile** tab. If a GPU is successfully locked for use by the solver process, the profile shows the GPU's CUDA device ID and its name.

If the GPU is not used, the Solution Profile indicates that, and the CPU cores are used.

When GPU is used:

- GPU must be enabled for HFSS/Maxwell.
- Both Windows and Linux are supported.
- Only complex symmetric matrices can be solved by CUDA.
- The matrix (number of unknowns to be solved) must be large enough. By default, its dimension should be larger than 2,000,000.
- The times using CUDA and CPU only are estimated. If GPU is faster, it will be used. If not, the solver falls back to multi-core CPU.
- HFSS-SBR+: GPU is used a) whenever computing far-field radiation and b) during coupling (S-parameter) computations whenever the receiving antenna is a FEBI antenna. In particular, GPU is used to accelerate the radiation of SBR ray footprints (equivalent currents) to scattered field observers. For example, if you have two antennas, where one is FEBI and the other uses a far-field representation, and the SBR+ setup includes a configured far-field radiation sphere, GPU is used for computing the installed far-field radiation pattern of both antennas and the coupling from the far-field antenna (source) to

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the FEBI antenna (receiver). However, coupling from the FEBI antenna (source) to the far-field antenna (receiver) is computed using multi-core CPU. The speed benefit of GPU over multi-core CPU is greatest when computing the SBR+ contribution to far-field radiation, especially if there are many observation angles, as in a fully populated sphere. GPU loses efficiency when computing coupling to FEBI antennas located near the scattering geometry.

- For HFSS SBR+ on bi-static RCS, GPU is enabled since you likely have many observation points (far-field points) for each incident wave, and GPU can make a really big difference in simulation time. GPU will help to a lesser extent when modeling installed radiation patterns for antennas with far-field patterns (since they are simple point sources, sourcing only a single set of SBR rays).

GPU and CPU:

- The estimation is based on the structure of the matrix, not just the dimension.
- The generation and model of GPU are considered. The newer the model is, the faster the GPU estimation is.
- The clock rate and number of cores of CPU are considered. The higher the rate is, the more the cores are, the faster the CPU estimation is.
- Finally, the faster device (GPU or CPU) is selected based on the estimation.

Why GPU is Not Used

- The estimation is based on the whole matrix, not just the size. Having larger matrices doesn’t mean GPU will be used.
- The bandwidth is not useful either. It is possible to have two matrices with exactly the same size and bandwidth, one favoring GPU and the other CPU.
- For HFSS SBR+ for monostatic RCS problems, the GPU is not used for simulation. The reason is that there is only one observation point for each plane wave excitation, so the GPU does not have as much potential to enhance the processing speed.
- Other HFSS SBR+ problems where GPU is not used include antenna problems where an antenna that is designated as an Rx or Tx/Rx type is a far-field source only. GPU is most effective in antenna problems involving many current sources, many far-field observation points and multiple frequencies.
- For HFSS SBR+ GPU will not kick in at all for antenna coupling problems with Rx antennas that are far-field type only.
- If GPU is of older models while CPU is not, GPU may not be used.
- If CPU has high clock rate, GPU may not be used.
- If CPU has many cores, GPU may not be used.
- If Use Automatic settings in Analysis Configuration is checked.
- Nvidia GPU drivers are downloaded from certain repository (e.g. RedHat) instead of directly from Nvidia website

Multiple GPUs

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When there are multiple GPU cards in the same machine, the first solver process (hf3d, 3dedy) locks the first GPU card available (not locked or used by other processes, not used for display, etc.), the second process locks the second card available, and so on. After all cards are locked, solver uses CPU only.

Selecting Optimal Configurations for Distributed Analysis

This section provides some basic guidelines for configuring a distributed analysis, including how to choose the number of tasks for Distributed Direct and Distributed Iterative solvers, and when to use two-level distribution. Also included are tips for the HFSS-IE solver and Domain Decomposition solver. In this section, assume that a fixed number of machines and cores are available, and that the goal is to use them as efficiently as possible.

In the majority of situations, configuring a Distributed Solution Option (DSO) in combination with the multiprocessing option to take optimal advantage of the available hardware results in improved speed.

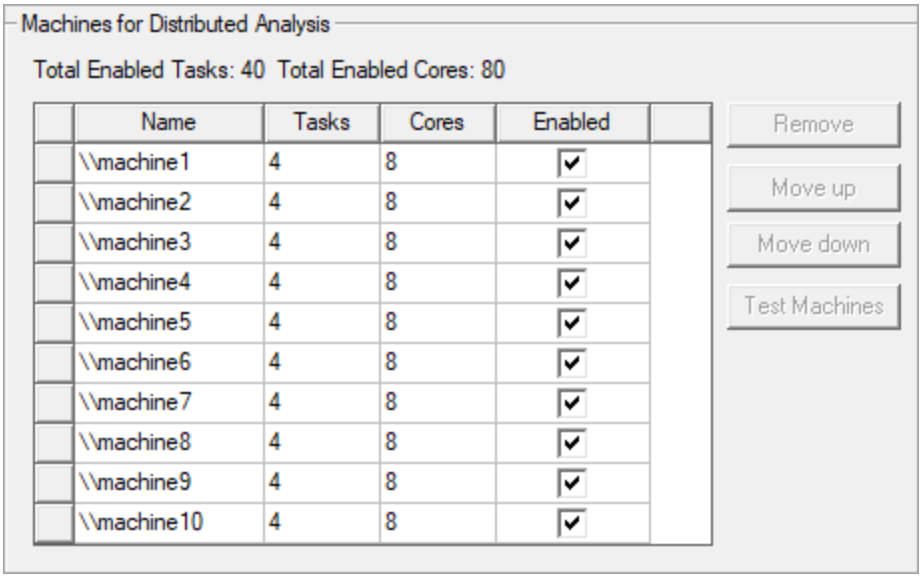
> **Note:**
>
> This topic assumes that there is enough hard drive space and memory for DSO simulations. For multiple DSO simulations on a single machine, the total memory needed is the sum of the memory used by each simulation. For example, consider a discrete frequency sweep in which each frequency point needs 3.5GB. A quad core system with 8GB of RAM would rely heavily on swap, which is highly inefficient. In this case, updating the number of tasks in the machine list is ideal.

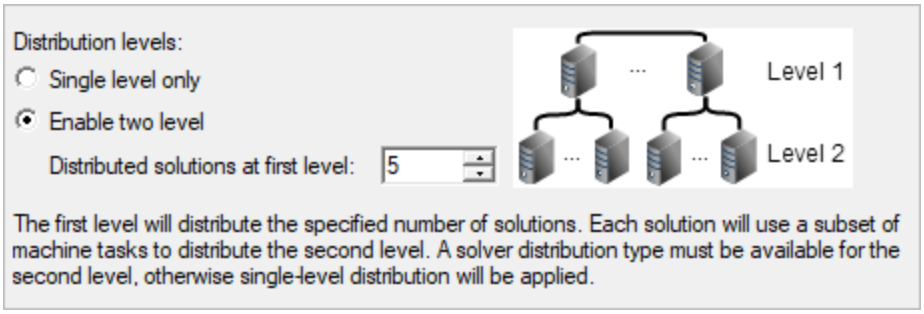
Two-Level Distribution Guidelines

Ansys recommends that you focus resources for the highest level of distribution (level one). In other words, allocate the resources so that a maximum number of frequencies or variations are solved in parallel. First, determine the minimum number of machines required to solve one frequency or variation based on your prior knowledge of the type of model being solved. Then allocate the resources so that a maximum number of solutions can be solved in parallel.

For example, suppose that a given model requires at least 100 GB of memory to solve using DDM. Assume that you have 10 machines, each with 60 GB and 8 cores. One solution will require two machines for the DDM solver. This means that, at most, 5 frequencies can be distributed. Each DDM solution requires at least 3 tasks, but since more cores are available you should set 4 tasks per machine, resulting in 2 cores for each Domain (see: Domain Decomposition Solver Guidelines for HPC Configuration for more details). A machine configuration based on the machines and number of tasks described above is shown in the figures below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Domain Decomposition Solver HPC Configuration Guidelines

The domain decomposition solver supports two types of domains: solver domains and finite array domains.

For problems with solver domains, there can be either FEM domains alone or FEM domains with FE-BI boundary and/or IE regions. The following cases offer basic guidelines for choosing the number of tasks and number of cores per task for each.

Case1: Solver Domain Problem *without* FE-BI Boundary or IE region.

In this case, the total memory requirement will typically decrease with more tasks. For optimal speed, ensure that you have 2-3 cores per task.

Case2: Solver Domain Problem *with* FE-BI boundary and IE region.

If the FEM portion is dominant geometrically, the rule of Case1 also applies here. Otherwise, more task input will trigger distributed IE solver in IE and BI domains. The Distributed IE solver

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will reduce memory requirements in each BI/IE domain with modest speed improvement for some cases.

Case3: Finite Array.

For most arrays, speed is optimal with 8-16 tasks. If you have more than 16 cores available, assigning 2-3 cores per task will most likely be better than assigning more tasks. For example, if you have a total of 24 cores, you should assign 12 tasks with 2 cores per task rather than assign 24 tasks with 1 core per task.

Distributed Direct Solver Guidelines for HPC Configuration

The dependence of Distributed Direct solver on number of tasks is very similar to the HFSS solver. For distributed direct solver, the number of tasks increases the total memory but also typically increases the speed of the simulation.

Some general guidelines:

1. If memory usage is critical, use 1 task/machine using all cores.
2. If memory usage is not critical, use 4 cores/task for optimal speed.

Distributed Iterative Solver Guidelines for HPC Configuration

The distributed iterative solver distributes excitations for faster simulations and to access less memory to solve a large problem. Each task corresponds to a number of excitations and, therefore, more excitations are solved in parallel when the number of tasks increases.

The first consideration for the number of tasks is the memory requirement. Each task uses a similar amount of memory, and the total memory usage doubles as the number of tasks doubles. Thus the number of tasks on each compute node should be restricted to avoid an "Out of Memory" failure.

The second consideration for the number of tasks is the number of excitations. HFSS designates one of the tasks as the master, which does not participate in the iterative process. ***Thus the number of tasks should be no more than the Total Number of Excitations + 1***.

In summary, assuming that you have enough memory, if the number of excitations N is less than the total number of cores M and you choose 4 cores per task, then the number of tasks is ceil (N/4) + 1, where ceil() stands for the ceiling function and the extra "1" is dedicated to the master task; if the number of excitations N is more than the total number of cores M and you choose 4 cores per task, then the number of tasks is ceil(M/4). For example, if you have a design with 64 excitations and you have 32 cores available, you could define a possible setup having 8 tasks with 4 cores per task.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Distributed HFSS-IE Guidelines for HPC Configuration

The dependence of distributed HFSS-IE solver on number of tasks is somewhat different compared with the domain decomposition solver. For HFSS-IE, the number of tasks increases the total memory but typically also increases the speed of the simulation. We suggest the following guidelines:

1. If memory usage is critical, 1 task per machine is recommended using all cores.
2. If memory usage is not critical, then using 4 cores per task is recommended for optimal speed.

Distributed Mesh Assembly Guidelines for HPC Configuration

The concurrent initial mesh generation workflow with Distributed Mesh Assembly relies on the MPI based distribution technology inside the MeshAssemblyManager. The decision whether to launch sequential or parallel mesh generation is based on a combination of the number of individual meshes present, the HPC setting, and the number of tasks available.

- If there is only a single mesh or if curvilinear contact exists: the normal initial mesh workflow with single G3dmesher will be launched.
- If there is more than one individual mesh and no curvilinear contact exists:
  - If **Use Automatic Setting** is *not* selected under the **Job Distribution** tab, and if **Mesh Assembly** is not enabled, then meshes will be generated sequentially. If **Mesh Assembly** is enabled and user assigns more than 1 tasks, then parallel mesh generation will launch.
  - If **Use Automatic Setting** is selected and user assigns more than 1 core, then parallel mesh generation will launch.
  - If **Use Automatic Setting** is selected, the number of MPI tasks will be the number of cores user assigned or the number of geometries to be meshed, whichever is smaller. If there are more geometries than cores, the geometries will be dynamically assigned to tasks. If there are more cores than geometries, the remaining available cores will provide some of the tasks with multi-thread capabilities.

All detailed progress information from the mesher is suppressed and the progress will report the number of meshes being finished. All mesh profiles will be available under profile report and mesh feedback will also be available under mesh feedback tool.

Large Scale DSO for Parametric Analysis

Large Scale DSO for parametric analysis operates through a non-graphical batch application called desktopjob. You can run the desktopjob command line to perform parametric analysis DSO. The command-line interface supported by this batch program is similar to the command line used for regular DSO jobs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Large Scale DSO is used for large scale parallel jobs, which either fail or scale poorly as regular DSO jobs. A Large Scale DSO job does not support the output of full parametric results, but produces reduced datasets corresponding to predefined rectangular plots. The extracted columns of data are saved as CSV files. Typically, there is one CSV file per-trace, per-variation. These CSV outputs can be used directly in downstream applications (for example, Microsoft Excel). They can also be imported as dataset solutions for post-processing. Non-rectangular plots of the design (such as statistical eye or digital plot) are not extracted. In order to produce a new output you must rerun the analysis.

> **Note:**
>
> For a machine with *n* cores, it should be expected that running *n*, single core, distributed simulations in parallel will encounter additional overhead due to the need to spawn *n* unique solve processes. Therefore, it should not be expected to observe an *n* times speed up over the time taken to run *n* analyses in series with a single core. The relative impact of this overhead increases as the size of the simulation and time to complete a single solve shrink.

The basic Large Scale DSO process involves

1. Preparing the model for Large Scale DSO Analysis.
2. Submitting the Large Scale DSO job through **Tools** > **Job Management** or via command line.
3. Monitoring the job's progress.
4. Postprocessing the results.

For details, refer to the following sections:

- Prerequisites for Large Scale DSO
- Job Management Interface for Large Scale DSO
- Command Line Syntax
- Deployment/Configuration
- Tutorial Example for HFSS
- Job Outputs
- Job Monitoring
- Process Limits for Large Scale DSO on Windows
- Known Issues/Troubleshooting for Large Scale DSO

Large Scale Distributed Solve Operation could submit a parametric setup to be solved in multiple machines, each machine may launch multiple EM-Desktop processes to solve the assigned variations (Design Points). Variations are distributed to each task (EM-Desktop process) equally, regardless of the machine hardware and each variation’s complexity. In practice, some

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tasks may finish earlier than others, in some extreme case some tasks may hours behind fastest task. DSO can redistribute tasks when a task finishes before other task. Variations are removed from slow tasks and reassigned to fast tasks. If you abort a task, they can be reassigned to the running task, when the running task finish its original assignment. For more information, see Large Scale DSO theory.

Large Scale DSO offers two new batchoptions related to the redistribution ability.

LargeScaleDSO/VarRedistribution, where 0 disables redistribution (default), and 1 enables it.

LargeScaleDSO/RedistributionLimit, is a positive integer specifying the minimum estimated remaining time (in minutes) for variations to redistribute to another task. The default is 3.

Aborting a Large Scale DSO Simulation

To abort the whole Job, select the **Abort** button on the Job Monitor dialog.

To abort using the Job progress bar, click the button next to the Job progress bar, Click the **Abort** menu item in the popup.

To abort all tasks in a Node(host), click the button next to the node progress bar., and click the **Abort** menu item in the popup. Aborted Variations will be redistributed to other running Nodes, if redistribution is turned on.

To abort an individual task, click the button next to the node progress bar, and click the “Detail” menu item in the popup. In the Task status dialog box, click the **Task** button in the grid. In the variation status dialog box, click **Abort** button. Aborted Variations will be redistributed to other running tasks, if redistribution is turned on.

To terminate a hanging EM-Desktop process, the hanging ansysedt.exe process won’t respond to the first abort command. Send a second abort command to terminate the hanging process. Remaining variations will be redistributed to other running tasks, if redistribution is turned on.

For an EM-Desktop process crashed or killed by Windows Task manager or other tools, the task status will be shown as aborted, Remaining variations will be redistributed to other running tasks, if redistribution is turned on.

Prerequisites for Large Scale DSO

General Prerequisites

- Ansys Electronics Desktop must be installed on the cluster which runs either a supported scheduler or Ansys RSM.
- The cluster is compatible with Large Scale DSO requirements.
- Every node of the cluster supports the disk space (in temp directory) and memory requirements of multiple engines that run in parallel.
- All the machines allocated to Large Scale DSO job must all come from the same platform, either Windows or LINUX.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- On the Windows platform, Ansys RSM is started as an admin account, rather than as a system account.

  > **Important:**
  >
  > Large Scale DSO does not support RSM Service running with system login credentials.

- On each machine of the cluster, the 'desktopjob' application is registered with Ansys RSM service using the command shown below:

  Windows: `<installation-directory>/<platform> desktopjob.exe -regserver`

  LINUX: `<installation-directory>/<platform> desktopjob -regserver`

DSO Behavior in HFSS

In general two level DSO occurs only when 2 different distribution technologies (engine and MPI) are used. Parametric distributions are implemented using the comengine. Solver distribution as well as frequencies and DDM are distributed by using the MPI technology.

HFSS and DSO

What's supported?

HFSS supports the following job distribution options

- Frequencies (MPI)
- Variations (engine)
- Transient excitations (MPI)
- Domain solver (MPI)
- Direct solver (MPI)
- Iterative solver (MPI)
- IE Solvers (MPI)

HFSS supports multi-level distribution. Following combinations can be used:

- You can distribute Variations, Domain solver, Direct solver
- Frequencies and any of the solvers.

What will we distribute?

HFSS will distribute Optimetrics unless:

- There is only 1 machine in the workgroup.
- Two-level distribution is disabled and MPI is required (see below).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Two-level distribution is enabled and MPI is required, but dividing the workgroup by the 2 level count would not leave enough machines for each MPI solve.

If Optimetrics variations are not distributed, we might distribute frequencies. Discrete and interpolating sweeps are distributable. Fast sweeps are not. As with Optimetrics, if MPI is required it takes precedence over variations. So we need to be able to distribute enough MPI machines for each frequency.

Next, we look whether the setup is requesting the iterative solver. If we are not distributing Optimetrics, we will distribute iterative solvers via MPI. If we are distributing Optimetrics and 2 level is enabled, then we need enough MPI solvers for each variation in order to distribute iterative solvers.

> **Note:**
>
> MPI distribution of domain and direct solvers is of higher precedence than Optimetrics. But, Optimetrics is of higher precedence than iterative solvers.

Finally, if we are a transient network problem there is no MPI. So, we will distribute transient excitations unless we are already distributing variations.

When is MPI required?

If the current solve setup (for example. “Setup1”) has domains enabled, MPI is required if any of these are true:

- If there is an array.
- If there are no FE-BI or IE regions.
- If the DSO configuration enables domain distribution and we have a FE-BI or an IE region.

- If Use Automatic Settings is checked or the DSO configuration enables distribution of transient excitations.

If the current solve setup does not enable domains, MPI is required if the Setup selected the direct solver and the configuration enables Direct solver distribution

How many MPI solvers are needed?

We can always use more, but the minimum requirement is

1. If we are using domains
    a. If it is an array problem, 3.
    b. If there is a FE-BI and an IE region, then 4.
    c. For all other direct solves, 3.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. If we are solving multiple transient excitations, the number of excitations.
3. Otherwise 2

Note:

- In the case of an uneven distribution, the smallest workgroup is used to determine capability. For example, if we have 8 machines and the user wants 2 level solve of a frequency sweep with 3 machines at level 1, that would divide the machines into (3,3,2). If this is an array problem, we would need 3 machines for each workgroup. So, we would disable 2 level distribution (rather than reconfigure to only have 2 workgroups).
- The user's selection of (say) frequency distribution or 2 level does not *require* that we do so. In the above example, if we are adapting we might choose to devote all 8 machines to the array solve at each adaptive pass as there is neither frequency nor 2 level possible.

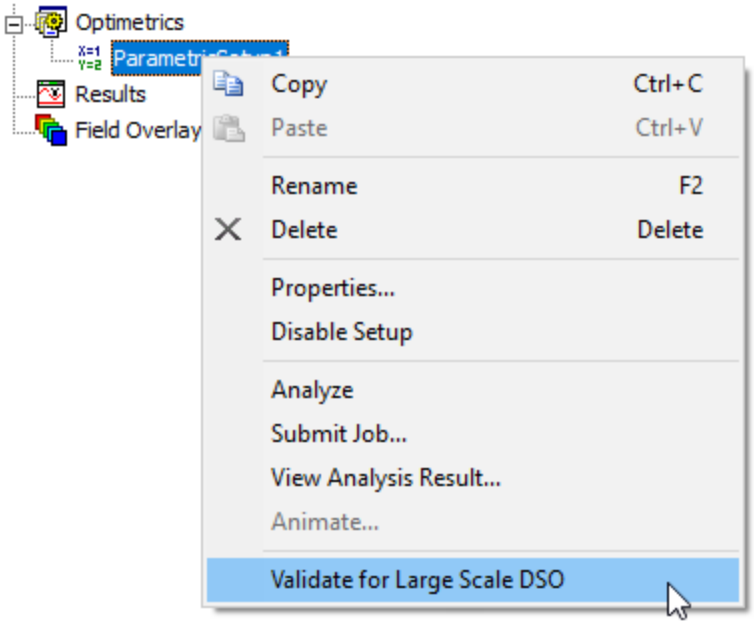
Job Management Interface for Large Scale DSO

Large Scale DSO jobs run only in non-graphical batch mode, irrespective of the scheduler environment. This is in contrast to a Regular DSO job, which can run in graphical mode. This consideration implies that projects corresponding to Large Scale DSO job must be saved and closed prior to job submission. Secondly, the command to submit a Large Scale DSO job is only available through **Tools** > **Job Management** or via a command window, while a Regular DSO job can be run in an RSM environment by right-clicking directly on the parametric setup. The Job Management window is accessed by running Ansys Electromagnetics product Desktop on the designated Postprocessing Node of the cluster. The Desktop provides UI commands for scheduler selection, job submission and job monitoring/control.

When you have selected the scheduler, perform the following steps to submit a Large Scale DSO job:

1. Set up and prepare the model on your local workstation. Right-click the desired solution setup and select **Validate for Large Scale DSO**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Correct any errors and save the project.
3. Close Electronics Desktop.
4. Copy the project (or folder, if the project references external files) from a personal workstation to a shared drive on cluster.

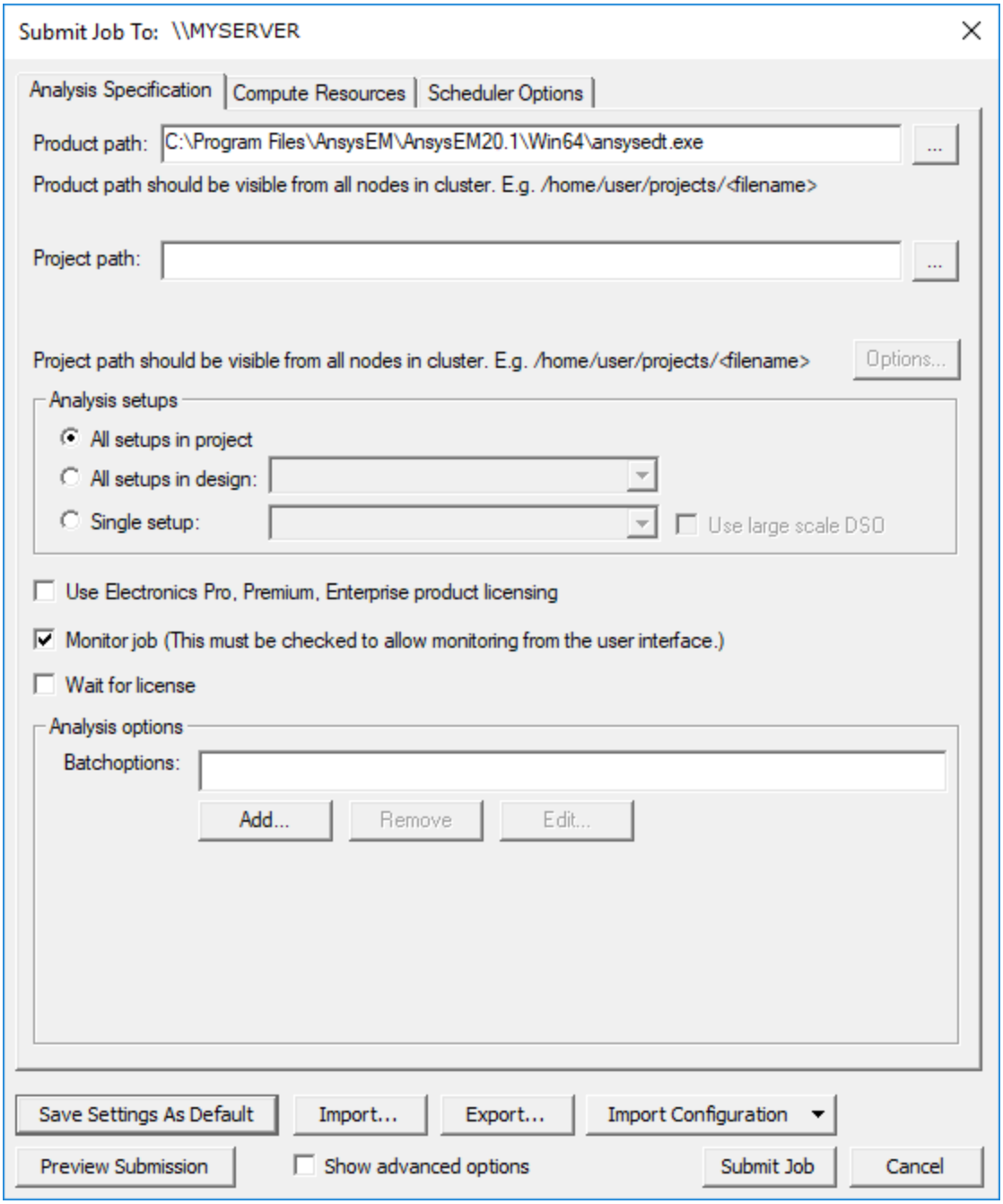
   In the RSM environment, you must specify a machine list (See: HPC and Analysis Options).

   In a Linux scheduler environment, a cluster must have a designated Postprocessing Node.

5. Open a remote desktop session (or equivalent) on the node corresponding to the first machine in the machine list (the designated Postprocessing Node on Linux).
6. Launch Electronics Desktop on that node and open the project.
7. Verify that the model has been prepared correctly.
8. Close the project.
9. Submit the job one of several ways:
   - Click **Tools** > **Job Management** > **Submit Job**.
   - Click **Project** > **Submit Job**.
   - Click **HFSS** > **Submit Job**.
   - Select the **Simulation** tab and click the **Submit** icon.

   The **Submit Job To** window appears, on the **Analysis Specification** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

Options vary slightly depending on the selected scheduler. See: HPC Integration.

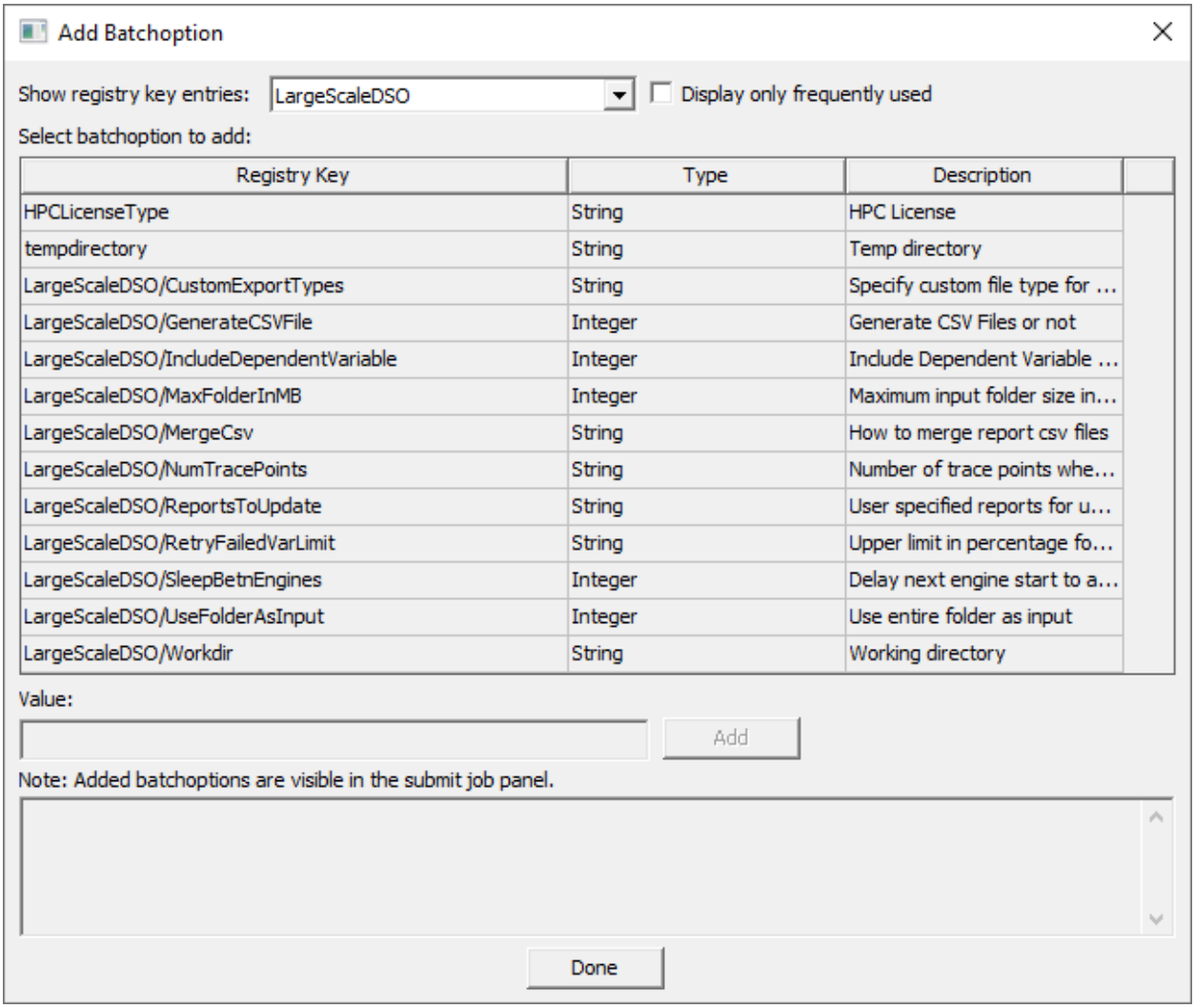
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. Specify information for all fields:
    - Most options will not be enabled until you select a **Project path**.
    - Under **Analysis setups**, select setups for analysis.

      For Large Scale DSO, select a **Single setup** from the drop-down menu and ensure the **Use large scale DSO** check box is selected.

    - Determine whether to **Use Electronics Pro, Premium, Enterprise product licensing**, whether to **Monitor job**, and whether to **Wait for license.** If you wish to monitor jobs through **Tools** > **Job Management**, you must enable job monitoring.
    - Add **Batchoptions**, if desired.

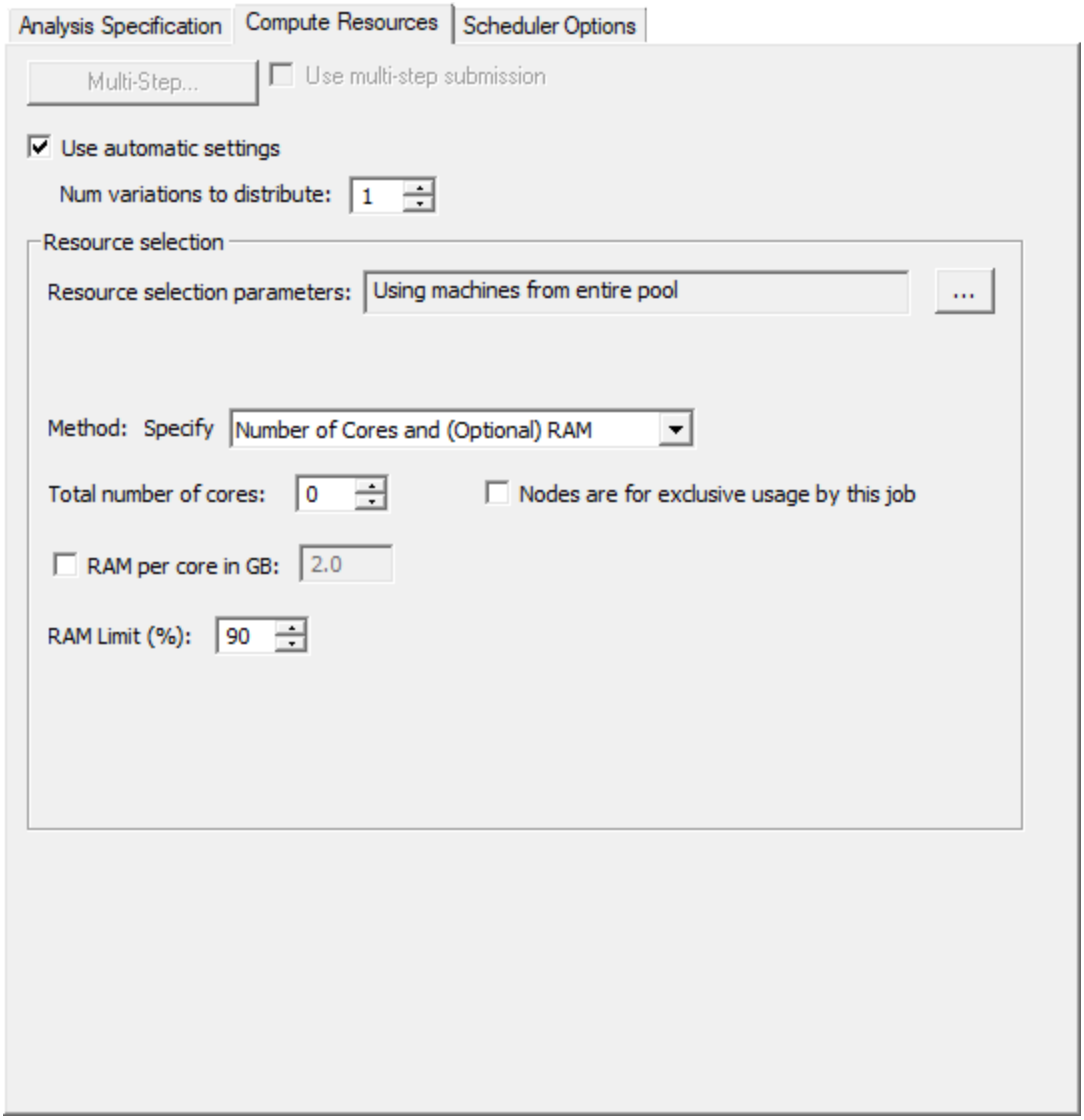
      The following shows batchoptions specific to Large Scale DSO.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11. Select the **Compute Resources** tab.

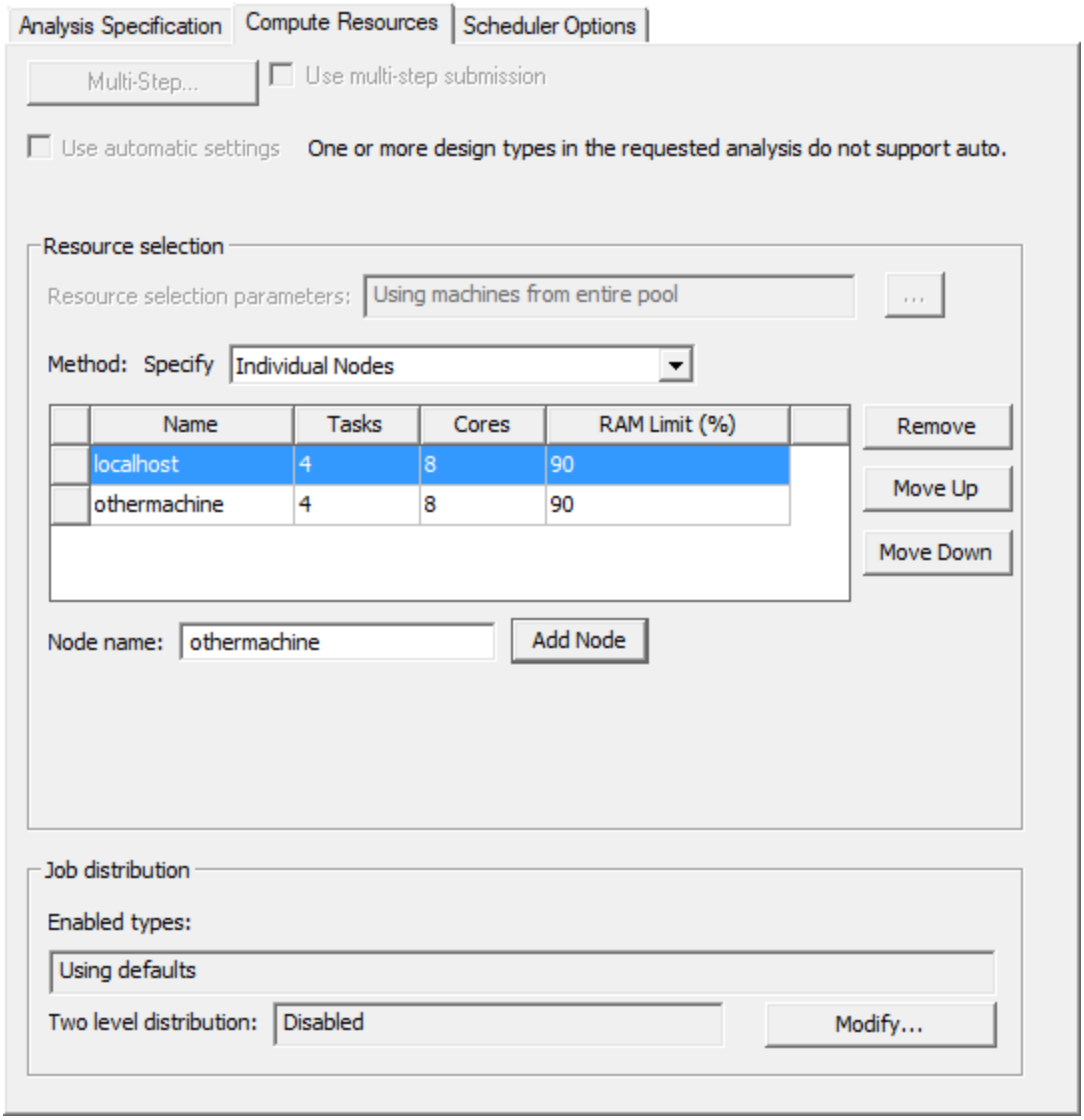
    When available, **Use automatic settings** may be selected. If it is selected, also select the Number of variations to distribute.



    The values you specify represent minimal requirements for each condition that can interact in leading to the total resources the Scheduler derives from them. A submission preview shows the number of resources assigned.

    When automatic settings are not available or not selected, additional options appear:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For RSM Large Scale DSO jobs submitted from the Job Submission panel, localhost must be the first node in the resource selection panel. Otherwise, the job will fail.

In the **Job distribution** area, you can enable or disable **Two-level distribution** by clicking **Modify**. See: Two-level Distribution Guidelines.

12. If desired, click **Preview Submission** to view a summary of the commands to be sent to the scheduler. The text can be copied to the clipboard.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13. To submit the commands to the scheduler, click **Submit Job**.

    > **Note:**
    >
    > The RSM environment does not support queuing, so clicking **Submit Job** starts it immediately.

14. If you enabled job monitoring, you can monitor the job via **Tools> Job Management > Monitor Jobs**. See: Monitoring Jobs.

Large Scale DSO Command Line Syntax

Large Scale DSO operates through a non-graphical batch application called `desktopjob`. You can run the `desktopjob` command line to perform parametric analysis DSO. The command-line interface supported by this batch program is consistent with the command line used for current DSO jobs. `desktopjob -help` lists all available command-line options as shown below:

Command Line Syntax:

```
desktopjob.exe <options> <project-path-on-shared-drive>
```

Note that the project path can be to an archive file.

Options:

- **-help**

  Prints the help text.

- **-cmd**

  Specifies the command to run.

  Available choices: dso

- **-ng**

  Runs the analysis in non-graphical mode.

- **-monitor**

  Outputs progress and messages to standard output/error.

- **-waitforlicense**

  Queues the job until licenses are available.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **-preserve**

  Preserves the local storage space of the distributed job for investigation into the job's run. If local storage directory (for example, the temp directory) is provisioned by scheduler, ensure it is also configured to preserve the job's local storage. *This storage should be deleted manually*.

- **-batchoptions**

  Overrides the Tools/Option entries through either a batchoptions file or batchoptions string.

  -batchoptions specific for Large Scale DSO include:

  To retry failed variations, new batch option LargeScaleDSO/FailedVarRetryCount.

  For each task to re-simulate its failed variations when its assigned variations are finished, specify the FailedVarRetryCount batch option with positive integer. Default 0; Value-Zero(0) will disable the re-simulation for failed variations. Value ranges from 0 or positive integer.

  Redistribution batch option, LargeScaleDSO/VarRedistribution .Value 0 disables redistribution (default). Value 1 enables redistribution.

  Redistribution limit batch option, LargeScaleDSO/RedistributionLimit. Minimum estimated remaining time (in minutes) for variations to redistribute to another task. Value must be positive integer. Default 3 minutes

  Example:

  ```
  -batchoptions <config-file-on-shared-drive>
  -batchoptions "'name1'='val1' 'n2'='v2' "
  ```

- **-jobid**

  Specify a custom job ID for the job. The job's output is organized into a folder with job ID name. This parameter is ignored when run under a scheduler.

- **-machinelist**
  - In the context of **Ansoft RSM**:

    Specify machines for distributed analysis. Machine list is specified either inline (as a comma separated machine names) or through a file. Multiple cores are specified by repeating the name of machine or by embedding number of cores in the machine name, using a colon separator.

    Example 1:

    ```
    -machinelist "list=m1,m1,m1,m2,m2,m3"
    ```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example 2:

```
-machinelist "list=m1:3,m2:2,m3"
```

Example 3:

```
-machinelist "list=m1:1:3,m2:2:2"
```

Example 4:

```
-machinelist "file=machines.txt"
```

- In the context of **a scheduler such as LSF**:

  Specify the portion of total machines for distributed analysis. Use remaining for overhead or shared memory multiprocessing.

  Manual Example:

  ```
  -machinelist "Num=10"
  ```

  Auto Example:

  ```
  -machinelist "NumCores=40"
  ```

  Note: In a scheduler environment, the machinelist argument can be omitted, but the results may not be as expected or desired. If there is no machinelist argument, the scheduler-assigned resources will be used, assuming one core per task. E.g., if the scheduler assigns 40 cores, there will be 40 variations solved in parallel, with only one core used per variation. Since one core per task may not make sense in many cases or may not be what the user desires, a warning message that machinelist is missing will be output, but the job will proceed.

- **-auto**
  - Run the leaf jobs in auto mode. See Submitting Large Scale DSO Job Examples.
- **-numdistributedvariations**:

  Specify the number of parallel leaf jobs to run in auto mode. Required when running under a scheduler. If not specified when not running under a scheduler, then the number of tasks for each machine must be specified in the machine list. The total number of tasks will be the number of parallel variations (leaf jobs) to run.

  Example:

  ```
  -machinelist NumCores=40 -auto -NumDistributedVariations 10
  ```

- **-usefolderasinput**

  Choose this option if the job's input represents an entire folder rather than just the project file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **-maxfolderInMB**

  Specify the maximum size input folder that is allowed for a valid job (in MBytes). By default, the maximum size allowed for input is 10MB. Specify a value of 0 to remove this size restriction and enable inputs of any size. This option applies when `-usefolderasinput` is used.

- **-workdir**

  Specifies the shared drive folder for status and result files generated by analysis. By default, the results folder of input project is used as the work directory.

- **-meregecsv: [acrossDPs | singleDP | both]**

  across DPs: Merge report csv files for all design-points (variations). One file is created per trace, across all variations.
  singleDP: Merge csv files within a single design-point (variation). One file is created per variation, per a set of traces that can be merged.
  both: Merge all traces that have the same primary sweep for all design-points (variations) into one csv file.
  Interpolation note: If primary sweep values are not uniformly spaced, mergecsv is enabled with traced values and are re-sampled uniformly using '-batchoptions' syntax as shown below:

```
-batchoptions "'LargeScaleDSO/NumTracePoints'=500"
-batchoptions
"'LargeScaleDSO/NumTracePoints'='PrimarySweepName:200'""
-batchoptions
"'LargeScaleDSO/NumTracePoints'='ReportName1:Trace1:100;ReportNam
e1:Trace1:100;ReportName1:TraceName2:200'"
```

- **-abort**

  Abort a running job identified through the job's working directory. Example: `-abort <projectresultsfolder-path>/>jobid>`. For a complete discussion of methods for aborting jobs or specific tasks, see the discussion of Aborting a Large Scale DSO Simulation under Large Scale DSO for Parametric Analysis.

- **-repackageresults**

  Choose this option to add simulations results to the input archive file. Note: this option only applies if an archive file is provided as input.

- **-batchsolve**

  Solves the specified parametric setup.

  Syntax for the setup:

  `<design-name>:Optimetrics:<parametric-setup>`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Large Scale DSO Job Outputs

A Large Scale DSO analysis does not support the output of full parametric results. Instead, it extracts a subset of results using predefined rectangular plots, which are created by the user before running the job. The extracted columns of data are saved as CSV files. Typically, there is one CSV file per-trace, per-variation. The outputs can be either imported as datasets for post-processing in the desktop also as function of parametric variations, or used directly in downstream applications (for example, Excel, or custom programs that parse .csv files).

> **Note:**
>
> Non-Rectangular plots of the design (for example, statistical eye and digital plot) are not extracted.

Output Location and Organization

The results of a Large Scale DSO job are located in the `<workdir>/<jobid>/results` folder. If `workdir` is not specified on the job command line, it is same as the input project's results folder. For example, the default `workdir` corresponding to `\\shared\projects\tee.aedt` is `\\shared\projects\tee.aedtresults`. Within this results folder, there is one folder per variation. The name of the variation's folder is an integer number corresponding to variation's index in the parametric table. For example, a variation folder named '4' has results for the fifth row of parametric table, while a variation-folder named '0' has results for the first row of the table. Each variation's folder contains a CSV file for each trace.

CSV File Contents

The initial header rows of the CSV file define the solved variation. For each such row, the first column contains a variable name and the second contains a variable value. The row following variation rows has the name of primary sweep and the name(s) of extracted quantities. Subsequent rows contain data—quantity values as a function of primary sweep.

These examples provide context:

- **Traces of an S-parameter Report** – The data portion of the CSV file contains two columns of data: the first contains Freq values and the second contains values for the trace's s-parameter component.
- **Trace of a Far Field Report** – Suppose there is a far field report with a trace (magrE), whose primary sweep is phi and whose secondary sweep is theta. Further suppose that two values of theta are chosen and all values of phi are chosen. For this trace, the data portion of CSV file contains three columns of data: the first containing phi values, the second containing magrE values for the first value of theta, and the third column containing values for second value of theta. The magrE output columns are titled as 'magrE_crv1' and 'magrE_crv2' respectively.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Advanced Sweeps** – In the case of a trace with special primary sweep (such as the trace of a time domain quantity), one CSV file is created per curve of trace, per variation. These CSV files always have two columns, irrespective of the number of values chosen for secondary/higher sweeps.

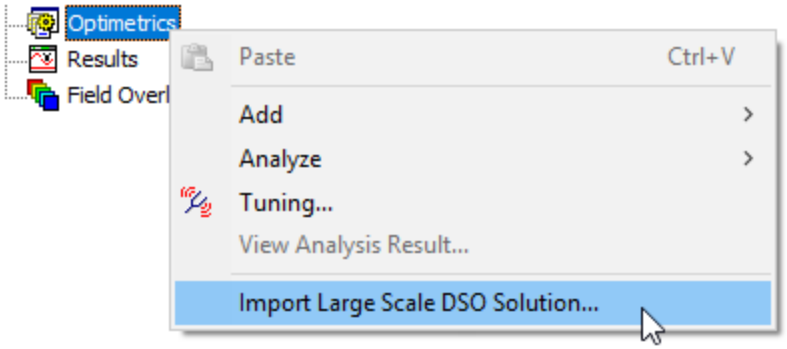
Postprocessing Large Scale DSO Dataset Solutions

In Ansys Electronics Desktop, the **Import Large Scale DSO Solution** command allows you to post-process Large Scale DSO dataset solutions.

Importing a Large Scale DSO Solution

To import a Large Scale DSO solution:

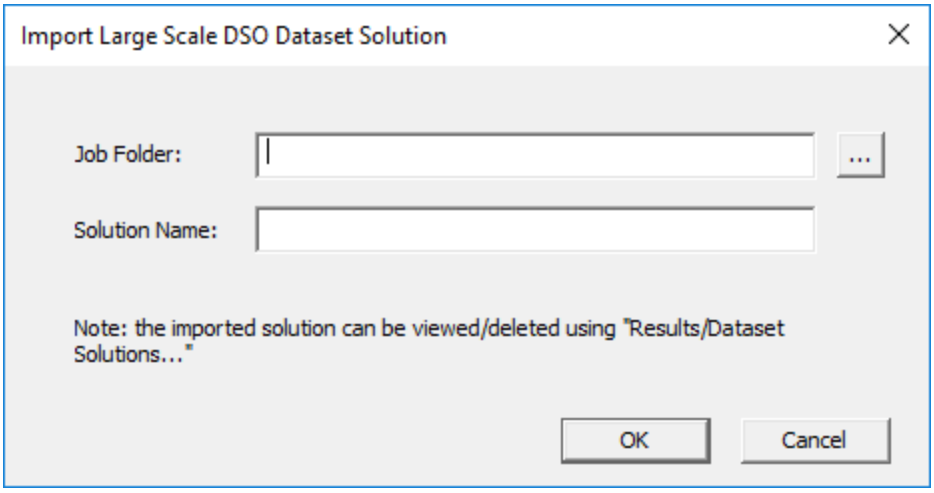
1. In the Project Manager, right-click **Optimetrics** and select **Import Large Scale DSO Solution**.



> **Note:**
>
> You can also access this window by right-clicking **Results** > **Dataset Solutions**.

The **Import Large Scale DSO Dataset Solution** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click the ellipsis button (**...**) and browse to select a job folder. To select a results dataset, double-click the results folder name. Results folders are organized by the scheduler prefix (for example, RSM) and job number.
3. Click **Open** to return to the **Import Large Scale DSO Solution** window.
4. Review the **Job Folder** path and **Solution Name**, then click **OK**.

   The dataset is imported.

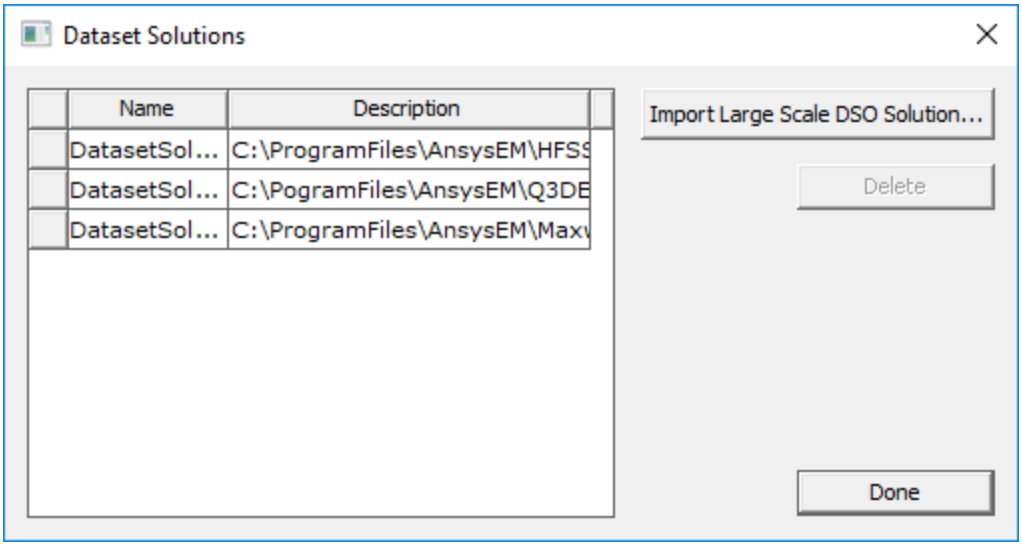
Viewing Imported Solutions

To view a list of imported datasets:

1. In the Project Tree, right-click **Results** and select **Dataset Solutions...**

   The **Dataset Solutions** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selecting a dataset enables the **Delete** button, which you can click to remove it.

2. Click **Done** to close the window.

Creating a Dataset Report

After importing one or more DSO solutions, you can create a dataset report.

To do so:

1. In the Project Tree, right-click **Results**. Select **Create Dataset Report** > **[Report Type]**.

   The **Report** window for the report type opens. See Creating Reports.

Cloning a Dataset Solution

If you reopen a project that was solved using Large Scale DSO, you can quickly clone the solution report:

1. In the Project Tree, under **Results**, right-click the report and select **Clone from Dataset Solution** > **[Solution Name]**.

   The cloned solution appears. You can now reuse the existing report definition rather than creating a new report.

Large Scale DSO Job Monitoring

The **Monitor Job window** allows you to monitor the progress and status of Large Scale DSO jobs, including information on variations solved so far, variations currently solving, and the number of variations remaining.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Additional Resources for Large Scale DSO Monitoring

Large Scale DSO avoids detailed intra-variation monitoring, as it increases network traffic for large-scale jobs. Additional monitoring resources include:

- **Cluster Monitoring Tools** – Standard cluster monitoring tools are ideal for job-neutral resource monitoring as they use negligible network bandwidth.
- **Detailed Monitoring of Analysis of a Variation** – For detailed monitoring, you may want to examine a job's log files. Large Scale DSO writes detailed logs about the machines where engines are running and the local storage location of per-engine distributed databases. You can log in to individual machines for deeper probing of each distributed engine.

  The following logs are available:

  - **Per-Node Logs** – There is one desktopjob.log file per node assigned to the job. This log contains information regarding the node such as name, local storage folder, and number of engines started on this node. It is located in <workdir>/<jobid>/r<nodeIndex>. For example, <workdir>/<jobid>/r0 contains the desktopjob.log corresponding to the engines running on the first node of job, while <workdir>/<jobid>/r2 contains the log corresponding to engines running on the third node.
  - **Per-Engine Logs** – There is one desktopjob.log file per distributed engine. It is located in <workdir>/<jobid>/r<nodeIndex>/r<taskIndex>. For example, <workdir>/<jobid>/r0/r0 contains the log corresponding to first engine running on first node, while <workdir>/<jobid>/r1/r2 contains the log corresponding to the third engine running on the second node. Engine unique information (such as local storage of this engine) is logged here.
  - **Parametric Analysis Log** – This log file is located in'<workdir>/<jobid>/r<nodeIndex>/r<taskIndex> and corresponds to Desktop's local-machine parametric batchsolve. It is available only at the end of analysis and contains information regarding the variations solved by this engine and any info/warning/error messages.
  - **Root Log** – This is the top-level desktopjob.log file that logs job distribution information such as hierarchical activation and the list of nodes assigned to this job.

For a complete discussion of methods for aborting jobs or specific tasks, see the discussion of Aborting a Large Scale DSO Simulation under Large Scale DSO for Parametric Analysis.

Large Scale DSO Deployment/Configuration

LINUX Cluster Configuration

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 'Temp directory' configuration.

  Temp directory is either on 'local storage' or on storage that has equivalent speed characteristics. The I/O rates of the storage should be invariant to network traffic

  Temp directory on a host has sufficient space to hold results database for the variations that are solved on it. (Note: This storage is freed at the end of the analysis.)

  The amount of required space depends on the number of engines per node and the cumulative variations solved on this node

  The amount of required space depends on the project's compression-options. For example, if 'Save Fields' of a parametric setup is OFF, the space requirement is smaller by the amount of space taken up by field solution data.

- Ansoft RSM environment: In the case of supported scheduler environments, there is no extra configuration needed. In the case of Ansoft RSM environment, following additional steps are needed:

  Ansoft RSM must be running on all the nodes of cluster. The credentials of 'RSM service' allow read/write to shared drive. Reason: the remote engine processes are launched using the credentials of RSM service

  Registration of 'desktopjob.exe' with RSM service: 'desktopjob' program must be registered with Ansoft RSM using 'desktopjob -regserver". To ensure that the registration is successful, check that the 'desktopjob' entry in '<RSM-installation-folder>/AnsoftRSMService.cfg' file is valid.

> **Note:**
>
> LINUX specific critical note: Edit AnsoftRSMService.cfg and replace 'desktopjob.bin' with 'desktopjob'

Major limitation: In the Ansoft RSM environment, Large Scale DSO can only be enabled for one product.

Troubleshooting hints (Ansoft RSM environment only): "shared drive read/write" requirement is a new constraint introduced in Large Scale DSO. So if user runs into a situation where Regular DSO jobs run and Large Scale DSO jobs fail, one possible cause for the failure: RSM service does not have privileges to read and write to project folder located on shared-drive.

Windows Cluster Configuration

All the above steps apply, except for steps that are stated as LINUX-specific. Additional instructions:

- Ansoft RSM and Ansys Electromagnetics products are either installed locally on each node of cluster OR installed on a single shared-drive available to all nodes of cluster.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Registration of 'desktopjob.exe' with RSM service.
- Network installation: desktopjob.exe is registered with RSM service once, on any of the nodes of cluster
- Local installation: Since each node has it's own RSM installation, desktopjob.exe must be registered with RSM on each node.

> **Important:**
>
> Ansoft RSM service must be started using the credentials of a non-system 'admin' account, which has read/write permissions to project's shared drive. If RSM service runs as 'system' user, large-scale-dso jobs will fail

Heterogeneous Cluster configuration

Limitation: Currently heterogeneous cluster (with both linux and windows nodes) is not supported. This is due to the shared drive requirement.

Large Scale DSO Tutorial Example

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

This section provides an example of the use of Large Scale DSO to distribute parametric variations of an HFSS model across the nodes of a cluster or to multiple cores of a single machine.

This example presumes that your configuration satisfies the Prerequisites for Large Scale DSO. For this example, pre-suppose that we have a Windows cluster. Further suppose that the shared drive folder, which contains the input projects and computed results is at the location *\\sjo7na1\hfssprojs*.

Major steps for Large Scale DSO Example

1. Prepare the model for Large Scale DSO Analysis
2. Submit the Large Scale DSO Job
3. Post process the results

Prepare the Model for Large Scale DSO Analysis

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Launch the desktop.
2. Create the input project on the shared drive.

   For this example, start with the standard HFSS OptimTee.aedt example and copy it to the shared drive.

   Copy "<installation-directory>\<platform>Examples\RF_Microwave\OptimTee.aedt" to "\\sjo7na1\hfssprojs\OptimTee.aedt"

3. set up the parametric table.

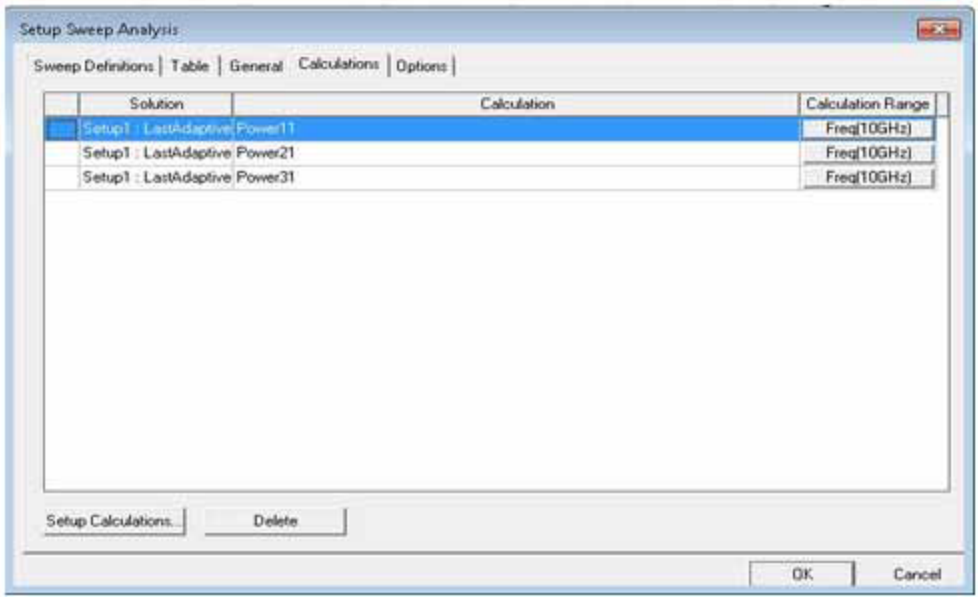
   For this example, use the existing 'ParametricSetup1' as the parametric setup to solve.

4. Prepare the outputs.

   Outputs from Large Scale DSO come from pre-defined rectangular plots that are created before the Analysis command is issued. Follow the steps below:

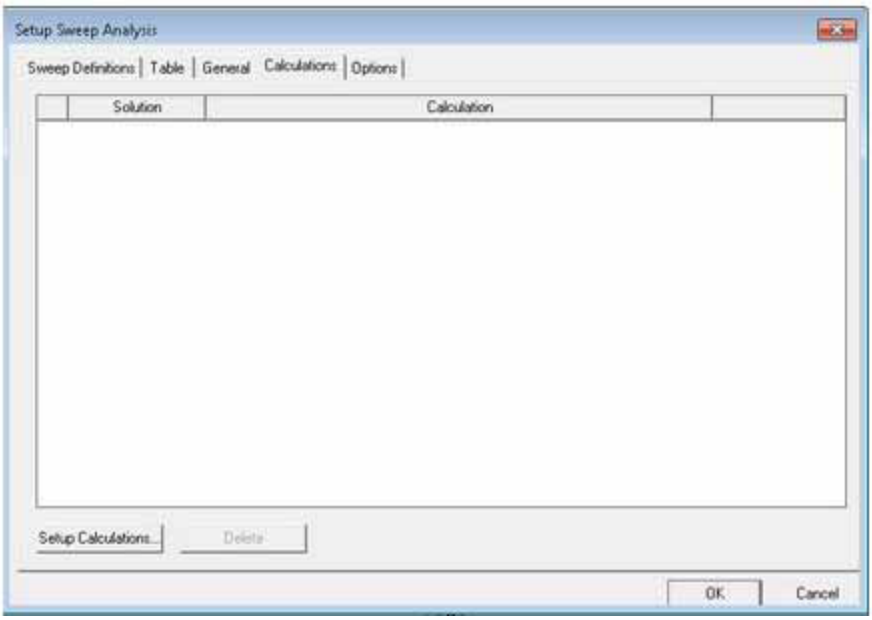
   a. Because these DSO outputs come solely from Rectangular Plots, delete all other postprocessing setups, and then turn off Save Fields And Mesh as shown below.

      Open ParametricSetup1 for editing. You will see that the OptimTee parametric setup contains three calculations, as shown below.



   b. Delete all three calculations. When you have done so, the Setup Sweep Analysis Calculations tab looks like this:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



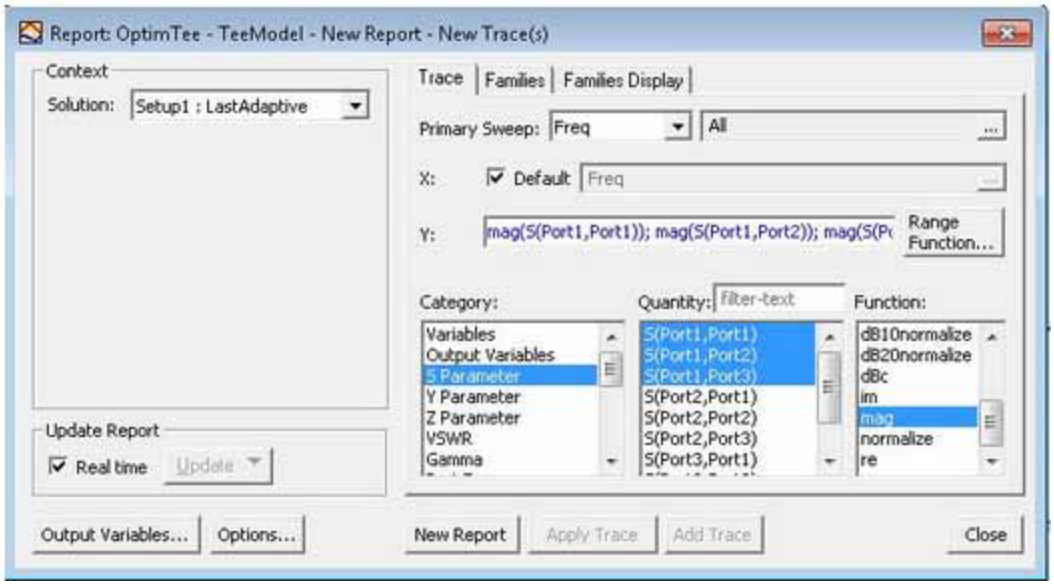
c. Click on the **Options** tab and uncheck Save Fields And Mesh, as shown below:

5. Use the Reporter to define outputs.

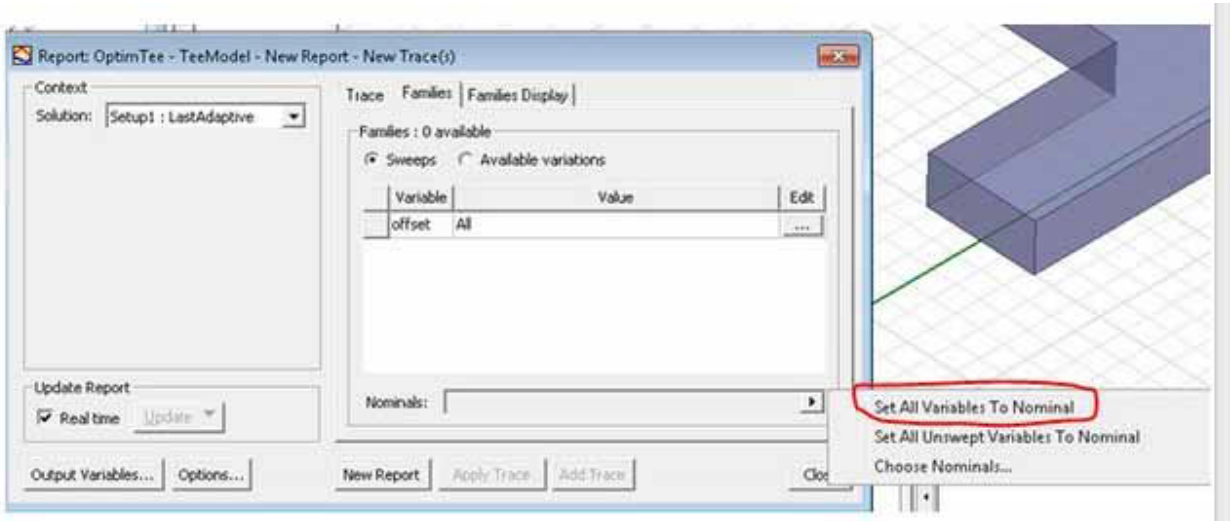
   For this example, you add six traces that correspond to six csv outputs of the Large Scale DSO job: mag(S11), mag(S12), mag(S13), Power11, Power12, and Power13.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. As shown below, select three quantities: mag(S11), mag(S12), and mag(S13).



b. Click on the Families tab and ensure that all variables are set to Nominal, as shown below.



Note:

For Large Scale DSO, outputs are not extracted correctly unless all variables on the **Families** tab are set to nominal.

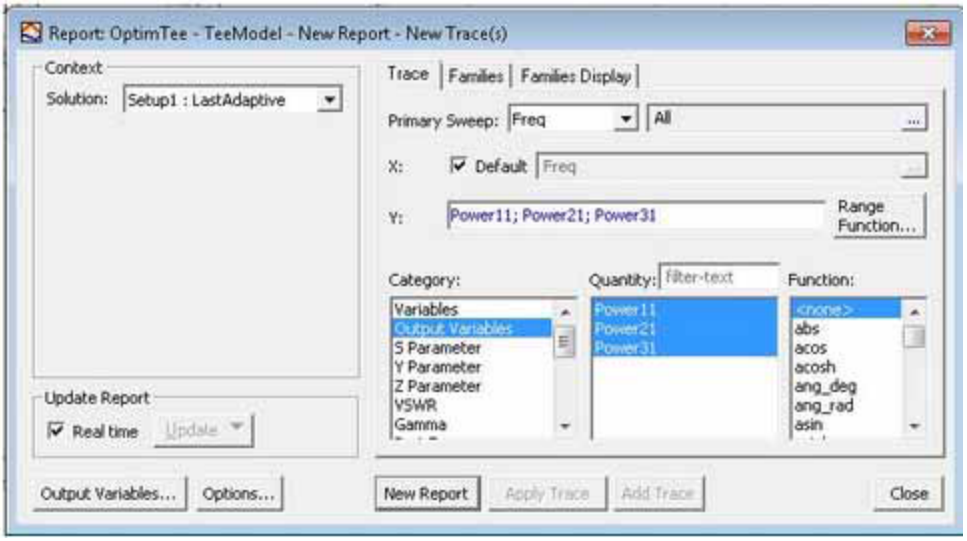
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Click the **New Report** button.

   Use the Report to create a Power distribution plot that has three power distribution traces.

d. To create an S-parameter plot that has three traces, click **Results> Create Modal Solution Data Report> Rectangular Plot**.

e. As shown below, select three quantities, Power11, Power21, and Power31.

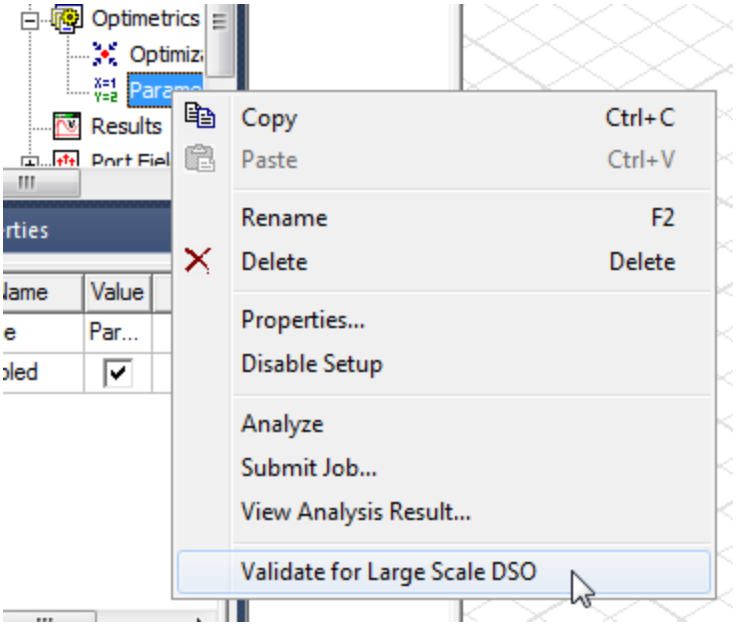


f. Click on the Families tab and ensure that all variables are set to Nominal.

g. Click the **New Report** button.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Right-click the parametric sweep, and select Validate for Large Scale DSO.



7. A dialog reports any errors. These may occur if the steps just taken are neglected:

   ```
   [error] Please remove all the calculations in Parametric Setup/Calculation page.
   ```

   ```
   [error] Please turn off "Save Fields And Mesh" in Parametric Setup/Options page.
   ```

   ```
   [error] No rectangular plot exist in the design. Please create a rectangular report.
   ```

8. Correct any errors, if necessary to pass validation.
9. Save and Close the project.

Next

Submit the Large Scale DSO Job: Examples

Submit the Large Scale DSO Job: Examples

This section includes examples of submitting a Large Scale DSO job using Ansoft RSM and a Scheduler.

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

Using the Ansoft RSM Environment

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Before submitting the job make sure that the input project is not open in any Ansys Electronics Desktop window.
2. For this example, suppose that there are two quad-core machines on your cluster with the names "m1" and "m2". Further suppose that there are two engines per machine, for a total of four parallel engines. Let the number of processors allocated to each engine be 1.
3. From a command prompt, issue the following command:

   ```
   "<installation-directory>\ANSYS
   Inc\v251\AnsysEM\desktopjob.exe" -cmd dso -machinelist
   "list=m1:2,m2:2" -batchoptions
   \\sjo7na\aedtprojs\hfssoptions.txt -batchsolve
   "TeeModel:Optimetrics:ParametricSetup1"
   \\sjo7na\aedtprojs\OptimTee.aedt
   ```

   where the file \\sjo7na\hfssproj\hfssoptions.txt has the following contents:

   ```
   $begin Config
     'Desktop/Settings/ProjectOptions/NumberOfProcessors'=1
     'HFSS/Preferences/SaveBeforeSolving'=0
     'HFSS/Preferences/HPCLicenceType'='pack'
   #end 'Config'
   ```

4. Suppose the above job is assigned ID "jobID".

Using a Scheduler Environment (such as LSF)

1. Suppose you want to solve variations using four parallel engines, each engine being assigned a single core.
2. From a command prompt, run the following command:

   ```
   bsub -n 4 "<installation-directory>\ANSYS
   Inc\v251\AnsysEM\desktopjob.exe" -cmd dso -batchoptions
   \\sjo7na\hfssprojs\hfssoptions.txt -batchsolve
   "TeeModel:Optimetrics:ParametricSetup1"\\sjo7na\hfssprojs\Opti
   mTee.aedt
   ```

   where the file \\sjo7na\hfssproj\hfssoptions.txt has the same contents as the RSM example above.

3. Suppose the above job is assigned an ID "jobid"

For example with 2 nodes of 44 cores each (with multiple distributed variations per machine):

MANUAL: `bsub -n 88 <installation-directory>\v<version>\<platform>\desktopjob -cmd dso -machinelist num=8 -monitor -useelectronicsppe=1 -ng -batchoptions " 'HFSS/NumCoresPerDistributedTask'=11 'HFSS/RAMLimitPercent'=90`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
'HPCLicenseType'='Pack'" -batchsolve
TeeModel:Optimetrics:ParametricSetup1 OptimTee-v232-1.aedt
```

AUTO: `bsub -n 88 <installation-directory>\v<version>\<platform>\desktopjob -cmd dso -machinelist numcores=88 -auto -NumDistributedVariations 8 -monitor -useelectronicsppe=1 -ng -batchoptions " 'HFSS/RAMLimitPercent'=90 'HPCLicenseType'='Pack'" -batchsolve TeeModel:Optimetrics:ParametricSetup1 OptimTee-v232-1.aedt`

For example, with 16 nodes of 44 cores each (note each distributed variation will span 2 machines):

AUTO: `bsub -n 704 <installation-directory>\v<version>\<platform>\desktopjob -cmd dso -machinelist numcores=704 -auto -NumDistributedVariations 8 -monitor -useelectronicsppe=1 -ng -batchoptions " 'HFSS/RAMLimitPercent'=90 'HPCLicenseType'='Pack'" -batchsolve TeeModel:Optimetrics:ParametricSetup1 OptimTee-v232-1.aedt`

Next

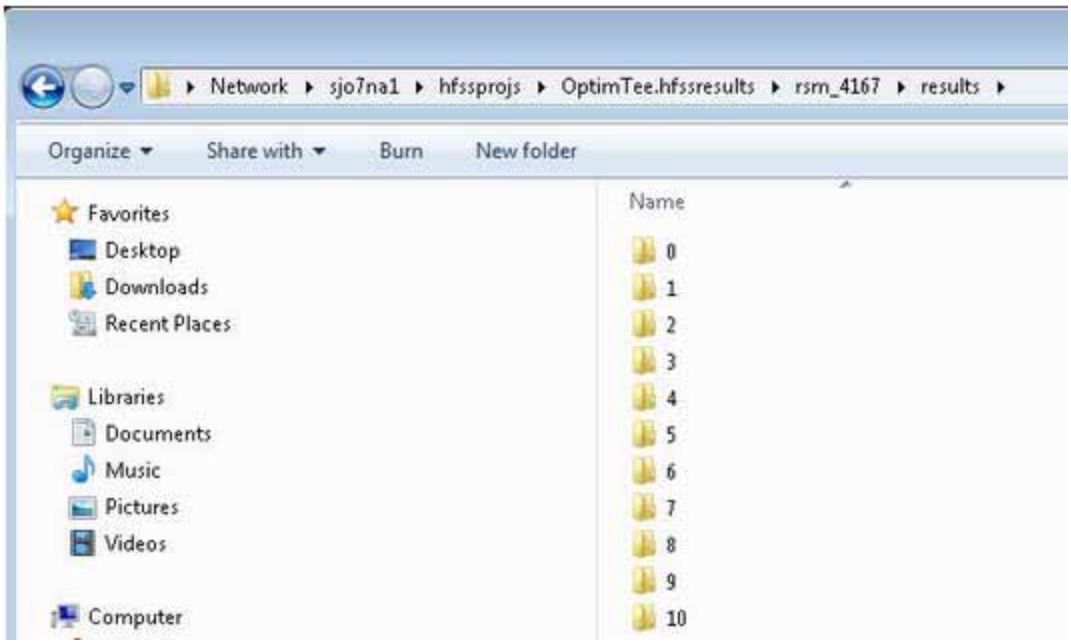
Large Scale DSO Example: Post Process the Results

Large Scale DSO Example: Post Process the Results

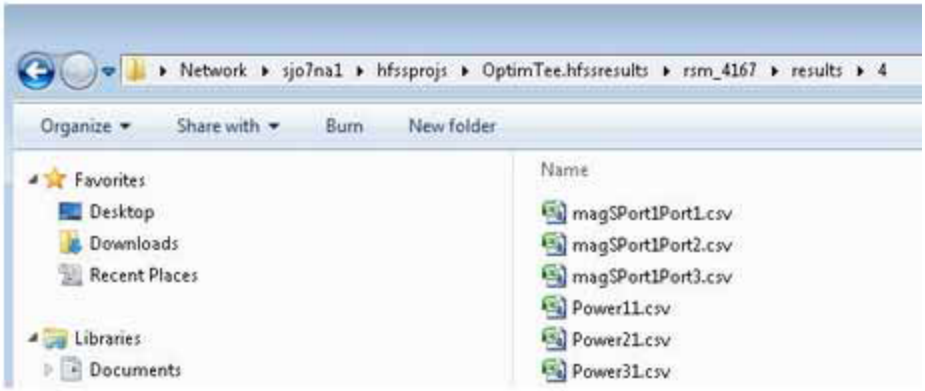
Once the job is done, output is available in the ~\OptimTee.aedt\jobid\results folder. Each variation creates a subfolder, which in turn has one csv file per trace of each report. See the detailed information regarding job monitoring and the location of the analysis logs.

The following figure shows the results for 10 variations as located in 10 folders.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



There are six csv files corresponding to three S-parameter traces and three power distribution traces, as shown below for the fourth variation.

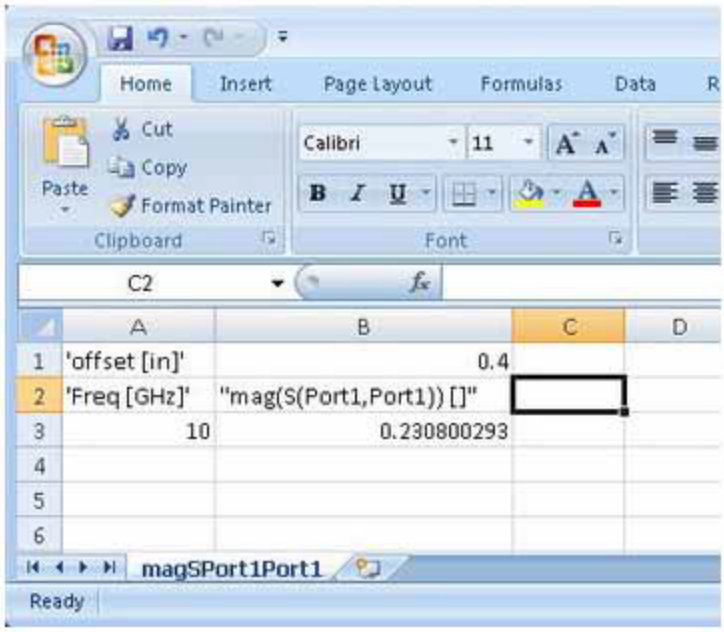


You have three options for postprocessing csv files.

- Import Large Scale DSO Dataset Solution
- Use Microsoft Excel or any other application that has csv post processing functionality.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Parse the csv output into your custom program, for any downstream flow.



Process Limits for Large Scale DSO on Windows

Windows has a default limit on the resource usage for non-interactive sessions, which limits the number of processes that can be started in parallel. This puts a limit on the number of Large Scale DSO tasks (Design Points) that can be started on a given Windows machine. The limit is defined by the third parameter of the SharedSection segment of Windows registry HKEY_LOCAL_MACHINE\SYSTEM\CurrentControlSet\Control\Session Manager\SubSystems\Windows.

`SharedSection=1024,20480,768` (In this example, the limit is set to 768)

To increase the number of tasks that can be run on a single machine, increase this limit gradually, as needed, in multiples of 1024 (but not exceeding 8192). Ansys in-house testing has shown success in running up to 64 tasks on a machine with the value of 4096.

For more information, refer to Microsoft documentation.

Large Scale DSO Known Issues/Troubleshooting

Node Order

For Large Scale DSO jobs that are submitted from job submission panel using RSM, localhost must be the first node in the resource selection node list, otherwise Large Scale DSO solve with RSM will fail. Set the order in the **Submit Job To** window, **Compute Resources** tab:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Name	Tasks
	localhost	1
	othermachine	1

Parallel Task Limitation for LS-DSO Parametric Variations

There some limitations of running short parametric runs in parallel and using a subset of cores and/or running across multiple machines instead of just one. LS-DSO does help in getting close to linear scalability for parametric runs in many scenarios, it has some overheads and typically they are small in comparison to total time of run. However, a few factors in particular scenarios can make them prominent.

The startup of each task involves copying the project and launching ansysedt to solve a subset of parametric table. Typically, one task solves several variations and this startup cost is negligible in comparison to total time. Consider a case where each task is solving only single variation. Additionally the actual solve time of one variation is relatively small, 3-5 minutes. These factors make the startup cost significant. There is also some variance in solve times of different variations. When all variations are solved in parallel, the total time is determined by slowest variation, even though the expectation intuitively might be relative to average time. That could make the perception of overhead worse in this case.

Another important factor is the number of parallel tasks on each machine relative to the total number of cores on the machine. Each task is effectively running an ansysedt of its own and solving subset of parametric table on a copy of project. The solve process does involve significant disk IO. If you run too many tasks on a single machine, they may end up competing with each other for single disk, and that could cause them to slowdown. The conflict in memory access may also become a factor.

In a particular case, 26 tasks were run on one 26 core machine, and this caused significant slowdowns. Spreading the tasks across machines helped scalability as an example with a 32 core machines in a Linux Cloud, we ran this job with 26 tasks on a single machine and then another one over two machines with 13 tasks per machine. The single machine job finishes in about 14 mins, while the job on two machine finishes in about 8.5 mins. This indicates that spreading tasks across machines helps with scalability by minimize file read/write conflicts.

Job Restart

There is no provision for stopping and restarting a job. A new job does not reuse solved results; it always solves all rows in the table. An abort or failure of a job restarts from the beginning, unless a new parametric table with the unsolved rows is created.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Linux-Only Issues

- Deployment/Installation errors (such as mainsoft-related) are not captured. If there is such an issue, the Large Scale DSO job will fail without useful messages in the logs.
- Report-based extraction fails if traces and parametric-setup are not prepared as per the Getting Started guides.
- Job status: The exit code of job doesn't indicate success or failure correctly. The error messages from multiple log files needs to be combined to determine the reason for failure. In many situations, the reason for a failure is apparent only after re-running the job after turning ON the 'debug logging'.
- In some LINUX scenarios, the analysis appears to finish successfully with valid results, except that the exit code is '134'. In this case, although the exit is abnormal, the failed exit code can be ignored.
- Load Balancing: For models with 'unbalanced variations table' (variations that take considerably different amount of time to solve are clustered in few regions of table), job will take longer time to solve than a Regular DSO as the job's overall completion time is determined by the slowest solving region. Workaround: rearrange the rows in the parametric table so that each region takes a similar time to solve.
- GM Specifics: the model used for 'Report-based extractor' jobs is NOT compatible with the 'Ansys-extractor-for-GM' jobs. A valid model for Ansys-extractor-for-GM cannot contain any of: reports, overlay plots, Optimetrics calculations.

Interactive Scheduler Jobs

This document includes information, guidelines, and caveats for users running interactive scheduler jobs on Linux.

In most cases, jobs run under a scheduler run as a batch job. These jobs may be submitted using the Ansys Electromagnetics Desktop job submission GUI, using cluster job submission commands on a command line or using a cluster GUI, if available.

Some customers use an alternative method for submission of scheduler jobs. For convenience, we call such jobs "interactive scheduler jobs." In this approach, the user submits an interactive job to the scheduler. From the interactive job prompt, the user launches an Ansys Electromagnetics Desktop product, which starts in interactive (GUI) mode, not batch mode. The user selects a project and then runs one or more analysis commands using the GUI. The intent is that these analysis commands should use all resources allocated to the job, whether on the same host as the GUI or on other hosts.

This approach is supported on Linux, where the user may set up an X Window System server for interacting with the Ansys Electromagnetics Desktop product GUI. The user needs to configure

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the cluster environment and/or the interactive environment so that the user may view and interact with the product GUI. This approach is not supported on Microsoft Windows.

Specifying Options for Interactive Scheduler Jobs

DSO Configuration for Interactive Scheduler Jobs

Design Type Options for Interactive Scheduler Jobs

Specifying Options for Interactive Scheduler Jobs

Batch scheduler jobs use command line options for specifying options rather than GUI controls. For interactive scheduler jobs, options may be specified on the command line, using GUI controls, or obtained from the registry. The machines specified in the **Configurations** tab of the **HPC and Analysis Options** window will be ignored for interactive scheduler jobs. If the command line used to launch the product contains a list of specific machines (using option -machinelist list=...), then the job will use the specified machines and cores. If the command line used to launch the product does not contain a specific machine list, then the machines and cores allocated to the job by the scheduler will be used for the job. In general, the job distribution settings for interactive scheduler jobs should be specified on the command line, instead of using the **HPC and Analysis Options** window.

- The UI shows the settings that will be used for analysis, even if they come from the command line, not the registry.
- If you make changes to the settings in the UI, and the changed settings will be used for the analysis, even if the changed settings conflict with the command line (including batchoptions).
- If you makes no changes to the settings in the UI, then the command line settings (including batchoptions) will be used for analysis.

Batchoptions settings that are automatically generated for batch jobs submitted using the AnsysEM job submission GUI will need to be manually included in the product command line. Some use cases require that certain settings be made on the command line when the product is launched, rather than using the GUI.

Batchoptions for Interactive Scheduler Jobs

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

For interactive scheduler jobs, only a limited set of batchoptions are supported. These batchoptions include the DSO configuration options, the design-type-specific options, and the following additional design-type-specific options that are not currently listed in the user interface or command line help windows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **CreateStartingMesh:** Create the starting mesh only. Do not solve any other steps. Often used for multi-step jobs where the mesh creation is solved in a separate step using fewer resources than the rest of the analysis.
- **NumCoresPerDistributedTask:** Specifies the number of cores that are allocated to the job to use for each distributed task, used when running jobs under a scheduler. The scheduler communicates the number of cores allocated to the job on each host. This setting is used to determine how to allocate the cores to each task.
- **RAMLimitPercent:** Specifies the percentage of system RAM that the analysis is expected to use on each host. Running out of memory can cause processes to be killed by the Operating System or other failures. The analysis will be gracefully terminated if it requires more than this limit. Typical limits are about 90%, to allow some RAM for the Operating System and other services. If there are multiple jobs running on a single host, then the RAM limit should be reduced accordingly. For example, if there are two jobs running on a host, then the limit could be 45%, which would allow 10% of the RAM for the Operating System and other services.
- **RAMLimitPerCoreInGB:** This is an alternative way to specify the RAM limit, instead of using the RAMLimitPercent. In this case, the RAM limit is the limit per core multiplied by the number of cores allocated to the job on each host. Typically used for jobs running under a scheduler to ensure that the scheduler memory limit and the ansysedt memory limit are consistent.
- **SolveAdaptiveOnly:** Solve adaptive passes only. Do not solve any other steps. Often used for multi-step jobs where the adaptive passes are solved in a separate step using fewer resources than the rest of the analysis.
- **TotalNumOfCores:** The total number of cores for the job. This option is only used for EKM (Engineering Knowledge Manager).
- **ValidateOnly:** Only validate the specified setup, design, or project. Do not analyze the specified setup, design or project.

Any other batchoptions will result in a warning message and will be ignored.

Command Line Example:

```
desktopjob.exe" -cmd dso -jobid RSM_27086 -machinelist
list=localHost:2:2:90%
-monitor -ng -batchoptions "
'LargeScaleDSO/MergeCsv'='acrossDPsAndTraces'"
-batchsolve TeeModel:Optimetrics:ParametricSetup2 E:\work\2018\LS_
DSO\OptimTee.aedt
desktopjob.exe" -cmd dso -jobid RSM_25248 -machinelist
list=localHost:2:2:90%
-monitor -ng -batchoptions "
'LargeScaleDSO/MergeCsv'='acrossDPsByRow'"
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
-batchsolve TeeModel:Optimetrics:ParametricSetup2 E:\work\2018\LS_
DSO\OptimTee.aedt
ansysedt -batchoptions " 'TempDirectory'='C:\\TEMP'
'HFSS/SelectedDSOConfiguration'='Local'
'Desktop/Settings/ProjectOptions/DoAutoSave'=1
'LargeScaleDSO/MaxFolderInMB'=100 "
```

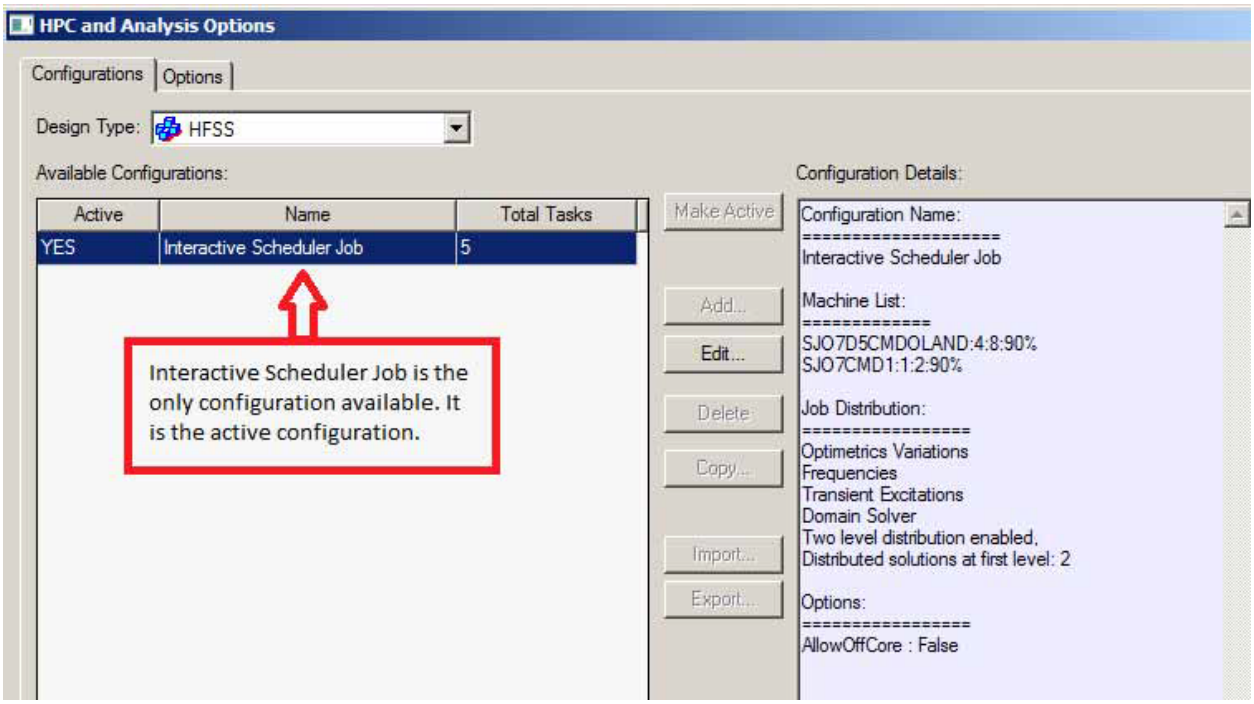
DSO Configuration for Interactive Scheduler Jobs

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

When running an interactive scheduler job, there is only one DSO configuration available for each design type. Each configuration is named "Interactive Scheduler Job". This configuration is always the active configuration for an interactive scheduler job. and it is the only configuration displayed in the list of available configurations shown in the "HPC and Analysis Options" dialog box. No configurations can be added or removed, but the "Interactive Scheduler Job" configuration may be modified using the "Edit" button, which pops up the "Analysis Configuration" dialog.

Because the "Interactive Scheduler Job" configuration is the only configuration accessible for interactive scheduler jobs and it is not accessible in other modes, there is no sharing of the "Interactive Scheduler Job" configuration settings with other modes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



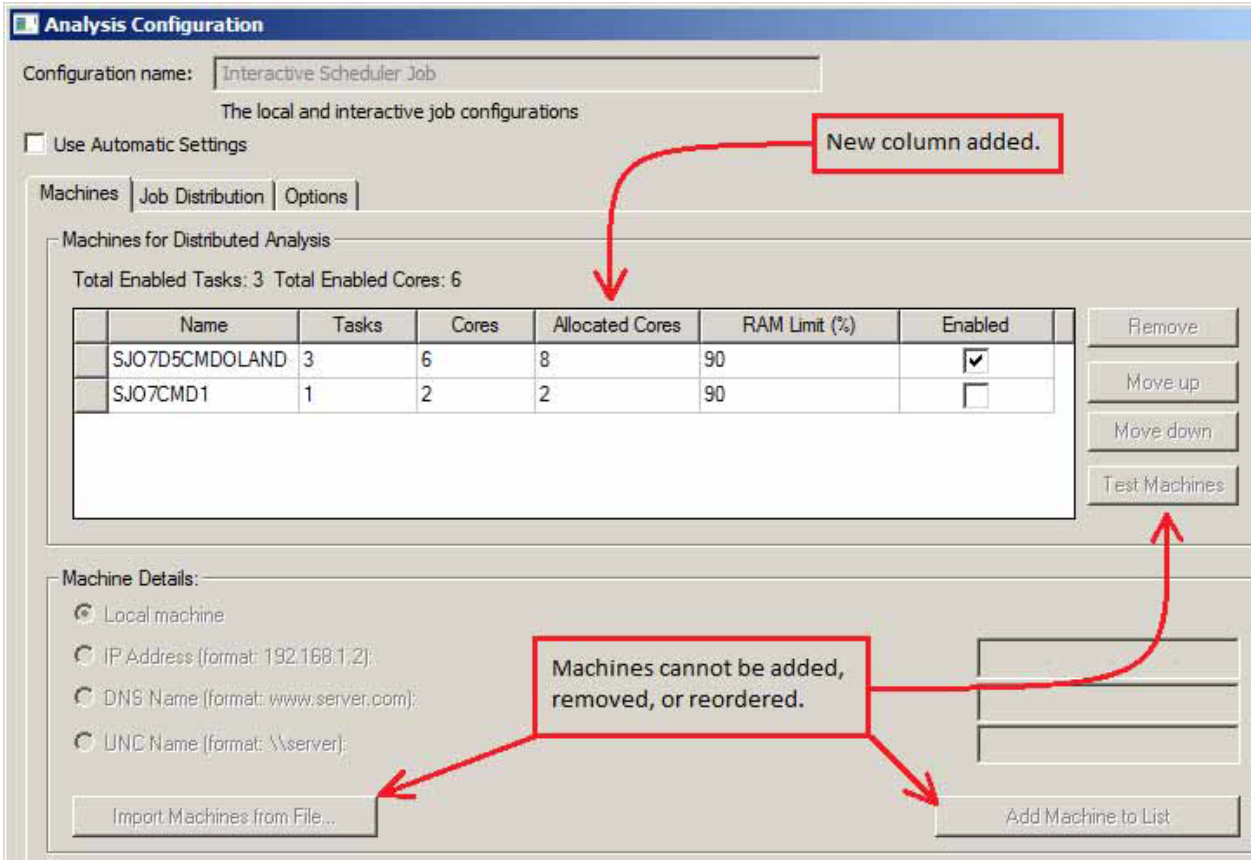
One major difference between interactive scheduler mode and batch mode or normal interactive mode is:

- In interactive scheduler mode, DSO configuration settings and design type options in the UI override command line options and batchoptions.
- When not in interactive scheduler mode, DSO configuration settings and design type options specified on the command line (including batchoptions) override UI settings.

Analysis Configuration - Machines Tab

The most obvious changes for interactive scheduler jobs are visible in the **Machines** tab of the **Analysis Configuration** window. The grid of machine information is prepopulated with the list of machines allocated to the job. It also contains an additional column in interactive scheduler mode. This column, "Allocated Cores", indicates the number of cores allocated to the job by the scheduler on each host. You cannot add or remove machines from the list, or modify the allocated cores for any machines. You can modify the tasks, cores, and RAM limit for any machine, or specify that a machine is enabled or disabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Analysis Configuration - Job Distribution Tab

The **Job Distribution** tab of the **Analysis Configuration** window only appears if the **Use Automatic Settings** check box is not checked. The state of this check box, and the settings shown on this tab are initialized from the command line or from the registry. The command line options that affect the **Use Automatic Settings** check box or the job distribution settings are:

- -distributed
- -local
- -auto
- -machinelist

The registry contains the last value of these settings for an interactive scheduler job for the same user on the same host. Settings on the command line override settings from the registry. Any changes in the GUI will override the initial settings, even if the initial settings are from the command line. Any changes in the GUI also update the registry settings for the "Interactive Scheduler Job" configuration for the current design type.

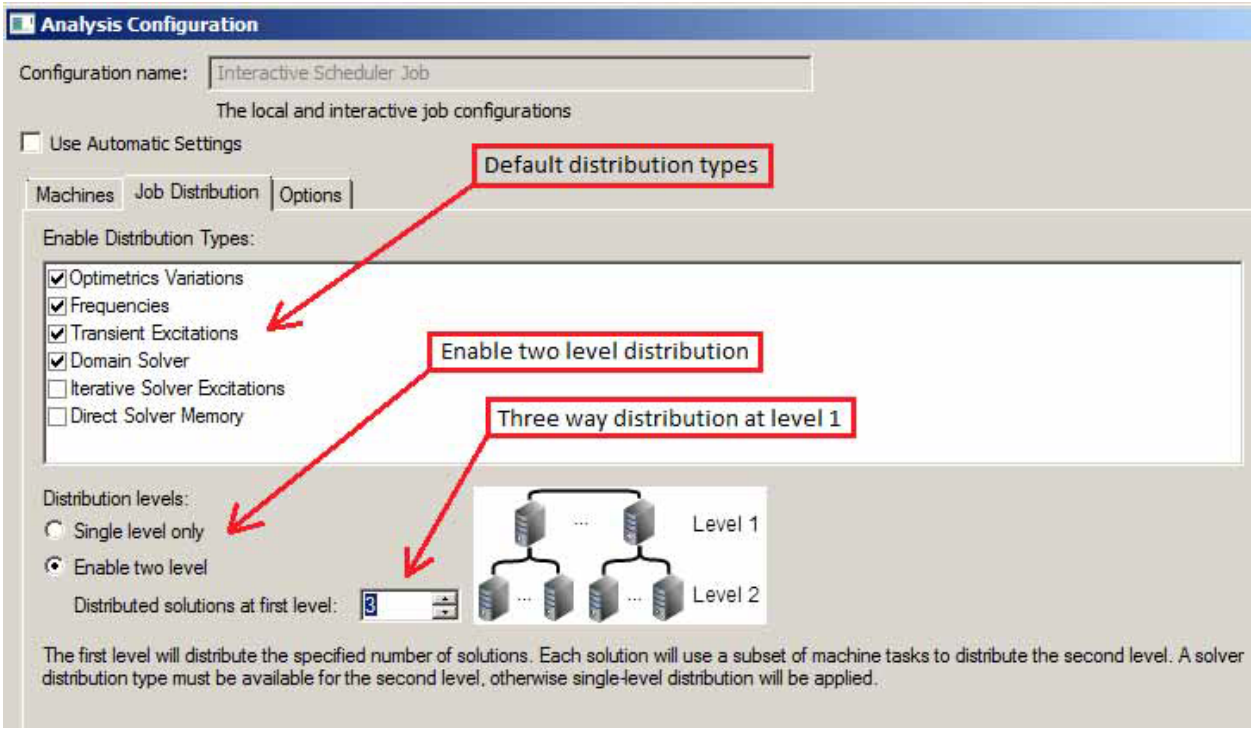
Example

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command line:

```
ansysedt -distributed includetypes=default maxlevels=2 numlevel1=3
```

Initial "Job Distribution" settings:



Analysis Configuration - Options Tab

You can use the **Options** tab of the **Analysis Configuration** window to examine or modify the DSO configuration options. The DSO configuration options are handled like the design type options, except that the Interactive Scheduler Job configuration settings are not shared with other modes.

The settings on this tab are initialized from the command line (using the -batchoptions command line option) or from the registry. The registry contains the last value of these settings for an interactive scheduler job for the same user on the same host. Settings on the command line override settings from the registry. Any changes in the GUI will override the initial settings, even if the initial settings are from the command line. Any changes in the GUI also update the registry settings for the "Interactive Scheduler Job" configuration for the current design type.

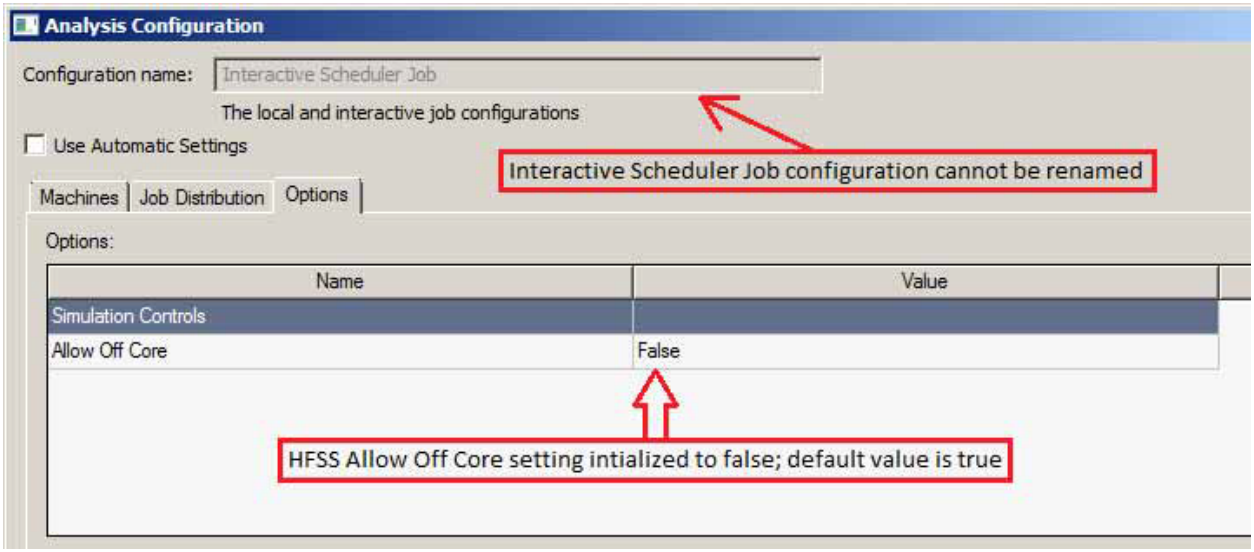
Example

Command line:

```
ansysedt -batchoptions 'HFSS/AllowOffCore'=0
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initial Analysis Configuration Options:



Design Type Options for Interactive Scheduler Jobs

> **Note:**
>
> Functionality featured in the example(s) in this section applies to multiple design types.

You can use the **Options** tab of the **HPC and Analysis Options** window to examine or modify the design type options. The settings on this tab are initialized from the command line (using the -batchoptions command line option) or from the registry. The registry contains the last value of these settings for the same user on the same host. Settings on the command line override settings from the registry. Any changes in the GUI will override the initial settings, even if the initial settings are from the command line. Any changes in the GUI also update the registry settings for the current design type.

These settings are shared between interactive scheduler mode and other modes.

Example

The following example includes two command line options, one that is applicable to HFSS ('EnableGPU') and one that is applicable to all design types ('HPCLicenseType'). These opitons would be included as part of the complete job submission command line syntax.

Command Line Excerpt:

```
-batchoptions 'HFSS/EnableGPU'=1 'HPCLicenseType'='Pack'
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

High Performance Computing (HPC) Integration

Ansys Electromagnetics products offer a direct integration with a number of High Performance Computing (HPC) software programs.

The list of currently supported schedulers includes:

- Ansys Cloud Direct
- Ansys Cloud Burst Compute
- Microsoft Windows HPC Server (Windows Only)
- Grid Engine (GE) (Linux Only)
- Platform's Load Sharing Facility (LSF) (Linux Only)
- PBS Pro or PBS Torque (Linux Only)

Electronics Desktop also supports custom integration.

A job scheduler may also be described as a batch system, a Distributed Resource Management System (DRMS) or Distributed Resource Manager (DRM). The features supported on each scheduler are included in the documents for each. For each job scheduler, the versions or revisions that have been tested are included.

A user may submit jobs using the command line tools or other tools provided by the scheduler. The Desktop includes a GUI to help the user submit jobs to a job scheduler. This generic Job Submission GUI is shared across the Ansys EM products. The general procedure is to specify the scheduler and head node, describe and submit the job, and monitor the results. Using “Ansys Cloud Direct” or “Ansys Cloud Burst Compute will require a login through a browser instead.

Scheduler Terminology

This help uses some specific terminology when discussing HPC and schedulers. See definitions below.

Term	Definition
Core	Unit of processing.
Compute Cluster	Network of machines on which jobs run. Typically consists of head nodes and many compute nodes.
Distributed Processing	Multiple engines are launched simultaneously on the same machine *or* on different machines.
Engine	Electronics Desktop application (aka. executable) launched during analysis commands to generate analysis results.
Job	Application (aka. program, executable) with command line options that uses resources to produce results. For example, `hfss.exe -ng -BatchSolve`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Term	Definition
Machine/Host/Node	Consists of one or more processors, memory, disk, etc.
Multiprocessing	A single engine uses multiple cores on the same machine.
Parallel Job	Job that runs on multiple cores belonging to the same or different machines.
Processor	Consists of one or more cores.
Resource	Machines, licenses, etc. that a job uses.
Serial Job	Job that runs on a single core.
Service	Program that runs in the background (for example, RSM Service) and listens on a port. The OS provides the programming interface through which Applications communicate with services, once the machine and port number are known. Launching an executable on a remote machine requires a service to run on the remote machine.

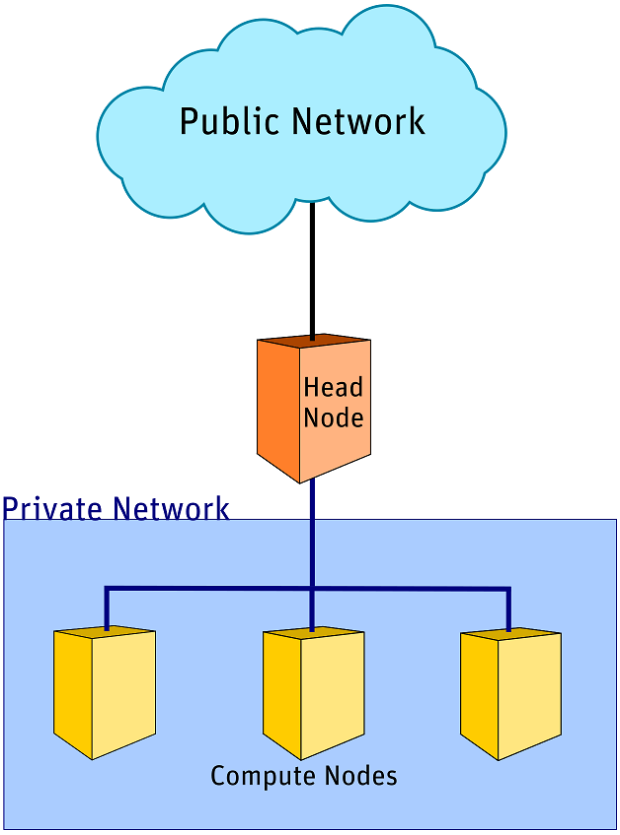
What a Scheduler Does

Schedules are responsible for the following:

- Enabling effective/efficient utilization of cluster's resources consistent with organization's goals
- Maintaining queue(s) of jobs
- Maximizing throughput of jobs by processing all jobs as quickly as possible (typically, one job per CPU)
- Allowing a choice of various scheduling policies (for example, First Come First Serve, Priority-Based, and Preemption)
- Providing a suite of tools or utilities (graphical or command line) for end users to submit, monitor, suspend, and abort jobs
- Managing a compute cluster by running various interacting services on head nodes and compute nodes
- Providing a programming interface to access services

Scheduler-Managed Compute Cluster

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Head node(s) typically maintain queues. Compute nodes are typically on a high speed network to improve scalability of parallel jobs. Services running on nodes interact with each other to manage resources. End user tools communicate with services to manage jobs.

Configuring Electronics Installation for HPC

In HPC applications, Ansys Electromagnetics Suite must be available on each cluster host where jobs may be run.

- On the Linux platform, Ansys EM may be installed on a shared drive that is accessible to all machines in the cluster.
- On the Windows platform, Ansys EM must be installed separately on each host of the cluster.

Ansys EM must be accessible using the same path on each host. All cluster users running Ansys EM jobs must have permission to read and execute the files in the installation directory and its subdirectories.

The temp directory selected during installation must be readable and writable by all user accounts used to run the Ansys Electromagnetics Suite. This temp directory path should be the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

same on all machines of the cluster and should be local to every machine. For example, C:\temp on Windows, /tmp on Linux.

Because HPC is offered as a direct integration, you need only install the Ansys EM software. No additional configuration is required.

Firewall Configuration

If firewall is turned *off* between the machines of the cluster, there is no need for any configuration.

If firewall is turned *on*:

- For a Windows cluster: Configure firewall by adding exceptions that allow Ansys EM programs and services to communicate with each other. If you are using a standard Windows Firewall, this is automatically done for you by the installation program. If you are using a third-party firewall software, it needs to be configured in a similar manner.
- For a Linux cluster: Open up the firewall for range of ports denoting ephemeral (or dynamic) ports. Check with your system administrator on how this can be done on each machine of cluster.

Installation Directory Examples

Microsoft Windows

Install the Ansys Electromagnetics Suite in directory "C:\Program Files\ANSYS Inc\v251\AnsysEM" on each node of the cluster. The same directory pathname must be used on all hosts.

Linux

Install the Ansys Electromagnetics Suite in a common directory that is accessible using the path /opt/ansys_inc/v251/AnsysEM on each execution node of the cluster.

Integration with Microsoft Windows® HPC Scheduler

The Windows HPC scheduler is only supported on Windows.

Jobs may be submitted in any of the following ways:

- Using Windows HPC GUIs from Microsoft: Job Manager or Cluster Manager
- Using Windows HPC command line tools
- Using Ansys Electronics Desktop UI commands for scheduler selection, job submission and job monitoring/control.

General Guidelines for Submitting Ansys EM Jobs

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A Job submitted to Windows HPC Cluster is defined by Job properties, Task List and Task properties. Priority, resource requirements, node preferences, etc. come from Job properties. In the case of Ansys Electromagnetics jobs, Task List consists of a single task. Properties of this task specify the command line that runs Ansys Electromagnetics desktop in non-graphical mode to perform analysis of a project.

Specifying the Number of Compute Resource Units for HPC Jobs

You can either select **Use automatic settings** on the *Compute Resources* tab, or you either enter the number of tasks and total cores per machine, or individual nodes. HFSS, HFSS 3D Layout, and Icepak have **Use automatic settings** selected by default. This release permits options in setting whether nodes are exclusive for the submission. See Windows HPC Non-exclusive Jobs.

Ansys EM Project File and Project Directory for use with Windows HPC Scheduler

Ansys Electromagnetics Suite tools write their results to a subdirectory of the directory containing the Ansys EM project file. The Project Directory (the directory containing the project file) must be accessible to all of the cluster hosts that may run Ansys EM jobs. The user account for the job must have permission to read the project directory, and to create and modify files and subdirectories of this directory. The pathname of the project file must be accessible to all cluster hosts using the same path name, which is generally expressed as a UNC pathname.

Example:

The project file is on the user's workstation (with hostname user1_PC) in directory C:\user1\projects\new\project1.aedt, and the directory C:\user1\projects is shared with sharename projects.

Correct

When submitting the job, you should use the following pathname to specify the project file:

```
\\user1_PC\projects\new\project1.aedt
```

Incorrect

If a local pathname is used, the cluster hosts will not be able to find the user's project on the workstation

```
user1_PC: ' C:\user1\projects\new\project1.aedt '
```

Submitting and Monitoring Ansys EM HPC Jobs

Jobs may be submitted to the Windows HPC Scheduler using any of the following methods:

- Using the *Submit HPC Job* dialog box
- Using the Windows HPC Job Manager GUI

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Using the Windows HPC Command Line Tools
- Using the Windows PowerShell

Client Utilities from the Microsoft HPC Pack, must be installed on the submit host to use any of these methods to submit a job to a cluster. The *Submit HPC Job* dialog box will be unable to contact the cluster head node if the client utilities are not installed.

This document covers the first method. See the Microsoft documentation for information on the other three methods.

- Submitting and Monitoring Jobs for Windows HPC
- Specifying the Number of Compute Resource Units for HPC Jobs

Jobs may be submitted from any Microsoft Windows host meeting the following requirements:

- For submitting jobs to the Windows HPC scheduler, the Desktop process must run on a node that is configured for submission of jobs to the Windows HPC cluster. That is, the Windows HPC Client Utilities must be installed on the node, and network communication from the Desktop node to the head node of the cluster must be allowed. For Ansys Electromagnetics Suite 2025 R1, Windows HPC Server 2008 R2 (or later) client utilities are required. Using a computer on the network is not supported for submission of jobs to the Windows HPC cluster.
- When submitting jobs to a Windows HPC cluster, the user must also specify the head node of the cluster to which the jobs will be submitted. When the user selects the "Windows HPC" scheduler in the "Choose scheduler" list, the Head Node edit control is enabled. The user may enter the Windows HPC cluster head node name into the edit box. Alternatively, the head node may be selected using a "Browse for Computer" browser by pressing the ellipsis [...] button.
- The Windows HPC Pack client utilities are installed on the submission host.
- Network communication between the submission host and the Windows HPC Cluster head node is permitted; there is a network connection between these hosts that is not blocked by any firewall or the like.
- The submission user is permitted to submit jobs to the Windows HPC Cluster.

Job Monitoring

1. Windows HPC Jobs may be monitored using the *Monitor Job* dialog box, which is brought up by the **Tools > Job Management > Monitor Jobs...** command. This dialog bos may also be brought up by checking the **Begin monitoring this job now** check box when a job is successfully submitting using the job submission dialog box. You can also select the **Simulation** tab of the ribbon and select the **Monitor** icon. You can monitor this job either automatically (by checking the option) or through the **Tools> Job Management> Monitor Jobs...** command. For more details, see Monitor Jobs window.

In addition to the above requirements to allow job monitoring the following is also necessary:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Network communication between the submission host and all Windows HPC Cluster nodes where the job may run is permitted; there is a network connection between these hosts that is not blocked by any firewall or the like.

Cluster Configuration

Any job running on a Windows HPC Cluster that is distributed over multiple compute hosts requires network communication between processes running on these hosts. The cluster must be configured to allow this communication. Any firewall or other security software must be disabled or configured to allow communication between any of the compute hosts were a job could run.

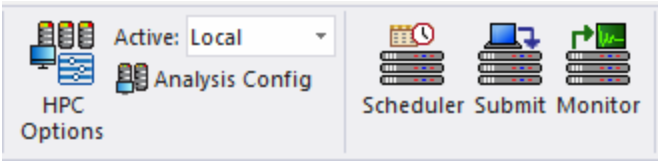
Job Submission User Profile on Cluster Compute Nodes

In order for a job to run correctly, the submission user's profile must be accessible and properly initialized on the cluster compute nodes where the job runs. If the Ansoft/temp subdirectory of the user's "My Documents" directory does not exist or is not accessible on the compute cluster nodes where a job runs, the batchoptions for the job will not be processed correctly, resulting in job failure. One way to ensure that this directory is created on each compute host is for the submission user to login to each compute host and run the product GUI one time.

Submitting and Monitoring Jobs for Windows HPC

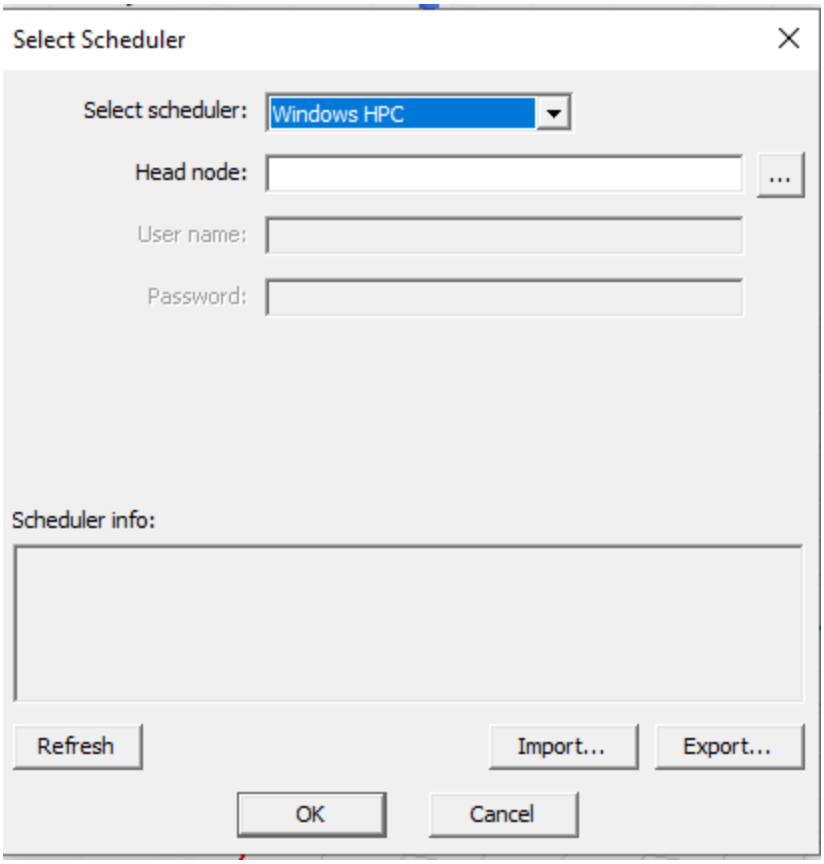
In order to submit jobs using **Windows HPC**:

- Click **Tools** > **Job Management** > **Select Scheduler**, or
- Select the **Simulation** tab of the ribbon, and click the **Scheduler** icon.



This opens the **Select Scheduler** dialog box. Specify **Windows HPC** as the scheduler.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For Windows HPC, specify the head node of the cluster.

After specifying the cluster head node, click **Refresh**. This verifies that the head node may be contacted, and displays the scheduler name, a brief description (including the head node name), and the version of the Windows HPC head node.

Pressing **Cancel** discards changes made in this dialog box. Pressing **OK** verifies that the head node can be contacted before accepting the changes. If no problem occurs, the dialog box will close. If there is a problem contacting the head node, the dialog box will not close and the changes will not be accepted.

After setting the job submission node, perform *one* of the following:

- Select **Tools** > **Job Management** > **Submit Job...**
- Click **Project** > **Submit Job...**
- Click **HFSS** > **Submit Job...**
- Select the **Simulation** tab and click the **Submit** icon.
- Use a shortcut menu to select **Submit Job**.

This opens the **Submit Job To** dialog box.

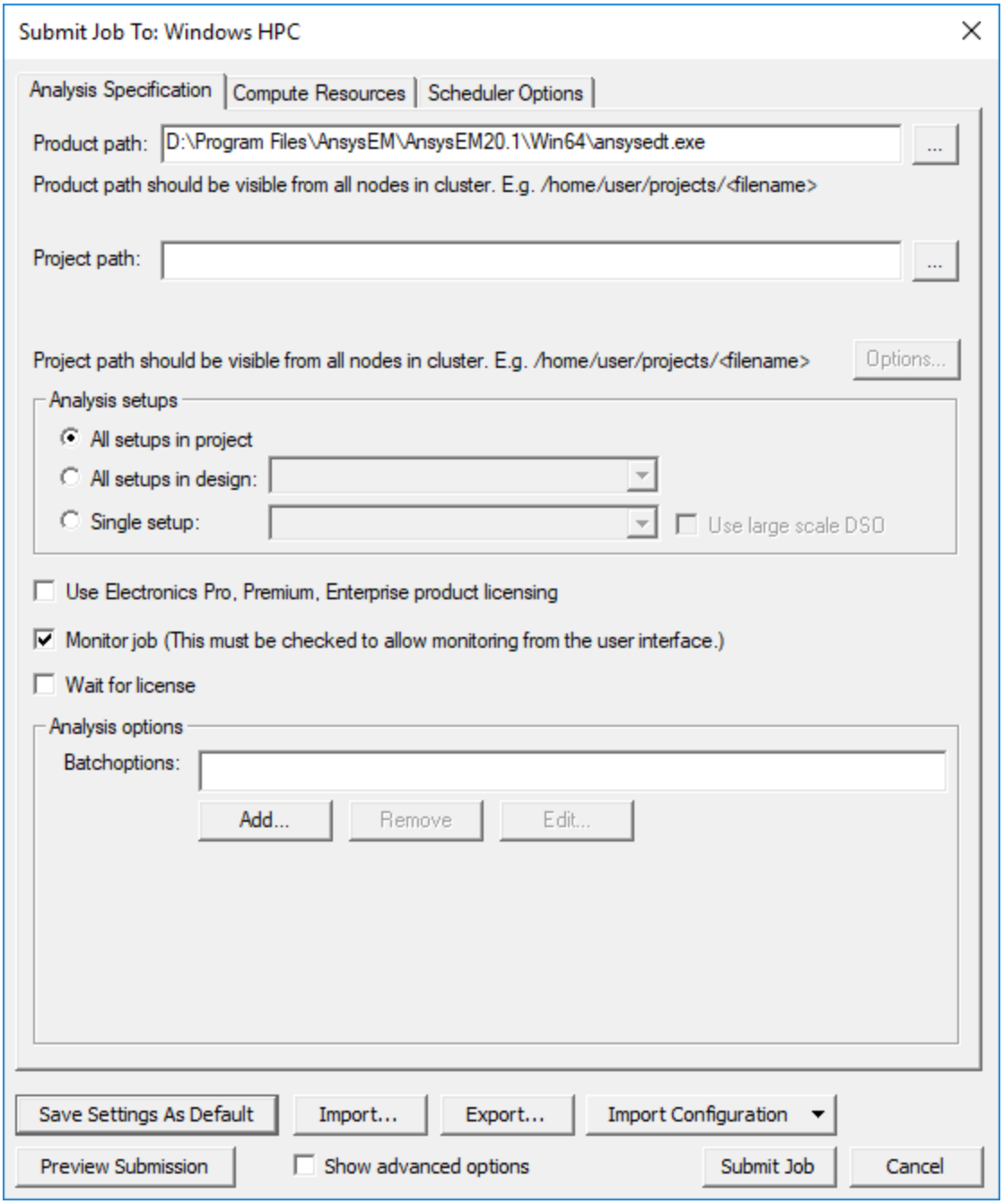
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Submit Job To** dialog box contains three tabs:

- **Analysis Specification**– specify the Product path, Project name, the setups, and analysis options such as batchoptions, or, for advanced users, Environment variables. If you select the Analysis or Optimetrics setup, the Analysis Specification is pre-populated.
- **Compute Resources**– this tab can be populated either by automatic settings, by predefined Analysis Configuration, or specifying parameters in the fields for resource selection, for job parallelization and enabled forms of parallelization.
- **Scheduler Options**– contains fields for Job name and priority. The customization options shown by checking advanced are not used for Windows HPC.

On the **Analysis Specification** tab, enter path names for the Product and Project. These must be UNC paths that are accessible from each compute host used for Ansys Electromagnetics jobs. The Project can be an archive. The submission user must have permission to write to the directory containing the project file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can select which setups are analyzed in the **Analysis Setups** area of the dialog box. There are radio buttons to select:

- All setups in the project
- All setups in a specified design (you select the design from the drop-down menu)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

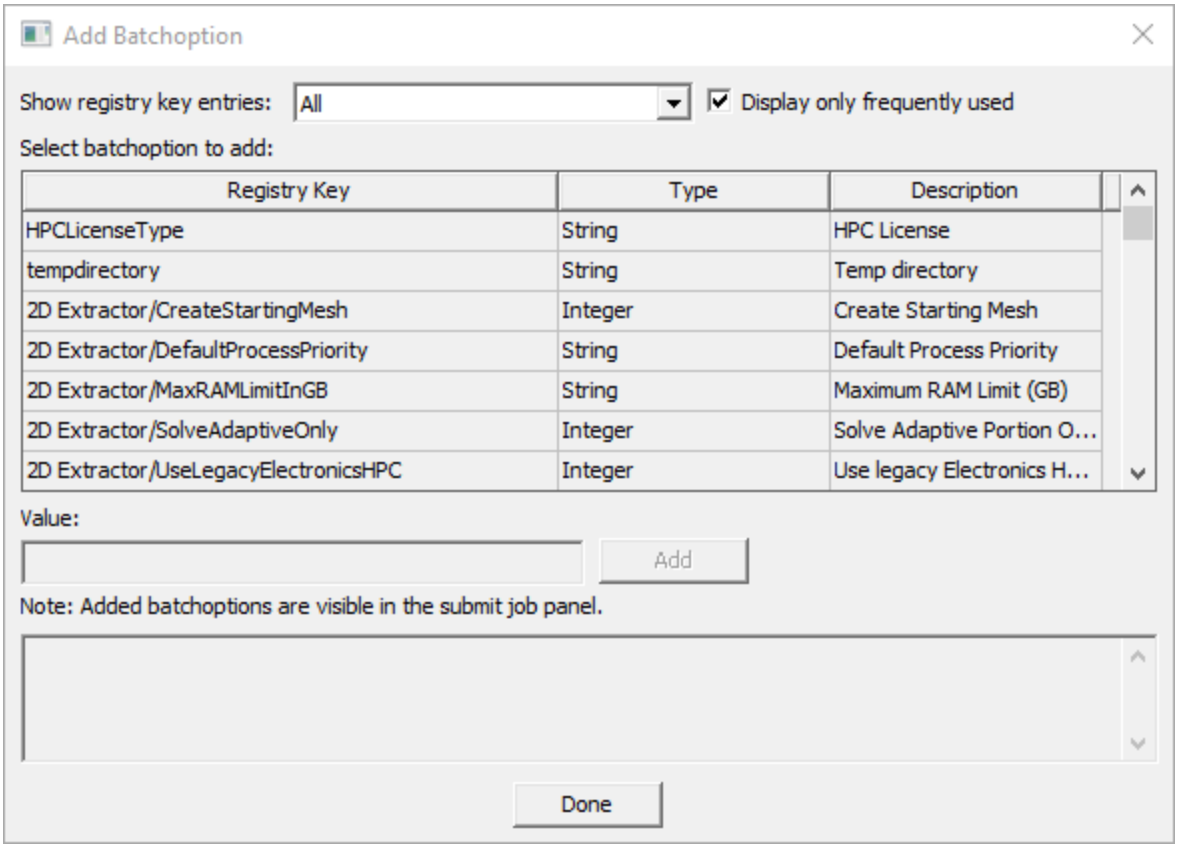
- Single setup (if you selected the Submit Job command from the shortcut menu, the setup name populates the field)

If you specify multiple setups, they will be processed sequentially in the order displayed in the edit box.

The Analysis options include:

- Monitor job – you must enable this option to monitor the job from the user interface.
- Wait for license – specify whether to wait until a license if available before starting a simulation.
- Batch options – optionally specify -batchoptions in the text field. See detailed discussion of -batchoptions beginning under Running Ansys Electronics Desktop from a Command Line.

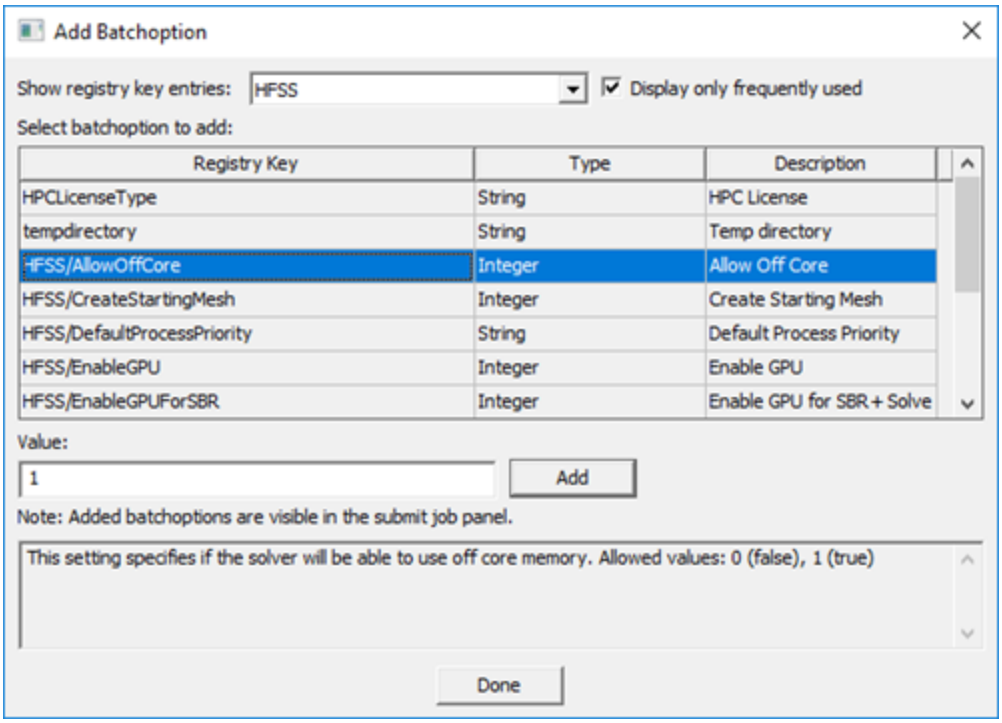
The **Add...** button opens the **Add Batchoption** dialog box.



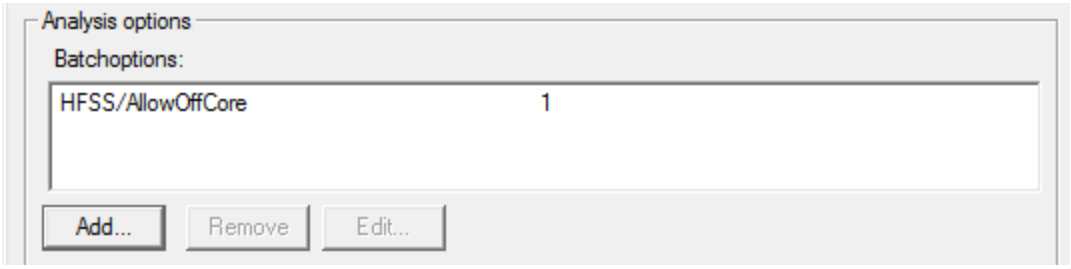
This dialog box provides access to all -batchoption commands. The drop-down menu lets you select specific categories, and you can choose to display only frequently used commands. You can edit and remove any batch options you specify.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select a Registry Key in order to show the current Value for the type. The lower field explains the meaning of the Type Value.

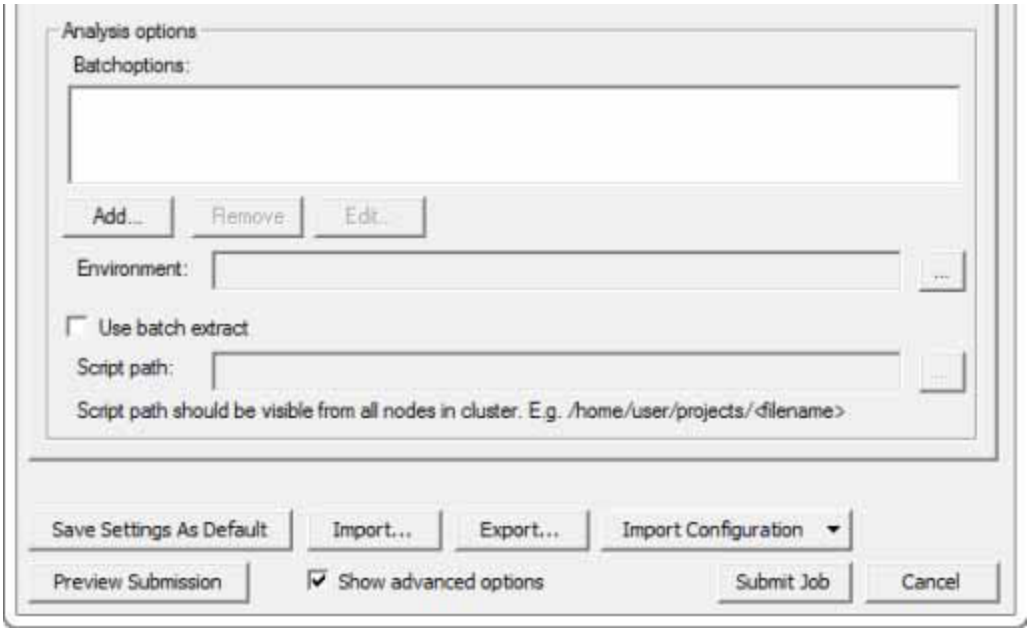


Any batchoptions for which you select **Add** will be visible in the **Submit Job** dialog box.

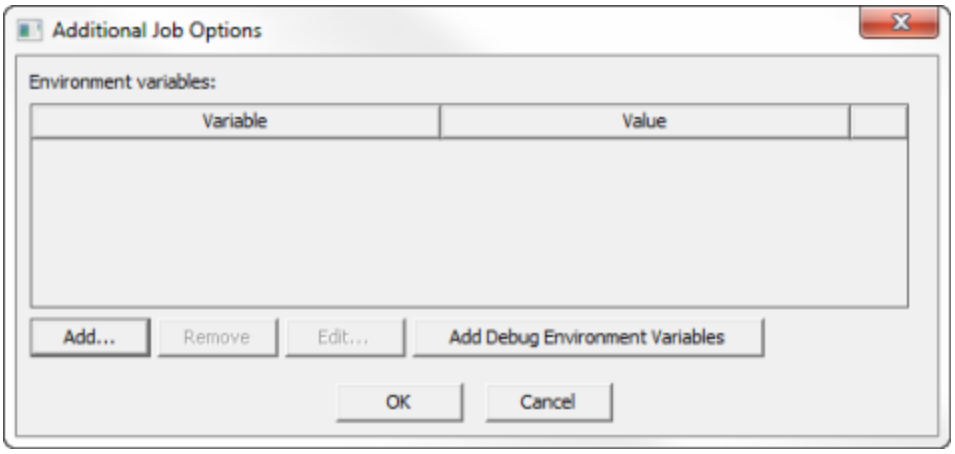


If you have the **Show advanced options** box checked in the **Submit Job** dialog box, the Environment field and the Use batch extract fields display.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

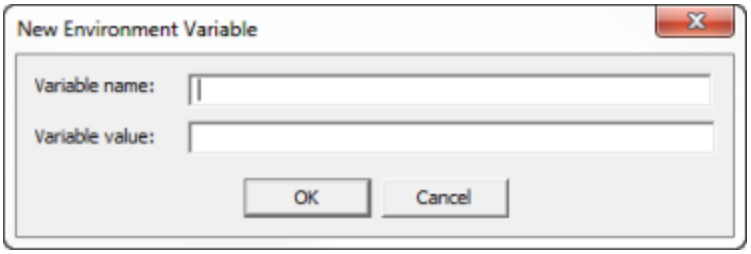


The **Environment** field lets you specify any environment variables. Click the ellipsis button [...] to display the **Additional Job Options** dialog box.



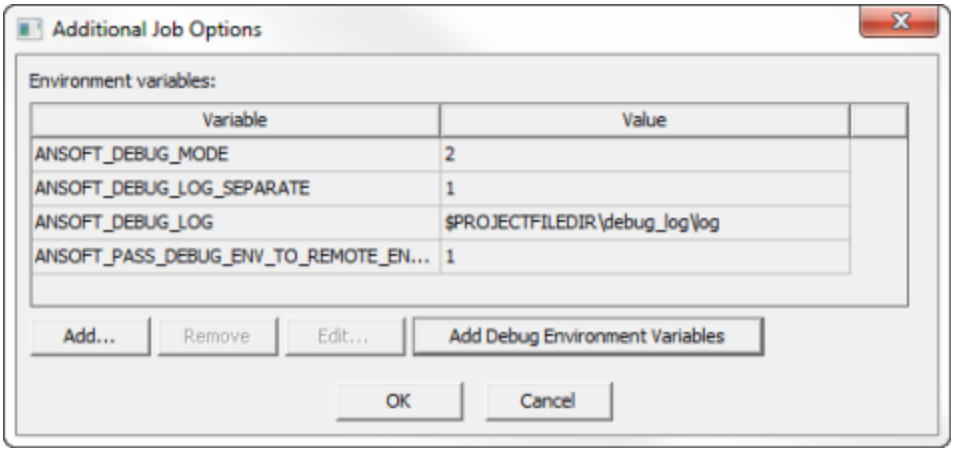
Click the **Add...** button to open the **New Environment Variable** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Here you can provide a Variable name and Variable value. Click **OK** to display the variable in the **Additional Job Options** dialog box.

Select a variable to enable the **Remove** and **Edit...** buttons. You can also click **Add Debug Environment Variables**.



Note:

For certain methods of resource selection for Windows HPC job submission, Ansys Electronics Desktop checks for nodes being online. In an auto-scaling cluster, nodes can fail the online check because they're not online until a job is running. An environment variable allows customers to bypass this check:

ANSYSEM_SKIP_NODE_ONLINE_CHECK

Set the variable value to 1 to disable the online check. Debug logging fully verifies what aspects of the cluster are causing this check to fail.

Job submission in Windows works without changing settings if environment variable ANSYSEM_SUBMIT_JOB_REQ_NODE_ONLINE is set with value 0. The installation doesn’t set this environment variable. ANSYSEM_SUBMIT_JOB_REQ_NODE_ONLINE accepted

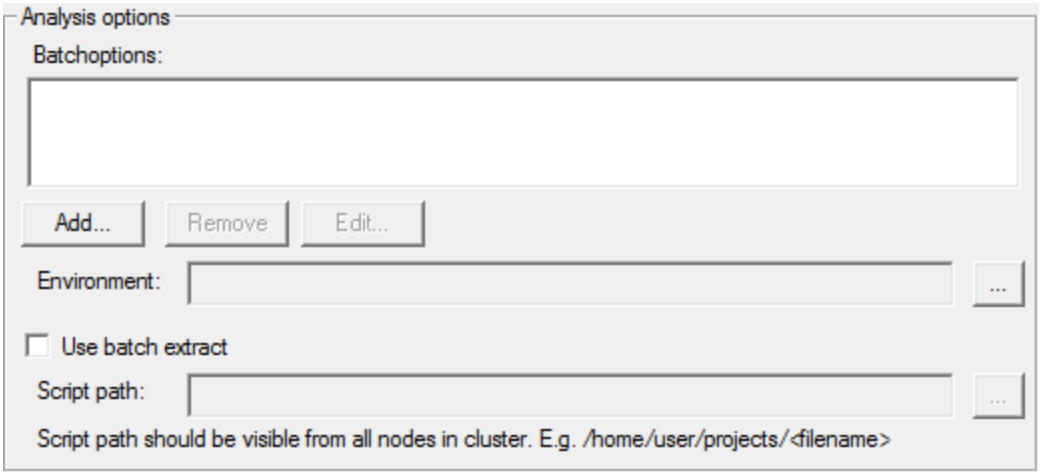
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

values: 0 or 1 (0 = disable; 1 = enable, i.e. require nodes to be online for submission). Default: 1 (if not set, Ansys Electronics Desktop requires nodes to be online for submission).

If you have enabled **Show Advanced Option**s, any variables that you add will be displayed in the **Environment** field of the **Submit Job** dialog box.

Use Batch Extract for Windows HPC

Enabling **Show Advanced Options** for Windows HPC also displays the **Use batch extract** check box and script path field.

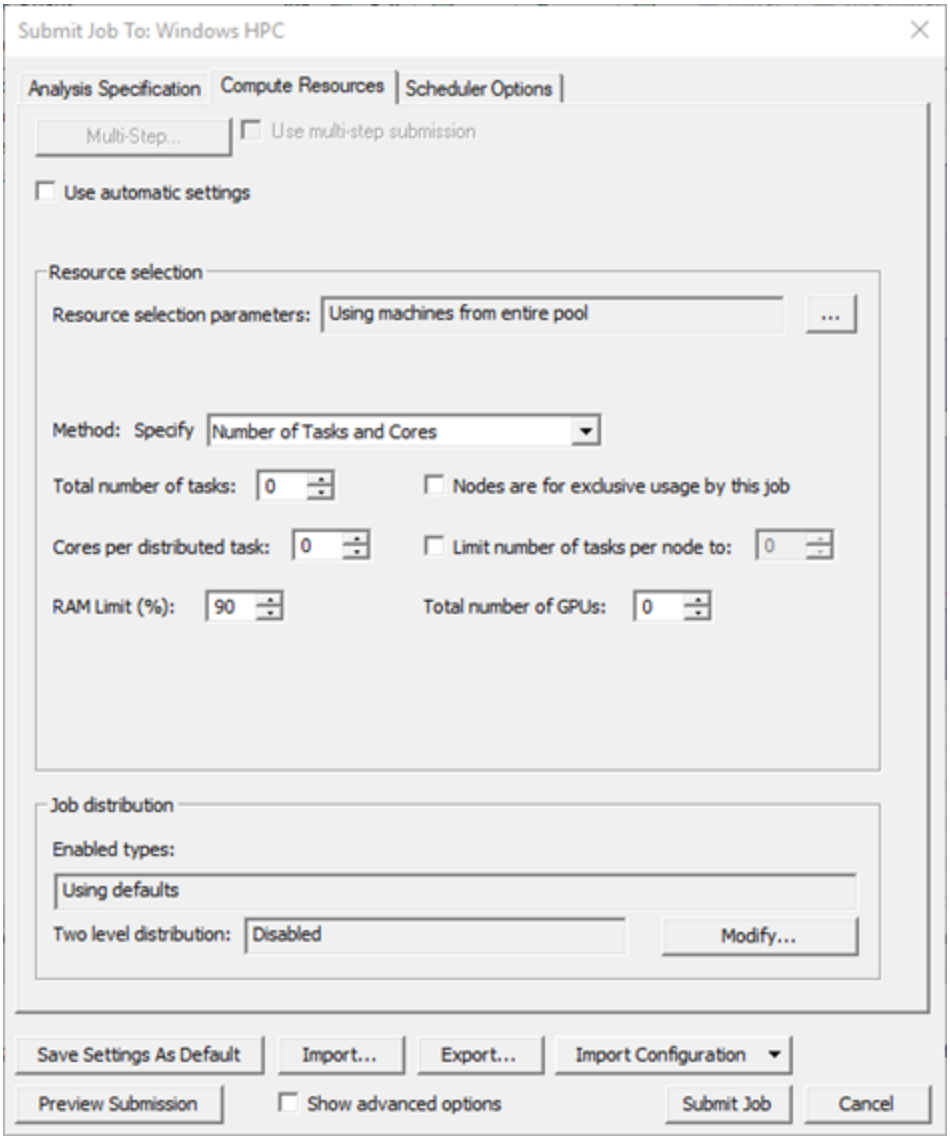


See Running Ansys Electronics Desktop from a Command line for an explanation of the solve information available through batch extract.

The **Preview Submission** button opens a window that shows the text commands that will be sent to the scheduler.

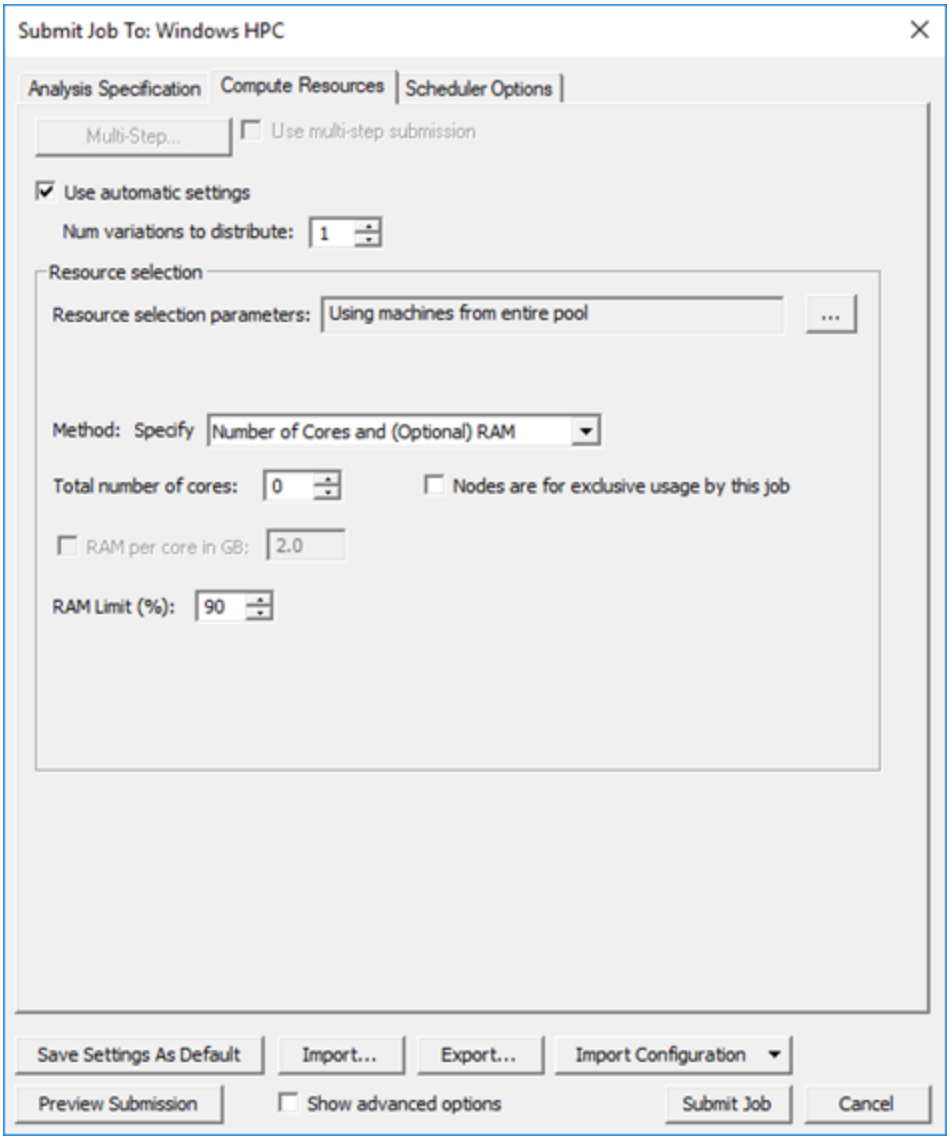
The following figure shows the **Compute Resources** tab of the **Submit Job To** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For Ansys Electronics Desktop configurations, the **Submit Job To** dialog box includes a **Use automatic settings** check box that simplifies the **Compute Resources** tab.

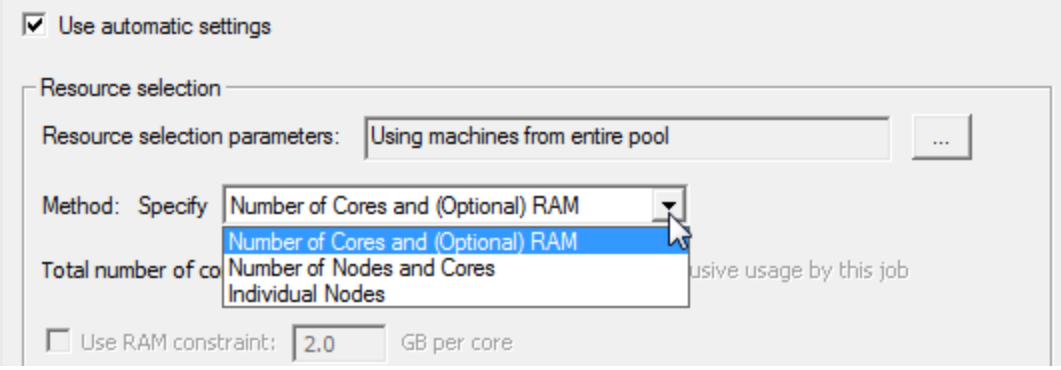
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



HFSS and HFSS 3D Layout have **Use automatic settings** selected by default. With **Use automatic settings** selected, the **Job distribution** field is removed. When using automatic settings, you can specify **Resource selection** parameters. The ellipsis button [...] opens the Compute Resource Selection Parameters dialog box. If you do not specify any parameters, the default is "Using machines from entire pool."

The Method field of the **Submit Job To** dialog box has a drop-down menu with two or three selections, depending on whether you select **Use automatic settings**.

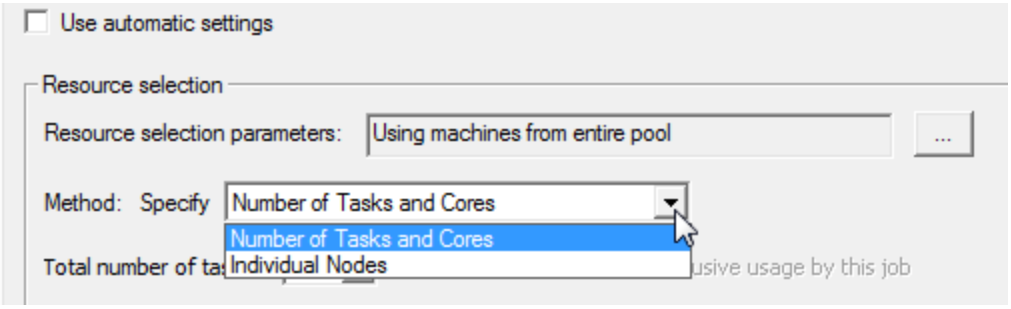
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

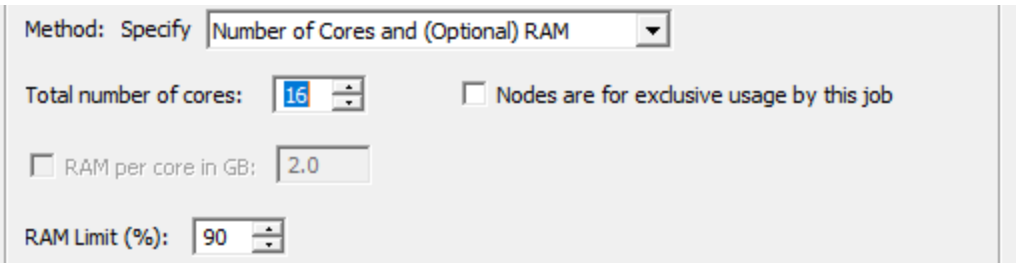
If you select Use automatic settings, Optimetrics variations will be solved sequentially. Other distribution types will be distributed automatically. It does distribute frequencies, domains, and use of multiple level domains.

If you uncheck or cannot access Use automatic settings, these two Methods are listed:



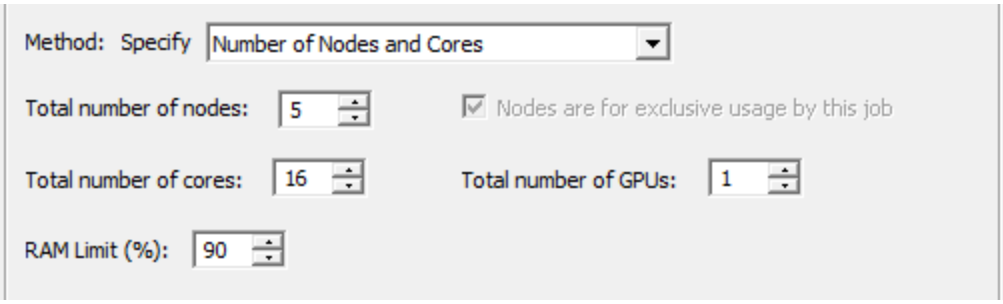
For Windows, there is a 64 core limit for a single machine. Each Method selection changes the available options listed:

- Specify Number of Cores and (Optional) RAM

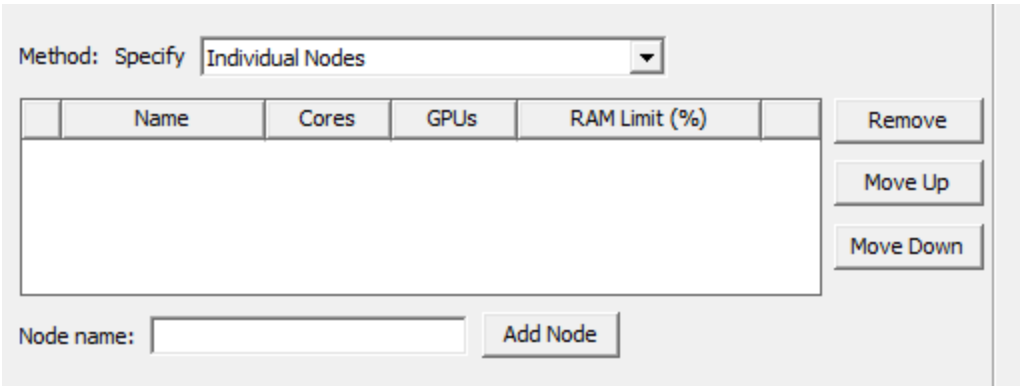


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Number of Nodes and Cores

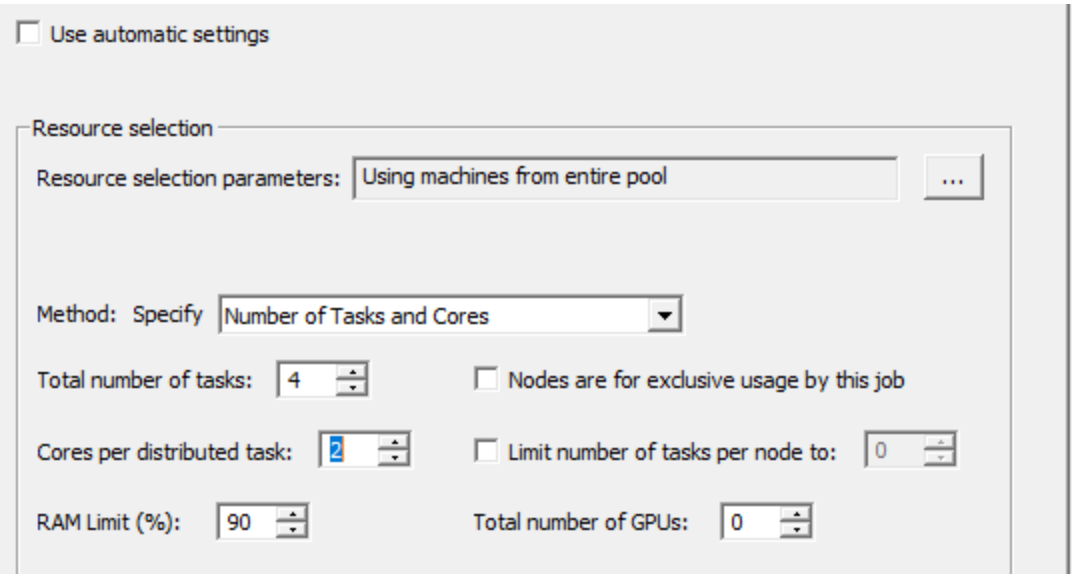


- Individual Nodes



- Number of Tasks and Cores ("Use automatic settings" is unchecked for this option. Checking "Use automatic settings" means that you do not have to specify tasks or core parameters):

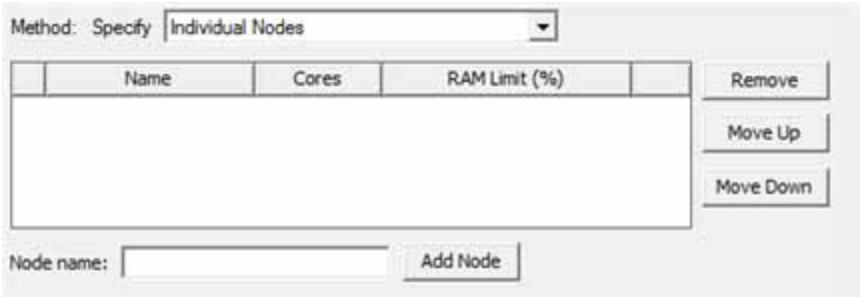
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Individual Node List

For Windows HPC jobs, you may either specify a node list, or specify job parallelization parameters, but not both.

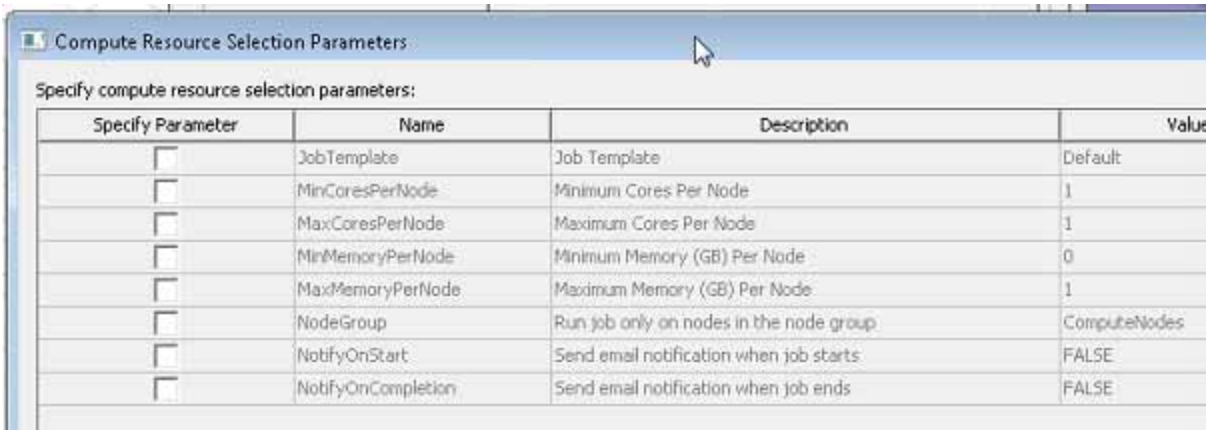
If you select the Individual Nodes Method, you may specify a node list, and the Job parallelization controls are disabled. In this case, the node list should only include cluster nodes that are valid for the job. For each node, you enter the node name and add the node. In the table, you can specify the number of cores and the RAM limit as a percentage. You can use the Remove, Move Up and Move Down buttons to edit and order the list.



Compute Resource Selection Dialog

By default, you can draw from the entire pool. You can also click the ellipsis button [...] to open a **Compute Resource Selection** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The resource selection parameters for Windows HPC jobs are:

- JobTemplate: Job Template - The JobTemplate may limit the job parameters or specify defaults values for job parameters
- MinCoresPerNode: Minimum Cores Per Node
- MaxCoresPerNode: Maximum Cores Per Node
- MinMemoryPerNode: Minimum Memory (GB) Per Node
- MaxMemoryPerNode: Maximum Memory (GB) Per Node
- NodeGroup: Run job only on nodes in the node group
- NotifyOnStart: If True, send email notification when job starts. Email notifications must be configured and enabled for the cluster by the administrator. (The cluster head node must run Windows HPC Server 2008 or above.)
- NotifyOnCompletion: If True, send email notification when job ends. Email notifications must be configured and enabled for the cluster by the administrator. (The cluster head node must run Windows HPC Server 2008 or above.)

Job Parallelization

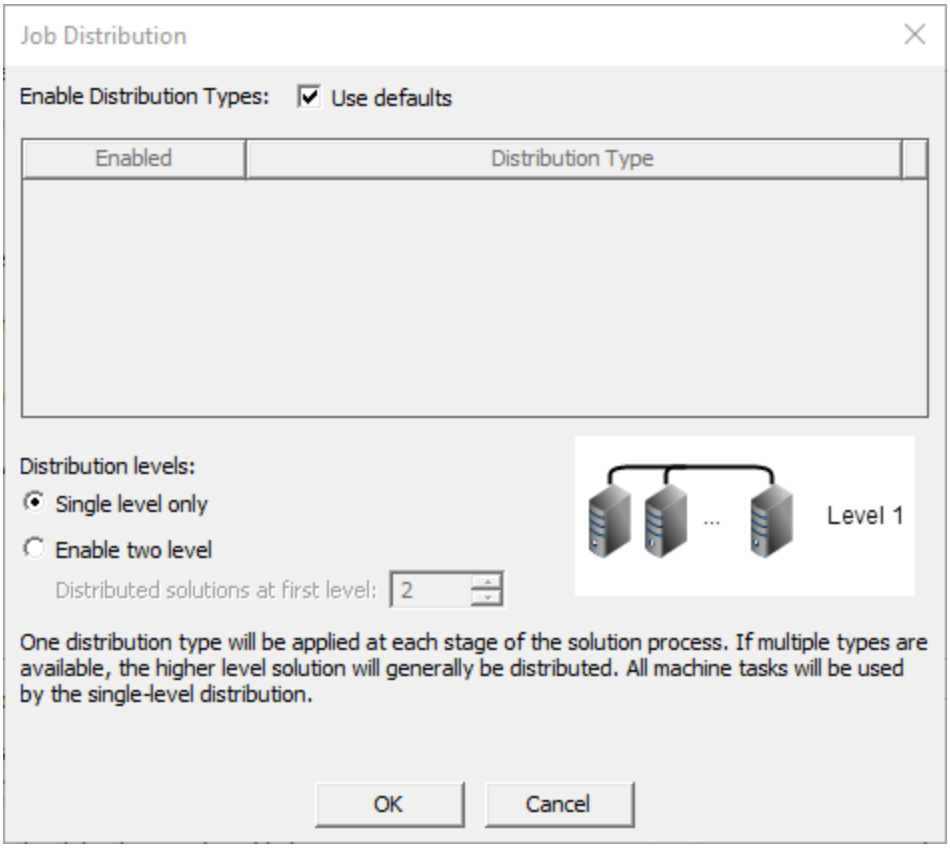
For Windows HPC jobs, you may either specify a node list, or specify the job parallelization parameters, but not both. The Job parallelization fields let you specify:

- Total number of tasks: The number of nodes requested for the job is the total number of tasks divided by limit on the number of tasks per node, rounded up if it is not an integer.
- Cores per distributed task. This determines the amount of multiprocessing per task.
- Whether nodes are for exclusive usage by this job
- Whether to limit the number of tasks per node to a value. If the "Limit number of tasks per node" check box is not checked, then the job is submitted with a job unit type of "Core".

Job Distribution

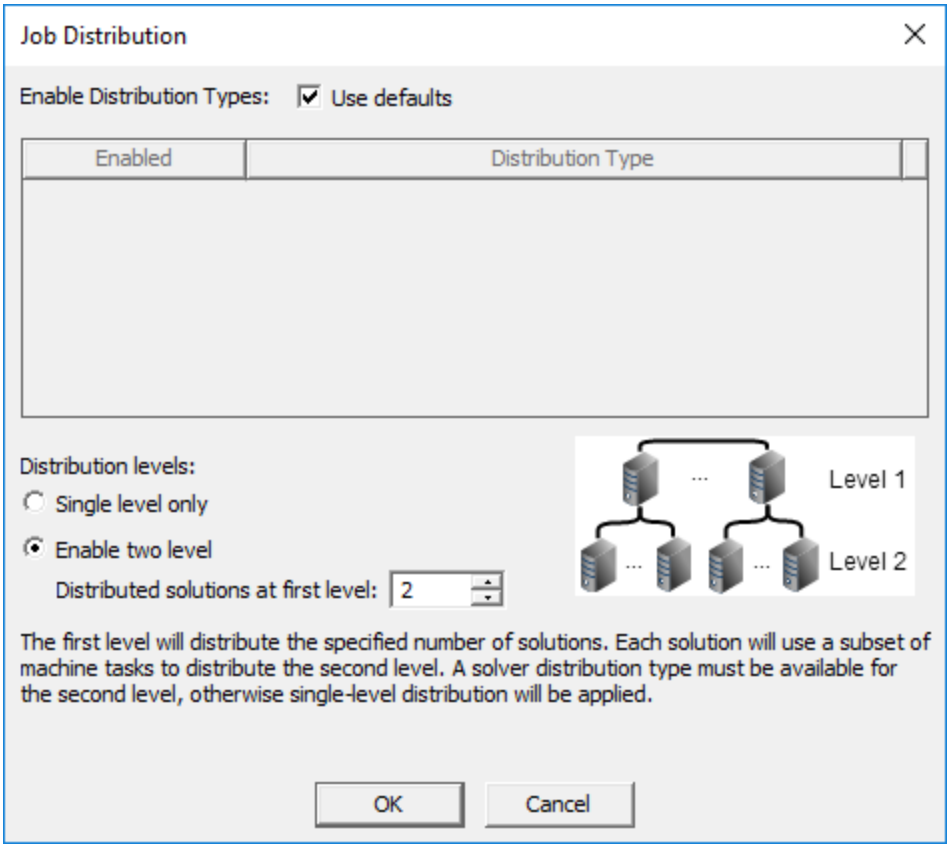
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Single level or two level distribution (*single level* is the default). Click **Modify** to display the **Job Distribution** dialog box and select the **Enable two level** option if applicable and desired.



- Second level distribution operates within DSO. If available and enabled you can specify the number distributed solutions for level 1.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

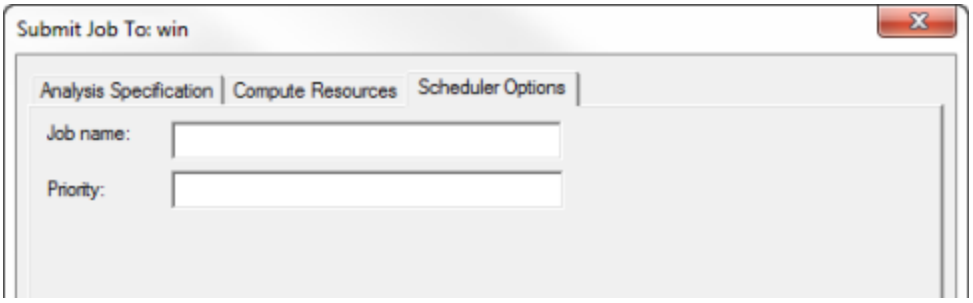


In response to a set of minimal constraints, the Scheduler may increase the resources assigned beyond the minimal values in order to meet the full set of requirements. For example, if you specify 7 distributed engines, with two processors per engine, and also limit the number of engines per node to 4, the scheduler may increase the number of cores used in order to meet the limit specified for engines per node. Notice that a preview of the Submit Job Results shows the number of resources assigned, and that the scheduler generated code includes an MPI specification.

Scheduler Options

The **Scheduler Options** tab provides for specifying the job name and/or the job priority. While the **Show advanced options** check box enables the display of Job submission options, no job submission options should be specified for Windows HPC.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Preview Submission

The Preview Submission button opens a window that shows a text description of the job to be submitted and the task used to start the product on one of the nodes.

The JOB PARAMETERS section contains information on parameter that apply to the job as a whole.

- The "Job resource parameters" section indicates whether the job has exclusive use of nodes, the job unit type, and the minimum and maximum number of units requested for the job, node group, and email notifications.
- The "Job attributes" section displays the job name and job priority.
- The "User Specified Compute Resource Attributes" displays the Resource selection settings.

The TASK PARAMETERS section contains information on parameters that apply to the Desktop task, which is the main task of the job.

- The "Desktop task resource parameters" section indicates the job unit type (which is the same as in the JOB PARAMETERS), and the minimum and maximum number of units requested for the Desktop task.
- The "Command Line section" displays the desktop task command line, including all arguments.
- The "Environment variables" section displays the environment variables that are set for the Desktop task; the same environment variables will also apply to all other tasks of the job.
- The "Working directory" section indicates the working directory in which the Desktop task will run.

Monitor Job

If you have checked the Monitor Job option on the **Submit Job To** dialog box, **AnalysisSpecification** tab, you can invoke the **Monitor Job** window by clicking **Tools** > **Job Management** > **Monitor Jobs...** This dialog box may also be brought up by checking the **Begin**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

monitoring this job now check box when a job is successfully submitting using the job submission dialog box. For more details, see: Monitor Jobs Window.

Windows® HPC Job Templates

The job templates are managed by the Windows HPC cluster administrator. Every cluster has at least one job template, the "Default" job template. Every job has an associated job template. If no job template is specified, then the "Default" job template is used. The job template controls two related aspects of the job submission process. When a job is submitted, there are a number of job parameters which may be specified. Each parameter has a set of valid values. For example, the Priority parameter has five valid values, Highest, AboveNormal, Normal, BelowNormal, and Lowest. The job template controls the default value of each parameter; this is the value that the parameter has if it is not specifically overridden by the submitter. For example, in the Default job template, the default value of the Priority parameter is Normal. The job template may also limit the allowed values of each parameter to a subset of the valid values. For example, a job template for privileged users could allow all five Priority values, which a job template for unprivileged users could limit the allowed Priority values to Normal, BelowNormal and Lowest.

Each job template is a Windows object with access controlled by an ACL (access control list). Instead of the usual "Read" or "Read & Execute" permissions, there is a "Submit Job" permission which corresponds to the right to submit a job with this job template. The cluster administrator may create job templates to limit or control access to cluster resources. For example, a job template with limited allowed job run times, or access to a limited set of compute nodes could be created by the cluster administrator. Specific users or user groups could be forced to use this limited job template by omitting access to the other job templates or by adding a deny access entry for the specified user or group to the other job templates. See the *Microsoft HPC Pack 2019: Job Templates* white paper from Microsoft for additional details:

https://learn.microsoft.com/en-us/powershell/high-performance-computing/job-templates?view=hpc19-ps

Job templates may also be created to allow users to run jobs with limited knowledge of the appropriate job parameters. The cluster administrator creates a job template which has reasonable default values for the type of job to be run, and informs users which job template to use for each type of job. The template could also limit some parameters to only the subset of all values that are useful for the type of job associated with the template.

Selecting Computation Resource Units (Job Unit Type)

The Job Unit Type is the smallest unit of processing resources used to schedule the job. This is one of the most important job properties. There are three options for the Job Unit Type: cores, nodes or sockets.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Cores**: Jobs are scheduled in units of cores, which may be also described as a CPU cores, logical processors, or CPUs. This is the smallest unit of granularity available. This selection allows the scheduler to start multiple tasks on a processor, if the total number of cores needed by the tasks is less than or equal to the number of cores on the processor. This selection may also allow the scheduler to distribute more of the computational load to processors with more cores than to processors with fewer cores. For Windows, there is a 64 core limit for a single machine.
- **Nodes**: Jobs are scheduled in units of nodes, hosts or machines. This is the coarsest level of granularity that may be selected. When this option is selected, only one task will run on any give node at any given time. This is useful in cases where it is not desirable to run multiple tasks on a single host. For example, if each task is multi-threaded, running multiple tasks on the same node may not be needed to fully utilize the computing resources on the node. This may also be preferred if the tasks are memory intensive, and multiple tasks would be competing for the limited memory resources.
- **Sockets**: A socket (which may also be called a NUMA node) is a collection of cores sharing a direct connection to memory. A socket will contain at least one core, and it may contain several cores. The socket concept may not necessarily correspond to a physical socket. Scheduling at the socket level may be useful in cases in which each task requires extensive use of the memory bus, and scheduling multiple tasks on the same socket would result in excessive bus contention.

Windows® HPC Non-Exclusive Jobs

Windows HPC allows three choices for the Cluster AffinityType setting:

- No Jobs: Affinity is not set for any jobs
- Non Exclusive Jobs (default): Affinity is not set for exclusive jobs
- All Jobs: Affinity is set for all jobs

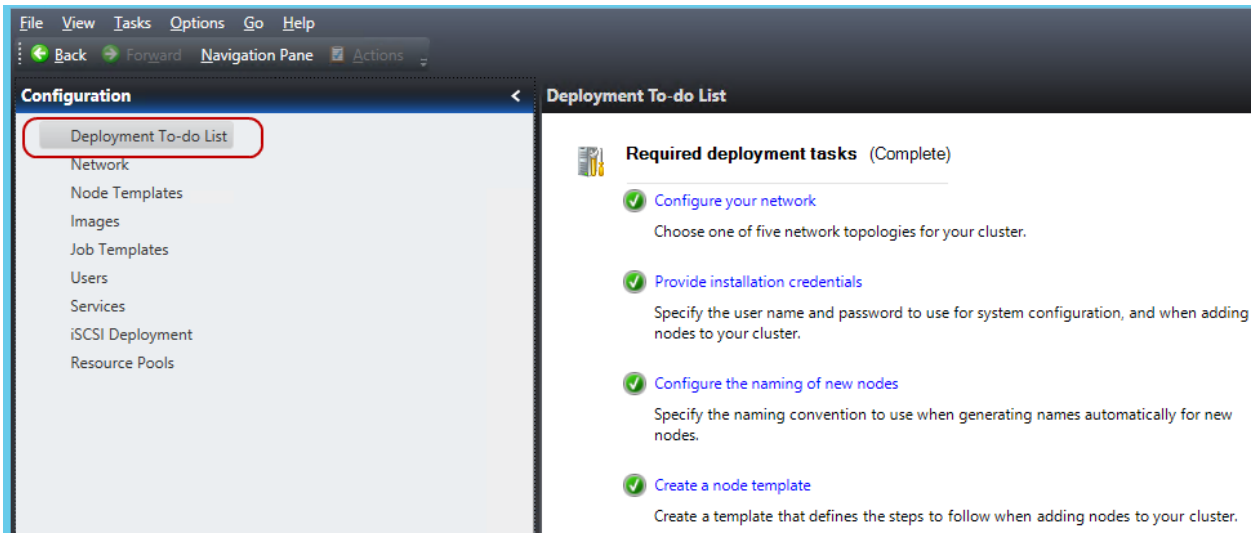
If the AffinityType is set to “No Jobs,” then we can allow non-exclusive jobs in many cases, but non-exclusive jobs will be disallowed if you specify an explicit host list (because all resources on all specified hosts are used), if you specify nodes and cores (to ensure that the job has exclusive access to the specified number of nodes), or if you specify multi-threading (to prevent a single task from being split across hosts). If the AffinityType is set to “Non Exclusive Jobs,” then exclusive jobs will be disallowed for all cases, the same as the restrictions for the previous release.

The previous job submission dialog always has the “exclusive” check box disabled for Windows HPC jobs, so that “exclusive” jobs cannot be submitted to a Windows HPC cluster. The “exclusive” check box is now enabled for Windows HPC jobs. This allows you to submit exclusive jobs to a Windows HPC cluster. If this check box is checked, but the submission parameters and Windows HPC cluster AffinityType do not allow submission of an exclusive job, then an error message is displayed when you submit or preview the job.

Setting the Windows HPC Cluster Affinity Type

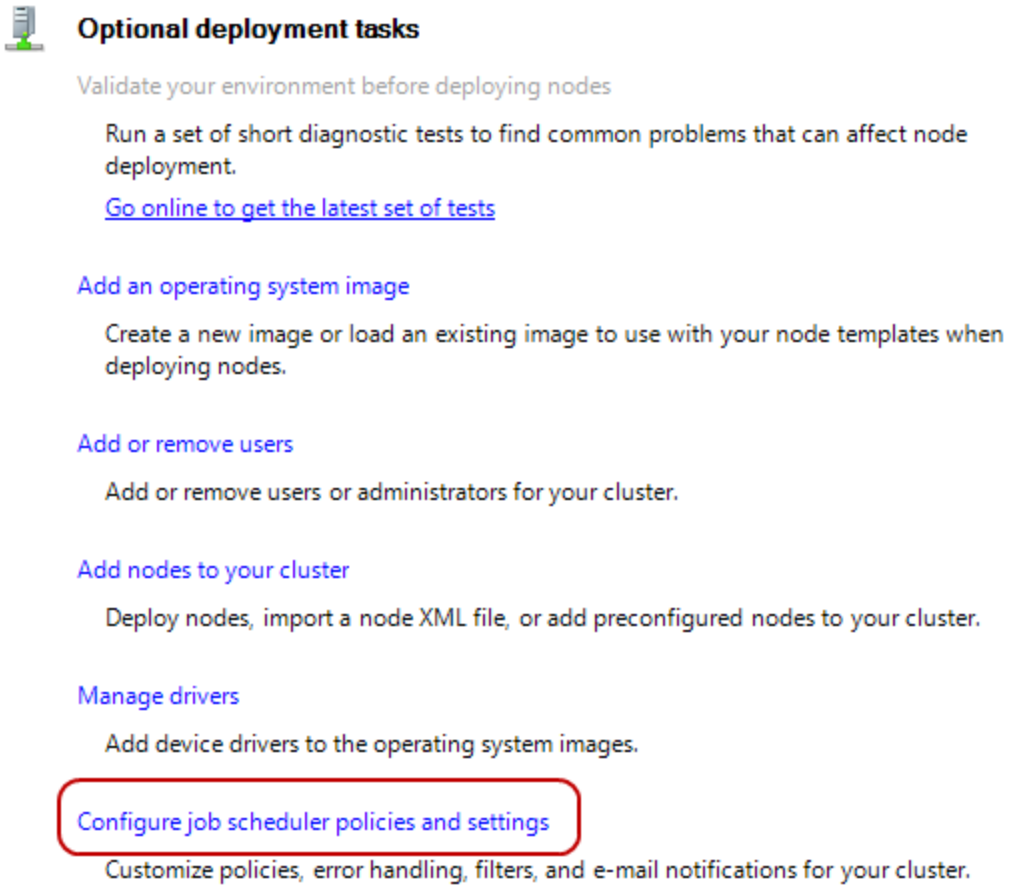
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A Windows HPC Cluster administrator is able to modify the Affinity Type setting for the cluster. The administrator should start the Windows HPC Cluster Manager. The first step is to select the Configuration button on the lower left pane, then select the Deployment To-do List in the upper left pane. Then, the upper right pane displays the Deployment To-do List, as shown below.



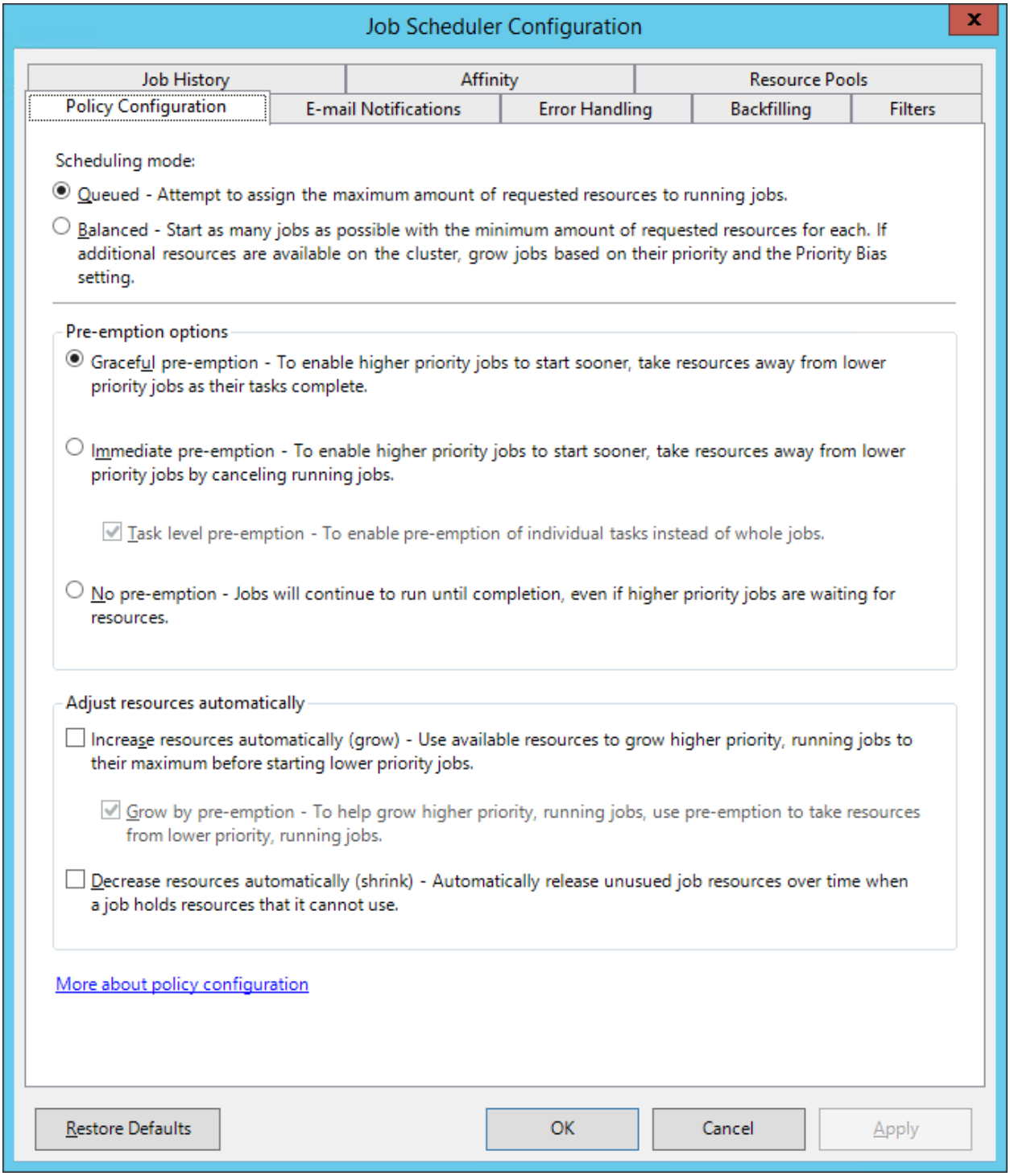
Next, the "Configure job scheduler policies and settings" should be selected. This is under the heading "Optional deployment tasks."

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



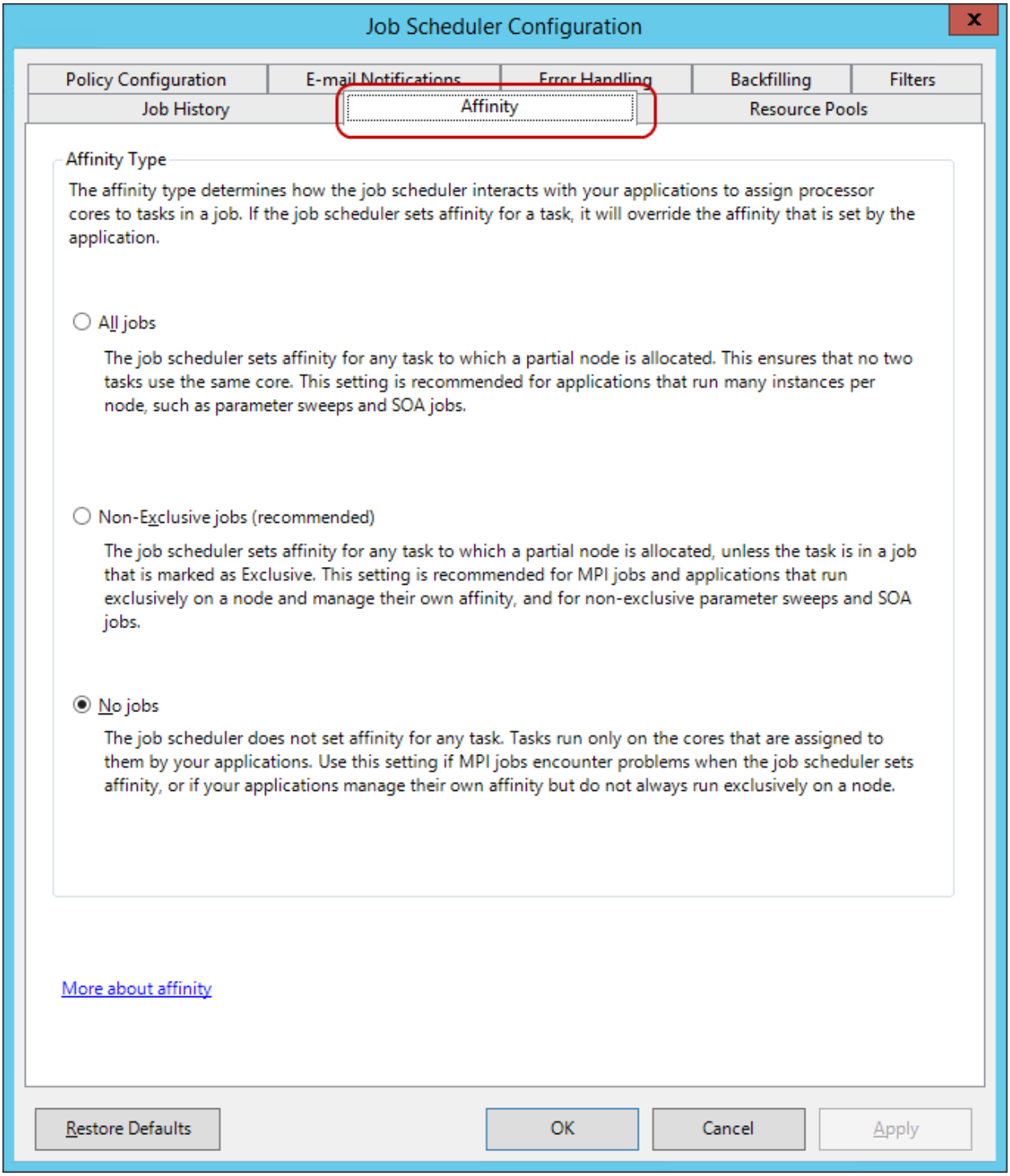
The *Job Scheduler Configuration* dialog box pops up, as shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Next, select the "Affinity" tab should be selected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The cluster administrator may now select a radio button to select the desired Affinity Type:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- All jobs
- Non-Exclusive jobs
- No jobs

In order to submit non-exclusive jobs, the "No jobs" option should be selected. The "Non-Exclusive jobs" option may be selected, but this will not allow non-exclusive Ansys Electromagnetics jobs to be submitted to the cluster. The "All jobs" option is not supported for Ansys Electromagnetics jobs. The OK or Apply buttons may be used to apply the selected option to the cluster.

Examples with "Submissions with Nodes are for exclusive usage by this job" Unchecked and Checked

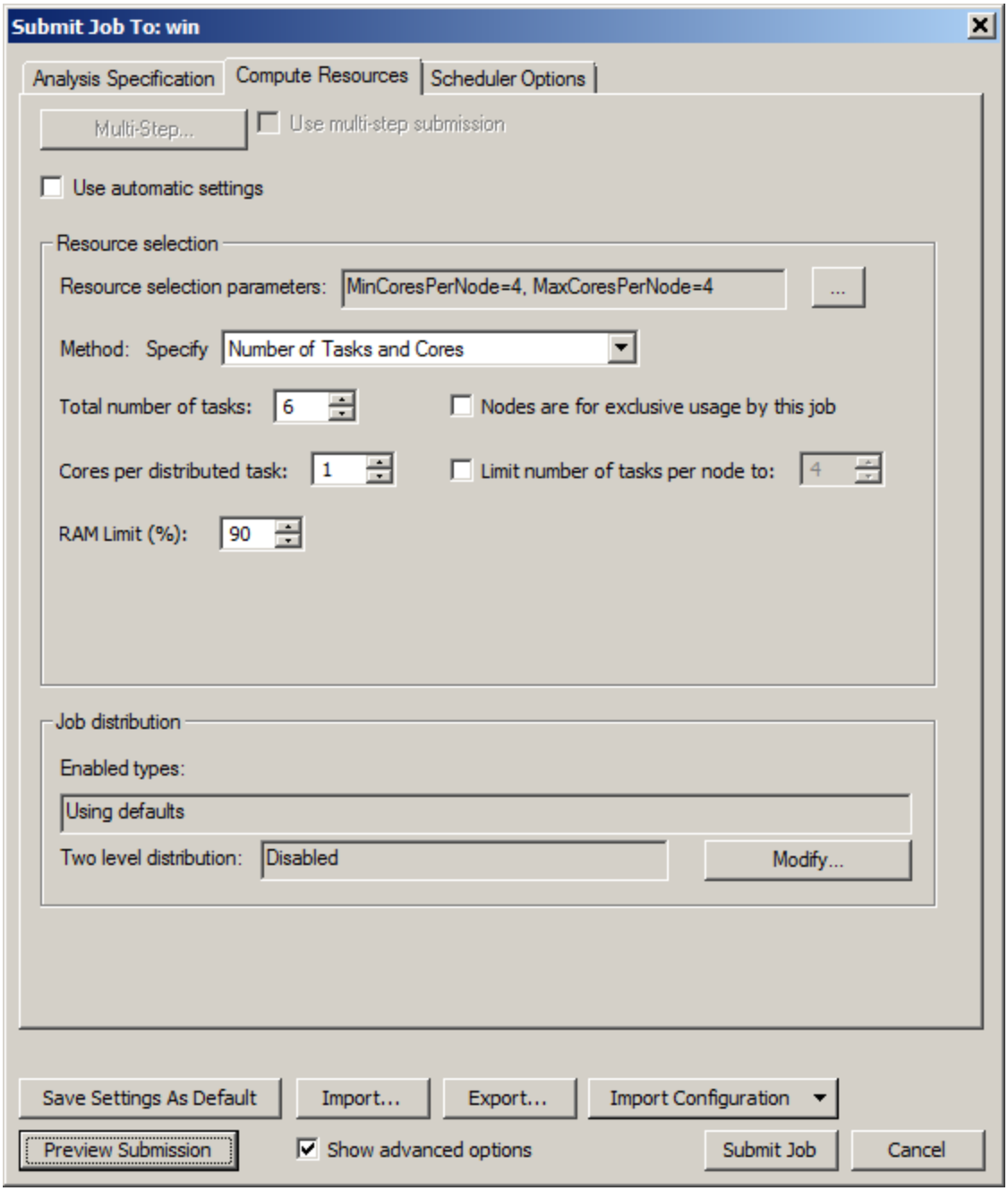
Most of the following screen shots are for the same project file submitted to a Windows HPC cluster with the AffinityType set to "No Jobs". The scheduler resource selection parameters are the same for all of these screen shots:

- MinCoresPerNode=4
- MaxCoresPerNode=4

The first set of screen shots show submission of a job to a Windows HPC cluster in which the "Use automatic settings" check box is not checked, and the resource selection method is "Specify Number of Tasks and Cores". The exclusive setting for the job is controlled by the "Nodes are for exclusive usage by this job" check box. In previous releases, this check box was always disabled and checked for Windows HPC jobs. Now, it is enabled, and the user may select whether it is checked or unchecked. If it is checked, then the behavior is the same as previous releases. If it is unchecked, then the job is submitted with the exclusive parameter for the job set to false.

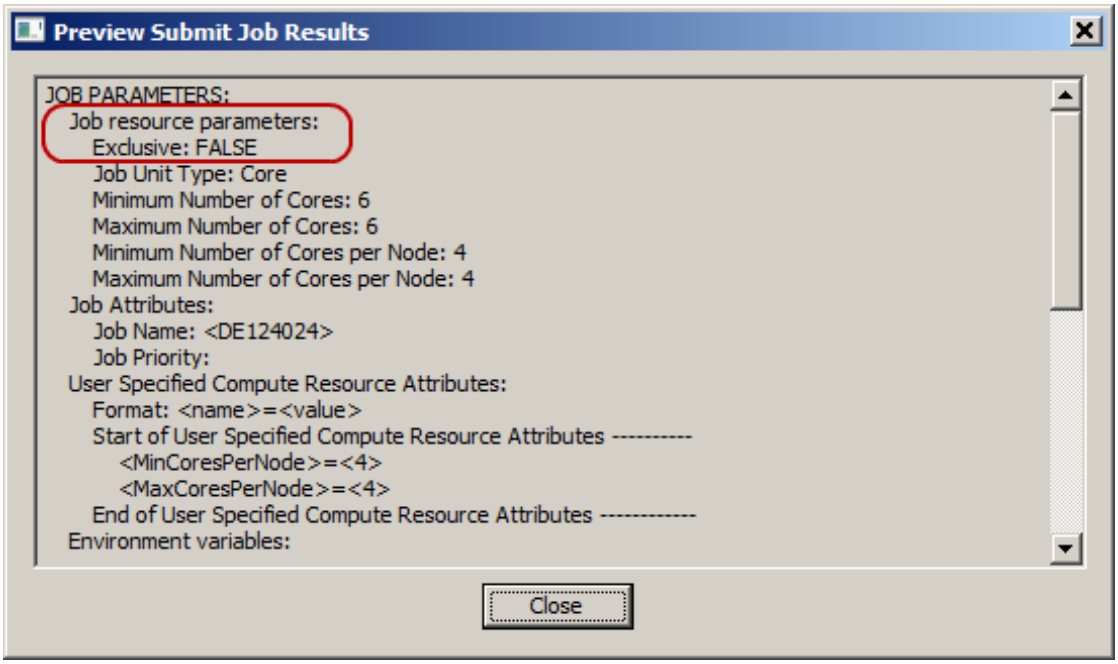
The following screen shot shows this configuration with a Submit Job with Task and Cores specified and nodes as non-exclusive.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The next figure shows Preview for the job submission indicating that the exclusive parameter for the job is false.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

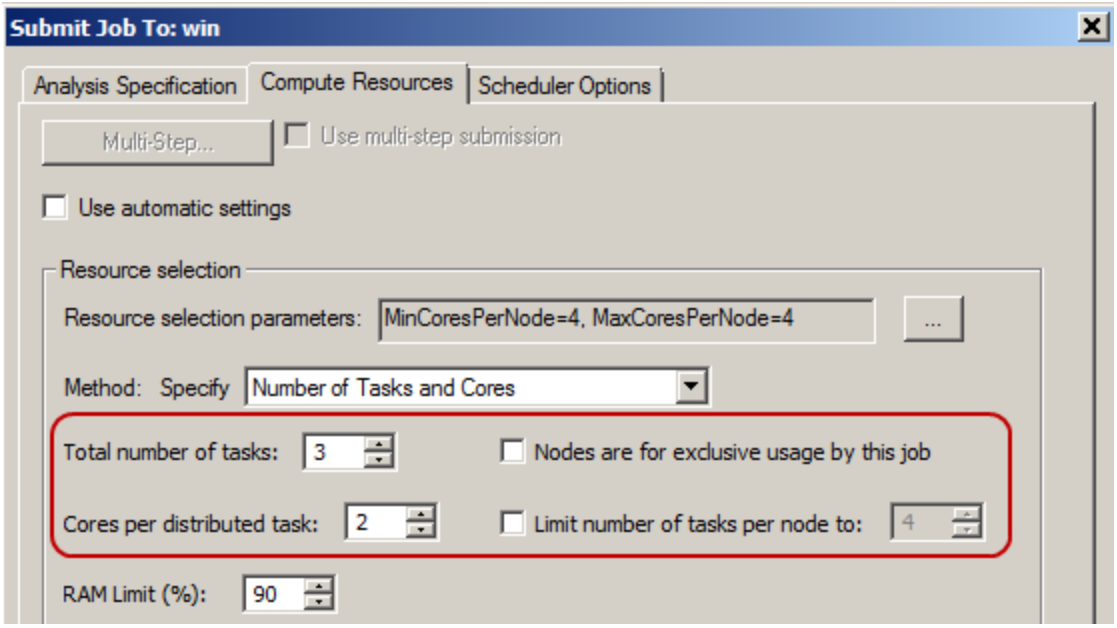


Jobs may only be submitted using this resource selection method with the exclusive parameter set to false if all three of the following conditions are met:

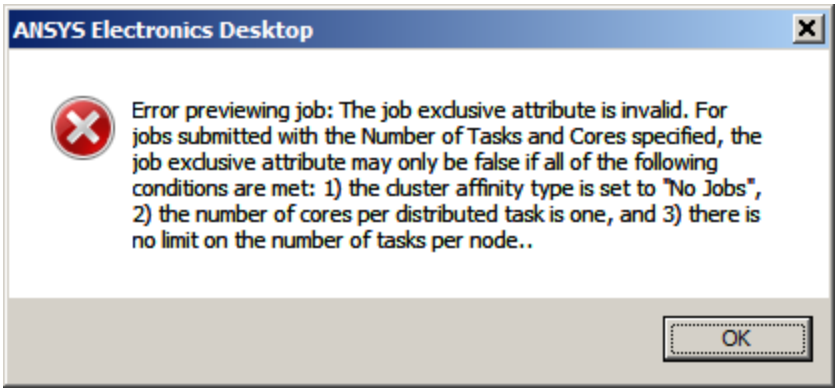
1. The cluster AffinityType is "No Jobs",
2. There is no multiprocessing, and
3. There is no limit on the number of tasks per node.

If any of these requirements are not met, then the job is not submitted or previewed, and an error message appears. The following figure shows this configuration.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



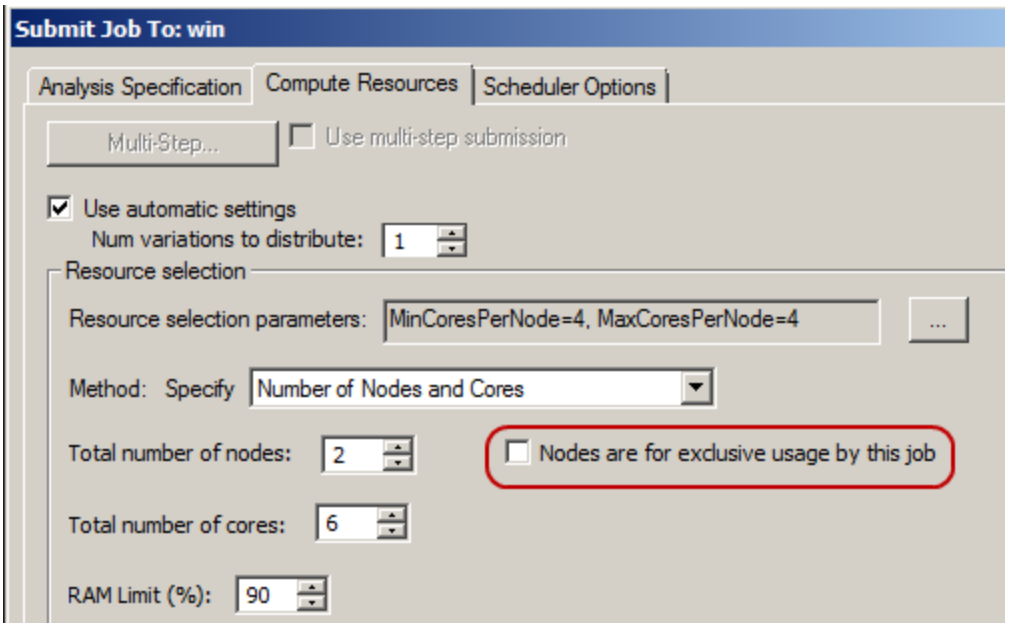
The following figure and screen shot ErrorMsgTasksAndCoresNonExMP.png shows the error message box shown to the user.



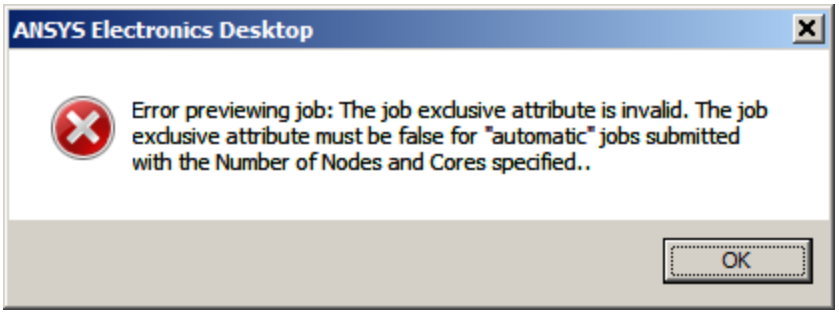
The next example shows submission of a job to a Windows HPC cluster in which the "Use automatic settings" check box is checked, and the resource selection method is "Specify Number of Nodes and Cores". If you select *Use automatic settings* with *Num variations to distribute* set to 1, Optimetrics variations will be solved sequentially. Other distribution types will be distributed automatically. It does distribute frequencies, domains, and use of multiple level domains. If you set Num variations to distribute to 2 or more, Optimetrics variations will be solved in parallel. Other distribution types will be distributed automatically. The exclusive setting for the job is controlled by the "Nodes are for exclusive usage by this job" check box. In previous releases, this check box was always disabled and checked for Windows HPC jobs. Now, it is enabled, and the user may select whether it is checked or unchecked. If it is checked, then the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

behavior is the same as previous releases. If it is unchecked, then the job is not submitted or previewed because only exclusive jobs are allowed when using this resource selection method, and an error message box is shown to the user in this case. The following figure shows this configuration:

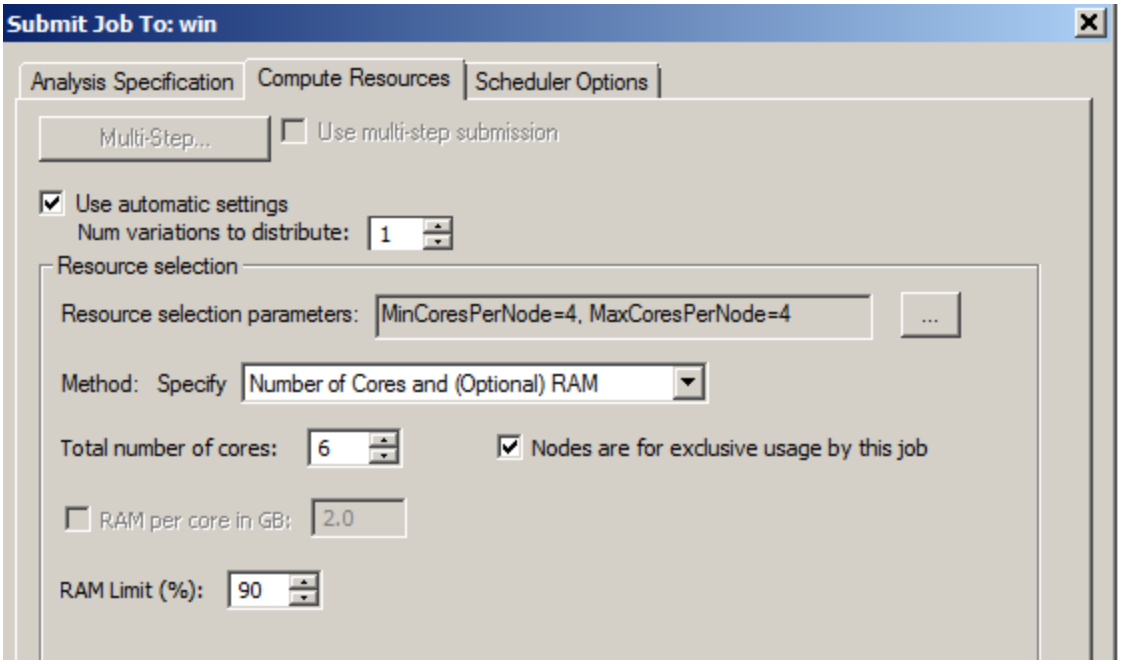


The following figure shows the error message.

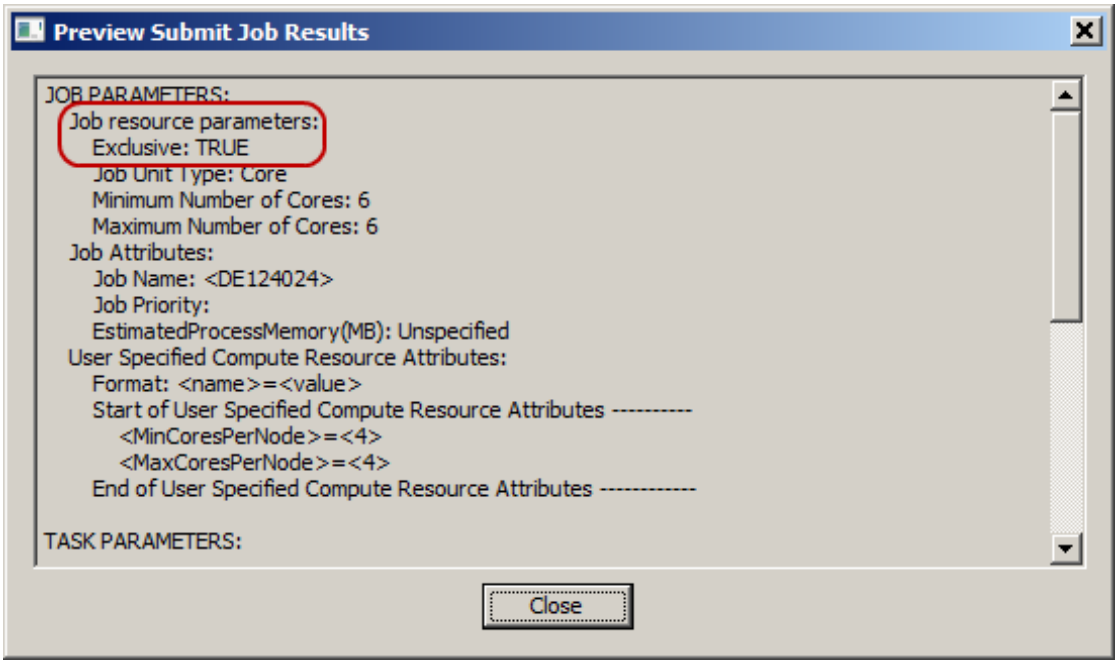


The next example show submission of a job to a Windows HPC cluster in which the "Use automatic settings" check box is checked, and the resource selection method is "Specify Number of Cores and (Optional) RAM". The exclusive setting for the job is controlled by the "Nodes are for exclusive usage by this job" check box. In previous releases, this check box was always disabled and checked for Windows HPC jobs. Now, it is enabled, and you may select whether it is checked or unchecked. If it is checked, then the behavior is the same as previous releases. The following figure shows this configuration.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

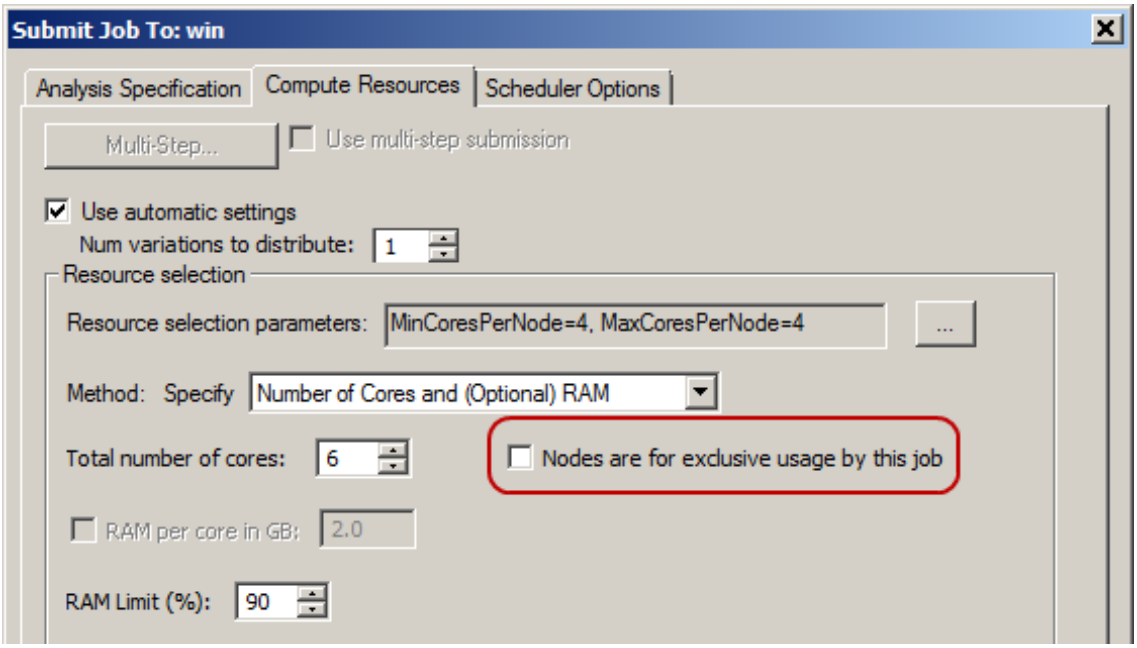


The following figure shows the Job Preview for this case, indicating that the exclusive parameter for the job is true.

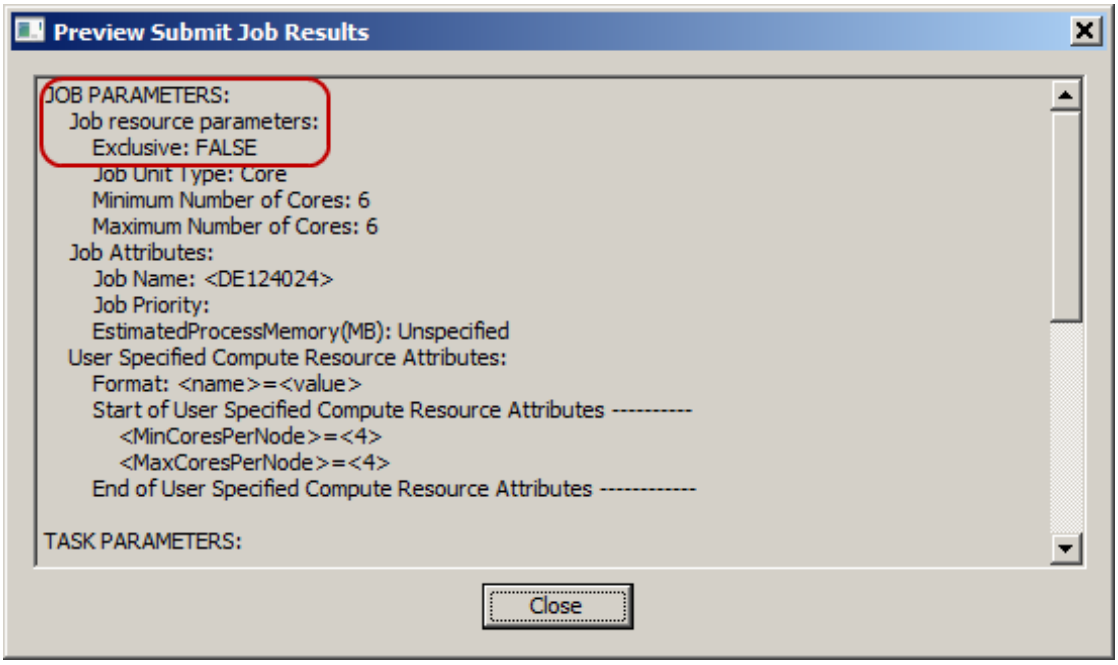


If the exclusive check box is unchecked, then the job is submitted with the exclusive parameter set to false. The following figure shows this configuration.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



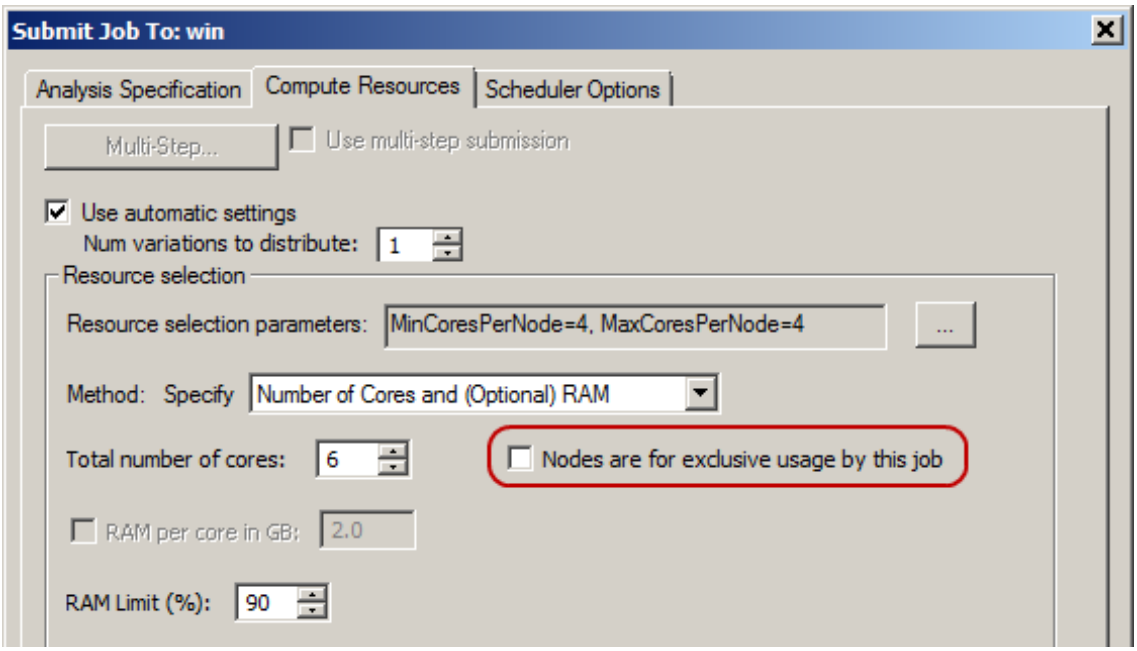
The following figurescreen shot PreviewCoresAndRAMNonEx.png shows the Job Preview for this case, indicating that the exclusive parameter for the job is false.



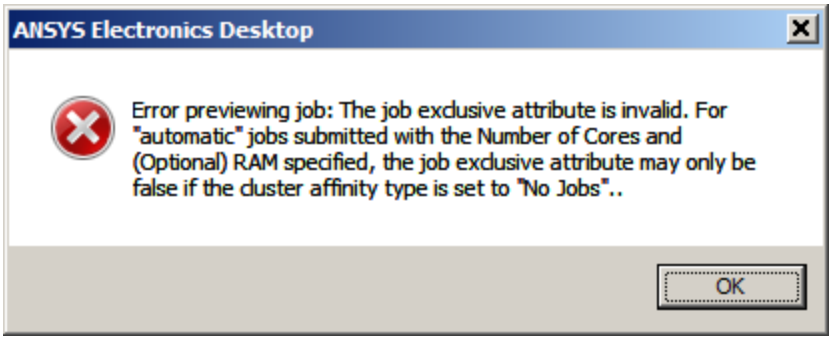
The last example resembles the previous one, except that the Windows HPC cluster AffinityType is set to "Non Exclusive Jobs" in this case. The job is submitted with the "Use

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

automatic settings" check box in the checked state, and the resource selection method set to "Specify Number of Cores and (Optional) RAM". Because the AffinityType is not "No Jobs", only exclusive jobs may be submitted to this cluster. If the exclusive check box is not checked, then the job is not submitted or previewed, and an error message box is displayed to the user. The following figure shows this configuration.



The following figure shows the error message.



Windows® HPC Job Credentials

Normally, a user will be prompted for the credentials used to submit a job. One way to simplify this process is to use the "cluscfg setcreds" command to set the user's credentials in the credentials cache. If this is done, then no password needs to be supplied for a job submitted for

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the specified user. Here is a cluscfg command that may be used to set the user credentials in the credentials cache:

```
cluscfg setcreds /password:* /scheduler:cluster_name
/user:domain\user_name
```

Here:

- cluster_name = the name of the cluster (hostname of the head node)
- domain = optional domain name; if omitted, the following \ should also be omitted
- user = user name

When this form of the command is used, the user is prompted for the password and also asked if the password should be remembered (cached).

See the following web page for more information on the cluscfg setcreds command:

http://technet.microsoft.com/en-us/library/cc947669(WS.10).aspx

Integration with Grid Engine (GE)

The Grid Engine (GE) scheduler (also known as OGE and SGE) is only supported on Linux. With GE, jobs may be submitted in any of the following ways:

- Using GE commands (qsub, etc.) or the SGE gui (qmon)
- Using the generic scheduler GUI in local mode
- Using the generic scheduler GUI in service mode

For additional information on supported schedulers, visit the Platform Support page. Then, locate and click the following link:

Ansys 2025 R1 - Job Schedulers and Queuing Systems Support

Ansys Electromagnetics products support Grid Engine (GE) for Serial analysis, Multi Processing and Distributed Analysis. Models with parametric sweeps can use Large Scale DSO. With GE, the Ansys EM job doesn't require graphics. Ansys EM job's progress can be monitored through GE commands or through Electronics Desktop's Job Management interface.

GE Job Management

You can use Ansys Electronics Desktop to submit batch jobs to GE and monitor those jobs.

This involves the following steps:

1. Use **Tools** > **Job Management** > **Select Scheduler** to select SGE/OGE/GE as the scheduler.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Use **Tools** > **Job Management** > **Submit Job** to submit a batch job to GE.

   GE-specific Settings:

   - On the **Compute Resources** tab, click the **Resource Selection Parameters** ellipses button (**...**) to specify the **ParallelEnvironment** parameter. If you do not specify anything, the scheduler will select a parameter.
   - On the **Compute Resources** tab, select **Specify node list** if you wish to specify the nodes. In a computing environment where the available cores are not uniform, you can use this to have control over which resources your job will use. If your Analysis configuration contains a node list, you can use Populate this Page from Analysis Configuration.
   - Memory resource behavior is dependent on the GE version as well as the particular scheduler settings. The output of the `qconf -sc` command shows all of the complexes available to GE schedulers. For a complex to be considered valid for memory resource selection, it must be of type "MEMORY", have relop "<=", be requestable (could be "forced" as well, instead of "YES"), and be consumable. At least one valid memory complex must be available to submit jobs using the Automatic Cores and RAM method.

     For Univa GE 8.3 and later, there is an additional column for whether resources are available to a preempting job after preemption of a running job. It is up to the cluster administrator to determine the appropriate Available After Pre-Emption (aapre) setting for memory complexes. This setting is ignored for memory complex validation.

     Determining the correct memory complex by default is error-prone. Because the correct choice of memory complex can vary from cluster to cluster, the memory complex selection is now exposed by default under the compute resource selection parameters, allowing the user to make the selection without having to set an environment variable. Only complexes that have been validated (meeting the requirements specified above) can be selected. The cluster administrator (or someone who has knowledge of the specifics of the cluster in use) should be able to determine the correct memory complex to use for Ansys Electronics jobs. To disable exposure of this selection option, the following environment variable can be set to "0": ANSOFT_SGE_ENABLE_MEM_RES_ATTRIB
   - On the **Scheduler Options** tab, you can **Customize Job Submission**. When the **Override job submission** radio button is selected, user-specified options replace most of the job submission options. When the **Additional job submission options** radio button is selected, user-specified options are appended to the bsub command.

   Click **Preview Submission** to view the qsub commands to be used to submit the job.

3. Use **Tools** > **Job Management** > **Monitor Job** to monitor the job's progress.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Installation of Ansys EM Tools on GE

Windows:

Install on every node of cluster

Setup 'temp directory' to a path that is same on all nodes. For example, c:\temp

LINUX:

Install on a single node, on a shared drive.

Setup 'temp directory' to a path that is same on all nodes. For example, /tmp

Ensure that the product is available using the same path on all nodes

Permissions:

All users of the cluster should have read/write permissions to temp directory

All users should have read/execute permissions to installation directory

When a desktop scheduler GUI is run the same node as the job submission node, no other configuration is necessary: installation is sufficient. You select the scheduler through the desktop GUI. You need to ensure that scheduler commands are available in the path before you launch desktop.

> **Note:**
>
> There is no need to install RSM unless the you are using the scheduler GUI on a post processing node that is different from the job submission node. In this case, RSM must be configured with the scheduler type and path.
>
> A post processing nodes is a node in the cluster that can run the Ansys Electromagnetics desktop in graphical mode. A job submission node is a node in the cluster in which job submission commands are available.

Turn OFF firewall between cluster nodes.

Scenario 1: The post-processing node and job-submission node roles are served by distinct machines.

In this case, perform the following configuration:

The job-submission node should be configured to run RSM service, which serves as a proxy to scheduler. The RSM Service should be running as 'root' in order to facilitate jobs running using the credentials of the job's owner. A **configuration file in the RSM installation folder should be edited** to specify information regarding the scheduler that manages jobs on this cluster. A

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

block labeled 'Scheduler' must be included within the 'AnsoftCOMDaemon' block. This block contains two string entries:

- SchedulerName: this contains the unique part of the scheduler proxy library name
- ConfigString: this contains a scheduler specific configuration string

The case of the SchedulerName string is significant on Linux because Linux file names are case sensitive. The case of the SchedulerName string is not significant on Microsoft Windows. In Ansys Electromagnetics Suite 2025 R1, the possible scheduler names are: lsf and sge. The ConfigString entry is a scheduler specific configuration string, described below.

In addition, the AnsoftRSMService must be started with appropriate environment variables set. Generally, the environment variables must be set the same as they would be set for using the scheduler via command lines.

SGE Details

For SGE, the ConfigString entry must contain the search path for the SGE commands. It may contain a single directory, the directory containing the SGE commands. Alternatively, it may be a path, with directories separated by the colon character ":", where the SGE command directory appears before any other directory containing files with the same name as any SGE commands.

Example ansoftrsmservice.cfg configuration file:

```
$begin 'AnsoftCOMDaemon'
$begin 'Managed COM Servers'
$end 'Managed COM Servers'
$begin 'Scheduler'
'SchedulerName'='sge'
'ConfigString'='/opt/sge6.2u4/bin/lx24-amd64'
$end 'Scheduler'
$end 'AnsoftCOMDaemon'
```

Scenario 2: The post-processing node and job-submission node roles are served by the same machine.

The **Select Scheduler...** command is used to gather details about the scheduler. In this case, the Desktop process should be started in an environment suitable for submitting jobs to the scheduler. See: Selecting a Scheduler.

The environment should be configured so that all SGE commands are found using the standard search path. In particular, search for the following commands in the search path should result in the SGE command being found: "qsub", "qdel", "qstat", and "qconf". No other command with the same name should appear before the SGE command in the search path.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GE Commands for Information about Jobs and Cluster Configuration

The following GE commands are especially useful for getting information about the cluster configuration or for getting information about running or completed jobs. This list only contains a few of the most common commands. Consult Grid Engine help for more information.

qconf -help: The first line displays the SGE version

qacct -j *job-id*: Displays a log of the completed job with the specified job ID (if accounting is enabled)

qstat -j *job-id*: Displays a log of the running job with the specified job ID

qconf -sc: Show all complex attributes

qconf -spl: Show a list of all parallel environments

qconf -sp *pe-name*: Show details of the parallel environment with the specified name

qconf -sql: Show a list of all queues

qconf -sq *queue-name*: Show details of the queue with the specified name

qconf -sconf: Show configurations

Submitting Ansys EM SGE Batch Jobs via the Command Line

The SGE qsub command may be used to submit Ansys EM jobs. Typical command formats are:

```
qsub qsub_args ansysEM_exe ansys_args
qsub qsub_args job_script
qsub qsub_args[ -]
```

where:

- *qsub_args* are the options of the qsub command,
- *ansysEM_exe* is the pathname of the Ansys EM tool executable to launch,
- *ansys_args* are the arguments to the Ansoft tool command, and
- *job_script* is a shell script containing the Ansys Electromagnetics desktop command to run.

In the first format, the Ansys EM desktop command and its arguments are specified on the **qsub** command line. In the second format, the pathname of a shell script containing the Ansys EM desktop command and its arguments is specified on the **qsub** command line. In the third format, the command is omitted or replaced with a hyphen; this indicates that the command or script will be taken from stdin.

Quoting Ansys EM Command or Arguments for SGE

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the Ansys EM tool executable pathname (*ansysEM_exe*) or any of the arguments of the Ansys EM tool command (*ansysEM_args*) contain characters which are interpreted by the command shell, then these special characters must be properly quoted to ensure that the correct command is launched by SGE. This is especially important when using the first form of the **qsub** command, as the Ansys EM desktop command is processed by the shell twice in this case. It is processed by the shell when the **qsub** command is processed, and again when the job is started.

Serial SGE Batch Jobs

In general, Ansys EM batch jobs may be submitted as SGE serial jobs without any special considerations.

See Monitoring Ansys EM SGE Batch Jobs for options for monitoring Ansys EM batch jobs.

Parallel SGE Batch Jobs

When an Ansys EM batch job is run as an SGE parallel job, the SGE scheduler will select the hosts for the distributed analysis job, and start the desktop process on one of these hosts. The desktop process will obtain the list of hosts from the SGE scheduler, and start analysis processes, as needed, using the SGE scheduler facilities. To run an SGE parallel job, the job must be submitted to an SGE parallel environment (PE).

If the qmaster tcp port is not configured as a service, but rather via the environment variable SGE_QMASTER_PORT, this variable must be set in the Ansys EM batch job environment. This is needed because the ANSOFT EM desktop uses the "qrsh -inherit" command to launch engine processes.

See Monitoring Ansys EM SGE Batch Jobs for options for monitoring Ansoft batch jobs.

Setting Up an SGE Parallel Environment (PE)

To allow Ansys EM batch jobs to distribute analysis engines to multiple hosts, the job must be run in a parallel environment (PE) in which the control_slaves parameter is set to TRUE. This setting is required to allow the Ansys EM desktop to start analysis engines on hosts other than the local host.

Here is a sample parallel environment configuration:

```
pe_name ans_test1
slots 999
user_lists NONE
xuser_lists NONE
start_proc_args /bin/true
stop_proc_args /bin/true
allocation_rule $round_robin
control_slaves TRUE
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
job_is_first_task FALSE
urgency_slots min
accounting_summary TRUE
```

The user_lists and xuser_lists parameters are ACLs (access control lists) used to control which users have permission to use the parallel environment. The user_lists setting gives permission to use the PE. The xuser_lists setting denies permission to use the parallel environment. The xuser_lists settings override the user_lists settings.

The start_proc_args and stop_proc_args parameters contain the pathname and arguments for the parallel environment startup and shutdown scripts. No startup or shutdown scripts are needed for parallel Ansys Electromagnetics batch jobs. The setting /bin/true may be used as the value for these scripts; this utility does nothing and returns an exit code indicating success (0).

The parallel environment allocation_rule parameter will affect how the analysis engine tasks are distributed across the hosts allocated to the job. The $round_robin setting distributes the tasks across the hosts in a round robin fashion, resulting in the load being relatively evenly distributed over all of the hosts. The $fill_up setting allocates all slots on a host before distributing the tasks to another host; the result is that most hosts are either fully utilized or completely unused. See the sge_pe man page for other settings for this parameter.

The control_slaves parameter must be set to TRUE, as described above.

The job_is_first_task parameter also affects how tasks are allocated. When submitting a job to run in a parallel environment, the number of parallel tasks, n, is specified on the command line. If this setting is TRUE, then the job process is considered one of the tasks, and only (n-1) additional tasks are allocated to the job. If the setting is FALSE, then the job process is not considered to be one of the tasks, and n additional tasks are allocated for the job.

See the sge_pe man page for more information about these and other PE parameters.

A parallel environment does not run tasks directly. Instead, the tasks are distributed to queues associated with the parallel environment. In order to complete the setup of a parallel environment, one or more queues need to be associated with the parallel environment. The queue pe_list parameter is used to specify the parallel environments (PEs) supported by the queue. This is an important step; **if no queues support a given PE, then jobs submitted to that PE will not run**.

Parallel Batch Job Command Line Considerations

The number of engines run on a host will depend on the total number of distributed engines, and the number of hosts allocated to the job. The memory required on a host depends on the number of engines running on the host and on the memory needed for each engine. The **qsub** command **-l resource=** *value*,... or **-q** *queue_list* > command line options specify that the parallel batch job run on machines with sufficient memory and other resources.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Monitoring GE Serial and Parallel Batch Jobs

You can monitor jobs through the Electronics Desktop user interface, or through the command line.

The suggestions below are for GE serial jobs and GE parallel jobs.

GE qstat Command

The SGE **qstat** command may be used to display information on jobs and queues. If the **-j** option is included, then information on jobs is displayed. If the option includes a job list (**-j [** *job_list* **]**), then the displayed information is limited to the jobs in the job list.

The **-u***user*,... option limits the output to jobs associated with users in the user list. If the **-u***user*,... option is not specified, then information on queues or jobs of the current user are displayed.

The **-t** option displays extended information about the subtasks of each displayed job. This is equivalent to the **-g t** option. The **-r** option displays extended information about the resource requirements of the displayed jobs.

See the SGE manual pages for more information.

Ansys EM Desktop -monitor Command Line Option for SGE

The **-monitor** command line option enables batch job output to the standard output and standard error streams. The warning, info, and progress messages are sent to the standard output stream. The error and fatal messages are sent to the standard error stream.

The SGE scheduler redirects the standard output and standard error streams of batch jobs to files specified in the **qsub -o [[** *hostname* **]:]** *path*,... and the **-e [[** *hostname* **]:]** *path*,... command line options, respectively. If either option is not specified, then the associated stream is redirected to the default file pathname.

The **qsub -j y[es] | n[o]** controls whether the standard output and standard error streams are merged. If the y or yes value is specified, then the standard error stream is merged into the standard output stream. If the **-e** *host_and_path* option is also specified in this case, the *host_and_path* setting is ignored. If the n or no value is specified, or if this option is not specified, then the standard error stream and standard output stream are not merged.

You can monitor the progress of a job by checking the standard output file for progress, info and warning messages, and checking the standard error file for error and fatal messages.

Example SGE qsub Command Lines

All of the following examples show how to submit Linux HFSS jobs on SGE, but similar command lines will work for all Ansys Electromagnetics products.

Serial job using command line:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
qsub -b y /opt/ansys_inc/v251/AnsysEM/ansysedt -ng -BatchSolve
 ~/projects/OptimTee.aedt
```

- The **-b y** option indicates that hfss is launched directly from the command line, instead of using a script.
- No queue is specified, so the default queue will be used.

Serial job with a hard runtime limit of 15 minutes:

```
qsub -b y -l h_rt=00:15:00 /opt/ansys_inc/v251/AnsysEM/ansysedt
 -ng -BatchSolve ~/projects/OptimTee.aedt
```

- The -l h_rt=00:15:00 option indicates that this job has a "hard" runtime limit of 15 minutes.

Serial job using a script, with a runtime limit specified in the script:

```
qsub ~/sge/scripts/OptimTee.csh
```

- The **-b y** option is absent, so the script ~/sge/scripts/OptimTee.csh will be run when the job starts.
- The script file OptimTee.csh may contain SGE directives in addition to the command(s) to run. In this example, a directive with a hard runtime limit if 15 minutes is included in the script.

Script file contents:

```
#!/bin/csh
#$ -l h_rt=00:15:00
/opt/ansys_inc/v251/AnsysEM/ansysedt -ng -BatchSolve
 ~/projects/OptimTee.aedt
```

- The SGE directive #$ -l h_rt=00:15:00 is equivalent to including -l h_rt=00:15:00 on the qsub command line.

Distributed processing job using 4 engines:

```
qsub -b y -pe pe1 4 /opt/ansys_inc/v251/AnsysEM/ansysedt
 -ng -BatchSolve -Distributed -machinelist num=4
~/projects/OptimTee.aedt
```

- The **-b y** option indicates that hfss is launched directly from the command line, instead of using a script.
- The **-pe pe1 4** command_line option indicates that this is a parallel job running under the pe1 parallel environment, and that 4 cores or processors are allocated to this parallel job.
- The "-machinelist num=n" option is now required for batch jobs.
- The **-Distributed** option indicates that this is a DSO job, so that multiple engines will be started. Because 4 cores are allocated to the job, the job will run 4 engines. The -Distributed option may now have additional options, such as includetypes=xxx, excludetypes=xxx, maxlevels=n, and numlevel1=n, where n indicates and integer, and xxx indicates a list of distribution types or "default".

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Recommended Practices for GE Clusters

The following subsections contain recommendations on how to set up an GE cluster for efficiently running Ansys Electromagnetics Suite serial and parallel jobs. These recommendations require the cluster administrator to make configuration changes.

Submitting Exclusive Jobs

Consumable Memory Limits

Serial Jobs in SGE

Parallel Jobs in SGE

Using Multithreading with Parallel Jobs

Submitting Exclusive Jobs

In many cases, clusters are used to run "large" Ansys Electromagnetics Suite batch jobs. That is, these are jobs that may require a large quantity of resources, such as processors, memory, disk space, or run time. One way to ensure that the resources needed by the batch job are available to the job is to run the job in an "exclusive" mode. That is, any host running the job is not available for use by any other jobs. There is no GE built in mechanism for specifying that a job is "exclusive". GE is extensible, and it is not difficult to configure the cluster to allow exclusive jobs. The steps below show one way to do this. This example requires GE 6.2u3 or later. Note that specifying a job as "exclusive" may delay the start of the job if there are not enough hosts available to run the job exclusively.

1. Use the command qconf -mc to add a new complex to the table of complexes. Recommended attributes are:
   - name : exclusive
   - shortcut : excl
   - type : BOOL
   - relop : EXCL
   - requestable : YES
   - consumable : YES
   - default : 0
   - urgency : 0
2. Set the value of "exclusive" to TRUE for each execution host using the command qconf -me hostname, where hostname is the name of the host. The values of all host configuration parameters may be displayed using the command qconf -se hostname. The "complex_values" line should look similar to:

   complex_values exclusive=TRUE, but other values may also be included.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. When submitting a job, the job will be "exclusive" if the value "excl" is included in the resource list specified by the qsub -l option. If the resource list does not include "excl" then the job will not be exclusive, and other jobs may run on the same host or hosts as this job.
4. Example qsub command line for exclusive serial job:

   ```
   qsub -b y -l excl /opt/ansys_inc/v251/AnsysEM/ansysedt -ng -
   BatchSolve -machinelist num=1 ~/projects/OptimTee.aedt.
   ```

   Although serial jobs use only one slot, no other jobs will run on the host where this job is running, even if additional slots are present.

5. Example qsub command line for exclusive parallel job using eight engines, each using a single thread of execution:

   ```
   qsub -b y -l excl -pe pe1 8 /opt/ansys_inc/v251/AnsysEM/ansysedt
   -ng -BatchSolve -Distributed -machinelist num=8
   ~/projects/OptimTee.aedt
   ```

   None of the hosts used for this job will be allowed to run other jobs while this job is running.

Consumable Memory Limits

GE contains several built-in complexes related to memory, including mem_total, for example, but none of these are "consumable". If a job is submitted with resource list including one of these non-consumable memory complexes (such as mem_total), then the job will run on a host or hosts only if sufficient memory is available. If a second job is submitted, the memory request for the second job is compared to the original total when determining if the job may run on a host. This may result in both jobs running out of memory. For example, if host A has mem_total=16G of memory, and two jobs are submitting with option "-l mt=16G", then both jobs could run on host A, if sufficient slots are available on host A.

GE allows complexes to be "consumable" to avoid this type of problem. If a complex is consumable and a job requests x amount of the complex in the -l resource list, then the available amount of the resource is decreased by x for subsequent jobs. For the same example as above, if the mem_total complex was consumable, then the first job would run on host A. This would decrease the available mem_total from 16G to 16G-16G = 0. The second job could not run on host A because there is no memory available for this job.

We do not recommend changing the behavior of the built-in complexes (such as mem_total) because other scripts may expect normal behavior of the built-in complexes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Recent versions of UGE (Univa Grid Engine) come with "m_mem_free" and "mem_free" complexes already configured, and if so then there is no more configuration required. You can just use mem_free when per-host memory request is desired, and m_mem_free when per-core (per-slot really) memory request is desired. SGE may already have "mem_free" which can be used for per-host memory request.

Below shows how to configure the mem_free consumable resource.

Recommended attributes are:

- name : mem_free
- shortcut : mf
- type : MEMORY
- relop : <=
- requestable : YES
- consumable : YES
- default : 0
- urgency : 0

Note:

Ansys Electronics Desktop has the capability in auto cores and RAM to automatically select the memory complex (if one is available) and create this command line option for the user, if you check the "Use RAM constraint" and enter a non-zero amount of GB to use per core.

Serial Jobs in GE

If a serial job is submitted with the option -l phys_mem=mem_neededincluded, then the job may only run on a host in which the remaining physical_memory is equal to or greater than the mem_neededvalue.

Example 1: Host A has physical_memory=16G, and host B has physical_memory=8G. If mem_neededis 8G, the job may run on either host A or host B. If mem_neededis 16G, then the job may only run on host A.

Example 2: Host A has physical_memory=16G, and host B has physical_memory=8G. Job 1 is already running on host A, and it was submitted with option -l phys_mem=8G. If job 2 is submitted with option -l phys_mem=16G, then job 2 cannot start until job 1 finishes, because only host A has 16GB of physical_memory. If job 2 is submitted with option -l phys_mem=8G,

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

then job 2 may start immediately, and run on either host A or host B, because both hosts have 8G of physical_memory remaining.

Parallel Jobs in GE

Because the consumable setting for physical_memory is YES (and not JOB), each slot of the job requires a physical_memory of mem_needed. The number of slots on a host assigned to the job is limited by the number of available slots on the host. It is also limited by the physical_memory available on the host; the number of slots assigned to the job cannot exceed the available physical_memory on the host divided by the mem_neededspecification.

Example 1: Execution host A and execution host B both have 4 slots per host (configured in the queue associated with the parallel environment). Host A has physical_memory=16G and host B has physical_memory=8G (shown by commands qconf -se Aand qconf -se B). If a job is submitted that requires 6 slots and 4G per slot, it will be able to run, with 4 slots on host A and 2 slots on host B. The qsub command might look like: qsub -l phys_mem=4G -pe pe_name 6 command args

Example 2: Same as example 1, except that 7 slots are requested. In this case, the job will never run. Although there are 8 slots available on hosts A and B, only two of the slots on host B are usable by this job because it only has physical_memory of 8G. With only 6 slots total available to this job (4 on host A and 2 on host B), the job can not start. In this case the command might look like: qsub -l phys_mem=4G -pe pe_name 7 command args

Using Multithreading with Parallel Jobs

For large jobs it may be useful to combine multiprocessing with distributed processing. Distributed processing refers to starting multiple processes, in which each process performs a portion of the analysis. These processes may run on the same host or on different hosts. The number of processes running at the same time is known as the number of "analysis engines". Multiprocessing refers to using multiple threads within a single process to decrease the run time of the process. Multiprocessing may also be called multi-threaded processing.

As a concrete example of combining multiprocessing with distributed processing, an analysis could run with four engines, where each engine uses two threads. In order to distribute the processing load so that no processor is overloaded, one slot is generally allocated per thread, so 8 slots would be needed for this example (4 engines * 2 threads per engine = 8 threads). The four engines could all run on a single host, or they could be distributed across 2, 3 or 4 hosts, depending on available slots. Each engine represents a single process, so the two slots for each engine must be allocated on the same host.

This section describes how to set up a GE cluster so that a specified number of slots per host may be requested when a job is submitted. This procedure will require the cluster administrator privileges. This capability may be used to submit parallel jobs in which one engine runs on each host, and the number of slots per host matches the number of threads used by each engine.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Let n be the largest number of slots available on any host used for the jobs. Create a separate parallel environment for each value of the number of slots per host from 1 to n. For example, pe_sph1 is a parallel environment in which one slot is allocated to the job per host, and pe_sph2 is a parallel environment in which two slots are allocated to the job per host, etc. The command qconf -ap pe_name may be used to create each new parallel environment. The allocation_rule parameter should be set to the number of slots per host, an integer from 1 to n. The control_slaves parameter should be set to TRUE, as described above. The slots parameter should be set to the maximum number of slots managed by this parallel_environment, which is typically set to a large number, such as 999. The other parameters should be set to values appropriate for the cluster. For example, the pe_sph2 parallel environment might have the following parameters:
    - pe_name : pe_sph2
    - slots : 999
    - user_lists : NONE
    - xuser_lists : NONE
    - start_proc_args : /bin/true
    - stop_proc_args : /bin/true
    - allocation_rule : 2
    - control_slaves : TRUE
    - job_is_first_task : FALSE
    - urgency_slots : min
    - accounting_summary : TRUE
2. When submitting a job, use the parallel environment where the slots per host matches the number of threads per engine.

The batchoptions setting -machinelist num=n is required. This should be set to match the number of slots per host. With any analysis, a portion of the analysis may not be distributed across multiple engines.

Example qsub command line for running distributed processing with four engines and multiprocessing with two threads per engine:

```
qsub -V -b y -pe pe_sph2 8 /opt/ansys_inc/v251/AnsysEM/ansysedt -ng -
BatchSolve -Distributed -machinelist num=4 -batchoptions
"projects/OptimTee.aedt"
```

The -Voption indicates that the all environment variables in the submission environment should be copied to the job environment.

- The -b y option indicates that hfss is launched directly from the command line, instead of using a script.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The -pe sph2 8 command_line option indicates that this is a parallel job running under the pe_sph2parallel environment so that two slots are allocated to this job from each host, and that 8 slots in total are allocated to this parallel job.
- The -Distributed option indicates that this is a DSO job, so that multiple engines will be started. The -Distributed option may now have additional options, such as includetypes=xxx, excludetypes=xxx, maxlevels=n, and numlevel1=n, where n indicates and integer, and xxx indicates a list of distribution types or "default".
- The -machinelist num=4option indicates that a total of four engines will be started.
- The entire hfss command is in double quotes, and the double quotes enclosing the -batchoptions value are escaped. Each of these double quotes is replaced by the sequence "\"".

Issue with qrsh (SGE)

Ansys EM parallel batch jobs use the SGE **qrsh** command to launch engine processes on remote hosts. If the **qrsh** command is not working correctly, then the parallel job is unable to launch engine processes on remote hosts. If this problem occurs, the batch log for the job typically includes one or more error messages indicating that a COM engine was unable to be started on a remote host. If this occurs, the user or cluster administrator should verify that the SGE qrsh command is working correctly, and correct the problem if the SGE **qrsh** command is not working correctly.

The **qrsh** command may be tested by running a simple command on a specified host, such as **qrsh -l hostname=host1 hostname** or **qrsh -l hostname=host1 ls /tmp**, where host1 is the remote host name. The first test should simply echo back the hostname of the remote machine. The second test should list the contents of the /tmp directory on the remote machine.

The failures of the SGE qrsh command are associated with the following global sge configuration parameters, listed below with values that may cause the failures:

```
qrsh_command /usr/bin/ssh -t
rsh_command /usr/bin/ssh -t
rlogin_command /usr/bin/ssh -t
```

If these parameter settings are removed, then the SGE built-in mechanisms are used for qrsh, rsh, and rlogin. No problems with the built-in versions have been reported. The SGE qconf -sconf global command may be used to view these parameter settings. The SGE qconf -mconf global command may be used to modify or remove these parameter settings.

Issue with MainWin Core Services for SGE

By default, SGE creates a temporary directory for each SGE batch job, and deletes this temporary directory and its contents when the job finishes. SGE sets the TMP and TMPDIR environment variables of the job environment to point to this temporary directory. Ansys EM desktop software starts the MainWin Core Services on startup, if they are not already running.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After the Ansys EM desktop software finishes, the MainWin Core Services time out and automatically shut down. The MainWin Core Services use the TMP and/or TMPDIR directories to store temporary data. If this temporary data is removed before the services shut down, then the services do not shut down automatically. Normally, SGE will remove the temporary directory and its contents before the services time out. The result is that these extraneous service processes run forever. If this problem occurs, each Ansoft batch job starts an additional set of these services that never shut down. This can result in an excessive number of processes running on the host where the Ansys EM desktop is started. The names of the service processes are:

- watchdog
- regss
- mwrpcss

Workaround for Issue with MainWin Core Services

One way to avoid this problem is to modify the environment in which the Ansys EM desktop runs so that the TMP and TMPDIR environment variables do not point to the directory which will be immediately removed by SGE when the job finishes. This can be done by copying the value of the TMPDIR environment variable to the ANS_SGE_TMPDIR environment variable, and unsetting the TMPDIR and TMP environment variables. The services ignore the ANS_SGE_ TMPDIR environment variable, but if this variable is set, then it will be used as the temporary directory for the rest of the Ansys EM software.

Here is an example bash wrapper script that may be used to work around this issue. In this example, the product is hfss, but the same approach will work for any Ansys EM product. In this example, the script is named sge_hfss and is in the AnsysEM software installation directory. When an Ansys Electromagnetics desktop job is submitted to the SGE scheduler, the script (sge_hfss, in this example) should be submitted instead of hfss. The script will modify the environment, as needed, then start hfss. When the analysis finishes, the script returns the exit status of hfss.

An alternative is to place the script in an arbitrary directory, and modify the script to include an absolute path to the product (hfss in this example).

Script contents:

```
#! /bin/bash

# This script will not correctly process arguments containing
# spaces or other characters special to the shell.

# Create hfss command line
# In this example, sge_hfss and hfss are in the same directory
# An alternative is to use an absolute path for the hfss command
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
cmd0=$0
cmd="${cmd0/%sge_hfss/hfss} $@"

# Fix environment variables
export ANS_SGE_TMPDIR=${TMPDIR}
unset TMPDIR
unset TMP

# Run the hfss command and return the exit status
${cmd}
exit $?
```

Integration with Platform's Load Sharing Facility (LSF)

The Load Sharing Facility (LSF) scheduler is only supported on Linux. Jobs may be submitted in any of the following ways:

- Job Submission GUI
- Using LSF commands (qsub, etc.)

For additional information on supported schedulers, visit the Platform Support page. Then, locate and click the following link:

Ansys 2025 R1 - Job Schedulers and Queuing Systems Support

> **Note:**
>
> If a temp directory is setup by the LSF cluster administrator, analysis engines use this temp directory, overriding the setting in the Ansys EM product.

LSF Job Management

You can use Ansys Electronics Desktop to submit batch jobs to LSF and monitor those jobs.

This involves the following steps:

1. Use **Tools** > **Job Management** > **Select Scheduler** to select LSF as the scheduler.
2. Use **Tools** > **Job Management** > **Submit Job** to submit a batch job to LSF.

   LSF-specific Settings:

   - On the **Compute Resources** tab, click the **Resource Selection Parameters** ellipses button (**...**) to specify the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Queue** – a drop-down menu lets you select Normal, chkpn_rerun_queue, idle, license, night, normal_allow_excl, owners, priorityu, or short.
- **MinCoresPerNode** – the minimum number of cores allowed on a node to be eligible for selection; translates to bsub -R select[ncpus>=N]
- **MaxCoresPerNode** – the maximum number of cores allowed on a node to be eligible for selection; translates to bsub -R select[ncpus<=N]
- **MinMemoryPerNode** – the minimum amount of physical memory (specified in integer GigaBytes) allowed on a node to be eligible for selection; translates to bsub -R select[maxmem>=M]
- **MaxMemoryPerNode** – the maximum amount of physical memory (specified in integer GigaBytes) allowed on a node to be eligible for selection; translates to bsub -R select[maxmem<=M]

If you do not specify parameters, the scheduler does so.

- For LSF, the only non-automatic method of **Resource Selection** is **Number of Tasks and Cores**. You can specify the number of tasks, whether they are for exclusive use by the job, cores per distributed task, a limit number of tasks per node, and RAM limit as percent.

3. Use **Tools** > **Job Management** > **Monitor Job** to monitor the job's progress.

LSF Job Submission Guidelines

Before submitting an LSF job, ensure the following are true:

- The project is available in a shared drive that is accessible to all machines in the cluster.
- The project is available using the same path on all machines of cluster.
- There is sufficient space in the project directory and temp directories.
- There is sufficient memory per engine.
- The number of compute resources (Distributed Analysis machines and Multi Processing cores) will achieve the desired scale factor and effective resource utilization.

Integration of Ansys EM Products with LSF

With LSF you do not need to set 'Distributed Analysis Machines' or 'Remote Machine' options. Instead, you submit a job to LSF, requesting the appropriate resources for the job (number of processors, memory per processor, etc.).

For example:

```
bsub -n 1 ansysedt.exe -Batchsolve -ng -local -machinelist num=1
OptimTee.aedt
bsub -n 4 ansysedt.exe -Batchsolve -ng -Distributed -machinelist
num=4 OptimTee.adsn
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The job is queued by LSF until the requested resources are available. Upon resource availability, LSF starts Electronics Desktop with the specified command line on one of the allocated machines. During analysis, Electronics Desktop dynamically obtains the allocated 'Distributed Analysis Machines' from LSF. Electronics Desktop interfaces with LSF to launch engines on remote machines without going through Ansys RSM.

Installing Ansys EM Tools on LSF Cluster

The LSF scheduler is supported on Linux only.

lsf.conf should contain this line:

```
LSF_UNIT_FOR_LIMITS=MB
```

The administrator should have this line in the lsb.params file to ensure that memory reservations are per-slot (per-core):

```
RESOURCE_RESERVE_PER_SLOT=Y
```

Jobs may be submitted in any of the following ways:

- Using LSF commands (bsub, etc.)
- Using the generic scheduler GUI in local mode
- Using the generic scheduler GUI in service mode

For additional information on supported schedulers, visit the Platform Support page. Then, locate and click the following link:

Ansys 2025 R1 - Job Schedulers and Queuing Systems Support

Setup:

1. Install on a single node, on a shared drive.
2. Setup 'temp directory' to a path that is same on all nodes (for example, /tmp).
3. Ensure that the product is available using the same path on all nodes.

Permissions:

- All users of the cluster should have read/write permissions to temp directory.
- All users should have read/execute permissions in the installation directory.
- Turn OFF any firewall between cluster nodes.

When a desktop scheduler GUI is run the same node as the job submission node, no other configuration is necessary; installation is sufficient. Select the scheduler through the Electronics Desktop GUI. Ensure that scheduler commands are available in the path before you launch Electronics Desktop.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> There is no need to install RSM unless you are using the scheduler GUI on a post-processing node that is different from the job submission node. In this case, RSM must be configured with the scheduler type and path.
>
> A post-processing node is a node in the cluster that can run Electronics Desktop in graphical mode. A job submission node is a node in the cluster in which job submission commands are available.

Per-slot Resource Reservation

Set the cluster for per-slot resource allocation if the automatic cores and RAM resource selection method is to be used. You can check the cluster to see if per-slot resource allocation is configured by using the "bparams -a" command. Search the output for "RESOURCE_ RESERVE_PER_SLOT" to determine the setting (either "Y" or "N"). If set to "N" then consult the LSF administration guide on how to change this to "Y".

Scenario 1: The post-processing node and job-submission node roles are served by distinct machines.

In this case, perform the following configuration:

The job-submission node should be configured to run RSM service, which serves as a proxy to scheduler. The RSM Service should be running as 'root' in order to facilitate jobs running using the credentials of the job's owner. A **configuration file in the RSM installation folder should be edited** to specify information regarding the scheduler that manages jobs on this cluster. A block labeled 'Scheduler' must be included within the 'AnsoftCOMDaemon' block. This block contains two string entries:

- SchedulerName: this contains the unique part of the scheduler proxy library name
- ConfigString: this contains a scheduler specific configuration string

The case of the SchedulerName string is significant on Linux because Linux file names are case sensitive. In Ansys Electromagnetics Suite 2025 R1, possible scheduler names are: lsf and sge. The ConfigString entry is a scheduler specific configuration string, described below.

In addition, the AnsoftRSMService must be started with appropriate environment variables set. Generally, the environment variables must be set the same as they would be set for using the scheduler via command lines.

LSF Details

For the LSF scheduler proxy library, the ConfigString entry in the ansoftrsmservice.cfg configuration file is ignored. It may be empty or omitted entirely.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The AnsoftRSMService must be started with the environment set as it would be set for submitting jobs to the LSF cluster.

- For Linux, the cshrc.lsf or the profile.lsf file may be sourced to set up the environment, depending on the shell.

Example ansoftrsmservice.cfg configuration file:

```
$begin 'AnsoftCOMDaemon'
$begin 'Managed COM Servers'
$end 'Managed COM Servers'
$begin 'Scheduler'
'SchedulerName'='lsf'
'ConfigString'=''
$end 'Scheduler'
$end 'AnsoftCOMDaemon'
```

Scenario 2: The post-processing node and job-submission node roles are served by the same machine.

The **Select Scheduler...** command (as described in the Job Management User Interface for LSF section) is used to gather details about the scheduler. In this case, the Desktop process should be started in an environment suitable for submitting jobs to the scheduler. See below for details.

The environment should be configured so that the following LSF environment variables are set appropriately for the LSF cluster in use: LSF_BINDIR, LSF_SERVERDIR, LSF_LIBDIR, and LSF_ENVDIR. In addition, the following LSF commands should be found in the LSF_BINDIR directory: "bsub", "bjobs", "bkill", "lsid", "lsrun", "lshosts", "bmgroup", "bparams" and "bqueues".

Using the bsub Command to Submit Batch Jobs

The LSF bsub command may be used to submit jobs. The typical command format is:

`bsub` *`bsub_args ansys_exe ansys_args`*

where:

- *bsub_args* are the options of the bsub command,
- *ansysEM_exe* is the pathname of the Ansys Electromagnetics desktop executable to launch, and
- *ansys_args* are the arguments to the Ansys Electromagnetics desktop executable.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

bsub Arguments

The **bsub** command has a large number of options that may be used to control the submission process. Only a few options that are often used with Ansys Electromagnetics jobs are listed here. The following options may be used to submit serial or parallel LSF jobs.

-n*min_proc*, *max_proc* or **-n***min_proc*

Submits a parallel job, specifying the number of processors (or slots) required for the job. Here, *min_proc* is the minimum number of processors, and *max_proc* is the maximum number of processors. If no maximum is specified, then exactly *min_proc* processors are requested. If PARALLEL_SCHED_BY_SLOT=Y in lsb.params, this option specifies the number of slots required to run the job, not the number of processors. If the **-n** command line option is not specified, then the job is submitted as a serial batch job.

-R "span[ptile=n]"

There are many ways to use the -R "res_req" option to the bsub command. We only cover -R "span[ptile=n]" here, because this option is very useful for Ansys Electromagnetics jobs. When this option is specified, the LSF scheduler will allocate n processors (or slots) on each host to this job, even if more processors are available on the host.

-x

All hosts running this job operate in exclusive execution mode. The job will only run on a host having no other jobs running on that host. No other batch jobs will be started on a host while this job is running on that host.

See LSF documentation for a complete list of options for the **bsub** command.

> **Important:**
>
> If the Ansys EM tool executable pathname (*ansys_exe*) or any of the arguments of the Ansys tool command (*anssys_args*) contain characters which are interpreted by the command shell, these special characters must be properly quoted to ensure that the correct command is launched by LSF. A similar problem may occur if any of the *ansoft_args* require single quote, double quote or space characters. Note that the Ansys Electronics Desktop command is processed by the shell twice: once when the **bsub** command is processed, and again when the job starts. See: Example Command Lines.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example LSF bsub Command Lines (Linux Only)

> **Note:**
>
> The following examples use HFSS as the product, but similar command lines will work for all Ansys EM products.

Serial Job

```
bsub -n 1 "C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt" -ng
-BatchSolve -machinelist num=4 ~/projects/OptimTee.aedt
```

The -n 1 option indicates that this job runs on one core.

Serial Job Requiring a Minimum of 4GB

```
bsub -n 1 -R "select[mem>4000]"
"C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt" -ng
-BatchSolve -machinelist num=4 ~/projects/OptimTee.aedt
```

The -R "select[mem>4000]" option indicates that this needs a minimum of 4 GB memory.

Multi-processing Job using 4 Cores

```
bsub -n 4 -R "span[ptile=4]"
"C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt" -ng -BatchSolve
-batchoptions -machinelist num=4 ~/projects/OptimTee.aedt"
```

- The -R "span[ptile=4]" option indicates that the four cores need to be on the same machine.
- The -batchoptions option indicates that HFSS should use four cores for multi-processing.
- The entire hfss command is in double quotes, and the double quotes enclosing the -batchoptions value are escaped. Each of these double quotes is replaced by the sequence "\"".

Distributed Processing Job using 4 Engines

```
bsub -n 4 "C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt" -ng -
BatchSolve -Distributed ~/projects/OptimTee.aedt
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The -n 4 option indicates that the four cores are needed for the job.
- The -Distributed option indicates that this is a DSO job, so that multiple engines will be started. Because 4 cores are allocated to the job, the job will run 4 engines. The -Distributed option can have additional options, such as includetypes=xxx, excludetyeps=xxx, maxlevels=n, and numlevel1=n, where n indicates and integer, and xxx indicates a list of distribution types or "default".

Distributed Processing and Multi-processing Job using 4 Cores, with 2 Cores for Multi-processing

```
bsub -n 4 -R "span[ptile=2]" ~/projects/OptimTee.csh
```

Shell Script (~/projects/OptimTee.csh):

If a command is included in the **bsub** command line, the entire command will be processed by the command shell two times. The command is processed when the **bsub** command is processed by the shell and is processed again when the command is started by the scheduler. This example shows how to use a shell script so that the command line will be processed only once. The command is placed in the shell script, and then the shell script pathname is placed in the **bsub** command line. Then, the command is only processed by the command processor when the job is started. When using this approach, the shell script should be accessible from all of the cluster hosts.

```
#!/bin/csh
"C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt" -ng -BatchSolve
-Distributed -machinelist num=2 -batchoptions
~/projects/OptimTee.aedt
```

- The **-n 4** option indicates that the four cores are needed for the job.
- The **-R "span[ptile=2]"** option indicates that the cores must be allocated in groups of two cores on the same machine.
- The **-machinelist num=2** option indicates that this is a DSO job and that a total of two engines will be started.
- The **hfss** command is placed in the shell script (~/projects/OptimTee.csh). In the **bsub** command line, the **hfss** command is replaced by the shell script pathname.

Monitoring LSF Batch Jobs

You can monitor jobs through the Electronics Desktop user interface, or through the command line.

The suggestions below are for batch jobs run under LSF.

Ansys Electronics Desktop -monitor Command Line Option

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The -monitor command line option enables batch job output to the standard output and standard error streams. The warning, info, and progress messages are sent to the standard output stream. The error and fatal messages are sent to the standard error stream.

LSF bpeek Command

The LSF bpeek command may be used to monitor job progress. The command **pbeek [-f]** *job_id* displays the standard output and standard error produced by the job with id *job_id* from the job start to the current time (the time when the command is executed). This command is only valid for jobs that have not yet finished. When used with the **-f** option on Linux, the output of the job is displayed using the command tail **-f**, so that ongoing progress may be monitored.

In order to display messages to standard output and standard error, specify the **-monitor** command line option on the Ansys EM tool command line. Then, these messages can be seen using the LSF bpeek command.

Terminating LSF Batch Jobs

To cancel or terminate an Ansys EM LSF batch job, we recommend using the job monitoring UI to terminate jobs cleanly, rather than using the bkill commands. Using this approach will allow the batch job to shut down in an orderly fashion.

Using the LSF **bkill** command without the **-s SIGTERM** option or simply terminating the job processes may cause some of the following problems:

- Some engine processes are not shut down and continue to run.
- LSF job is not fully removed.
- Project .lock file is not removed.
- MainWin core service processes (watchdog, mwrpcss and/or regss) are not stopped.

Some of these may interfere with submission of additional LSF batch jobs. For example, it may be necessary to manually remove the project lock file to submit another batch job for the same project. MainWin core service processes may also interfere with starting subsequent Ansoft batch jobs. Normally, these processes should timeout and end 15 seconds after the Ansys Electromagnetics product shuts down. Any MainWin core service processes (watchdog, mwrpcss and/or regss) that continue to run for more than 15 seconds after the product has stopped may be hung. The hung processes may need to be manually killed, after ensuring that these processes are associated with an Ansys EM job that has finished or terminated.

Stop a job cleanly - ensures that the results obtained until now are preserved:

```
bkill -s TERM <jobid>.
```

Stop an job abruptly - results are most likely lost. You have to manually remove the project lock file:

```
bkill <jobid>
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LSF Known Issues and Workarounds

The following are known issues. Workarounds are noted when available:

- Desktop or remote machine cannot have multiple IP addresses. This is unsupported.
- Core dump files may appear when a job has finished running. Results are still computed correctly. Workaround: Limit size of core dumps to 0 through the following job submit option: `bsub -C 0 -n <number-of-cores> -q <queue-name>`
- Firewall should be turned off on all machines in the cluster.
- Sometimes LSF ends a job (for example, a job may be preempted due to a high priority job). This may result in the presence of a .lock file in the project directory. You must manually delete the .lock file before continuing with further analysis.
- When an LSF job is ended, MainWin services (watchdog, regss, and mwrcpss) could keep running. The result is that later jobs cannot start on the machine. The fix is to end these processes before starting a new job.
- Analysis fails abruptly when running out of resources (cpu/memory/disk). Ensure sufficient resources are provided.

LSF Troubleshooting

The following are general troubleshooting steps:

1. Ensure the LSF `lsrun` command is enabled.
2. Look for user errors.

   For example:

   - Are the executable path and project path correct and complete?
   - Are there sufficient resources (CPU/Memory/Disk) allocated to the job?
   - Is the project available on the execution host?
   - Does the job submitter have read/writer permissions on the project directory and read/execute permissions on the installation directory?
   - Is the project locked?
3. Determine whether this is a standalone product issue.
   - Run Electronics Desktop on the machine outside of the scheduler and see if it opens and analyzes.
4. Examine outputs and logs.
   - Output of the LSF batch job. Obtain this using LSF commands: "`bacct -l <jobid>`"
   - Batch log (typically <projectname>.log, located in the project directory.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Enable additional debug logs using the steps below.

   In the job submission window, set the following environment variables:

   - ANSOFT_DEBUG_MODE = 1
   - ANSOFT_DEBUG_LOG = <*path to directory accessible by all machines in the cluster*>
   - ANSOFT_DEBUG_LOG_SEPARATE = 1
   - ANSOFT_LSF_LOG = <*path to a specific .log file in the directory set under ANSOFT_DEBUG_LOG*>
6. For each pair of machines between which remote analysis fails, run `ping remote-machine` and note the output.
7. For each machine in the network, dump network interfaces (for example, run `ifconfig -a`) and note the output.

Integration with PBS (Portable Batch System)

The PBSPro and PBS/Torque schedulers are only supported on Linux. Jobs may be submitted in any of the following ways:

- Job Submission GUI
- Using PBS commands (qsub, etc.) or the PBS gui (xpbs)

For information about supported schedulers, visit the Platform Support page. Then locate and click the following link:

Ansys 2025 R1 – Job Schedulers and Queuing Systems Support

PBS Job Management

You can use Ansys Electronics Desktop to submit batch jobs to PBS and monitor those jobs.

This involves the following steps:

1. Use **Tools** > **Job Management** > **Select Scheduler** to select PBS as the scheduler.
2. Use **Tools** > **Job Management** > **Submit Job** to submit a batch job to PBS.

   PBS-specific Settings:

   - On the **Compute Resources** tab, click the **Resource Selection Parameters** ellipses button (**...**) to specify either the **Queue** or **QueueAtServer** parameter.

     For **Queue**, you may select a queue for the job from the list of queues configured for the default server. Only queues that are enabled and that do not have the from_ route_only attribute set to true are listed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For **QueueAtServer**, you may specify a queue at the default server or at another server by entering text into this field in one of three formats:

- queue_name
- @server_name
- queue_name@server_name

The queue_name format submits with the name queue_name. The other formats submit with the name server_name. The *destination* value of the -q *destination* option on the qsub command line is the user-specified string. *This string will not be validated by the scheduler proxy library*.

Either **Queue** or **QueueAtServer** may be specified, but both may not be specified. If neither **Queue** nor **QueueAtServer** is specified, the job is submitted to the default queue at the default server.

3. Use **Tools** > **Job Management** > **Monitor Job** to monitor the job's progress.

Non Standard Installations for PBS

PBSPro

If the environment variable PBS_DEFAULT is set, then the value of this environment variable will be used as the name of the default server, instead of obtaining the default server name from the PBSPro configuration file. The default pathname of the PBSPro configuration file is /etc/pbs.conf. The environment variable PBS_CONF may be used to specify a different pathname for the PBSPro configuration file.

PBS/Torque

If the environment variable PBS_DEFAULT is set, then the value of this environment variable will be used as the name of the default server, instead of obtaining the default server name from the PBS/Torque server file. The name of the PBS/Torque server file is server_name, and it is installed in the TORQUEHOME directory. By default, TORQUEHOME is /var/spool/torque. To specify a different TORQUEHOME directory, the environment variable ANSOFT_TORQUEHOME should be set to the pathname of the desired directory.

PBS Limitations

General Limitations

- There is no support for GPUs when submitting jobs via the GUI.
- Support for PBSPro and PBS/Torque is only available on Linux; Windows is not supported.
- Staging of input or output files is not supported for jobs submitted using the GUI.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- All jobs submitted via the GUI are independent jobs. Neither job dependencies nor job arrays are supported.
- If the user specified server is not the default server, then there is no check for sufficient resources before submitting the job.
- If the user specified server is not the default server, then the limit on the number of tasks per node is ignored for both PBSPro and PBS/Torque. For PBS/Torque, only one task will be allocated for each node. For PBSPro, the scheduler may allocate any number of tasks to a node, provided that the node has sufficient cores for all of the tasks.
- For jobs submitted to a routing queue, the check for sufficient nodes and cores only verifies that there are sufficient nodes and cores associated with the server.
- The queue attributes “resources_max” and “resources_min” are not checked when determining whether there are adequate resources to run the job.

PBSPro Limitations

- The PATH in the submission user’s default environment must include the directory containing the PBSPro commands.
- Failover is not supported.
- HPC Basic Profile Jobs are not supported.
- Globus vnodes are not supported. When checking for sufficient nodes and cores for the job, only nodes of type PBS are considered.
- For jobs submitted to an execution queue, the only vnode attribute used to determine if a vnode is available to the job is the “queue” attribute.

PBS/Torque Limitations

- When checking for sufficient nodes and cores for the job, only nodes of type cluster are considered.
- The “exclusive” check box has no effect for PBS/Torque.
- For PBS/Torque, even if a job is submitted to an execution queue at the default server, there is no check for sufficient nodes and cores available to the queue. All of the server’s execution nodes are assumed to be available for the job.
- For PBS/Torque, there are significant limitations when submitting a job in which the number of tasks and number of cores per task are specified. Unlike PBSPro, there is no capability to specify that the cores should be allocated in “chunks”. Instead, the submission command includes the number of groups of nodes and the number of processors per node (ppn) for each node in the group. To determine the size of each group and the ppn setting for each group, the server nodes are examined from largest number of cores to smallest. This may not be optimal because some of these nodes may not be usable by the queue specified for the job, or because the nodes with the largest number of cores may be busy. Similar issues could occur for PBSPro, but they should be less likely, because only the nodes usable by the queue are considered.

Submitting Jobs Via Ansoftrsmservice on a Different Host

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Before starting the ansoftrsmsevice as a daemon on a job submission host, the 'Scheduler' section of the ansoftrsmservice.cfg must be specified.

This section contains two settings, 'SchedulerName', which must be set the string 'pbs', and 'ConfigString', which must be set to the pathname of the directory containing the PBSPro or PBS/Torque commands.

Here is an example ansoftrsmservice.cfg file, showing the format of this file and an example 'ConfigString' setting:

```
$begin 'AnsoftCOMDaemon'
$begin 'Managed COM Servers'
$end 'Managed COM Servers'
$begin 'Scheduler'
'SchedulerName'='pbs'
'ConfigString'='/share/pbs/default/bin'
$end 'Scheduler'
$end 'AnsoftCOMDaemon'
```

Submitting Ansys EM PBS Batch Jobs

The PBS qsub command may be used to submit Ansys EM batch jobs. The typical command format is:

qsub *qsub_args script*

where:

- *qsub_args* are the options of the **qsub** command,
- *script* is the pathname of the job script.

The job script is a shell script containing the Ansys batch command or commands to be run. If a batch command line contains any characters that are special to the shell running the script, then these special characters should be quoted, as needed. The job script may also contain PBS directives on lines before the first executable line of the script. Any **qsub** options on the command line will take precedence over the PBS directives in the job script.

When a PBS batch job is started, the job script runs as the job user in a new shell. In this shell environment, the path must include the directory containing the PBS commands.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

You should ensure that the PATH variable set in the shell startup script (e.g., .cshrc, .profile, .bashrc) includes the directory containing the PBS commands. For example:

```
export PATH=/opt/pbs/default/bin:$PATH
```

If the PATH variable is not set correctly, the job runs only locally, the batch log file shows the list of allocated hosts as empty, and the error file shows an error (sh: qstat: command not found.)

Further PBS directives need to be on top of the job script file. This is discussed in the PBS documentation.

Serial PBS Batch Jobs

In the PBS documentation, serial batch jobs are also called single-node jobs. In general, any job submitted without specifying the -l nodes=value command line argument, will run as a serial or single-node job.

See Monitoring PBS Batch Jobs for options that can facilitate monitoring of Ansys Electromagnetics batch jobs.

Parallel PBS Batch Jobs

In the PBS documentation, parallel batch jobs are also called multi-node jobs. When an Ansys Electromagnetics batch job is run as an PBS parallel job, the PBS scheduler will select the hosts for the distributed analysis job based on the qsub command line arguments, the PBS resource directives from the job script, and the status of the hosts when the job is run. The desktop process will be started on one of these hosts. The desktop process will obtain the list of hosts allocated to the job from the PBS scheduler, and start analysis processes on these hosts, as needed, using the PBS scheduler facilities. To run a PBS parallel job, the job must be submitted with a -l nodes=value **qsub** command line argument or with a -l nodes=value PBS directive in the job script.

Monitoring PBS Batch Jobs

You can monitor jobs through the Electronics Desktop user interface, or through the command line.

The suggestions below are for batch jobs run under PBS.

PBS qstat Command

The PBS **qstat** command may be used to display information on jobs and queues. In this section, several qstat command line options that may be used to monitor job progress are described.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **qstat -a** command displays information about all jobs in the system.

The **qstat -r** command displays information about all running jobs in the system.

The **qstat -s** command resembles the **qstat -r** command; the only difference is that a comment from the scheduler or batch administrator is also shown for each job.

The **qstat -au** *userid* command displays information about all jobs owned by user userid.

The **qstat -f** *jobid* command displays all available information about the job with id *jobid*.

See the PBS manual pages for more information.

Ansys EM -monitor Command Line Option for PBS

The Ansys EM **-monitor** command line option enables batch job output to the standard output and standard error streams. The warning, info, and progress messages are sent to the standard output stream. The error and fatal messages are sent to the standard error stream.

The PBS scheduler redirects the standard output and standard error streams of batch jobs to files specified in the **qsub -o** [*hostname*:]*pathname* and the **-e** [*hostname*:]*pathname* command line options, respectively. If either option is not specified, then the associated stream is redirected to the default file pathname for that stream.

The **qsub -j join** option controls whether the standard error stream for the job will be merged with the standard output stream for the job. A join value of **oe** indicates that the interleaved standard output and standard error will be sent to the standard output file or stream. A join value of **oe** indicates that the interleaved standard output and standard error will be sent to the standard error file or stream. A join value of n indicates that the standard output and standard error streams will not be joined. If the **qsub -j join** option is not specified, then the standard error and standard output streams will not be joined.

A user can monitor the progress of a job by checking the standard output file for progress, info and warning messages, and checking the standard error file for error and fatal messages.

qsub Arguments

The PBS **qsub** command has a large number of options for control of the submission process. In this section, we review the -l nodes=*value* command line option with Ansoft parallel batch jobs.

This option or directive has the following format:

`-l nodes=`*`node_spec`* `[+`*`node_spec...`*`][#`*`suffix`*`]`

where *node_spec* is one of the following

`nodename[:`*`pc_spec`*`[:`*`pc_spec...`*`]]`

Host name of the specified node, followed by optional **ppn** or **cpp** specifiers.

`[`*`N`*`][:`*`property`*`[:`*`property...`*`]][:`*`pc_spec`*`[:`*`pc_spec...`*`]]`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optional number of nodes, followed by optional node properties, followed by optional **ppn** or **cpp** specifiers. If the number *N* is omitted, then the default value of 1 host is used.

Here, the optional **ppn** or **cpp** specifiers *pc_spec* are of form:

`ppn=X`

Number of processes (tasks) per node. Default is 1 if not specified.

`cpp=Y`

Number of CPUs (threads) per process. Default is 1 if not specified.

The optional global suffix, #suffix, which applies to all hosts has one of the following values:

`#excl`

This suffix requests exclusive access to the allocated nodes.

`#shared`

This suffix requests shared access to the allocated nodes.

The total number of requested processes is determined by adding up the product of the number of nodes and the number of processes per node for each *node_spec*. In general, this should match the number of distributed engines specified in the Ansys Electromagnetics desktop -Machinelist num=*num_distributed_engines* command line option.

The number of CPUs per process (**cpp**) specified in the PBS **qsub** command line or in the PBS directives in the script file should generally match the number of processors per engine specified in the Desktop **-batchoptions** value.

See the PBS documentation for a complete list of options for the **bsub** command, and further information on running multi-node jobs.

Example PBS qsub Command Lines

All of the following examples show how to submit Linux hfss jobs on PBS, but similar command lines and job scripts will work for all Ansys EM products. Most of the following examples are PBS "Single-node jobs." The last example is a PBS "multi-node jobs"; this example demonstrates how to specify the allocation of threads, tasks and nodes to a job.

Serial job:

```
qsub ~/pbs_scripts/OptimTee.sh
```

Job Script File:

```
#!/bin/sh
/opt/ansys_inc/v251/AnsysEM/ansysedt -ng -BatchSolve
~/projects/OptimTee.aedt
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Serial job that needs a minimum of 4GB memory and two hours of real (wallclock) time:

```
qsub ~/pbs_scripts/OptimTee.sh
```

Job Script File:

```
#!/bin/sh
#PBS -l walltime=2:00:00
#PBS -l mem=4gb
/opt/ansys_inc/v251/AnsysEM/ansysedt -ng -BatchSolve
~/projects/OptimTee.aedt
```

Multi-processing job using 4 cores:

```
qsub ~/pbs_scripts/OptimTee.sh
```

Job Script File:

```
#!/bin/sh
#PBS -l ncpus=4
/opt/ansys_inc/v251/AnsysEM/ansysedt -ng -BatchSolve -batchoptions
-machinelist num=4
~/projects/OptimTee.aedt
```

- The #PBS -l ncpus=4 directive indicates that four cores or CPUs are allocated to this job.
- The -batchoptions option indicates that Ansys Electronics Desktop should use four cores for multi-processing.

Distributed processing job using 4 engines on a single host:

```
qsub ~/pbs_scripts/OptimTee.sh
```

Job Script File:

```
#!/bin/sh
#PBS -l ncpus=4
/opt/ansys_inc/v251/AnsysEM/ansysedt -ng -BatchSolve -Distributed -
machinelist num=4
~/projects/OptimTee.aedt
```

- The #PBS -l ncpus=4 directive indicates that four cores or CPUs are allocated to this job.
- The -Distributed option indicates that this is a DSO job, so that multiple engines will be started. Because 4 cores are allocated to the job, the job will run 4 engines. The -Distributed option may now have additional options, such as includetypes=xxx, excludetyeps=xxx, maxlevels=n, and numlevel1=n, where n indicates and integer, and xxx indicates a list of distribution types or "default".

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Distributed processing and multi-processing job using 8 cores on two nodes, running 4 engines (two per node) with 2 cores for multi-processing:

```
qsub ~/pbs_scripts/OptimTee.sh
```

Job Script File:

```
#!/bin/sh
#PBS -l nodes=2:ppn=2:cpp=2#excl
/opt/ansys_inc/v251/AnsysEM/ansysedt -ng -BatchSolve -Distributed
-machinelist num=4 -batchoptions ~/projects/OptimTee.aedt
```

- The PBS directive **#PBS -l nodes=2:ppn=1:cpp=2#shared** indicates that two nodes are requested [2], two processes (engines) run on each node [ppn=2], and each process will use two cores [cpp=2]. The hosts allocated to this job may not be used for any other jobs while this job is running [#excl].
- The **-machinelist num=4** option indicates that this is a DSO job and that a total of four engines will be started. This option is required for all batch jobs.

Integration with Ansys Remote Simulation Management (RSM)

Ansys Electromagnetics supports its own Remote Simulation Management (RSM) software along with other High Performance Computing (HPC) software management programs (see High Performance Computing (HPC) Integration).

When do you need RSM?

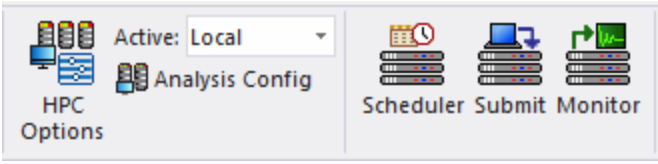
RSM is generally required if you want to run remote or distributed simulations. However, if you have a separate scheduling system that Ansys Electromagnetics supports, and you plan to run batchsolve simulations only, you may not need to install RSM. For details of installation and configuration of RSM, see the *Ansys Electromagnetics Installation Guides*.

Job Management UI for RSM

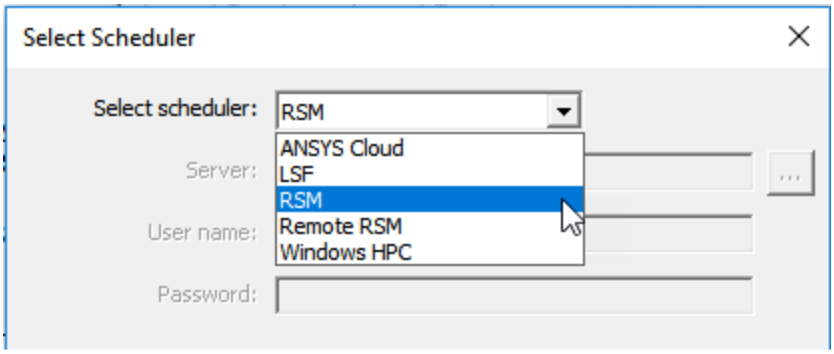
You can use the Job Management UI to submit batch jobs to RSM. The Job Management UI is accessed by running Ansys Electronics Desktop on the designated 'Postprocessing node' of the cluster. The Desktop provides UI commands for Scheduler selection, Job submission and Job monitoring/control. You access the Scheduler User Interface by clicking **Tools** > **Job Management** > **Select Scheduler...**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also select the **Simulation** tab of the ribbon, and click the **Scheduler** icon.



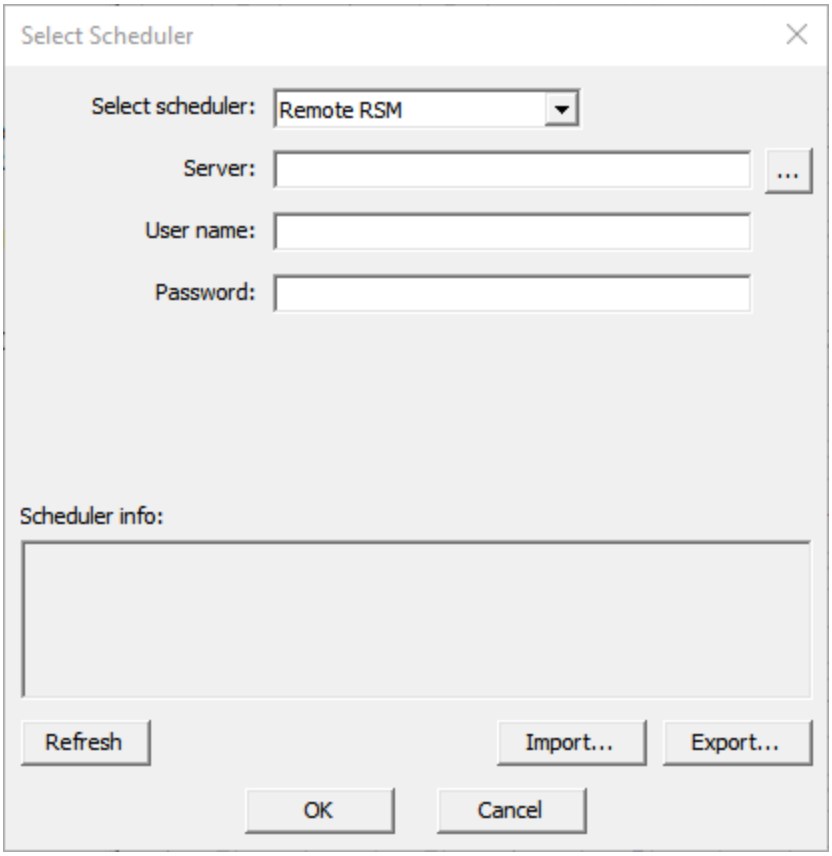
This displays the selection dialog box. The **Select scheduler** drop-down menu lists potential schedulers (which can include RSM, LSF, Windows HPC, or SGE, depending on the environment).



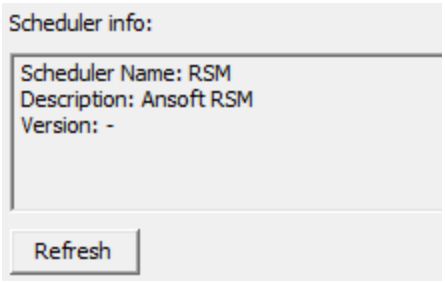
If you select a scheduler that is not supported in your environment, you receive a warning.

If you select Remote RSM and your environment has been configured , you can select a computer, user name, and password.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After selecting a scheduler, click **Refresh** to display information for that scheduler.



Once you have selected a scheduler supported in your environment, you can go through the following steps to submit a batch job.

1. Set up and prepare model on local workstation.
2. Copy the input project (or folder, if the project references external files) from a personal workstation to a shared-drive on cluster (say project is copied to /home/projects/spool/test.adsn).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the RSM environment, you are required to specify a machine-list. See: HPC and Analysis Options. For example, say the machine-list is: 3 cores from 'm1' and 3 cores from 'm2', for a total of 6 engines. You select the list on the **Compute Resources** tab of the **Submit Job to RSM** window, as described below.

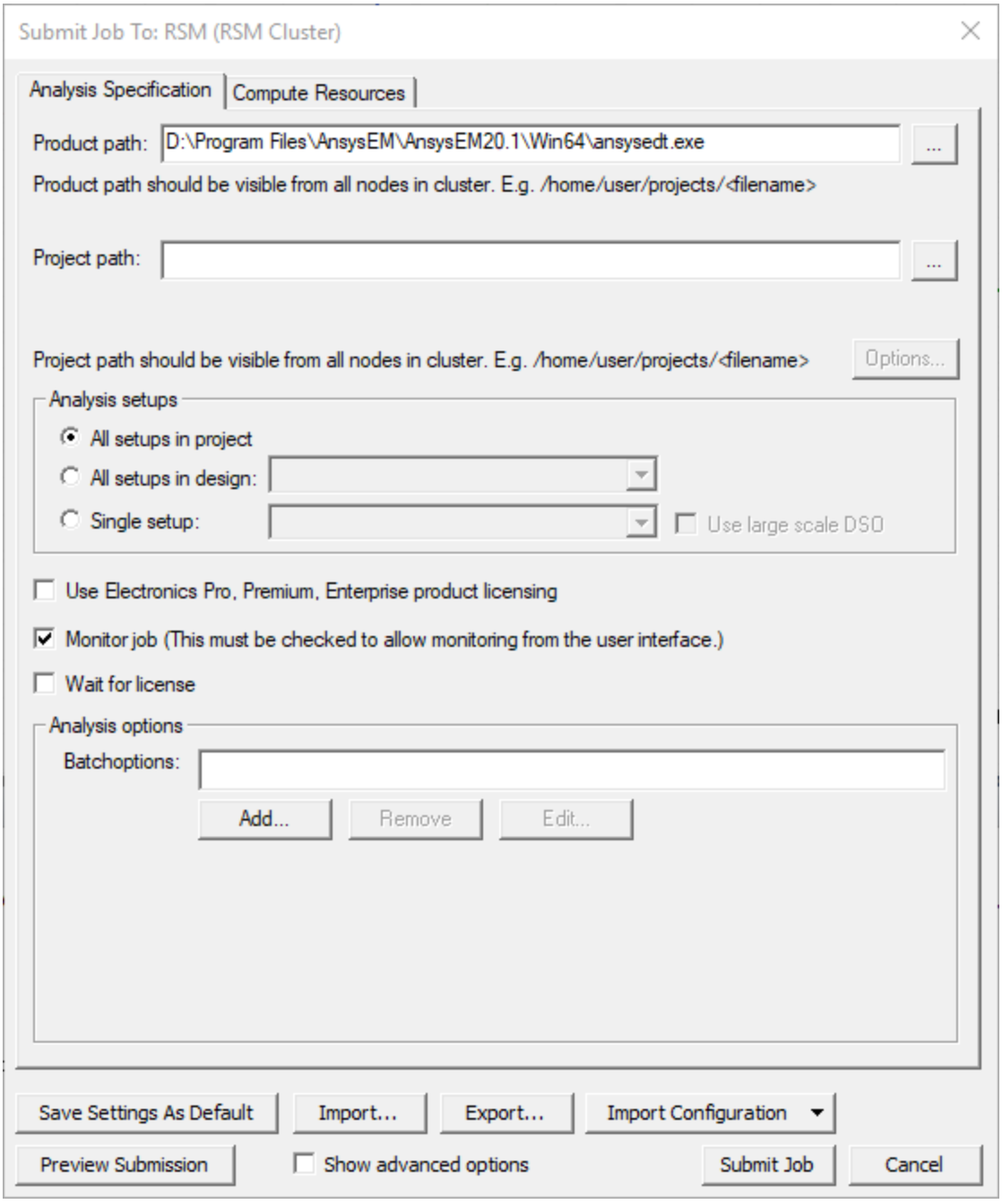
3. Open a remote-desktop session (or equivalent such as vnc session) on the node corresponding to the first machine of job's machine-list, 'm1' in this case. Launch Desktop graphically on 'm1'.
4. After setting the job submission node, select **Tools** > **Job Management** > **Submit Job...** or **Project** > **Submit Job...** or **[ProductName]** > **Submit Job...** to open the **Submit Job To:** window. You can also access **Submit Job** from the shortcut menus for the Project Name, Design name, Analysis Setup, or Optimetrics Setup.

   The **Submit Job To:** window contains two tabs:

   - **Analysis Specification**– specify the Product path, Project name, the setups, and analysis options such as batchoptions, or, for advanced users, Environment variables. If you select the Analysis or Optimetrics setup, the Analysis Specification is pre-populated.
   - **Compute Resources** – specify the amount of compute resources and how to select specific resources from those available, and automatic settings, if supported by the design types.

   The standard Job Submission window displays.

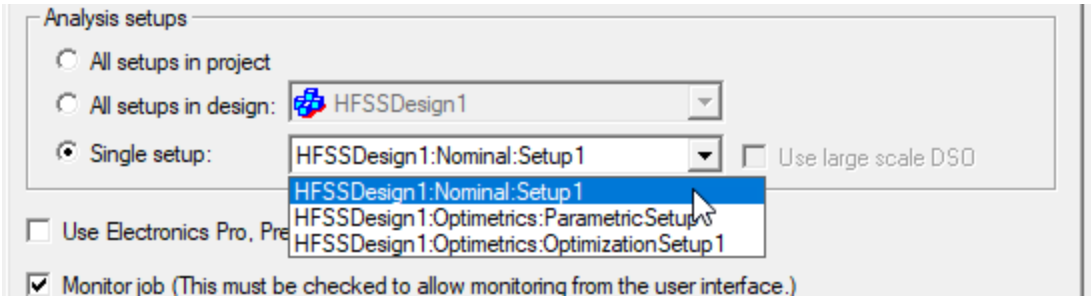
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Use the ellipsis button [**...**] to open a browser to select the project. The project can be an archive.
6. In the **Analysis setups** area, you can select radio buttons for **All setups in the project**, **All setups in the design**, or a **single setup**. For instance, the OptimTee example includes setups for Nominal, Parametric, and Optimization. If you accessed the **Submit Job** window from the right-click menus for Setup or Optimetrics Setup, this field can be

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

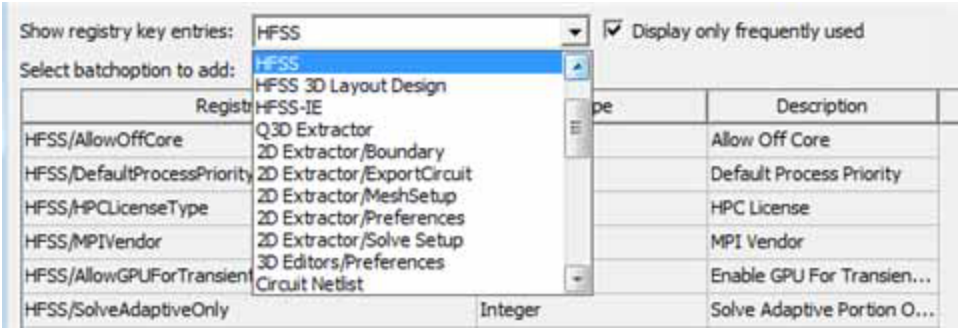
pre-populated.



For Parametric setups, you have the option to select **Use Large Scale DSO**. For details on how and when you use this feature, see Job Management Interface for Large Scale DSO.

7. The Analysis options include check boxes for Monitoring the job, whether to wait for a license, and a field for adding Batchoptions via a graphical interface, or as text.
    - If you intend to monitor the job through a user interface, you must select **Monitor job**. You can then monitor this job through the **Tools** > **Job Management** > **Monitor Jobs...** command or by checking the dialog that opens when you submit the job.
    - The Batchoptions field allows you to add additional -batchoptions parameters, either as text, or by using a dialog with selection menus. Click the **Add** button to view the **Add Batchoption** dialog box.

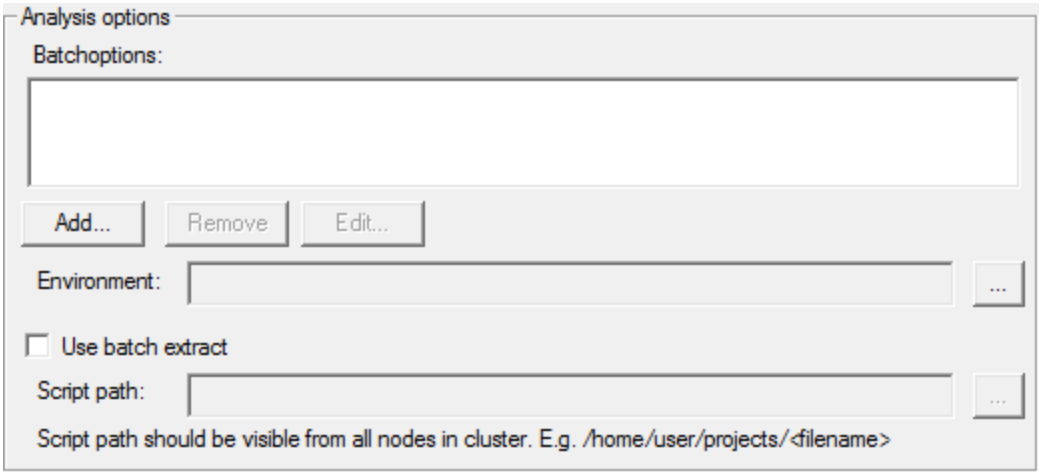
The **Show registry key entries** field lets you filter the entries displayed, by means of drop-down menu selection, and a check box to **Display only frequently used entries**.



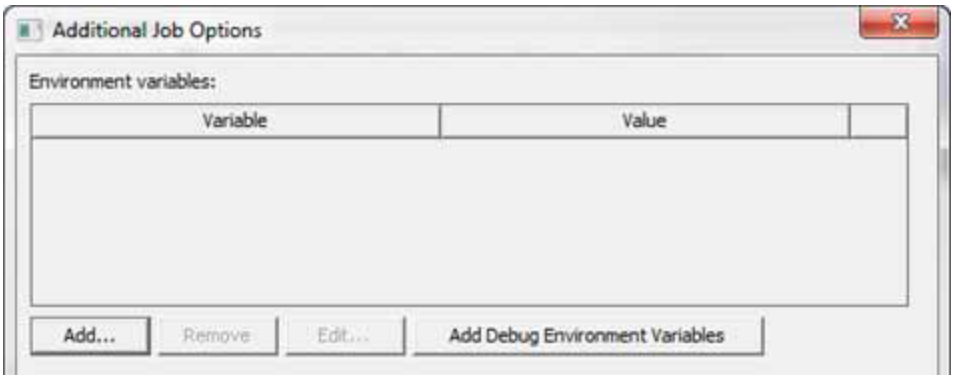
- When you have selected a batchoption, you can type the value in the field, and click the **Add** button to add the option to the batchcommand.
- In the **Submit Job To:** window, you can enable **Show advanced options** to display additional fields for environment variables, and whether to **Use batch**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

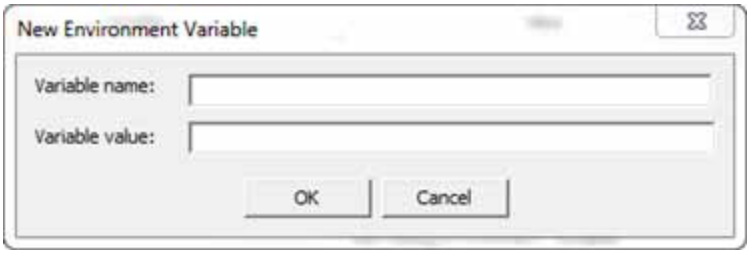
extract.



The Environment field is for environment variables, for instance, for debugging features or other variable controlled features. Click the ellipsis [**...**] button to open a dialog box for **Additional Job Options**.

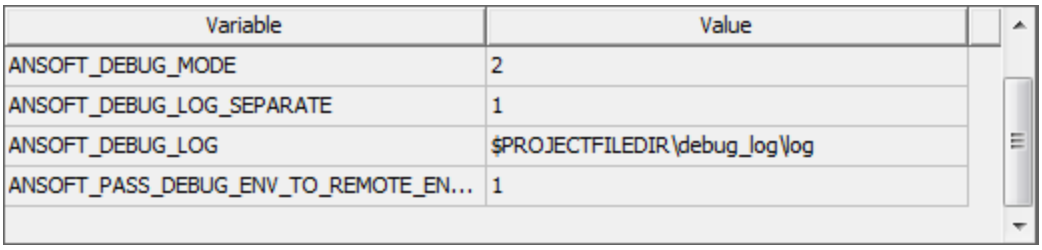


The **Add...** button opens a **New Environment Variable** dialog box in which you can include a variable name and value.



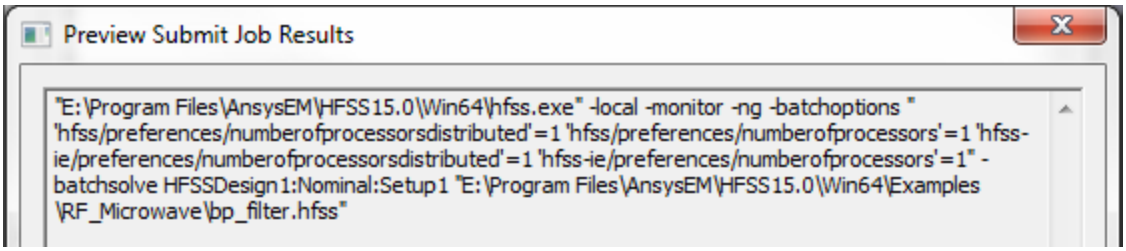
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clicking the **Add Debug Environment Variables** button automatically adds a set of debug variables. This can be useful in working with Ansys Application Engineering support.



Selecting a variable in the dialog enables the **Remove** and **Edit** buttons. The **Edit** button opens a dialog box where you can change the variable and value.

8. To see the command-line to be submitted to the scheduler, click **Preview Submission.** This opens a dialog box showing the command to be sent to the scheduler.

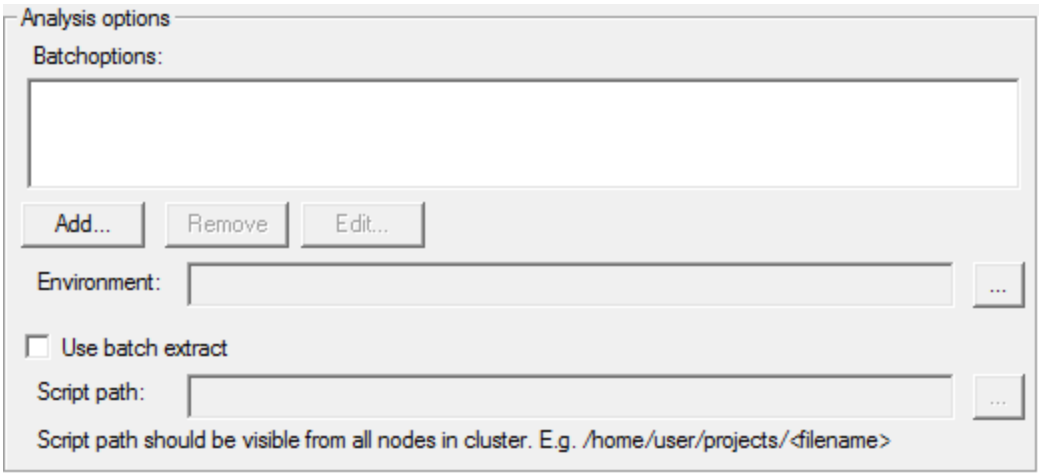


The text can be copied to the clipboard, if desired.

Use Batch Extract for RSM

Selecting **Show advanced options for RSM** also shows the Use batch extract fields.

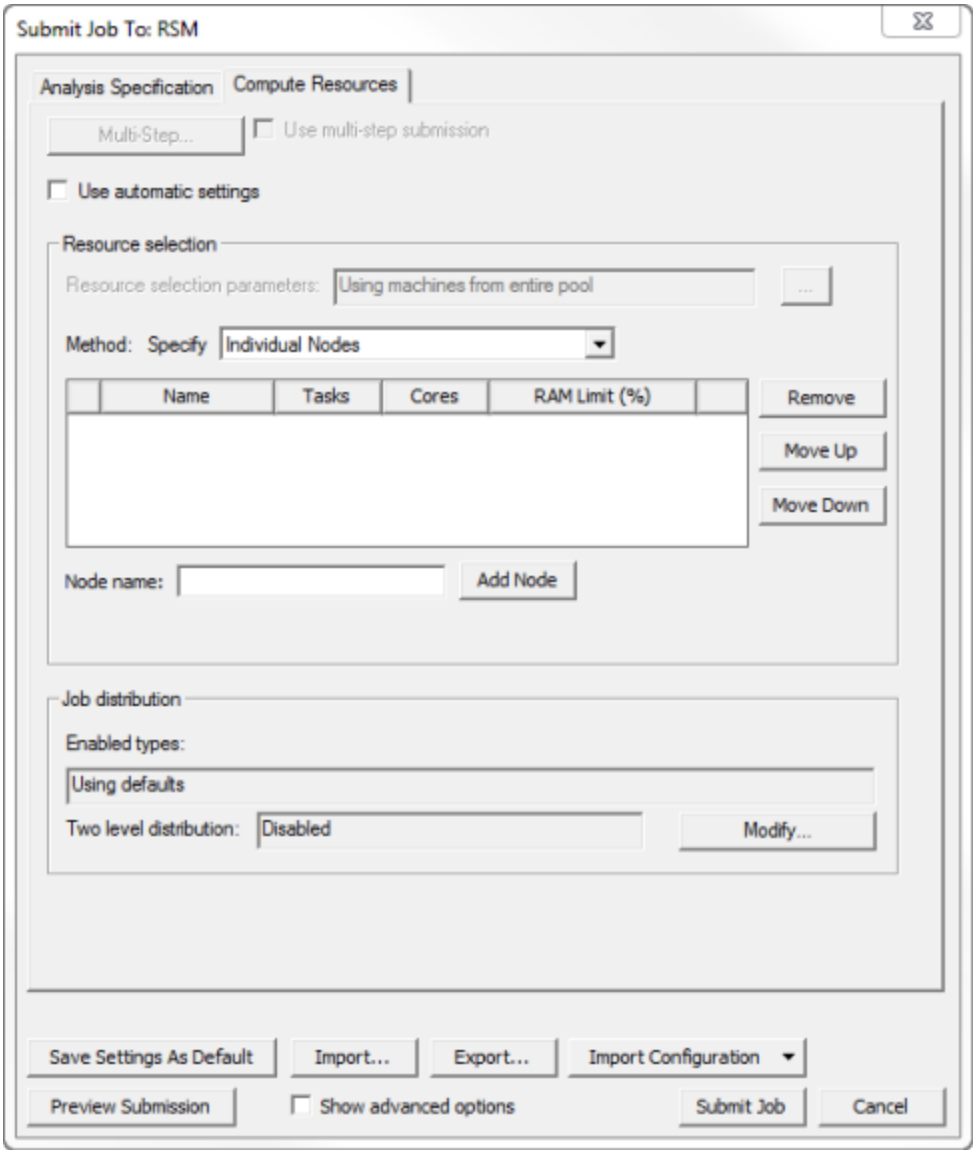
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



See the discussion on Running Ansys Electronics Desktop from a Command line for a discussion of the solve information available through batch extract.

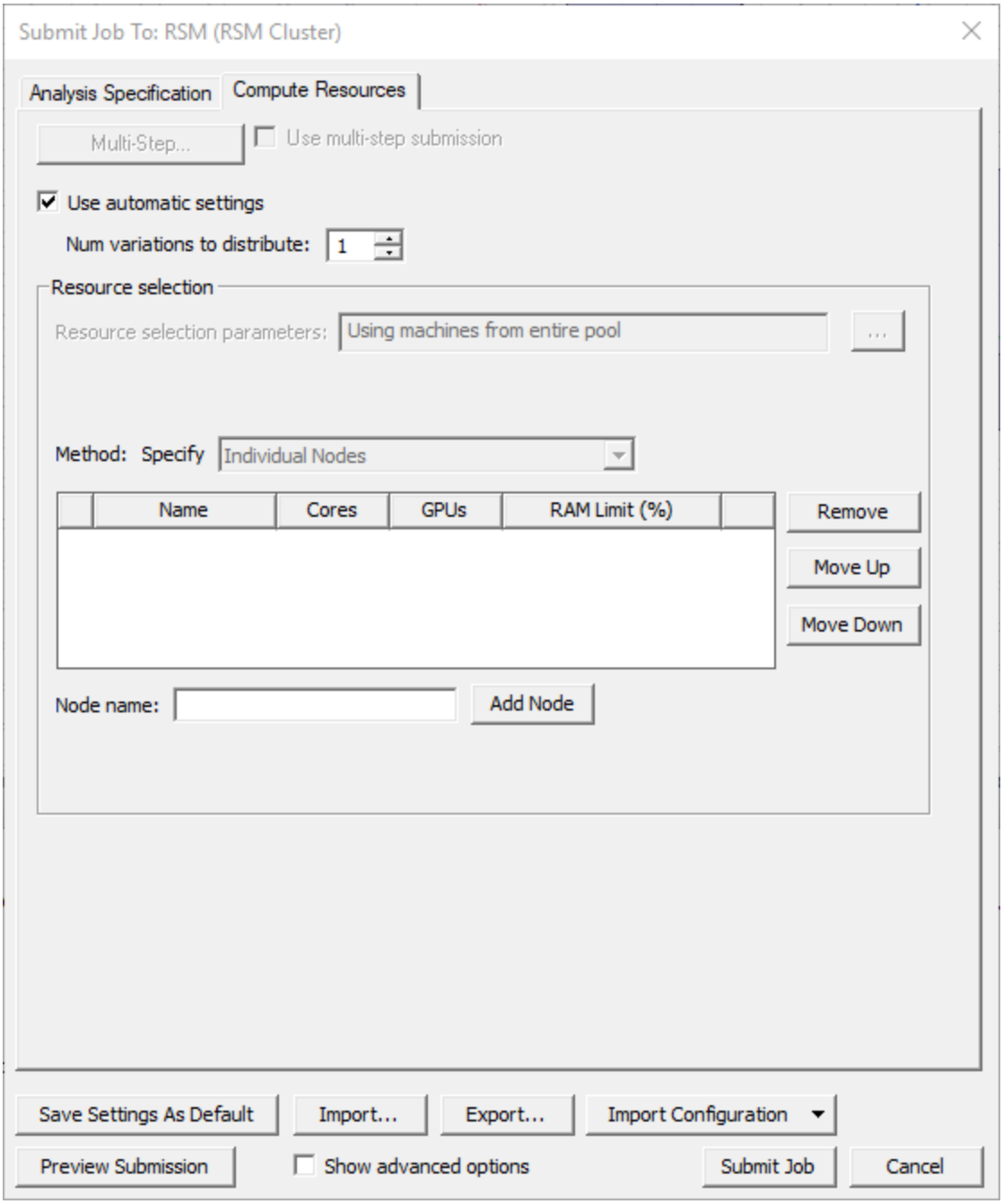
9. The **Compute Resources** tab of the **Submit Job to: RSM** window displays other parameters. Depending on the resources available for a scheduler environment, some of the fields may be disabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With **Use automatic settings** selected, the **Job distribution field** is removed and the **Use automatic settings** check box and **Num variations to distribute** field appear.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

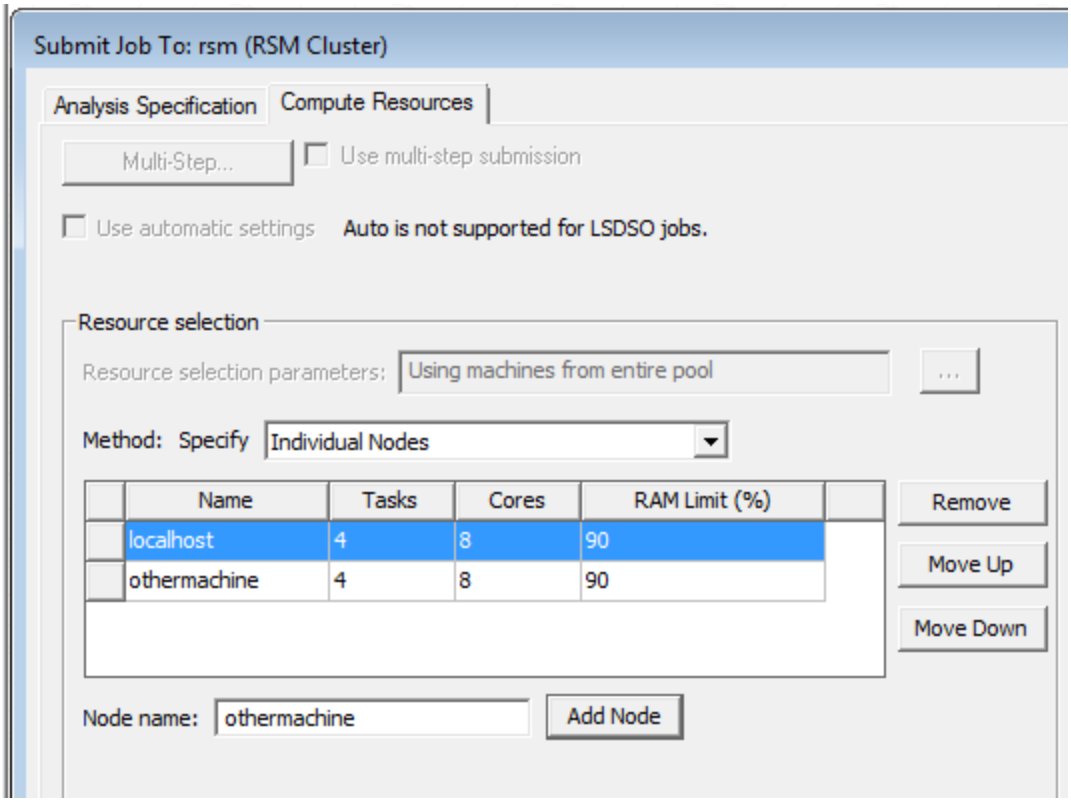
Note:

If you select **Use automatic settings** with **Num variations to distribute** set to 1, Optimetrics variations will be solved sequentially. Other distribution types will be distributed automatically. It does distribute frequencies, domains, and use of multiple level domains. If you set **Num variations to distribute** to 2 or more, Optimetrics variations will be solved in parallel. Other distribution types will be distributed automatically.

Otherwise:

- Specify node list

  Here you can specify a node list. In a computing environment where the available cores are not uniform, you can use this to control which resources your job will use. For use with Large Scale DSO for RSM, for jobs that are submitted from job submission panel, localhost must be the first node in the resource selection node list, other wise LSDSO solve with RSM will fail.

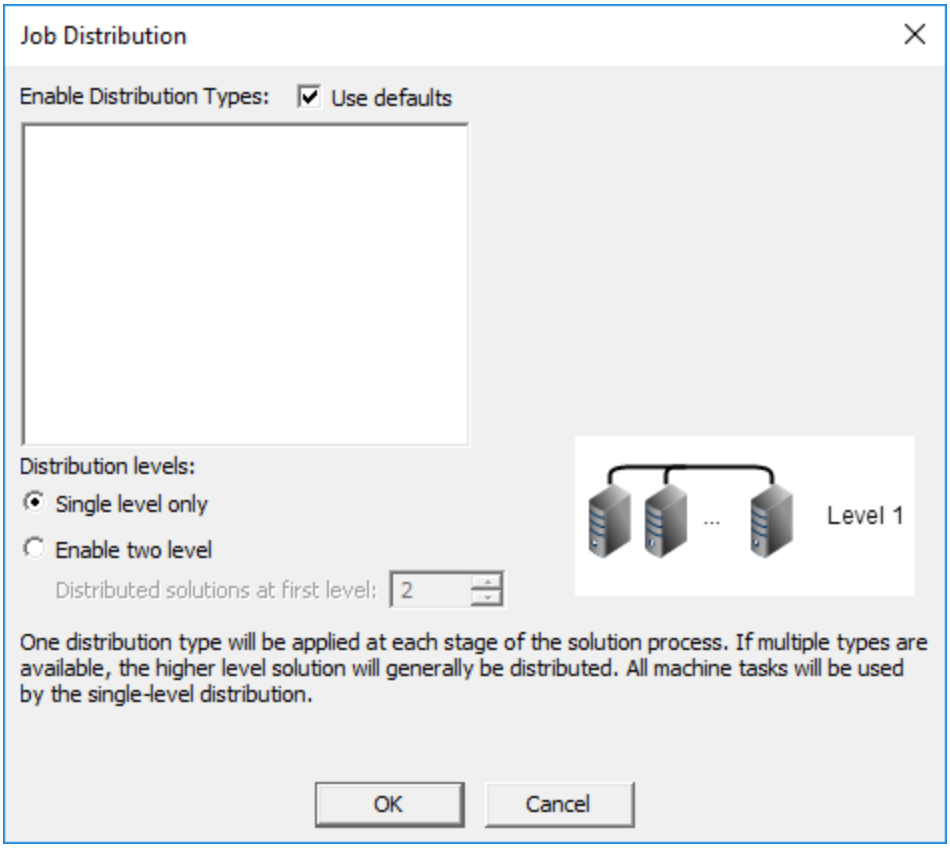


Job Distribution

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you disable User automatic selection, you can modify the Job distribution settings.

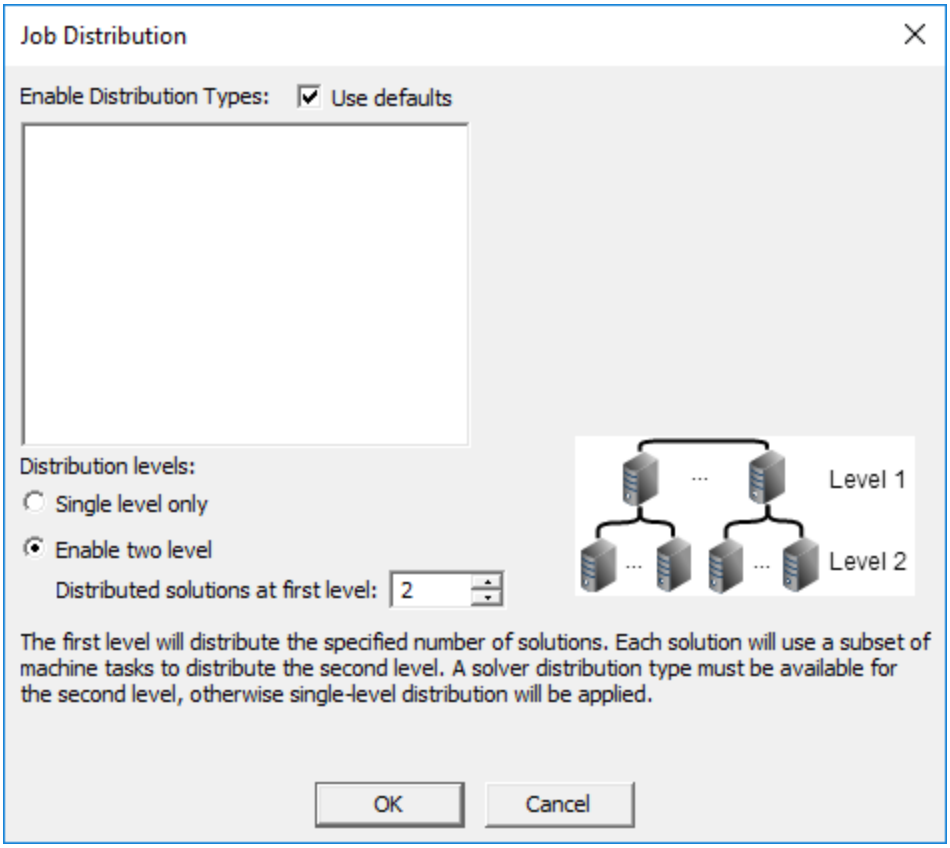
- Single level or two level distribution (*single level* is the default). Click **Modify** to display the *Job Distribution* dialog box and select the **Enable two level** option if appplicable and desired.



Enabled distribution types can be modified here.

Second-level distribution operates within DSO. If available and enabled you can specify a number of engines for level 1.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



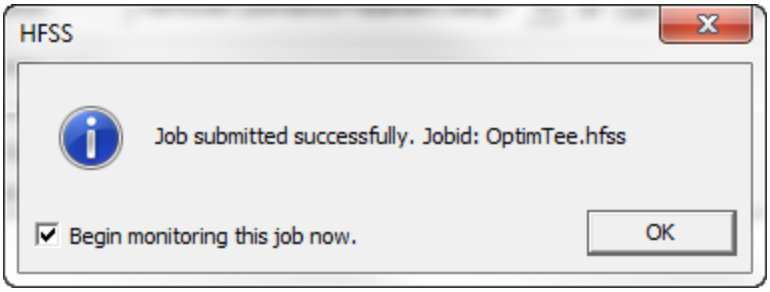
In response to a set of minimal constraints, the Scheduler may increase the resources assigned beyond the minimal values in order to meet the full set of requirements. For example, if you specify 7 distributed engines, with two processors per engine, and also limit the number of engines per node to 4, the scheduler may increase the number of cores used in order to meet the limit specified for engines per node. Notice that a preview of the Submit Job Results shows the number of resources assigned, and that the scheduler generated code includes an MPI specification.

1. To submit the command with the specified parameters, click **Submit Job**.

> **Note:**
>
> The RSM environment does not support for queuing, so **Submit Job** will immediately start running the job.

A dialog box displays. You can select **Begin monitoring this job now** and click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. You can monitor this job either automatically (by selecting the option above) or through the **Tools** > **Job Management** > **Monitor Jobs...** command. For more details, see Monitor Jobs window.

Process for Changing the Listening Port used by AnsoftRSM Service

To change the listening port used by the AnsoftRSMService, you need to change the configuration file, ansoftrsmservice.cfg, as follows:

You must specify the ListenPort within a 'CommDetails' block, which must be within a 'Default:CommDetails' block, which must be within the top level block of the file, the 'AnsoftCOMDaemon' block. The following example shows the listen port changed from 32958 to 32957, with these blocks at the beginning of the file:

```
$begin 'AnsoftCOMDaemon'
  $begin 'Default:CommDetails'
    $begin 'CommDetails'
      ListenPort='32957'
    $end 'CommDetails'
  $end 'Default:CommDetails'
  . . . .
$end 'AnsoftCOMDaemon'
```

For the second level block, ensure that there is a single colon character and no spaces or tabs separating the two parts of the block name 'Default:CommDetails'. The third level block, with name 'CommDetails' is also required. Use caution when editing this file by hand, because any typos in the block or value names may cause the data to be ignored.

Changing the AnsoftRSMService Listening Port

For Remote Analysis or Distributed Analsysis, processes may need to be started on multiple hosts. If the Ansys Electromagnetics Desktop needs to start a process on a remote host, the AnsoftRSMService is used to start these remote processes. By default, the AnsoftRSMService listens for socket connections from the Ansys Electromagnetics Desktop on port 32958.

This section describes how to change the port number used by the AnsoftRSMService.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To change the port number, both the AnsoftRSMService and the Ansys Electromagnetics Desktop must be configured to use the new port number. The same port number must be used for Ansys Electromagnetics Desktop and for the AnsoftRSMService process running on each host used for the analysis.

AnsoftRSMService Configuration

The AnsoftRSMService port number is configured in the ansoftrsmservice.cfg configuration file. This configuration file is located in the platform specific subdirectory of the RSM installation directory.

The default location of this directory is "C:\Program Files\ANSYS Inc\v251\EM_RSM" on Windows.

The default location of this directory is /opt/ansys_inc/v251/EM_RSM on Linux.

To modify the AnsoftRSMService configuration, first stop the ansoftrsmservice, then modify the ansoftrsmservice.cfg configuration file, then restart the ansoftrsmservice.

> **Note:**
>
> When using Linux, root maintains exclusive control over systemd-managed services. Once the startonboot command has been given, only root users can start and stop AnsoftRSMService, using the commands:
>
> ```
> sudo ansoftrsmservice start
>
> sudo ansoftrsmservice stop
> ```

The beginning of the configuration file should appear as follows:

```
$begin 'AnsoftCOMDaemon'
$begin 'Default:CommDetails'
$begin 'CommDetails'
'ListenPort'='32958'
$end 'CommDetails'
$end 'Default:CommDetails'
```

If there are additional lines between the following two lines, then they should not be modified:

```
$begin 'AnsoftCOMDaemon'
$begin 'Default:CommDetails'
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To change the port number, modify the ListenPort setting within the single quotes from 32958 to the desired port number. The single quotes should not be removed or changed.

For previous versions of the software, the ansoftrsmservice.cfg file may not contain the lines:

```
$begin 'Default:CommDetails'
$begin 'CommDetails'
'ListenPort'='32958'
$end 'CommDetails'
$end 'Default:CommDetails'
```

If these lines are not present, then add them to the ansoftrsmservice.cfg file after the first line of the file, and change the ListenPort to the desired port number. For example, to change the port number to 32000, the beginning of the ansoftrsmservice.cfg file should look like the following after the changes:

```
$begin 'AnsoftCOMDaemon'
$begin 'Default:CommDetails'
$begin 'CommDetails'
'ListenPort'='32000'
$end 'CommDetails'
$end 'Default:CommDetails'
```

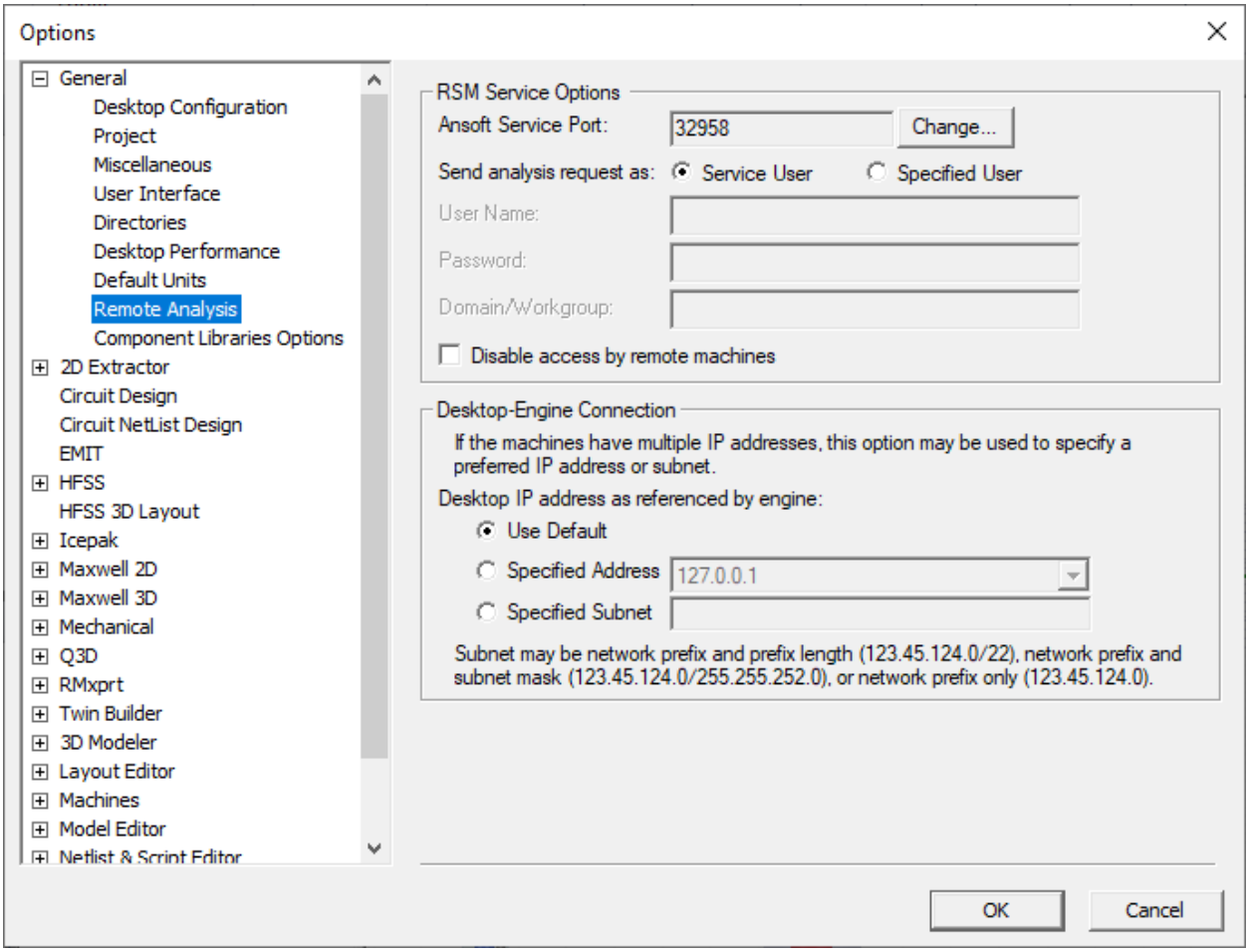
Ansoft Electromagnetics Desktop Configuration

The port number for connecting to the AnsoftRSMService is configured from the **General Options** window.

To access these options:

- Select **Tools** > **Options** > **General Options** to open the **General Options** window.
- If it is not already expanded, expand the **General** tree and click **Remote Analysis**.

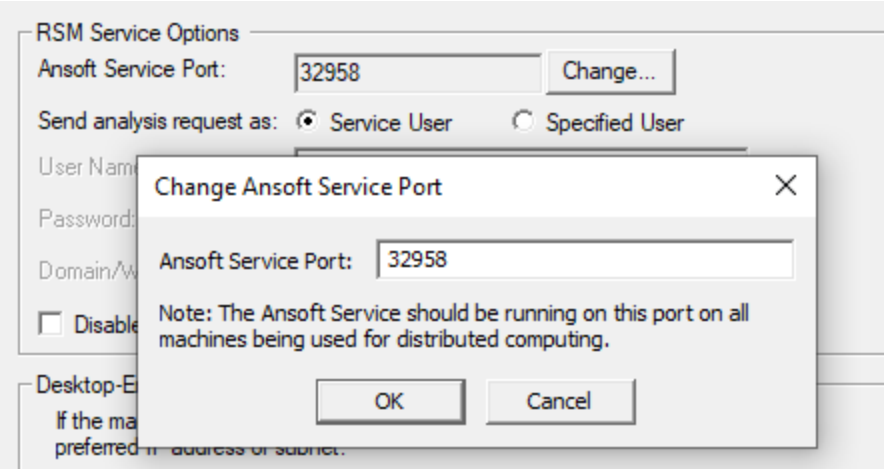
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Under **RSM Service Options**, the **Ansoft Service Port** field shows the current AnsoftRSMService port number.

1. Click **Change** to modify this setting. This opens the **Change Ansoft Service Port** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Enter the new port number in the **Ansoft Service Port** field.
3. Click **OK** to close the dialog box and accept the new port number.
4. Click **OK** to close the **General Options** window and accept the changes.

Integration with SLURM Scheduler (Linux Utility)

For Linux customers, Electronics Desktop supports the SLURM (Simple Linux Utility for Resource Management) scheduler for job submission, monitoring, and control (abort). In addition, this provides Windows-to-Linux support for the same functionality. Using SLURM as a starting point, customers should be able to develop their own integrations for other schedulers. Electronics Desktop can check job states in SLURM scheduler within the current job monitoring GUI. Persistence mode provides more efficient management between Desktop and scheduler proxies. Windows-to-Linux submission also provides convenient access to generic scheduler in Linux environments through Remote RSM Service.

When you submit a job to SLURM scheduler or Remote RSM to a SLURM cluster, you can use their own job options to override job options in the **Scheduler Options** tab. If you want to do that, the override job options must have "--export=None".

For additional information on supported schedulers, visit the Platform Support page. Then, locate and click the following link:

Ansys 2025 R1 - Job Schedulers and Queuing Systems Support

See also: Integrating Ansys EM Tools with Third Party Schedulers.

Configuration of the generic scheduler with SLURM

Requirements

Files must be under ${ANSYS_EM_INSTALL_DIR}/schedulers directory. Desktop and RSM service use the same required file list below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The required files are listed here:

- ${ANSYS_EM_INSTALL_DIR}/schedulers/libgeneric_scheduler.so
- ${ANSYS_EM_INSTALL_DIR}/schedulers/proxies.cfg
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/persistence.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/SLURM_GetInfo_linux.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/SLURM_GetRunInfo_linux.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/SLURM_SubmitJob_linux.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/SLURM_ControlJob_linux.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/SLURM_GetJobState_linux.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/SLURM_LaunchProcess_linux.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/utils/ComputeResourceList.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/utils/ErrorAndDebug.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/utils/__init__.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/utils/RunCommand.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/utils/SiteCustomize.py
- ${ANSYS_EM_INSTALL_DIR}/schedulers/scripts/utils/slurm_srun_wrapper.sh

Environment Variables

There are some environment variables for the Generic Scheduler proxy. All environment variables apply on both Desktop and RSM Service.

`ANSYSEM_GENERIC_SCRIPT_DIR`

This variable adds a customized directory to be searched for proxy scripts. The path can be any form of following addresses. Persistence script path will not change.

- absolute address to any folder with read access
- relative address to folder under the Electronics Desktop installation directory
- relative address to folder under the schedulers directory
- relative address to folder under the schedulers/scripts directory

`ANSYSEM_GENERIC_EXEC_PATH`

This variable adds customized directories to be searched for proxy executables. Mono executable and IronPython executable should be specified when they are not in the Ansys Electronics Desktop installation. This environment variable must be set for RSM Service.

`ANSYSEM_GENERIC_PROXY_PERSISTENCE`

If set, this variable redirects the persistence script location to be used; otherwise, the default persistence script path is used.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`ANSYS_EM_EXEC_DIR`

Set this variable to the same value with ANSYS_EM_INSTALL_DIR (not the RSM service path). To ensure the RSM service can accurately locate the essential script files required for MPI integration, it is necessary to set this variable within the RSM service environment.

Configuration File

The configuration file "proxies.cfg" is located at ${ANSYS_EM_INSTALL_DIR}/schedulers/. The file can have multiple scheduler configurations. The file content and explanation are below.

[slurm] ***# Unique scheduler name***

DisplayName=SLURM ***# Display name in GUI***

 # Proxy Scripts

SubmitJob=scripts/SLURM_SubmitJob_linux.py

ControlJob=scripts/SLURM_ControlJob_linux.py

GetSchedInfo=scripts/SLURM_GetInfo_linux.py

GetJobState=scripts/SLURM_GetJobState_linux.py

GetRunInfo=scripts/SLURM_GetRunInfo_linux.py

LaunchProcess=scripts/SLURM_LaunchProcess_linux.py

 # Persistence Mode

Persistence=Yes

Persistence Mode

Persistence mode uses a persistent process to communicate with Ansys Electronics Desktop and proxy scripts. If persistence mode is disabled, Ansys Electronics Desktop uses proxy scripts directly. Enabling persistence mode may help speed up communication with proxy scripts. **Note**: disabling persistence will cause large delays in GUI functionality. The primary use case of disabling persistence is to support troubleshooting.

- To enable persistence mode, make sure "Persistence=Yes" is in configuration file before launching Ansys Electronics Desktop.
- To disable persistence mode, make sure "Persistence=No" is in configuration file before launching Ansys Electronics Desktop.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Remote RSM Service Support

In ansoftrsmservice.cfg, fill your scheduler information from configuration file proxies.cfg as below. `SchedulerName` must be generic. Change `"Proxy"` as needed.

```
$begin 'Scheduler'
  'SchedulerName'='generic'
  'ConfigString'='{"Proxy":"slurm"}'
$end 'Scheduler'
```

Then, set the environment variable “ANSYSEM_GENERIC_EXEC_PATH” with Mono and IronPython path. Multiple paths can be filled with delimiter “:”. Ansys Electronics Desktop has built-in components in AnsysEM installation path, and you can replace the installation path like:

```
ANSYSEM_GENERIC_EXEC_PATH= <INSTALL_DIR>/ansys_
inc/v251/AnsysEM/common/mono/Linux64/bin:<INSTALL_DIR>/ansys_
inc/v251/AnsysEM/common/IronPython
```

Also set environment variable “ANSYS_EM_GENERIC_COMMON_TEMP” to a directory widely accessible to users using Remote RSM Service on the cluster. This environment variable specifies a temporary folder to use. Thus, it must be accessible for most users.

```
ANSYS_EM_GENERIC_COMMON_TEMP=/tmp
```

Then, launch **ansoftrsmservice** in the AnsysEM RSM installation folder. (Be sure not to confuse AnsysEM (Ansoft) RSM with Ansys RSM since these are two very different things.) It is recommended that the user for **ansoftrsmservice** should be a dedicated service user account for only running **ansoftrsmservice**. This account should have limited permissions for security reasons. The service user should not have a login. It is recommended to disable login, so that the only way to run as the user is to “su” to that user from the root (or other accounts with sudo permissions). Other users can use any account credentials in the Desktop Select Scheduler GUI to submit jobs as the requested user.

Using the generic scheduler with SLURM

To select the scheduler and get scheduler information:

1. Click **Tools** > **Job Management** > **Select Scheduler...**, or select the **Simulation** tab on the ribbon, and click the **Scheduler** icon.

   This opens the **Select Scheduler** dialog box.

2. Select **SLURM** from the drop-down menu; or, if the Remote RSM service is used, fill in scheduler master node address, and use valid account credentials.
3. Click **Refresh**.

   Scheduler Information should show.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **OK** to select the scheduler.

   Submit Jobs and Monitor Jobs with SLURM

   Job submission and monitoring follow the same process and method as using other schedulers. Currently, submission using a node-list is not supported.

5. Use **Tools** > **Job Management** > **Monitor Job** to monitor the job's progress.

Using MPI Tight Integration

By default, jobs would be submitted with SSH as remote spawn command method. If MPI tight integration preferred, you should submit the job with an additional batch option setting in the Submit Job Browser:

Enable MPI Tight Integration:

```
<design-type>/RemoteSpawnCommand = "Scheduler"
```

Enable SSH:

```
<design-type>/RemoteSpawnCommand = "SSH"
```

Using the Command Line to Submit HPC Jobs

Ansys Electronics Desktop can be run from the command line. When using the command line to perform HPC jobs, take the following into consideration.

Distributed Jobs

An Ansys EM batch job which distributes the analysis over several hosts may also be called a distributed job. To submit a distributed job, the following Ansys EM desktop command line options should be used:

- The -Distributed option should be present, and the -Local option should be absent. When running as a batch job under one of the schedulers with direct integration, this option is a directive to the job to 1) obtain the list of hosts allocated to the job, directly from the scheduler, and to 2) use the scheduler to launch the analysis engines on the hosts allocated to the job. The -Distributed option may now have additional options, such as includetypes=xxx, excludetypes=xxx, maxlevels=n, and numlevel1=n, where n indicates and integer, and xxx indicates a list of distribution types or "default".
- The -Machinelist num=*num_distributed_engines* option must be included, where *num_distributed_engines* is the total number of analysis engines to be started on the hosts assigned to the job.

Other examples:

- "Serial Job on a Single Processor" on the next page
- "Distributed Job Using Four Processors" on the next page

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- "Multiprocessing Job Using Four Cores" below
- "Distributed Analysis and Multi-Processing in the Same Job" on the facing page

Serial Job on a Single Processor

Suppose Ansys Electronics Desktop is installed at "C:\Program Files\ANSYS Inc\v251\AnsysEM\" and you are using RSM for DSO:

```
"C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt.exe" -ng -
BatchSolve -machinelist num=2
-monitor \\shared_drive\projs\OptimTee.aedt
```

User is using LSF for remote-analysis/DSO

```
bsub -n 1 "C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt.exe" -ng
-BatchSolve -machinelist num=3 -monitor -local \\shared_
drive\projs\OptimTee.aedt
```

Distributed Job Using Four Processors

Ansoft RSM

```
"C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt.exe" -ng -
Batchsolve -monitor -Distributed
-machinelist list="10.1.1.221, 10.1.1.222, 10.1.1.223, 10.1.1.224"
\\shared_drive\projs\OptimTee.aedt
```

LSF

```
bsub -n 4 "C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt.exe" -ng
-Batchsolve -monitor
-Distributed -machinelist num=4
\\shared_drive\projs\OptimTee.aedt
```

Multiprocessing Job Using Four Cores

Multi-processing job using 4 cores

```
bsub -n 4 -R span[ptile=4] "C:\Program Files\ANSYS
Inc\v251\AnsysEM\ansysedt.exe" -ng -monitor
-Local -BatchSolve -machinelist num=4 -batchoptions \\shared_
drive\registry.txt \\shared_drive\projs\OptimTee.aedt
```

This requests 4 cores to come from the same machine, as multi-processing needs cores to be on the same machine

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Distributed Analysis and Multi-Processing in the Same Job

Distributed-processing using 4 engines and multi-processing using 4 cores, using a total of 16 cores

```
bsub -n 16 -R "span[ptile=4]" "C:Program Files\ANSYS
Inc\v251\AnsysEM\ansysedt.exe" -ng
-BatchSolve -Distributed -machinelist num=4
-batchoptions \\shared_drive\registry.txt
\\shared_drive\projs\OptimTee.aedt
```

Integrating Ansys EM Tools with Third-Party Schedulers

This document indicates how to create a dynamically linked library to allow integration of Ansys EM tools with an arbitrary scheduler environment. Each scheduler proxy library is used for a single specific scheduler environment. If the library is installed with a valid name and in the correct location, it is then automatically loaded and used by Ansys EM tools.

- Introduction
- Common Requirements for Running Jobs
- Using a Shared Library (Linux) or a DLL (Microsoft Windows)
- Scheduler Proxy Interfaces
- Using an IronPython Program for Scheduler Integration

Introduction

Ansys EM Software Tools may be run as serial or parallel jobs on a cluster under control of a scheduler. Serial jobs are run using a single analysis engine at any one time on a single host. If the tool performs multiple analyses (for a frequency sweep or a parametric analysis, for example), the analyses are performed one after the other. Parallel jobs are run using multiple analysis engines running in parallel on the same host or on separate hosts. For parts of the analysis (such as meshing), the parallel job may use only a single analysis engine on a single host. Other parts of the analysis (such as a frequency sweep, parametric analysis or DDM) may be distributed to multiple analysis engines running in parallel.

- Serial Jobs
- Parallel Jobs

Serial Jobs

When an Ansys EM batch analysis runs as a serial job, the analysis engines run on the same host as the desktop process. The desktop process does not need to interact with the scheduler to get the names of hosts allocated to the job or to start processes on other hosts.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parallel Jobs

For a parallel job, the desktop process starts multiple analysis engines that run in parallel. These engines may be started on the host where the desktop process is running, or on other hosts allocated to the job. The desktop process interacts with the scheduler to obtain information on the hosts that are allocated to the job, and to start engines on the local host or on other hosts allocated to the job. This document provides information on how to facilitate this interaction between the desktop process and the scheduler controlling the cluster.

For some popular job schedulers in a standard configuration, Ansys EM provides an "out of the box" integrated solution that will work with the scheduler. In this case, the Ansys EM installation includes code that will determine if the analysis is running as a scheduler job and communicate with the scheduler when needed. For other schedulers, the code to obtain information about the hosts allocated to a job and to distribute portions of the job to hosts assigned to the job is not provided in the installation. In order to facilitate using Ansys EM Software Tools with other schedulers, the user may provide a way for Ansys EM Tools to interact with the scheduler. Currently, two general approaches are available to users.

In the first approach, the user creates a shared library (on Linux) or a dynamically linked library (on Microsoft Windows) to provide communication between the Ansys EM Tool and the scheduler. This library is loaded by the Ansys EM Tool at runtime, and if the Ansys EM Tool is running as part of a scheduler job, the Ansys EM Tool interacts with the library to get information from the scheduler, and to start additional processes on specified hosts. Each such library implements the same set of extern "C" functions needed to mediate the interactions between the Ansys EM Tool and the scheduler.

In the second approach, the user creates an IronPython program to provide communication between the Ansys EM Tool and the scheduler. This program is loaded by the Ansys EM Tool at runtime, and if the Ansys EM Tool is running as part of a scheduler job, the Ansys EMs Tool uses the IronPython program to get information from the scheduler, and to start additional processes on specified hosts. Each python script contains a class implementing a specified interface, which contains functions needed to mediate the interactions between the Ansys EM Tool and the scheduler. The details of the interface are described below. The IronPython interface is equivalent to the extern "C" functions used in the first approach.

Common Requirements for Running Jobs

The following requirements must be met for serial and parallel jobs to run successfully. They apply whether using "out of the box" scheduler integration or scheduler integration using a library or using an IronPython program. *Host requirements* apply to all hosts that may be allocated to an Ansys EM serial or parallel batch job.

Installation Requirements

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Ansys EM installation directory must be accessible from all cluster hosts using the same path. One way to achieve this is to place the Ansys EM installation on a shared drive that is accessible to the cluster hosts using the same pathname. On Windows, this may require the use of UNC names to refer to the installation directory. Another option is to install the Ansys EM tool locally on each cluster host using the same local directory path.

Project File and Directory Requirements

The directory containing the project file must also be available from all cluster hosts using the same path. The project file and the containing directory must be readable and writable by the user account used to run the job. The controlling process for a distributed job is called the Desktop process, and it reads from and writes to the project file and other files in the same directory and its subdirectories. Although only the Desktop process reads from and writes to this directory, the Desktop process may be started on any of the hosts allocated to the job, so all hosts should have access to this directory using the same path.

For cloud submission such as Cloud Direct, Ansys HPC Platform Services, and Ansys Cloud Burst Compute there are no requirements for shared drives or shared directories (installation or project).

Using a Shared Library (Linux) or a DLL (Microsoft Windows)

This section describes how to create a dynamically linked library to allow integration of Ansys Electromagnetics Suite 2025 R1 with an arbitrary scheduler environment. Each scheduler proxy library is used for a single specific scheduler environment. If the library is installed with a valid name and in the correct location, then it will automatically be loaded and used by Ansys Electromagnetics Suite 2025 R1.

Installation Details

The scheduler proxy library must be installed in the schedulers subdirectory of the Ansoft installation directory. For example, if the Ansys EM installation directory is "C:\Program Files\v251\AnsysEM", then the scheduler proxy library must be installed in directory "C:\Program Files\ANSYS Inc\v251\AnsysEM\schedulers.

The scheduler proxy library base name must match "libprefix_scheduler" on Windows and "liblibprefix_scheduler" on Linux. The extension must be a valid extension for a dynamically loaded library on the platform where it is used. The scheduler proxy library name prefix libprefix shall be unique, so it does not conflict with other scheduler proxy libraries in the same directory. To avoid confusion, the scheduler proxy library name should be all lower case on OSs where file names are case sensitive.

Build Information for Scheduler Proxy Library

This section contains the recommended compiler and linker settings for building a scheduler proxy library.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 64-Bit Microsoft Windows
- Linux

64-Bit Microsoft Windows

The proxy library should be compiled and linked as a 64-bit DLL, using the following recommended compiler and linker options:

Compiler Options

- Use of MFC: Use Standard Windows Libraries
- Character Set: Use Multi-Byte Character Set [/D "_MBCS"]
- Runtime Library: Multi-threaded DLL [/MD]
- Calling Convention: __cdecl [/Gd (default)]

Linker Options:

- Create a DLL [/DLL]
- 64-bit code [MACHINE:X64]

Linux

The proxy library should be compiled and linked as shared library (*.so) file. The following compiler and linker options are recommended when building using gcc/g++:

Compiler Options

- Generate position-independent code, suitable for use in a shared library: [-fpic]
- Generate code compatible with pthreads library: [-pthread]

Linker Options:

- Create a shared object file: [-shared]
- Generate position-independent code, suitable for use in a shared library: [-fpic]
- Generate code compatible with pthreads library: [-pthread]

Implementation Details for Custom Scheduler Integration

Function Name Prefix

Each exported function will have a scheduler specific function name prefix. The function name prefix will be the same as the library name prefix, except that it is converted to upper case. For example, if the library name prefix is "lsf", then the function name prefix is "LSF". In the examples below, we use FN_PREFIX to denote the function name prefix.

The scheduler proxy library must provide implementations of the following extern "C" functions:

- IsProductLaunchedInYourEnvironment
- GetTempDirectory

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- GetMachineListAvailableForDistribution
- GetMessageStringToRegisterForSigTerm
- LaunchProcess
- GetUseRsmForEngineLaunch
- GetThisJobID
- GetSchedulerDisplayName

IsProductLaunchedInYourEnvironment

Purpose

Determine if the program is running in the context of the scheduler for which this library was written.

Signature

```
extern "C" bool FN_PREFIX_IsProductLaunchedInYourEnvironment();
```

Arguments

None.

Return Value

Returns true if the current process is running as a job of the scheduler. Otherwise, false is returned.

> **Note:**
>
> For many schedulers, the presence of certain environment variables or their values may be checked to determine if the current process is running as a job of the scheduler.

GetTempDirectory

Purpose

Get the pathname of the temporary directory provided by the scheduler for the current job. The pathname is an empty string if the scheduler does not provide a temporary directory for the current job.

Signature

```
extern "C" bool FN_PREFIX_GetTempDirectory(char * buffer,
unsigned int* length);
```

Arguments

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

buffer: Pointer to a character buffer to contain the temporary directory path name or NULL.

length: Pointer to a location to contain the length of the buffer. Must be a valid pointer to an unsigned int.

Return Value

If argument buffer is NULL, then then required length of the buffer is stored in the location to which argument length points, and true is returned.

If argument buffer is not NULL, then the value to which argument length points (the buffer length) is checked. If it is large enough to contain the pathname of the temporary directory, including the terminal null byte, then the pathname is copied to the buffer and true is returned. If the buffer length is insufficient for the pathname of the temporary directory, then the buffer is unchanged, and false is returned.

> **Note:**
>
> To get the pathname of the temporary directory, the infrastructure first calls this function with a NULL buffer, and obtains the required length of the buffer for the pathname. After creating a buffer of the appropriate size, the infrastructure calls this function again, passing the pointer to the buffer in the buffer argument and a pointer to the size of the buffer in the length argument.

GetMachineListAvailableForDistribution

Purpose

Get the list of hosts allocated to the current job. A host will appear in the list multiple times if the scheduler has allocated multiple processors or cores on the host to the job. The number of times the host appears in the list is equal to the number of processors or cores of the host that are allocated to the current job. The list is a text string containing a space separated list of hostnames.

Signature

```
extern "C" bool FN_PREFIX_GetMachineListAvailableForDistribution
(char * buffer, unsigned int* length);
```

Arguments

buffer: Pointer to a character buffer to contain the list of machines available for distribution or NULL.

length: Pointer to a location to contain the length of the buffer. Must be a valid pointer to an unsigned int.

Return Value

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If argument buffer is NULL, then then required length of the buffer is stored in the location to which argument length points, and true is returned.

If argument buffer is not NULL, then the value to which argument length points (the buffer length) is checked. If it is large enough to contain the lists of hosts, including the terminal null byte, then the list is copied to the buffer and true is returned. If the buffer length is insufficient for the list of hosts, then the buffer is unchanged, and false is returned.

Note:

- To get the list of hosts for distribution, the infrastructure first calls this function with a NULL buffer, and obtains the required length of the buffer for the list. After creating a buffer of the appropriate size, the infrastructure calls this function again, passing the pointer to the buffer in the buffer argument and a pointer to the size of the buffer in the length argument.
- The hostnames in the list provided by this function shall be used in calls to LaunchProcess(). These host names must be in a format that is accepted by that function. See the section below on LaunchProcess.

GetMessageStringToRegisterForSigTerm

Purpose

Obsolete. The string copied to the buffer should be an empty string.

Signature

```
extern "C" bool FN_PREFIX_GetMessageStringToRegisterForSigTerm
(char * buffer, unsigned int* length);
```

Arguments

buffer: Pointer to a character buffer to contain the string or NULL.

length: Pointer to a location to contain the length of the buffer. Must be a valid pointer to an unsigned int.

Return Value

If argument buffer is NULL, then the required length of the buffer is stored in the location to which argument length points, and true is returned.

If argument buffer is not NULL, then the value to which argument length points (the buffer length) is checked. If it is large enough to contain the string, including the terminal null byte, then the string is copied to the buffer and true is returned. If the buffer length is insufficient for the string, then the buffer is unchanged, and false is returned.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

To get the string, the infrastructure first calls this function with a NULL buffer, and obtains the required length of the buffer for the string. After creating a buffer of the appropriate size, the infrastructure calls this function again, passing the pointer to the buffer in the buffer argument and a pointer to the size of the buffer in the length argument.

LaunchProcess

Purpose

Launch a local or remote process to run an analysis engine. This function is called by the Ansys Electromagnetics desktop application to launch an engine process on a specified host. The hostname is one of the names in the list provided by the GetMachineListAvailableForDistribution function. If the hostname does not refer to the local host, then this function shall use the scheduler to launch the engine on the specified host. If the hostname refers to the local host, then the engine may be started as a child process, or it may be started using the scheduler.

Signature

```
extern "C" int FN_PREFIX_LaunchProcess(const char* hostName,
const char* exePathName, const char* arg1, const char* arg2);
```

Arguments

hostName: The name of the host where the process is to be launched.

exePathName: The pathname of the analysis engine executable to be started.

arg1: The first argument of the analysis engine command line.

arg2: The second argument of the analysis engine command line.

Return Value

Returns 0 on success. Returns a non-zero value if an error occurs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- The hostName argument will be one of the hostnames provided by the function GetMachineListAvailableForDistribution().
- If the hostName argument is the same as the current host, then the analysis engine process may be started as a child process. If the hostName argument is not the same as the current host, then the analysis engine process will be started on the remote host using the facilities available in the scheduler environment. The command line of the analysis engine process is exePathName arg1 arg2. The command line arguments arg1 and arg2 may contain newlines, tabs, spaces or other characters that are interpreted by the command processor, such as single quote (') or double quote (") characters, or dollar signs ($). Newlines or tabs may be replaced by spaces, if the newline or tab characters cannot be easily handled. If the analysis engine command is processed by a shell, then it may be necessary to quote any special characters in the exePathName or in the arguments so that the special meaning is removed. If a scheduler command is used to request the scheduler to launch the command to start the engine process, the analysis engine command may be processed by the shell twice: once when the scheduler command is processed, and a second time when the analysis engine process is started. If this is the case, then the quoting of special characters needs to account for two passes through the command processor.

GetUseRsmForEngineLaunch

Purpose

This function is optional. If this feature is not needed, then the function need not be implemented. Most schedulers should not need this feature.

For some schedulers, it may be desirable for the Ansoft RSM service to launch the engine processes instead of using the scheduler proxy library. For example, if the scheduler proxy library is limited to launching one process per host, then the scheduler proxy library may be used to launch one Ansoft RSM service executable per host, and the Ansoft RSM executable will launch all of the engine processes.

If the Ansoft RSM service should be used to launch engine processes for this scheduler, then this function shall be implemented and it shall return true.

If the Ansoft RSM service should not be used to launch engine processes for this scheduler, then this function is not required. If it is implemented, it should return false. If it is not implemented, it will be treated the same as if it was implemented and returns false.

Signature

```
extern "C" bool FN_PREFIX_GetUseRsmForEngineLaunch(void)
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Arguments

None.

Return Value

Returns true if the Ansoft RSM service should be used to launch engine processes for this scheduler. Returns false if the Ansoft RSM service should not be used to launch engine processes for this scheduler.

> **Note:**
>
> This function is optional. If not implemented, then it is treated the same as if it was implemented and returns false.

GetThisJobID

Purpose

Get a string identifying the job currently running in the scheduler environment. This string is displayed to the end user to identify the job.

Signature

```
extern "C" bool FN_PREFIX_GetThisJobID(char * buffer, unsigned
int* length);
```

Arguments

buffer: Pointer to a character buffer to contain the Job ID or NULL.

length: Pointer to a location to contain the length of the buffer. Must be a valid pointer to an unsigned int.

Return Value

If argument buffer is NULL, then then required length of the buffer is stored in the location to which argument length points, and true is returned.

If argument buffer is not NULL, then the value to which argument length points (the buffer length) is checked. If it is large enough to contain the string identifying the current job, including the terminal null byte, then the job ID is copied to the buffer and true is returned. If the buffer length is insufficient for the job ID, then the buffer is unchanged, and false is returned.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- To get the job ID, the infrastructure first calls this function with a NULL buffer, and obtains the required length of the buffer for the job ID. After creating a buffer of the appropriate size, the infrastructure calls this function again, passing the pointer to the buffer in the buffer argument and a pointer to the size of the buffer in the length argument.
- For many schedulers, the job ID may be obtained from the value of an environment variable.

GetSchedulerDisplayName

Purpose

Get a string identifying the scheduler associated with the current scheduler proxy library. This string is displayed to the end user to identify the scheduler.

Signature

```
extern "C" bool FN_PREFIX_GetSchedulerDisplayName(char * buffer,
unsigned int* length);
```

Arguments

buffer: Pointer to a character buffer to contain the scheduler display name or NULL.

length: Pointer to a location to contain the length of the buffer. Must be a valid pointer to an unsigned int.

Return Value

If argument buffer is NULL, then required length of the buffer is stored in the location to which argument length points, and true is returned.

If argument buffer is not NULL, then the value to which argument length points (the buffer length) is checked. If it is large enough to contain the scheduler display name, including the terminal null byte, then the scheduler display name is copied to the buffer and true is returned. If the buffer length is insufficient for the scheduler display name, then the buffer is unchanged, and false is returned.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- To get the scheduler display name, the infrastructure first calls this function with a NULL buffer, and obtains the required length of the buffer for the scheduler display name. After creating a buffer of the appropriate size, the infrastructure calls this function again, passing the pointer to the buffer in the buffer argument and a pointer to the size of the buffer in the length argument.
- The scheduler display name is generally a fixed string.

Scheduler Proxy Interfaces

Scheduler proxy supports following new graphical interface functions. The scheduler specific prefix of each function is not shown in this listing.

void Initialize(const std::string& config):

Initialize the proxy library for scheduler interaction. The **config** argument contains scheduler specific initialization information.

int CheckEnvironment(std::string& msg):

Check the environment in which the proxy library is running.

- Returns 0 (success) if the environment is appropriate for submitting jobs to the scheduler.
- Returns a non-zero error code if the environment is incorrect. If a non-zero error code is returned, an error message to display to the user is written to the msg argument.

int GetSchedulerInfo(std::string& msg, std::string& schedulerName, std::string& schedulerDescription, std::string& schedulerVersion):

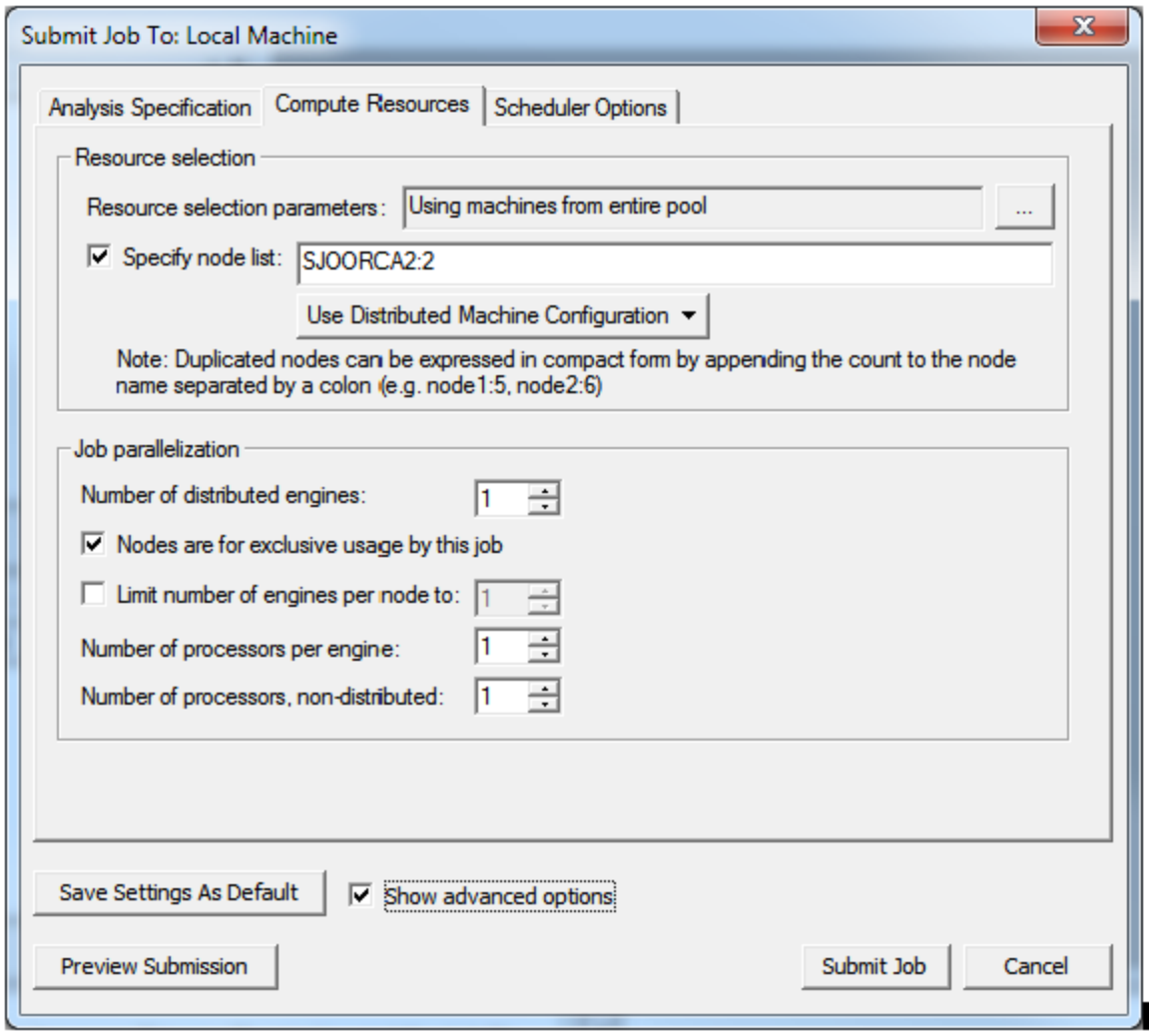
This function returns some basic information about the scheduler with which the scheduler proxy library interacts.

- On success, 0 is returned, and the scheduler name, scheduler description, and scheduler version are written to the **schedulerName**, **schedulerDescription** and **schedulerVersion** arguments.
- On failure, a non-zero error code is returned, and an error message to display to the user is written to the msg argument.

int GetComputeResourceAttributes(std::string& msg, AttributeDefinitionsStruct& attributeDefs):

The *Compute Resource* tab or the *Submit Job To* dialog box allows the user to specify scheduler specific resources. This function returns the information used to create and populate the *Compute Resource* tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Each line in the dialog box is defined by a single attribute definition in the **attributeDefs** argument. An attribute definition defines the name and description of an attribute, as well as information about the allowed values and the default value. In general, only the most commonly specified job attributes are included in the **attributeDefs** argument.

- On success, 0 is returned, and the attribute definitions are written to the **attributeDefs** argument.
- On failure, a non-zero error code is returned, and an error message to display to the user is written to the msg argument.
- If the scheduler proxy library does not support any attributes using this approach, the **attributeDefs** argument will contain no attribute definitions, and 0 will be returned.

int AbortJob(std::string& msg, const std::string& jobID, bool force, const SubmissionUserStruct& submissionUser):

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This function requests the scheduler to abort a job identified by the **jobID** argument. If the force argument is true, then errors should be ignored (the exact behavior is scheduler specific). The **submissionUser** argument contains information about the client user (the user running the Desktop process). The request to abort the job should run in the context of this user. If no user is specified, then the request to abort the job runs as the user of the process or thread running the function.

- If the request is successfully submitted, then 0 is returned.
- If there is an error, then a non-zero error code is returned, and an error message to display to the user is written to the msg argument.

int SubmitUniformJob(std::string& msg, std::string& jobID, const CmdLineStruct& cmdLineInfo, const JobParallelizationStruct& jobParallelization, const UniformComputeResourcesStruct& computeResources, const JobOptionsStruct& jobOptions, const JobAtributesStruct& jobAttributes, const SubmissionUserStruct& submissionUser,const IJobParameters* jobParametersCB):

This function submits a job to the scheduler.

- On success, 0 is returned, and the job identifier of the newly submitted job is written to the **jobID** argument.
- On failure, a non-zero error code is returned, and an error message to display to the user is written to the msg argument.

This function is used to submit jobs to the scheduler in which the resources allocated to the job are uniformly distributed across the nodes allocated to the job. All other arguments are input arguments, and they are described below:

The **cmdLineInfo** argument contains the command line arguments. The first argument is the command name.

The **jobParallelization** argument contains information on how the job should be parallelized. It contains the following integral parameters:

- the total number of distributed engines,
- the number of cores to allocate for each distributed engine,
- the maximum number of engines to allocate to a single node (optional), and
- the number of cores to allocated for the non-distributed portion of the analysis.
- It also contains a boolean parameter indicating whether nodes used for this job should be exclusively allocated to this job.

The **computeResources** argument is a reference to an object of type **UniformComputeResourcesStruct**. This **struct** contains zero or more resource attribute settings for the job. Each resource attribute setting consists or a resource name and a resource value. The resource name is the name of one of the resources defined in the **AttributeDefinitionsStruct** filled in by the **GetComputeResourceAttributes()** function. The

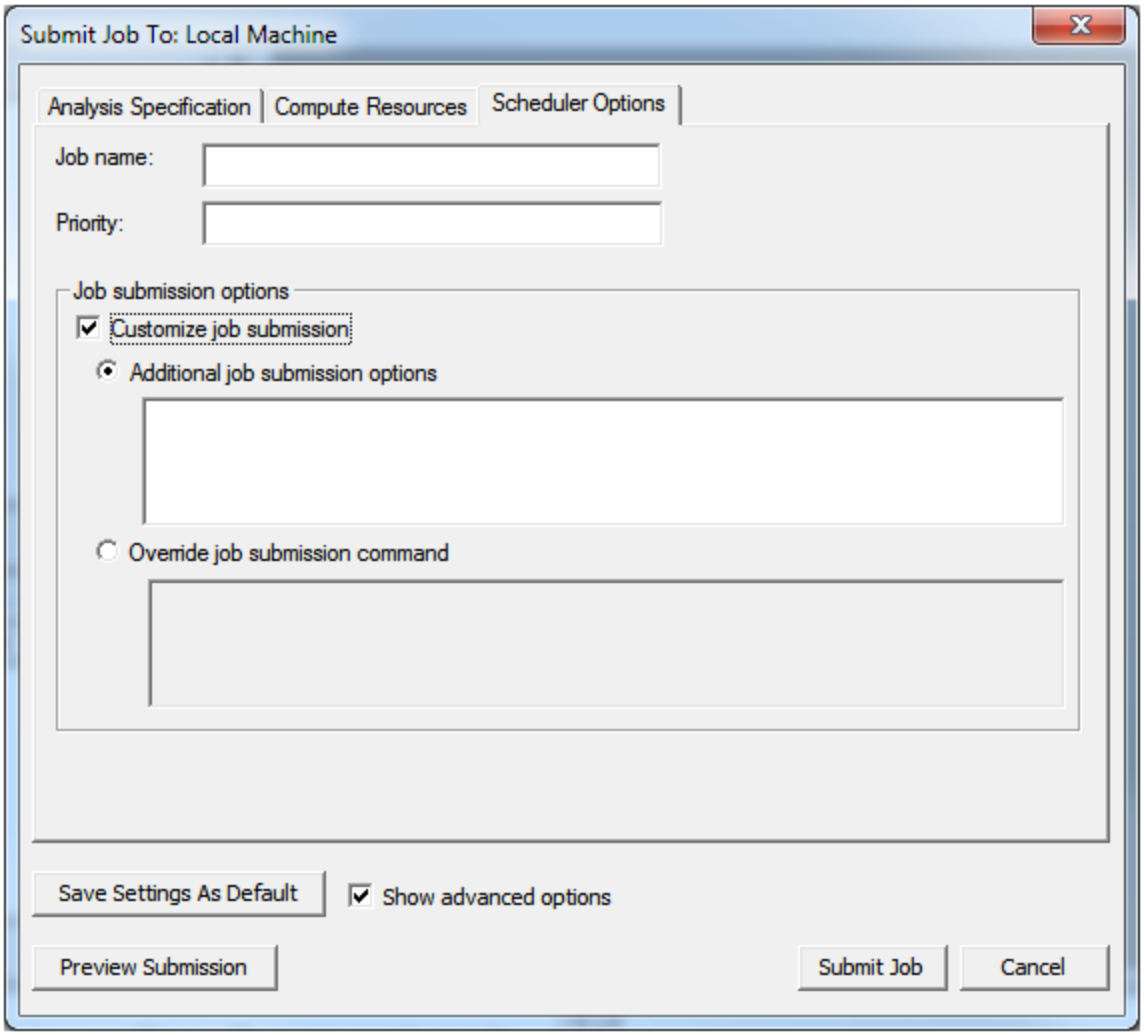
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

resource attribute value is the value specified for the resource attribute by the user using the *Compute Resource* tab of the *Submit Job To* dialog box. If no resource attributes are specified by the user in this dialog box, then the **computeResources** argument will contain no resource attribute settings.

The **jobOptions** argument contains the environment variable settings for the job.

The **jobAttributes** argument contains job submission attributes which are not necessarily related to the compute resources allocated to the job. The job name and the requested job priority are included in this data structure.

The *Scheduler Options* tab of the *Submit Job To* dialog box allows the user to either specify additional job submission options or to specify all submission options, replacing the settings from the other *Submit Job To* dialog box controls.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The user-specified submission options are included in this data structure, as well as a boolean setting indicating whether the user specified options are in addition to the automatically generated options, or whether they replace the automatically generated submission options.

The **submissionUser** argument contains information about the client user (the user running the Desktop process). The job is submitted to the scheduler to run as this user.

The **jobParametersCB** argument is a pointer to an object that implements the **IJobParameters** interface. This interface allows the scheduler proxy library to get additional information about the job. Specifically, the **GetWorkingDirectory()** interface function returns the working directory to be used for the job.

The **cmdLineInfo** argument contains the command line arguments. The first argument is the command name.

int SubmitNonUniformJob(std::string& msg, std::string& jobID, const CmdLineStruct& cmdLineInfo, const JobParallelizationStruct& jobParallelization, const NonUniformComputeResourcesStruct& computeResources, const JobOptionsStruct& jobOptions, const JobAtributesStruct& jobAttributes, const SubmissionUserStruct& submissionUser,const IJobParameters* jobParametersCB):

This function submits a job to the scheduler.

- On success, 0 is returned, and the job identifier of the newly submitted job is written to the **jobID** argument.
- On failure, a non-zero error code is returned, and an error message to display to the user is written to the msg argument.

This function is used to submit jobs to the scheduler in which the nodes to use and the number of engines to run on each node are specified by the user. All other arguments are input arguments, as for the **SubmitUniformJob()** function. These input arguments are the same as for the **SubmitUniformJob()** function, except that the **computeResources** argument is a reference to a **NonUniformComputeResourcesStruct**, as described below:

The **computeResources** argument is a reference to an object of type **NonUniformComputeResourcesStruct**. This object contains a vector of pairs, where each pair consists of the name of a node in the cluster, and the number of engines to run on the node.

int PreviewUniformJob(std::string& msg, std::string& preview, const CmdLineStruct& cmdLineInfo, const JobParallelizationStruct& jobParallelization, const UniformComputeResourcesStruct& computeResources, const JobOptionsStruct& jobOptions, const JobAtributesStruct& jobAttributes, const SubmissionUserStruct& submissionUser, const IJobParameters* jobParametersCB):

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This function is similar to the **SubmitUniformJob()** function, but instead of submitting the job, text representing how the job will be submitted is written to the preview argument. Typically the preview text includes the job submission command and the contents of the job script created for the job. For some schedulers, this content may not be meaningful, so the text returned could be different.

- On success, 0 is returned, and the job preview text is written to the preview argument.
- On failure, a non-zero error code is returned, and an error message to display to the user is written to the msg argument.

The other arguments are input arguments with the same meaning as for the **SubmitUniformJob()** function. The **submissionUser** argument is ignored for this function.

Extending Scheduler Integration with Tight-Integration for MPI

AnsysEM supports tight-integration with a few popular schedulers by leveraging built-in tight-integration capabilities of intel MPI. Specifically on Linux, this works out of the box with LSF, SGE and Ansys Cloud Burst Compute. For other Linux based schedulers, this note describes how scheduler integration developers can extend scheduler integration to provide tight integration for MPI based solves.

MPI Based Solves

Like many other simulation applications, AnsysEM also uses an MPI-based message passing approach to provide portable parallel execution on shared, distributed and distributed shared memory computers.

An MPI solve involves running a mpiexec (or mpirun) command that runs the simulation's executable on each allocated/specified host. For most applications, the entire simulation is just a single MPI solve. However, an AnsysEM simulation consists of multiple MPI solves which are organized as following:

1. Multiple MPI solves in series: Each AnsysEM simulation runs multiple MPI solves.
2. Different host list: Each such MPI solves may use different amount of resources (mostly cores per host) that is specified using the host list and count of ranks.
3. Multiple MPI solves in parallel: In some case, multiple MPI solves may be launched in parallel from different hosts. So multiple MPI solves may be running simultaneously.
4. Multi-processing: Finally, within each MPI based solve, each rank can use multi-processing as well.

To launch an MPI based solve AnsysEM code uses an MPI integration related wrapper file, called mpirun.fl, which in turn invokes MPI's mpirun/mpiexec. Ansys EM installation includes all the needed files for the supported MPI implementation's.

Scheduler integration for MPI

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When a scheduler runs a simulation, each scheduler job usually runs just a single MPI solve. Each time MPI needs to launch a process on a remote allocated host, it will do so either with password-less ssh (loose integration), or with a scheduler specific script/command (tight-integration).

Custom Tight Integration

For custom tight integration, you provide a script that uses scheduler's utilities to launch processes on a remote host. The script needs to be a drop-in replacement for ssh. For AnsysEM to use this script for MPI solves, you need to specify the following environment variables and provide bootstrap selector and bootstrap script file.

Environment Variables	Value	Description
ANSYSEM_HPC_MPI_USE_SCHEDULER_CMD	A keyword that identifies the scheduler	Scheduler for which to enable tight-integration
ANSYSEM_HPC_MPI_CUSTOM_SCRIPT_FILE	Fully qualified path to the custom script file	Script that implements a function that returns the switch to be used for MPI implementation's command line.

Script Files

Script Files	Content	Description
Bootstrap selector BASH script file	implements a bash function that returns the switch to be used for MPI implementation's command line.	**Intel MPI**: Implements `ansysem_hpc_intel_mpi_bootstrap` either returns -bootstrap or -bootstrap-exec with a suitable argument
Bootstrap BASH script file	Shell script that is a ssh drop-in replacement	Launches a process on a remote host. Accepts the same arguments as ssh

Example: PBSPro

For custom tight integration with PBSPro, you provide a script that uses pbsdsh utility to launch processes on a remote host. The script is a drop-in replacement for ssh. It assumes that the first two arguments are -x and -q. and provide bootstrap selector and bootstrap script file

For an AnsysEM batch solve runnning under PBSPro to use this script for MPI solves, the job script will set the following environment variables.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Job script excerpt:

```
# The selector script will specify the MPI options for PBSPro
export ANSYSEM_HPC_MPI_USE_SCHEDULER_CMD=pbspro
# Bootstrap selector BASH script file
export ANSYSEM_HPC_MPI_CUSTOM_SCRIPT_FILE=${INSTALL_DIR}/ansys_
inc/v251/AnsysEM/schedulers/utils/pbspro-intelMPI-integration.sh
# Special environment variable to enable PBS Node file to become
available in Bootstrap script file.
export ANSYSEM_HPC_MPI_SCHEDULER_NODEFILE=$PBS_NODEFILE
```

pbspro-intelMPI-integration.sh

```
#!/bin/bash
function ansysem_hpc_intel_mpi_bootstrap()
{
  local custom_scheduler="$1"
  local bootstrap_switch=""
  if [[ -n "$custom_scheduler" ]]
  then
    if [[ "$custom_scheduler" == "pbspro" ]]
    then
      exec_file=$(dirname $ANSYSEM_HPC_MPI_CUSTOM_SCRIPT_FILE)
      exec_file_name=pbspro_dsh_wrapper.sh
      bootstrap_switch="-bootstrap-exec $exec_file/$exec_file_name"
    fi
fi
  echo "$bootstrap_switch"
}
```

pbspro_dsh_wrapper.sh

```
#!/bin/bash
# Bootstrap BASH script file for PBSPro
# Assumes that the first two arguments are -x and -q
# For debugging, set it to a suitable path
# DEBUG_DIR=$(HOME)/Ansoft/Test/DebugLog
PBS_BINDIR=/opt/pbs/bin
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
shift 2
host_name=$1
# Assumes that the next argument is host name shift 1
# Compute the index for the first entry
index1=$(( $(grep -n -m 1 $host_name $ANSYSEM_HPC_MPI_SCHEDULER_
NODEFILE | cut -f1 -d:) - 1))
if [[ -n "$DEBUG_DIR" ]]
then
  echo "running pbsdsh ON $host_name as index $index1 $@" >>
  ${DEBUG_DIR}/wrapper.log
  cat $ANSYSEM_HPC_MPI_SCHEDULER_NODEFILE >> ${DEBUG_
  DIR}/wrapper.log
fi
$PBS_BINDIR/pbsdsh -v -n $index1 -- "$@"
```

Intel MPI custom bootstrap feature

From Intel MPI docs:

```
-bootstrap-exec <bootstrap server>
```

Use this option to set the executable to be used as a bootstrap server. For example:

```
> mpiexec -bootstrap-exec <bootstrap_server_executable> -f hostfile
-env <VAR1> <VAL1> -n 2 test.exe
```

This -bootstrap-exec switch lets you specify a shell script that is a ssh drop-in replacement. For custom tight-integration, AnsysEM uses Intel MPI, and leverages its special switch. This -bootstrap-exec switch lets you specify a shell script that is a ssh drop-in replacement. Using it requires a small patch to an MPI integration related wrapper file, called mpirun.fl. This file, which is available in AnsysEm installation, is used by AnsysEM code to launch an MPI based solve (similar to using mpirun). Each time Intel MPI needs to launch a process on a remote allocated host, it will call a scheduler (or user) specified script using arguments that are same as those for password-less ssh.

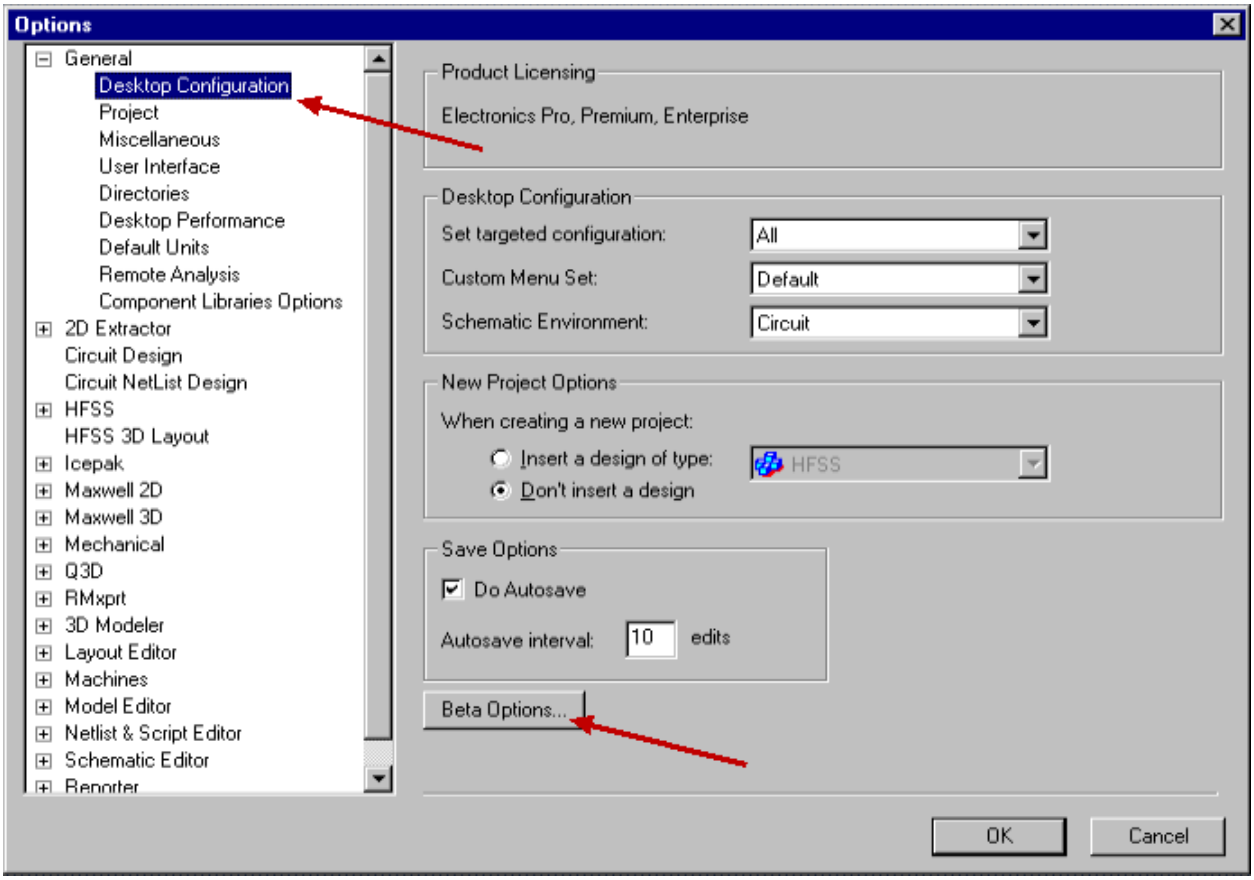
Support for Open MPI on Linux [Beta]

Support for Open MPI on Linux is available as a beta feature for HFSS, HFSS 3D Layout, Icepak, Maxwell (3D and 2D), and Q3D designs.

To enable the beta feature:

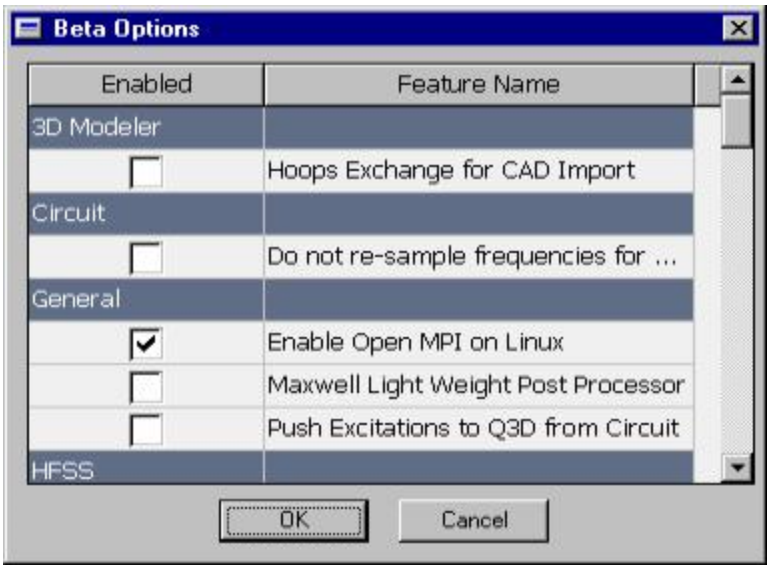
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Access the *Options* dialog box using one of the following two mehtods:
   - From the menu bar, click **Tools** > **Options** > **General Options**.
   - On the **Desktop** ribbon tab, click **General Options**.
2. In the tree on the left side of the dialog box, expand **General** and select **Desktop Configuration**.
3. Click **Beta Options**:



4. Select **Enable Open MPI on Linux**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. When making a change to any beta option, you may be prompted to restart the Ansys Electronics Desktop application. If prompted, click **Yes** to do so.

   The **Open MPI** option will now be available in the **MPI Vendor** drop-down menu within the *Options* tab of the *HPC and Analysis Options* dialog box. You only need to perform this procedure once; the Open MPI option will remain available for future sessions and projects.

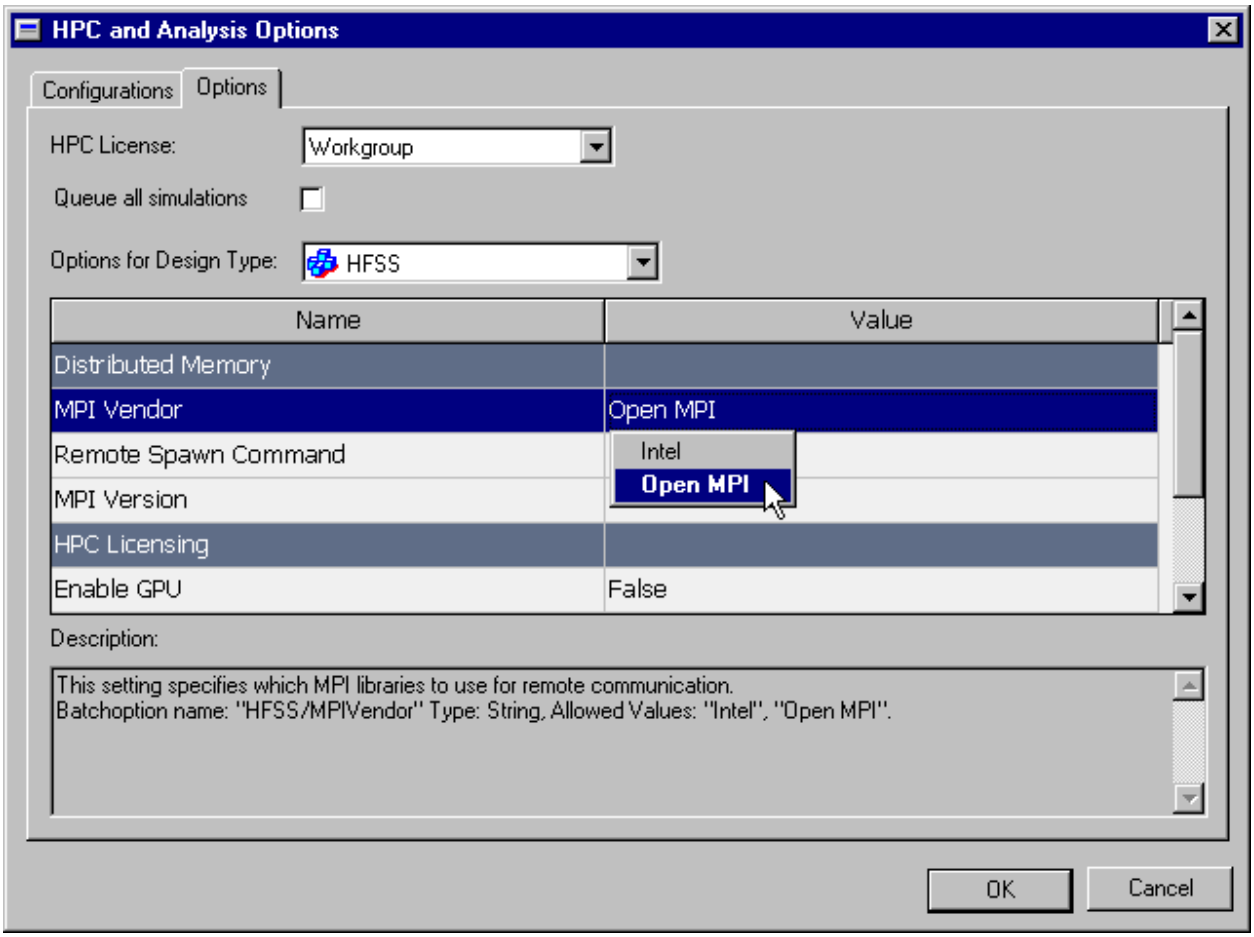
To choose Open MPI as the MPI Vendor:

1. Open the project in the Ansys Electronics Desktop application (interactive mode).
2. From the **Simulation** ribbon tab, click **HPC Options**
3. In the *HPC and Analysis Options* dialog box, select the **Options** tab.

   The contents of this tab vary by design type.

4. Select **Open MPI** from the **MPI Vendor** drop-down menu:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Click **OK**.

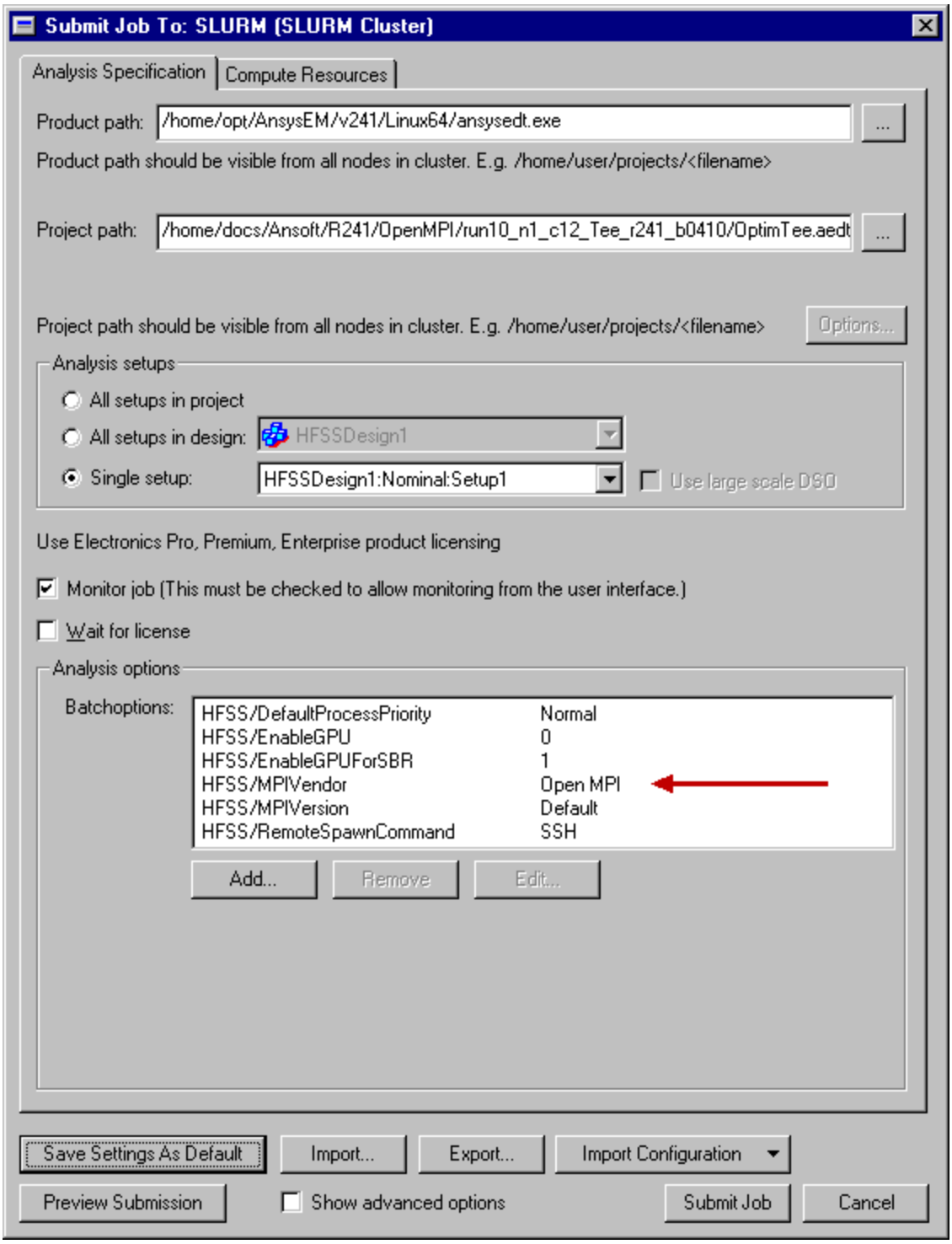
   You can then run the simulation on a Linux cluster as follows.

6. In the Project Manager, right-click the solution setup to solve and choose **Submit Job** from the shortcut menu.
7. Specify the desired parameters and click **Submit Job**.

   In the *Submit Job* dialog box, *Open MPI* is included as a batch option for each supported design type along with other applicable batch options.

   The following example is for an HFSS design:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Testing Scheduler Integration

One way to test these functions is to run the analysis for an Ansys EM product in batch mode. When running in batch mode, a batch log file is created in the same directory as the project file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The batch log file has the same base name is the project file, with an extension of ".log". For example, if the project file is TestProject123.aedt, then the batch file is TestProject123.log. The batch log file contains useful information about the analysis run.

See the product specific help for details on running the product in batch mode, and for the command line options to use for distributed analysis.

- Testing IsProductLaunchedInYourEnvironment
- Testing GetSchedulerDisplayName and GetThisJobID
- Testing GetTempDirectory
- Testing GetMachineListAvailableForDistribution
- Testing LaunchProcess
- Testing GetUseRsmForEngineLaunch

Testing IsProductLaunchedInYourEnvironment

This function should be tested first. If the Ansys EM application is not able to load and run this function, or if it returns false, then none of the other functions will be called. If the batch analysis is running in a scheduler environment, and this function returns true, then there will be an "info" message near the beginning of the batch log indicating that the analysis is running as a scheduler job. This message will include the scheduler display name returned by the function GetSchedulerDisplayName, and it will also include the job ID returned by the function GetThisJobID. If the batch analysis is not running in a scheduler environment, then none of the messages will include a scheduler display name or job ID.

If this message does not appear when running in a scheduler environment, ensure that the scheduler proxy library is named correctly, that it is built correctly, that it is installed in the correct directory, and that the function name prefix is the same is the library prefix converted to upper case.

Testing GetSchedulerDisplayName and GetThisJobID

As described above, when running a batch job in a scheduler environment, the scheduler display name and the job ID will appear in an "info" message near the beginning of the batch log. The values returned by these functions are copied to this message verbatim, so they can be directly compared to the expected values.

Testing GetTempDirectory

Many schedulers create a temporary directory for each job and delete the directory after the job finishes. One way to verify that this function is working correctly is to determine the pathname that the scheduler uses for the temporary directory and to monitor the contents of the temp directory as the job is running. If the analysis engines write files to this directory as the job runs, then this function is working.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Testing GetMachineListAvailableForDistribution

This function is used for distributed analysis. The analysis may be distributed across several machines if portions of the analysis are independent. For example, frequency sweeps, parametric analysis and domain decomposition allow different portions of the analysis to be distributed across machines. The analysis in a batch job will be distributed to multiple processors or hosts if the analysis includes a setup that may be distributed (e.g., frequency sweep, parametric analysis) and the **-Distributed** option is included in the desktop command line. The list of machines is displayed in an "info" message near the beginning of the batch log. The list in the info message can be directly compared to the expected list of machines.

To verify that the machine list is constructed correctly for a variety of cases, it may be necessary to test several jobs with different resource requirements and verify that the machine list is correct in each case. For example, one may run batch analyses with the following resource requirements:

- One processor on one host
- Several processors on one host
- One processor on each of several hosts
- Several processors on each of several hosts

Testing LaunchProcess

This function is used to launch analysis engines in the case where the analysis is distributed across multiple hosts. The analysis may be distributed across several machines if portions of the analysis are independent. For example, frequency sweeps, parametric analysis and domain decomposition allow different portions of the analysis to be distributed across machines. The analysis in a batch job will be distributed to multiple processors or hosts if the analysis includes a setup that may be distributed (e.g., frequency sweep, parametric analysis) and the **-Distributed** option is included in the desktop command line. The list of machines is displayed in an "info" message near the beginning of the batch log. The batch log may also contain info messages when portions of the analysis distributed to different machines start or finish. These messages usually include the name of the host when the analysis ran or will run. One can verify that the analysis is actually running on the expected host or hosts using the Linux ps command or the Windows Task Manager.

In general, one analysis engine is started for each occurrence of each host in the list of machines available for distribution. For example, if the list of hosts is "hostA hostA hostA hostB hostB", then a total of 5 engines would be started, three on hostA and two on hostB. In some cases, an additional engine is started to perform the portion of the analysis which is not distributed; if this is the case, the non-distributed engine is idle during the portion of the analysis which is distributed. If this occurs in the case where the list of hosts is "hostA hostA hostA hostB hostB", then a total of 6 engines would be started, but at most 5 engines would be active at any given time. When each analysis engine is running, it may start additional child processes to do a

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

portion of the analysis, but these are not counted as additional analysis engines because the parent of the sub-engine is inactive (waiting for the sub-engine results) when the sub-engine is active.

Testing should be sufficient to demonstrate that the scheduler proxy library can start multiple engine processes on the desktop host, and can also start multiple engine processes on other hosts.

Testing GetUseRsmForEngineLaunch

In most cases, this function will not be implemented or tested. If this function is implemented and returns true, then the Ansys Electromagnetics desktop application will not start the analysis engines using the LaunchProcess function directly. Instead, the Ansys Electromagnetics desktop application will start one AnsoftRSMService process on each host using the LaunchProcess function, and the engine processes will be started by these AnsoftRSMService processes. One may check for these processes using the Linux ps command or the Windows Task Manager. One AnsoftRSMService process should run on each host. These processes will be named ansoftrsmservice.exe or AnsoftRSMService.exe. These processes will be started on each host before any analysis engine is started on the host, and will remain running until the job is complete.

Troubleshooting Custom Scheduler Integration

- None of the Proxy Functions are Called
- Troubleshooting IsProductLaunchedInYourEnvironment Function
- Troubleshooting GetSchedulerDisplayName
- Troubleshooting GetThisJobID
- Troubleshooting GetTempDirectory
- Troubleshooting GetMachineListAvailableForDistribution
- Troubleshooting LaunchProcess
- Troubleshooting GetUseRsmForEngineLaunch

None of the Proxy Functions are Called

There are several problems which could result in none of the proxy functions being called.

The scheduler proxy library must be installed in the schedulers subdirectory of the Ansys Electronics Desktop installation directory.

The scheduler proxy library name must match "*_scheduler.dll" on Windows and "lib*_ scheduler.so" on Linux. If the library name does not match this format, then the library will not be loaded. In addition, the function name prefix must be the same as the library name prefix converted to upper case. For example, if the library name prefix is "abc", then the function name prefix is "ABC". In this example, the library name is "abc_scheduler.dll" on Windows, and

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

"libabc_scheduler.so" on Linux. In this example, the full name of the IsProductLaunchedInYourEnvironment function is ABC_IsProductLaunchedInYourEnvironment on Windows and Linux, and it must have extern "C" linkage.

Verify that the compile and link flags follow the guidelines in the section "Build Information", above. Incorrect compile or link flags may prevent the library from being loaded by the Ansys Electromagnetics product.

If there is a problem with calling the IsProductLaunchedInYourEnvironment function, then none of the other functions will be called. The other functions are only called if the IsProductLaunchedInYourEnvironment function is successfully called and returns true.

Troubleshooting IsProductLaunchedInYourEnvironment Function

Verify that the conditions specified in the section "None of the Proxy Functions are Called" are met.

Verify that this function returns true when called in an environment running under the scheduler, and that it returns false when called in an environment not running under the scheduler.

Troubleshooting GetSchedulerDisplayName

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

Verify that the scheduler display name is a valid ASCII string.

Verify that, if argument buffer is NULL, then the required length of the buffer is stored in the location to which argument length points, and true is returned. The required buffer length must include space for the string null terminator.

Verify that, if argument buffer is not NULL and the value to which argument length points (the buffer length) is large enough to contain the display name, including the terminal null byte, then the display name is copied to the buffer and true is returned.

Troubleshooting GetThisJobID

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

Verify that the job ID is a valid ASCII string.

Verify that, if argument buffer is NULL, then the required length of the buffer is stored in the location to which argument length points, and true is returned. The required buffer length must include space for the string null terminator.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Verify that, if argument buffer is not NULL and the value to which argument length points (the buffer length) is large enough to contain the job ID, including the terminal null byte, then the job ID is copied to the buffer and true is returned.

Troubleshooting GetTempDirectory

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

Verify that the temporary directory name is a valid ASCII string.

Verify that, if argument buffer is NULL, then the required length of the buffer is stored in the location to which argument length points, and true is returned. The required buffer length must include space for the string null terminator.

Verify that, if argument buffer is not NULL and the value to which argument length points (the buffer length) is large enough to contain the temporary directory pathname, including the terminal null byte, then the temporary directory pathname is copied to the buffer and true is returned.

Troubleshooting GetMachineListAvailableForDistribution

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

Verify that the list of hosts is a valid ASCII string containing a space separated list of host names. A host name will appear in the list a number of times equal to the number of processors or cores available to the job on that host.

Verify that, if argument buffer is NULL, then the required length of the buffer is stored in the location to which argument length points, and true is returned. The required buffer length must include space for the string null terminator.

Verify that, if argument buffer is not NULL and the value to which argument length points (the buffer length) is large enough to contain the list of hosts, including the terminal null byte, then the list of hosts is copied to the buffer and true is returned.

Troubleshooting LaunchProcess

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

The hostName argument is a host name from the list returned by the GetMachineListAvailableForDistribution function. Verify that the LaunchProcess function can accept host names in the format returned by the GetMachineListAvailableForDistribution function.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The exePathName argument is the pathname of the analysis engine executable to be started. This pathname may contain spaces or other characters special to the shell. Ensure that the LaunchProcess function is able to handle such cases.

The arg1 and arg2 arguments may contain newlines, tabs, single quotes, spaces, dollar signs, and other characters which may be special to the shell. Ensure that the LaunchProcess function is able to handle such cases. If needed, the newline characters may be replaced by other whitespace characters. One or both of these arguments could also be an empty string; verify that the empty string is correctly passed to the engine process command line.

If a scheduler command is used to launch the engine process on a remote machine, the engine command line may be processed by the shell twice, once when the scheduler command is processed by the shell, and again when the engine command is processed by the shell. In such cases, the quoting of characters special to the shell will need to be take these two passes through the shell into account. In some implementations, it may be necessary or convenient to use different approaches for launching engine processes on the local machine and on remote machines; if this is done, verify that the approach used to determine whether the hostName argument represents the local machine is correct.

Troubleshooting GetUseRsmForEngineLaunch

In most cases, this function will not be implemented. If it is implemented, then follow the suggestions below.

Verify that the IsProductLaunchedInYourEnvironment function returns true when running in the scheduler environment.

If the RSM should be used for launching engines, verify that this function returns true. Otherwise, verify that this function returns false.

Using an IronPython Program for Integration with a Scheduler

This section describes how to create an IronPython program for integration with a scheduler. Each such program is used for a single specific scheduler environment. If the program is installed with a valid name and in the correct location, then it will automatically be loaded and used by Ansys EM tools.

Installation Details

The IronPython program must be installed in the schedulers subdirectory of the Ansys EM installation directory. For example, if the installation directory is "C:\Program Files\ANSYS Inc\v251\AnsysEM", then the IronPython program must be installed in directory C:\Program Files\ANSYS Inc\v251\AnsysEM\schedulers".

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The program file extension must be ".py". Select the program name so that it does not conflict with other IronPython programs in the same directory. If the Operating System or file system treat file names in a case sensitive manner, the file extension ".py" must be lower case.

Python Programming Notes

The scheduler program will be run in the IronPython environment both on Microsoft Windows and on Linux. There are some differences between IronPython and CPython. The version of IronPython in use is 2.7.0.40

Implementation Details

The program must contain the following:

Import the ISchedulerPluginExtension interface as follows:

```
from Ansys.Ansoft.SchedulerPluginDotNet import
ISchedulerPluginExtension
```

Define a class which implements the ISchedulerPluginExtension interface. In this document, this class is named SamplePluginExtension, but any class name may be used. The class member functions are described in the next section. The class definition will look similar to the following:

```
class SamplePluginExtension(ISchedulerPluginExtension):

def GetName(self):
return "SamplePluginExtension"

def GetDescription(self):
return "Example python script plugin extension"
. . .
```

Include the following line in the program so that the class that you have defined, SamplePluginExtension, is loaded by the infrastructure:

```
ExtensionRegistrar.RegisterPluginExtension(SamplePluginExtension())
```

The infrastructure will make the EntensionRegistrar object available in the environment where the program is loaded.

Each of the functions to be implemented in the SamplePluginExtension class is described below.

- GetName
- GetDescription
- IsProductLaunchedInYourEnvironment
- GetSchedulerDisplayName

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- GetThisJobID
- GetUseRsmForEngineLaunch
- GetTempDirectory
- GetMessageStringToRegisterForSigTerm
- GetMachineListAvailableForDistribution
- LaunchProcess

GetName [IronPython]

Purpose

Return a short string containing the name of the plugin extension. This string is used to identify the scheduler plugin extension in logs or program output.

Signature

```
GetName(self)
```

Arguments (excluding self)

None.

Return Value

Returns a string containing the name of the plugin extension.

> **Note:**
>
> The plugin extension name is generally a fixed string.

GetDescription [IronPython]

Purpose

Return a string containing the description of the plugin extension. This string is used to identify the scheduler plugin extension in logs or program output.

Signature

```
GetDescription(self)
```

Arguments (excluding self)

None.

Return Value

Returns a string containing the description of the plugin extension.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The plugin extension description is generally a fixed string.

IsProductLaunchedInYourEnvironment [IronPython]

Purpose

Determine if the program is running in the context of the scheduler for which this program was written.

Signature

```
IsProductLaunchedInYourEnvironment(self)
```

Arguments (excluding self)

None.

Return Value

Returns True if the current process is running as a job of the scheduler. Otherwise, False is returned.

> **Note:**
>
> For many schedulers, the presence of certain environment variables or their values may be checked to determine if the current process is running as a job of the scheduler.

GetSchedulerDisplayName [IronPython]

Purpose

Get a string identifying the scheduler associated with the current plugin extension. This string is used to identify the scheduler.

Signature

```
GetSchedulerDisplayName(self)
```

Arguments (excluding self)

None.

Return Value

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Returns a string containing the description of the scheduler for which this plugin extension was written.

> **Note:**
>
> The scheduler display name is generally a fixed string.

GetThisJobID [IronPython]

Purpose

Get a string identifying the job currently running in the scheduler environment. This string is displayed to the end user to identify the job.

Signature

```
GetThisJobID(self)
```

Arguments (excluding self)

None.

Return Value

Returns a string containing the Job ID for the current job.

> **Note:**
>
> For many schedulers, the job ID may be obtained from the value of an environment variable.

GetUseRsmForEngineLaunch [IronPython]

Purpose

For some schedulers, it may be desirable for the AnsoftRSM program to launch the engine processes instead of using the scheduler plugin extension directly. For example, if the plugin extension is limited to launching one process per host, then the plugin extension may be used to launch one AnsoftRSM executable per host, and the AnsoftRSM executable will launch all of the engine processes.

If AnsoftRSM should be used to launch engine processes for this scheduler, then this function shall return True.

If AnsoftRSM should not be used to launch engine processes for this scheduler, then this function shall return False.

Signature

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
GetUseRsmForEngineLaunch(self)
```

Arguments (excluding self)

None.

Return Value

Returns True if AnsoftRSM should be used to launch engine processes for this scheduler. Returns False if the plugin extension should be used to directly launch engine processes for this scheduler.

> **Note:**
>
> If this function returns True, then the plugin extension will directly launch only one process on each host.

GetTempDirectory [IronPython]

Purpose

Get the pathname of the temporary directory provided by the scheduler for the current job. The pathname is an empty string if the scheduler does not provide a temporary directory for the current job.

Signature

```
GetTempDirectory(self)
```

Arguments (excluding self)

None.

Return Value

Returns a string containing the pathname of the temporary directory provided by the scheduler for the current job. Returns an empty string if no temporary directory is provided by the scheduler for the current job.

> **Note:**
>
> If the return value is an empty string, then the temporary directory specified on the command line or in the registry will be used.

GetMessageStringToRegisterForSigTerm [IronPython]

Purpose

Obsolete. This function should return an empty string.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Signature

```
GetMessageStringToRegisterForSigTerm(self)
```

Arguments (excluding self)

None.

Return Value

Returns an empty string.

> **Note:**
>
> This function should always return an empty string.

GetMachineListAvailableForDistribution [IronPython]

Purpose

Get the names of the hosts allocated to the current job. A host name will appear in the output string multiple times if the scheduler has allocated multiple processors or cores on the host to the job. The number of times the host name appears in the string is equal to the number of processors or cores of the host that are allocated to the current job. The output is a text string containing the host names separated by space characters.

Signature

```
GetMachineListAvailableForDistribution(self)
```

Arguments (excluding self)

None.

Return Value

A string containing the names of the hosts allocated to the job, separated by space characterss. The number of times the host appears in the string is equal to the number of processors or cores of the host that are allocated to the current job.

> **Note:**
>
> The hostnames in the string provided by this function shall be used in calls to LaunchProcess(). The host names must be in a format that is accepted by that function. See the section below on LaunchProcess.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LaunchProcess [IronPython]

Purpose

Launch a local or remote process to run an analysis engine. This function is called by the Ansys Electromagnetics desktop application to launch an engine process on a specified host. The hostname is one of the names provided by the GetMachineListAvailableForDistribution function. See the GetMachineListAvailableForDistribution section above. If the hostname does not refer to the local host, then this function shall use the scheduler to launch the engine on the specified host. If the hostname refers to the local host, then the engine may be started as a child process, or it may be started using the scheduler.

Signature

```
LaunchProcess(self, hostName, exePathName, arg1, arg2)
```

Arguments (excluding self)

hostName: The name of the host where the process is to be launched.

exePathName: The pathname of the analysis engine executable to be started.

arg1: The first argument of the analysis engine command line.

arg2: The second argument of the analysis engine command line.

Return Value

Returns 0 on success. Returns a non-zero value if an error occurs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

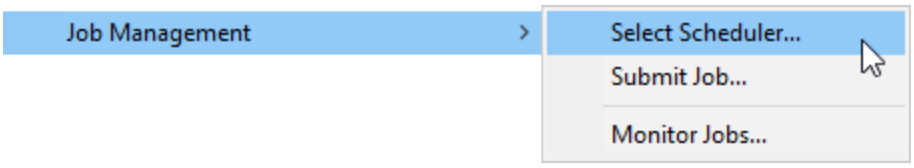
Note:

- The hostName argument will be one of the hostnames provided by the function GetMachineListAvailableForDistribution().
- If the hostName argument is the same as the current host, then the analysis engine process may be started as a child process. If the hostName argument is not the same as the current host, then the analysis engine process will be started on the remote host using the facilities available in the scheduler environment. The command line of the analysis engine process is exePathName arg1 arg2. The command line arguments arg1 and arg2 may contain spaces or other characters that are interpreted by the command processor, such as backslash (\), single quote (') or double quote (") characters, or dollar signs ($). If the analysis engine command is processed by a shell, then it may be necessary to quote any special characters in the exePathName or in the arguments so that the special meaning is removed. If a scheduler command is used to request the scheduler to launch the command to start the engine process, and that command is processed by a command shell, then the analysis engine command may be processed by the shell twice: once when the scheduler command is processed, and a second time when the analysis engine process is started. If this is the case, then the quoting of special characters needs to account for two passes through the command processor.
- The command line arguments arg1 and arg2 may be empty strings. These arguments must be preserved, even if they are empty strings. In some versions of the IronPython subprocess module, empty argument strings are discarded, resulting in an incorrect number of command line arguments. A workaround for this issue is to replace an empty string argument by a string consisting of a single space character.

Selecting a Scheduler

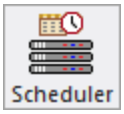
Access the **Select Scheduler** window one of three ways:

- Click **Tools** > **Job Management** > **Select Scheduler...**



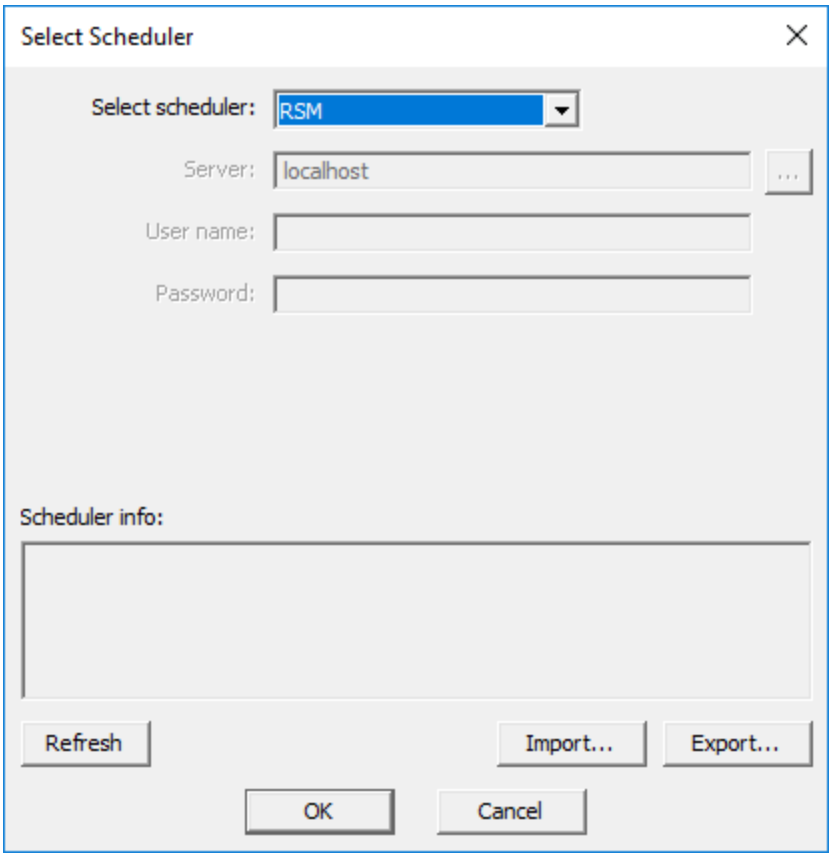
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select the **Simulation** tab and click the **Scheduler** icon.



- From a command window, use the -showselectscheduler command.

The **Select Scheduler** window appears:



From the **Select Scheduler** window:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Use the **Select scheduler** drop-down menu to select a scheduler.

   > **Note:**
   >
   > If you select a scheduler that is unsupported in your environment, you will receive a warning message.
   >
   > See HPC Integration for a list of currently supported schedulers.

2. If applicable for the scheduler type, enter server and user information. If Ansys Cloud Direct or Ansys Cloud Burst Compute is selected then the “Login” button will open the default browser and prompt for login.

   Information about the selected scheduler appears in the **Scheduler info** field.

3. Click **OK** to complete your selection.

Submitting a Job

Ansys Electromagnetics Desktop supports its own Remote Simulation Management (RSM) and Ansys Cloud Direct along with other High Performance Computing (HPC) software management programs (See: HPC Integration). The **Simulation** tab of the ribbon includes icons for setting HPC Options, creating and selecting analysis configuration, selecting the scheduler, submitting jobs, and monitoring jobs.

There are two ways that the GUI may be used to submit jobs. The first (and most common) mode requires that the Desktop (UI) process run on a host which is also a submission host for the job scheduler. This mode is called local mode or working mode.

The second mode is only supported on Linux in the Ansys Electromagnetics Suite. In the second mode, an administrator configures the RSM Service to act as an interface to the job scheduler, and starts the RSM Service on a submission host for the cluster. The user runs the Desktop (UI) process on another host (which may be called the post-processing host). To submit a job, the user specifies the host where the RSM Service is running, and the Desktop process connects to the RSM Service over the network to submit the job. In this mode, some configuration is required, and the RSM Service typically must run as a privileged user (for example, root), so that it can launch processes as any user. This mode is useful for cases in which the submission hosts are not able to run graphical processes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

For certain methods of resource selection for Windows HPC job submission, Ansys Electronics Desktop checks for nodes being online. In an auto-scaling cluster, nodes can fail the online check because they're not online until a job is running. An environment variable allows customers to bypass this check:

ANSYSEM_SKIP_NODE_ONLINE_CHECK

Set the variable value to 1 to disable the online check. Debug logging fully verifies what aspects of the cluster are causing this check to fail.

Job submission in Windows works without changing settings if environment variable ANSYSEM_SUBMIT_JOB_REQ_NODE_ONLINE is set with value 0. The installation doesn't set this environment variable. ANSYSEM_SUBMIT_JOB_REQ_NODE_ONLINE accepted values: 0 or 1 (0 = disable; 1 = enable, i.e. require nodes to be online for submission). Default: 1 (i.e. if not set, Ansys Electronics Desktop requires nodes to be online for submission).

To submit a job:

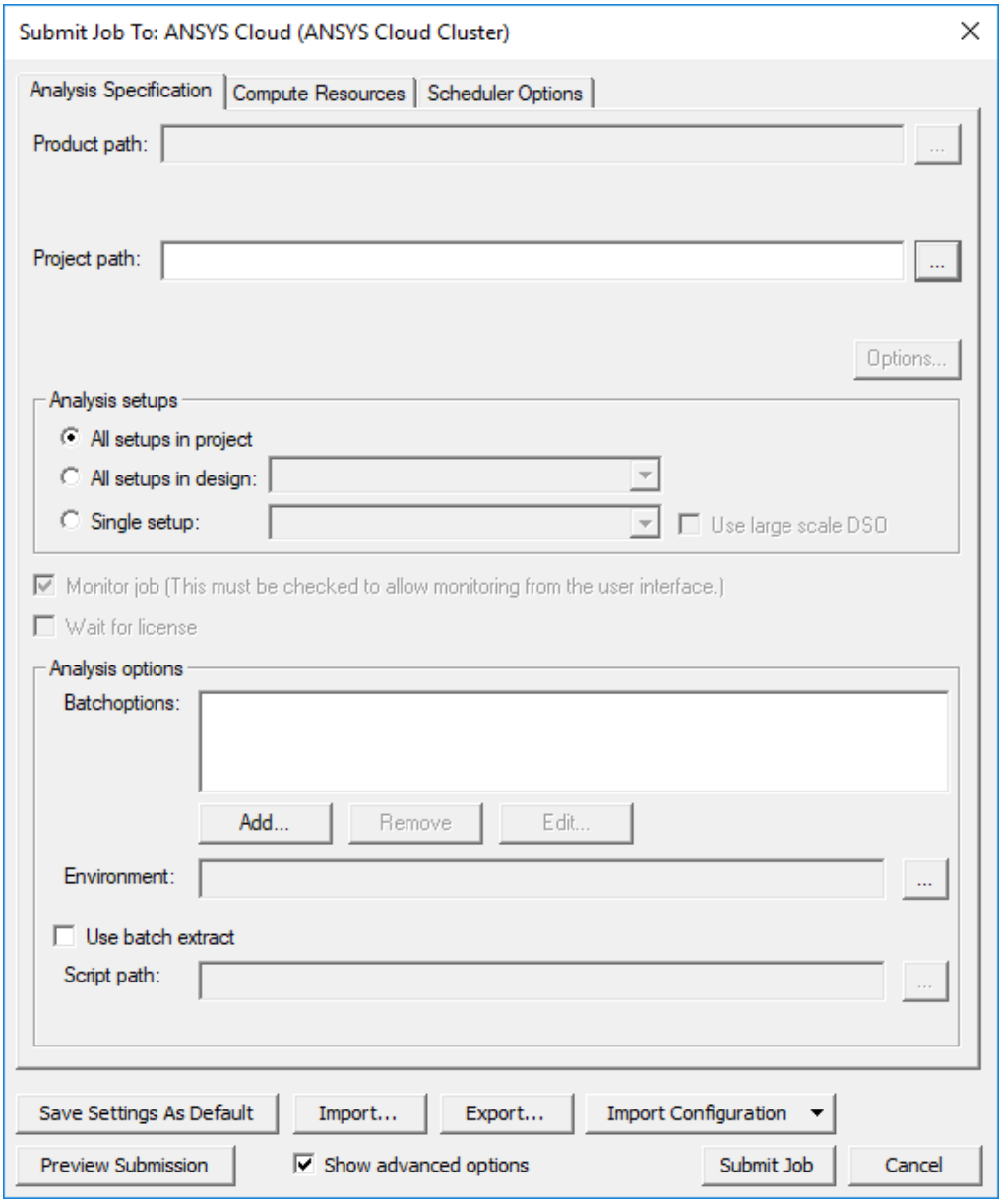
1. Prepare your design.

   For HFSS, Ansys recommends you use **HFSS** > **Design Settings** to turn on S parameter export for frequency so that they can be downloaded quickly after the simulation is complete. They must be exported to default project path for download to work correctly.

2. Open the **Submit Job To** window one of the following ways:
   - On the **Simulation** tab, click the **Submit** icon.
   - Select **Tools** > **Job Management** > **Submit Job**.
   - Select **HFSS** > **Submit Job**.
   - Right-click a solution setup and select **Submit Job**. *In this case, the information in the **Submit Job To** window will be pre-populated from the setup*.
   - From a Command window, use the -showsubmitjob command.

   The **Submit Job To** window appears. The window header indicates your selected scheduler.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This window contains up to three tabs:

- **Analysis Specification** – allows you to specify the product path, project name, setup and analysis options, batchoptions, and environment variables (for advanced users).
- **Compute Resources** – allows you to specify whether to use automatic settings, and to set resource selection and job distribution parameters.
- **Scheduler Options** – allows you to specify the job name and priority. *This tab does not appear if you have selected local RSM as the scheduler*.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. On the **Analysis Specification** tab, specify your desired options.

   The **Product path** and **Project** fields support mapped drives. Click the ellipses (**...**) to select files.

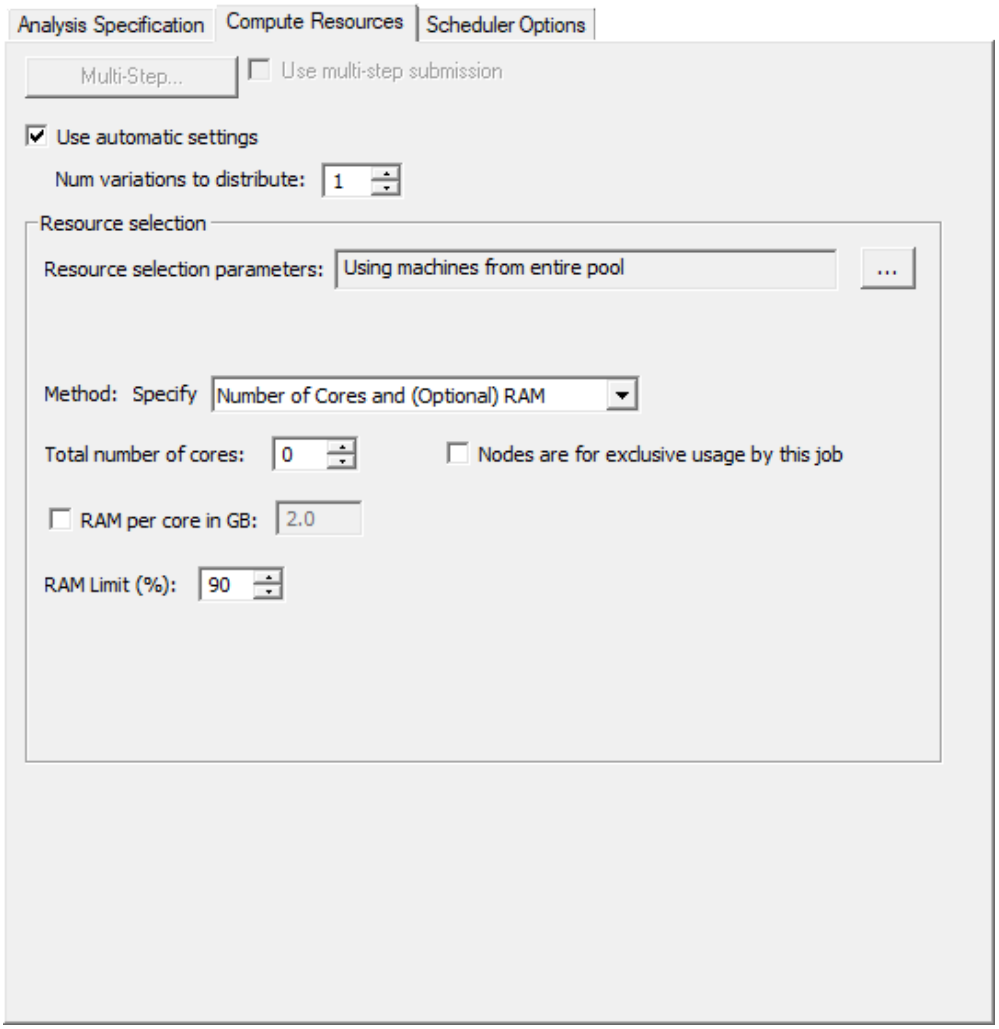
   The project can be an archive. The project file pathname must be a UNC path that is accessible from each compute host used for Ansys Electromagnetics jobs. After clicking the **Project path** field's ellipsis button (**...**), a check box allows you to **Use converted UNC path if mapped drive specified**. If you select a project or product on a mapped drive, and check the option, the converted UNC path equivalent to the mapped drive pathname is used.

   In the **Analysis Setups** area, select the radio button for **All setups in project**, **All setups in design**, or a **single setup**.

   For Parametric setups, you have the option to select **Use Large Scale DSO**. See: Large Scale DSO for Parametric Analysis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Select the **Compute Resources** tab.



Options may vary slightly depending on your selected scheduler.

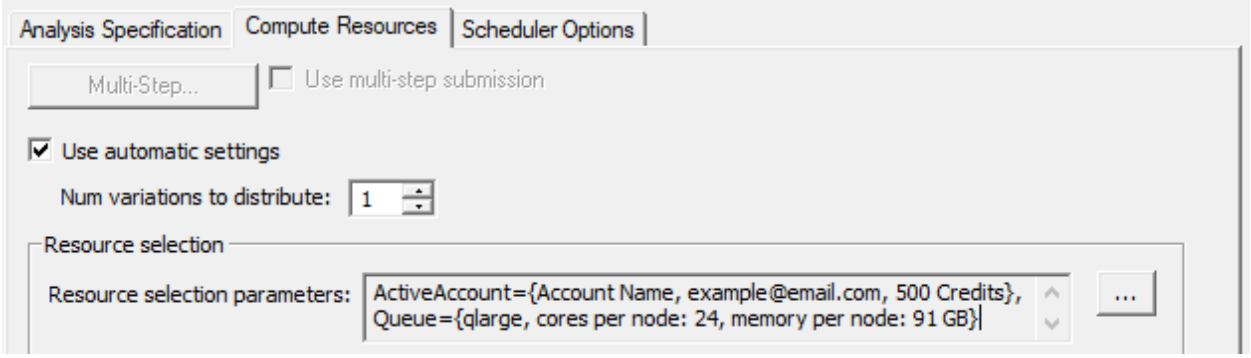
5. If applicable, determine whether or not to **Use multi-step submission**.
6. Determine whether or not to **Use automatic settings**.

> **Note:**
>
> Ansys strongly recommends using Automatic settings.

7. Use the buttons in the **Resource selection** area to allocate resources. See: Job Management for Large Scale DSO. If you are using Ansys Cloud Burst Compute select an active account using the ellipsis […] button to the right of the **Resource selection parameters:** text box and ensure that it has sufficient Core Hour Balance. This balance

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will be displayed in the **Resource selection parameters:** text box.



8. If you opted not to use automatic settings, you can also set **Job distribution** parameters. See: Distributed Analysis.
9. If applicable, select the **Scheduler Options** tab and set the **Job name** and **Priority**.

10. If desired, import or export job configurations.
11. If desired, click **Save Settings as Default** to save the current settings and overwrite defaults. These settings are saved on a per-scheduler basis.
12. If desired, click **Show advanced options** to see options for advanced users. See: Running Electronics Desktop from a Command Line.
13. If desired, click **Preview Submission** to view the commands to be sent to the scheduler. Ansys Cloud Direct submissions contain queue (pool) configuration details, including the job's hourly cost in Ansys Elastic Units (AEUs).

    The preview text can be copied to the clipboard.
14. Click **Submit Job** to submit the job to your selected scheduler.

Using Advanced Job Submission Options

From the job submission window, you can select **Show advanced options** to enable additional analysis options and job submission options.

This topic covers the following:

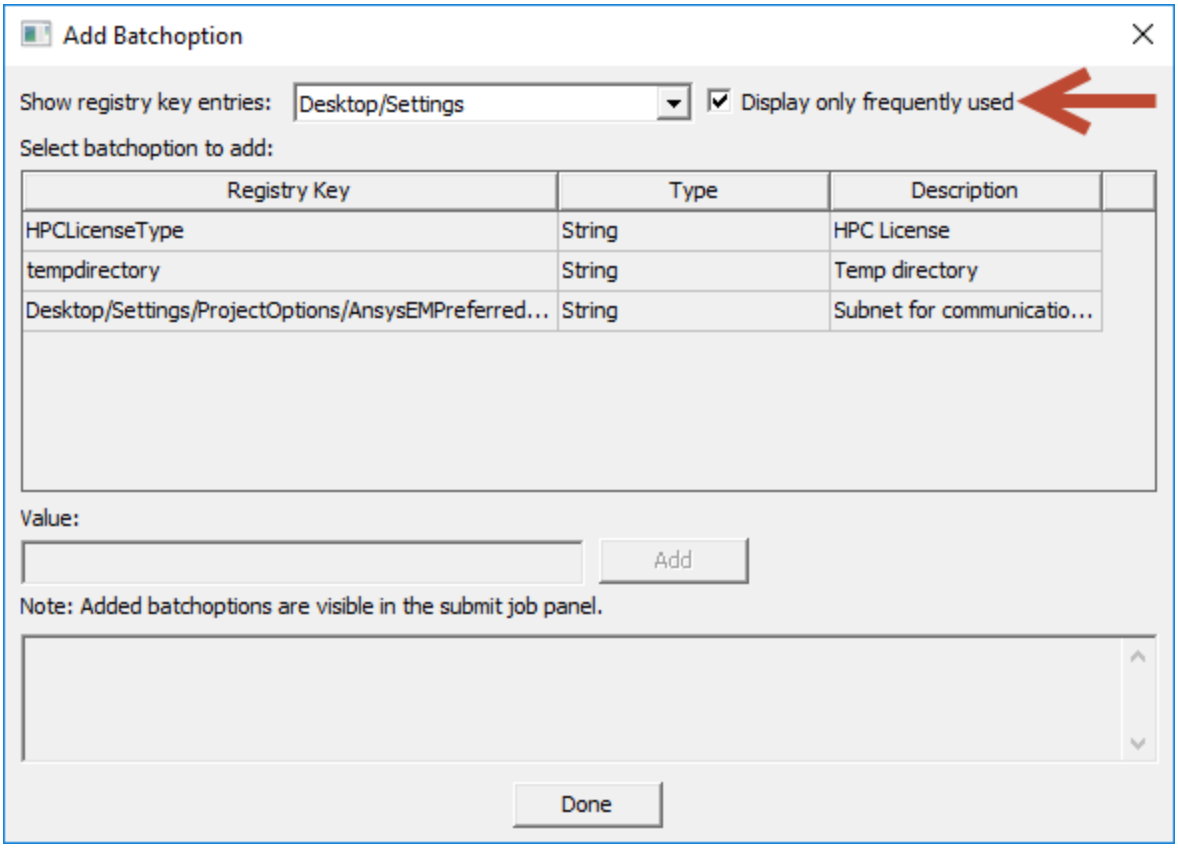
- Batchoptions
- Environment Variables

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Batch Extract
- Customize Job Submission

Batchoptions

In the **Submit Job To** window, under **Batchoptions**, click **Add** to open the **Add Batchoption** window.

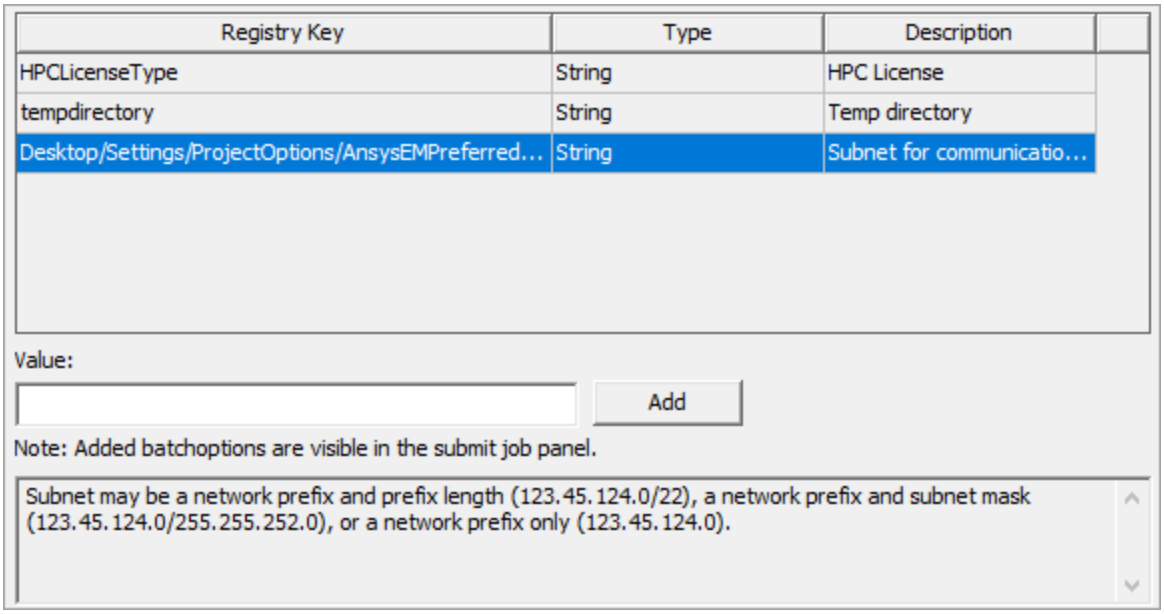


The **Show registry key entries** drop-down menu allows you to select categories of registry keys to display.

Note that **Display only frequently used** is selected by default. Deselect this to view all options for the selected category.

Select a registry key to activate the **Value** field, where you can enter a value. Selecting a key also populates the bottom of the window with a description of that key:

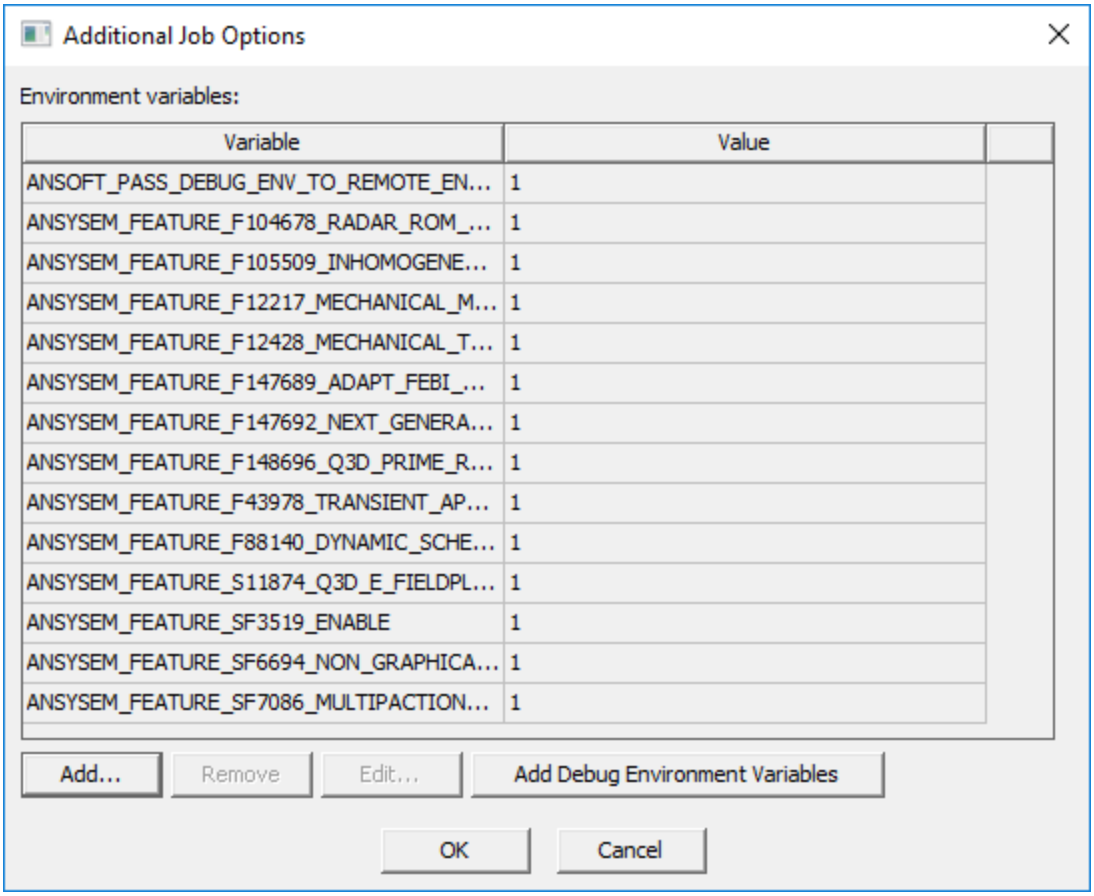
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Environment Variables

In the **Submit Job To** window, under **Environment**, click the ellipses (**...**) to open the **Additional Job Options** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This window lists all currently activated environment variables.

Active variables display a value of 1. Inactive variables display a value of 0. Non-binary variables may contain project paths, integer values, etc.

You can perform the following actions:

- **Add** – Add an environment variable. You will be prompted to enter the variable name and value.
- **Remove** – Select and remove an environment variable from the list.
- **Edit** – Select an environment variable and enter a new value.
- **Add Debug Environment Variables** – Activates a selection of debugging variables and adds them to the list. The log files created are only useful to development and if a customer or an application engineer needs to set these environment variables they should be working with a developer directly or indirectly who will know what needs to be set.

Batch Extract

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Submit Job To** window, select **Use batch extract** to enable the **Script path** field. Click the ellipses (**...**) to browse and select a VBscript or Python script to execute along with the job. See: Running Ansys Electronics Desktop from the Command Line for a description of Batch Extract.

Customize Job Submission

In the **Submit Job To** window's **Scheduler Options** tab, advanced options for some schedulers allow you to **Customize job submission**. When the **Override job submission** radio button is selected, user-specified options replace most of the job submission options. When the **Additional job submission options** radio button is selected, user-specified options are appended to the bsub command.

Using the Command Line to Submit HPC Jobs

Ansys Electronics Desktop can be run from the command line. When using the command line to perform HPC jobs, take the following into consideration.

Distributed Jobs

An Ansys EM batch job which distributes the analysis over several hosts may also be called a distributed job. To submit a distributed job, the following Ansys EM desktop command line options should be used:

- The -Distributed option should be present, and the -Local option should be absent. When running as a batch job under one of the schedulers with direct integration, this option is a directive to the job to 1) obtain the list of hosts allocated to the job, directly from the scheduler, and to 2) use the scheduler to launch the analysis engines on the hosts allocated to the job. The -Distributed option may now have additional options, such as includetypes=xxx, excludetypes=xxx, maxlevels=n, and numlevel1=n, where n indicates and integer, and xxx indicates a list of distribution types or "default".
- The -Machinelist num=*num_distributed_engines* option must be included, where *num_distributed_engines* is the total number of analysis engines to be started on the hosts assigned to the job.

Other examples:

- "Serial Job on a Single Processor" on the next page
- "Distributed Job Using Four Processors" on the next page
- "Multiprocessing Job Using Four Cores" on the next page
- "Distributed Analysis and Multi-Processing in the Same Job" on page 9-221

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Serial Job on a Single Processor

Suppose Ansys Electronics Desktop is installed at "C:\Program Files\ANSYS Inc\v251\AnsysEM\" and you are using RSM for DSO:

```
"C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt.exe" -ng -
BatchSolve -machinelist num=2
-monitor \\shared_drive\projs\OptimTee.aedt
```

User is using LSF for remote-analysis/DSO

```
bsub -n 1 "C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt.exe" -ng
-BatchSolve -machinelist num=3 -monitor -local \\shared_
drive\projs\OptimTee.aedt
```

Distributed Job Using Four Processors

Ansoft RSM

```
"C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt.exe" -ng -
Batchsolve -monitor -Distributed
-machinelist list="10.1.1.221, 10.1.1.222, 10.1.1.223, 10.1.1.224"
\\shared_drive\projs\OptimTee.aedt
```

LSF

```
bsub -n 4 "C:\Program Files\ANSYS Inc\v251\AnsysEM\ansysedt.exe" -ng
-Batchsolve -monitor
-Distributed -machinelist num=4
\\shared_drive\projs\OptimTee.aedt
```

Multiprocessing Job Using Four Cores

Multi-processing job using 4 cores

```
bsub -n 4 -R span[ptile=4] "C:\Program Files\ANSYS
Inc\v251\AnsysEM\ansysedt.exe" -ng -monitor
-Local -BatchSolve -machinelist num=4 -batchoptions \\shared_
drive\registry.txt \\shared_drive\projs\OptimTee.aedt
```

This requests 4 cores to come from the same machine, as multi-processing needs cores to be on the same machine

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Distributed Analysis and Multi-Processing in the Same Job

Distributed-processing using 4 engines and multi-processing using 4 cores, using a total of 16 cores

```
bsub -n 16 -R "span[ptile=4]" "C:Program Files\ANSYS
Inc\v251\AnsysEM\ansysedt.exe" -ng
-BatchSolve -Distributed -machinelist num=4
-batchoptions \\shared_drive\registry.txt
\\shared_drive\projs\OptimTee.aedt
```

Job Import and Export

The bottoms of the scheduler selection and job submission window contain buttons for **Import...**, **Export...**, and **Import Configuration**. Import and Export may be used to save and then restore a frequently used collection of job submission settings, to save multiple sets of settings, or to transfer settings from one machine to another. Electronics Desktop uses Ansoft Electronics Registry Settings (*.areg) files for these purposes:

- **Export...** – exports most of the settings of this window (all tabs) to an *.areg file.
- **Import...** – updates most of the settings in this window (all tabs) from an *.areg file.
- **Import Configuration** – updates DSO settings in this window from any DSO configuration, as shown in the **Configurations** and **Options** tabs of **HPC and Analysis Options**.

> **Important:**
>
> The Design Type of the DSO configuration must match the design type of one of the designs in the project, so the **Project path** must be specified before using the **Import Configuration** button.

Scripting

The SubmitJob scripting command uses job submission settings that have been exported from the **Submit Job** window to an *.areg file. The path to this *.areg file is the first argument to the SubmitJob scripting command. See: Job Submission Scripting.

Multi-Step Job Submission

Multi-step job submission allows you to divide the simulation of a project on a cluster as two or more jobs, each of which has unique resource specifications. For example, creating the initial mesh and doing adaptive refinement can use a single machine, while frequency sweeps can easily be distributed over many machines. Breaking up the simulation into multiple steps allows

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

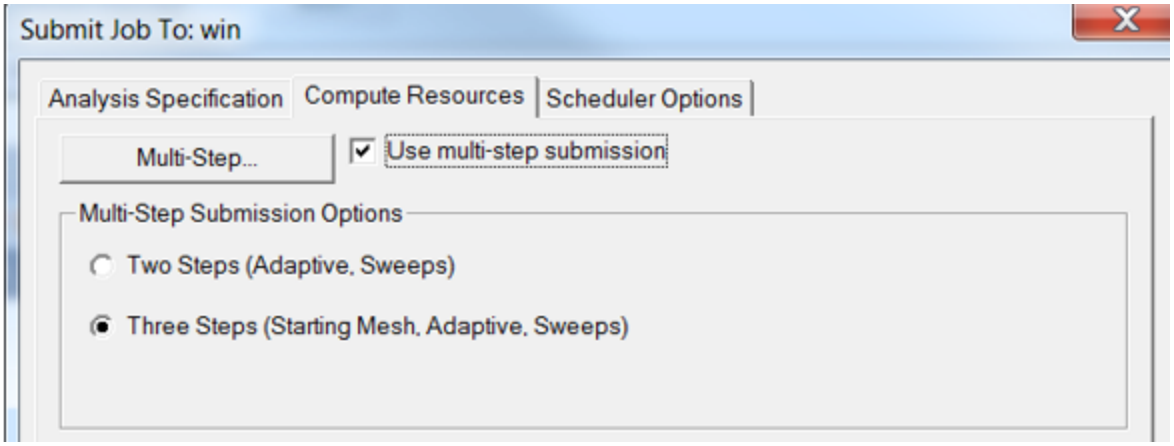
the first job to do initial meshing and adaptive passes, only reserving a single compute node, or maybe even reserving just a partial node. The second job can then do the frequency sweep(s), reserving and using multiple nodes. Note that while the first job runs, because it may only be using one node, other nodes are available for other jobs. The Ansys Electronics Desktop job submission GUI allows you to submit multi-step jobs, and specify compute resources individually for each step. Electronics Desktop can also be used to monitor the execution of multi-step jobs.

Limitations

Multi-step job submission is only allowed for a single setup of a project. Only certain design types (or setup types for a given design type) offer this functionality. Ansoft RSM does not support multi-step jobs because it does not have queuing capabilities.

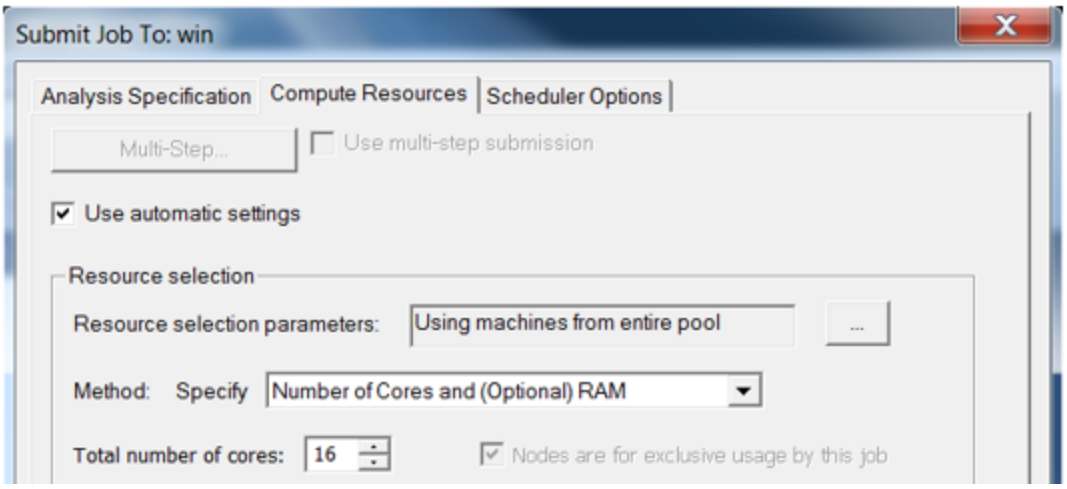
How-to Specify Multi-Step Job Submission

From an open project, right-click a setup in the project tree, and select **Submit Job...** on the shortcut menu. This pre-populates the **Submit Job To** window's **Analysis Specification** tab for the selected setup. Select the **Compute Resources** tab. If the design setup and selected scheduler allow for multi-step submission, and your computing resource supports it, the **Compute Resources** tab shows the **Multi-Step...** button and the **Use multi-step submission** check box is enabled.



If the **Multi-Step** button and check box are not enabled, it could be because you have not selected a single setup, the design type of the setup does not support Multi-Step, or the scheduler type (e.g., RSM) does not support it.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

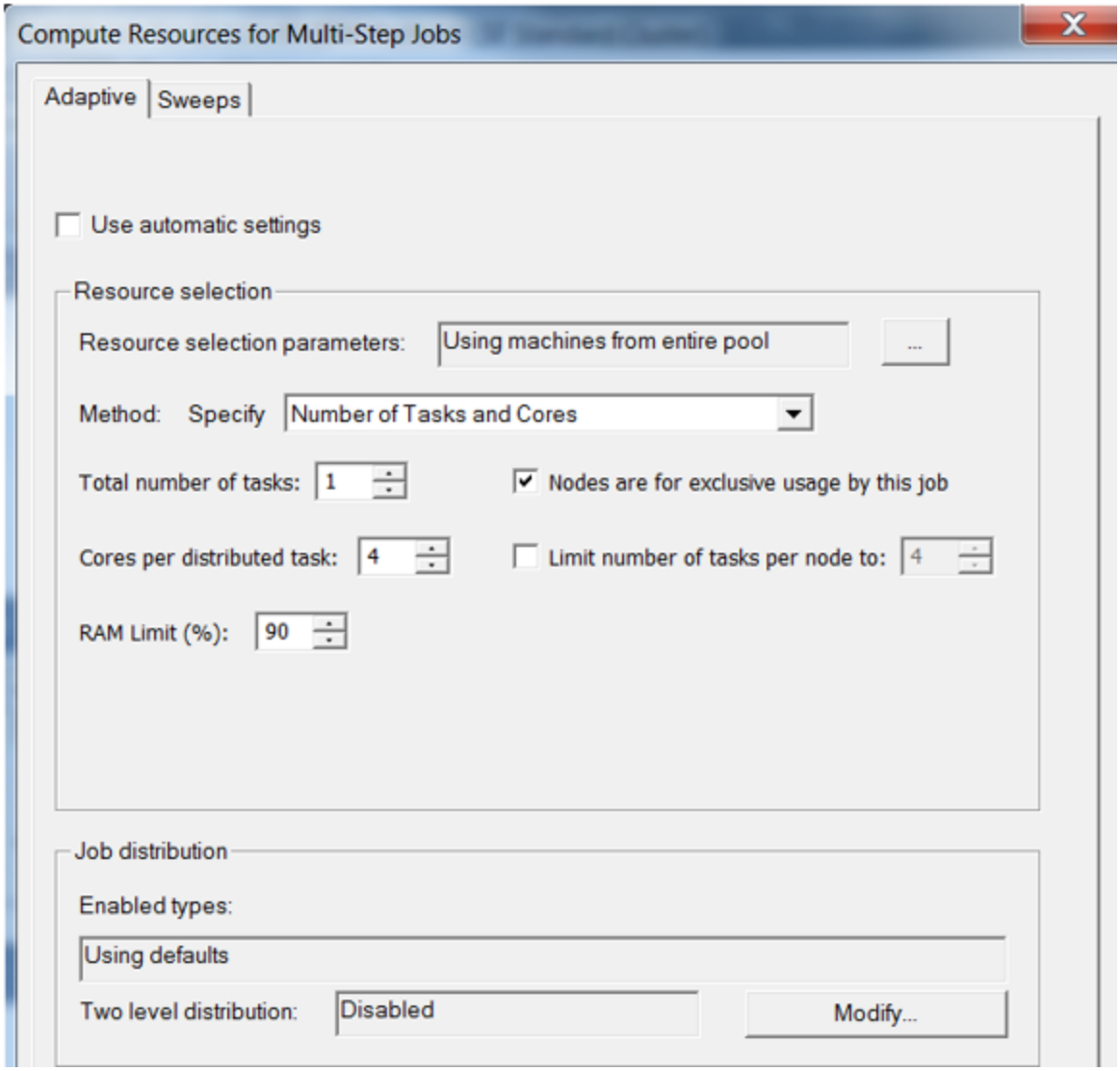


On the **Compute Resources** tab, check the **Use multi-step submission** box and select the appropriate submission option:

- Two Steps for Adaptive and Sweeps
- Three Steps for Mesh, Adaptive, and Sweeps

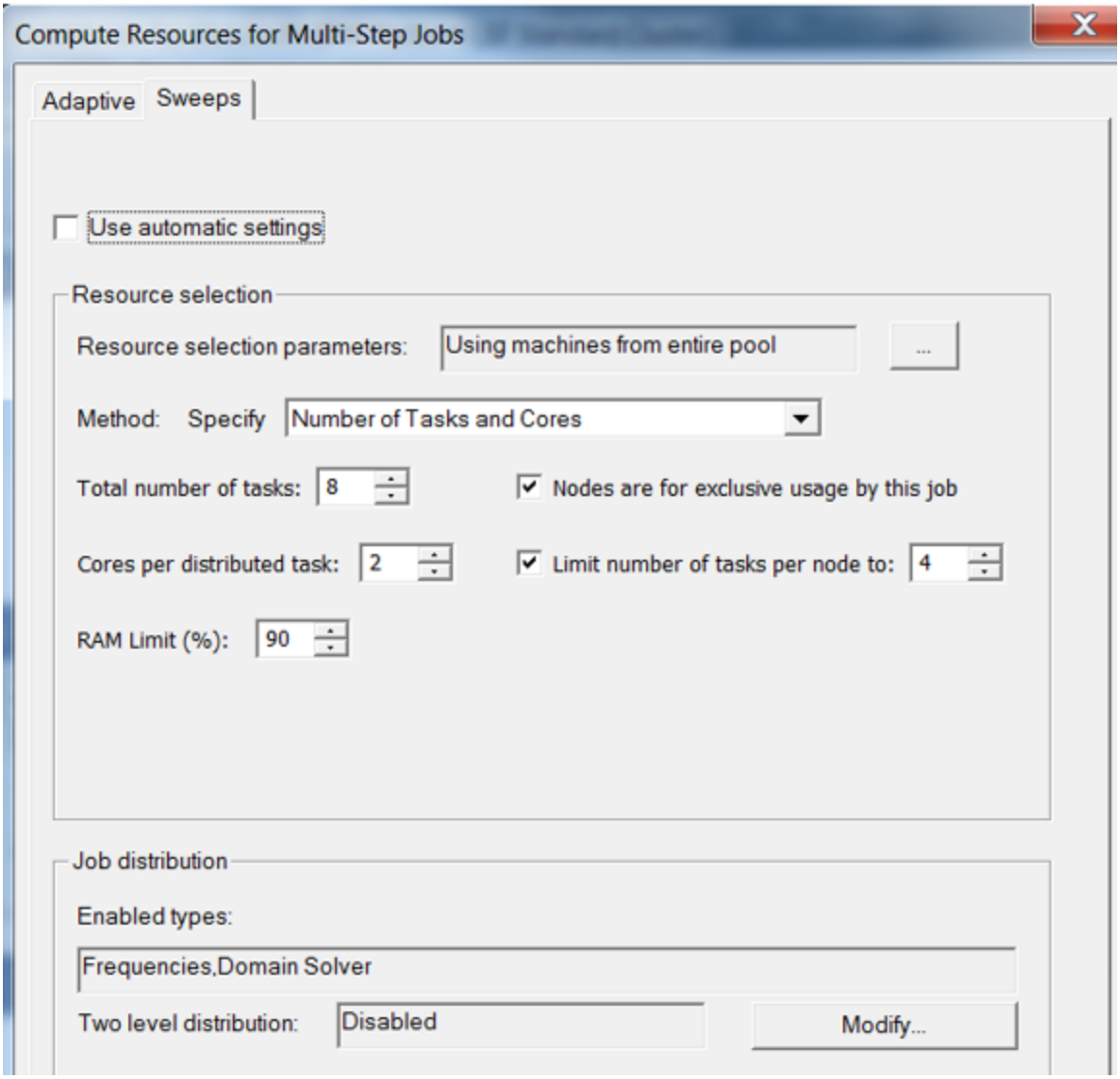
Then click **Multi-Step…**, which will bring up a **Compute Resources for Multi-Step Jobs** dialog box where there is a tab for each step used, that is for potential for Mesh, Adaptive, and Sweeps.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Notice, for example, that the **Sweeps** tab lists the same resources choices but they can be assigned differently.

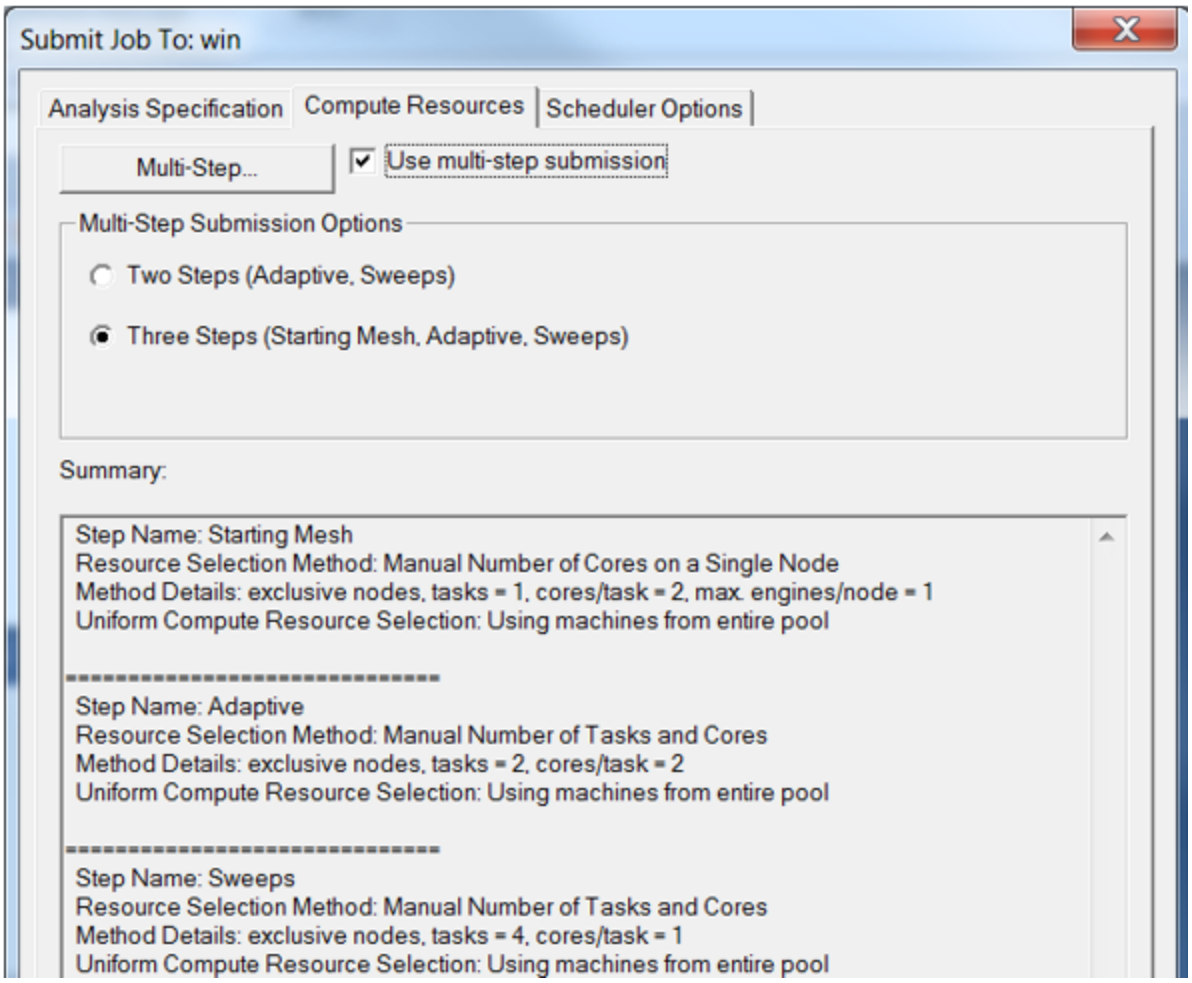
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



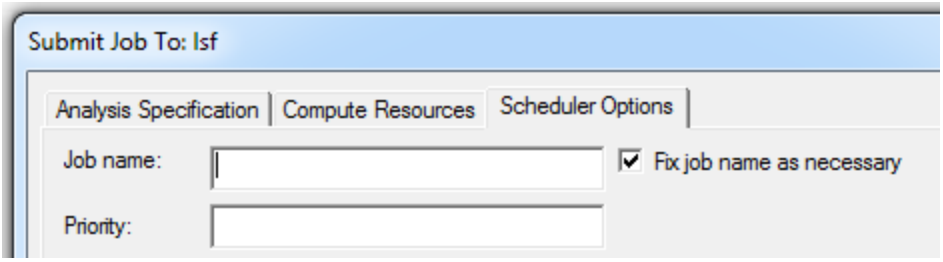
Set the desired compute resources for each step and click **OK**.

The Summary field of the **Compute Resources** tab shows a text summary of resource specifications used for each step.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



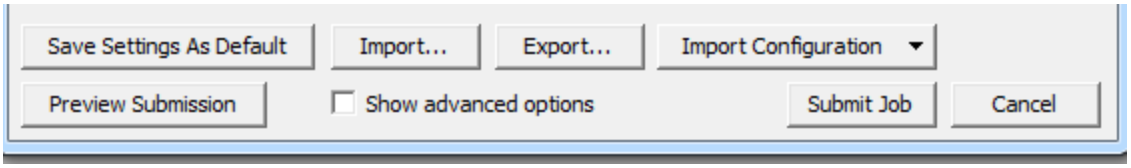
If desired, set the job name on the **Scheduler Options** tab. When the jobs are submitted, each will have a name given by job name (if any) appended with the name of the step for the job. For example, if the job name entered is “MultiStep”, then the individual jobs might be named “MultiStep-Adaptive” and “MultiStep-Sweeps”.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

There is a new **Fix job name as necessary** check box for the job name. This applies only to certain schedulers where there are constraints on the job name. It is useful to ensure proper submission in the case where appending the step name results in an invalid job name.

When you click **Preview Submission**, you see a preview for each step, all in the same output window. Any errors or warnings for a step, generated during preview validation, are displayed with the text for the preview of that step.



When you click **Submit Job**, each job is submitted individually, and dependencies are set with the scheduler so that subsequent jobs wait for the prior step's job to complete before starting. Any errors in job submission for any step prevent further steps from being submitted.

The window reporting successful submission has been modified slightly for multi-step jobs. It will show the jobs IDs of all jobs that are part of the multi-step job sequence.

Multi-Step Job Monitoring

You can monitor the job step in progress. When one job completes, the status shows "Completed", but monitoring restarts once the next job step is running.

Aborting a Job Step

You can also abort the job step in progress. With multi-step jobs submitted from the Electronics Desktop GUI, this will also cause remaining jobs (which would otherwise remain queued in the scheduler) to be canceled.

Archive Projects for Multi-Step Job Submission

Note that you can submit archive projects. Monitoring is based on the archive for the first step, then on the extracted project for subsequent steps.

Submitting a Job without Opening the Project

You can also submit a job without opening the project. This can be done by choosing **Tools** > **Job Management** > **Submit Job…** and then manually entering the project path (You can also use **Browse** can to select the project.) Note that you must also select a single nominal setup before the **Use multi-step submission** check box is enabled on the **Compute Resources** tab.

Windows to Linux Job Submission

Given a set of prerequisites, Ansys Electronics Desktop can permit Windows to Linux job submission as part of HPC.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Prerequisites for Job Submission

Directory Shared between Windows and Linux

For all jobs submitted to a Linux cluster, the project file is required to be in a directory that is accessible from all execution hosts used by the job. For submission of jobs from a Windows host to a Linux cluster, the project file must also be accessible from the Windows host where the GUI runs. There must be a directory shared with both Windows and Linux hosts, and the project file may be in a subdirectory (at any level) of the shared directory.

Network Access from Windows Host to Linux Job Management Host

The job is submitted to the cluster from a Linux host configured for submission of jobs to the Linux cluster. We call this Linux host the “Job Management” host. The information about the job to be submitted is transmitted to the Job Management host over the network. As a result, the Windows host where the GUI runs must have network access to the Job Management host. If this communication is blocked, then job submission from a Windows host to the Linux cluster will not be possible. Communication could be blocked if there is a firewall or if the Linux cluster is only on a private network, for example.

RSM Service Running on Job Management Host

The **ansoftrsmservice** must be running on the Linux Job Management host. Before the ansoftrsmservice is started, it must be configured for submission of jobs to the cluster. The SchedulerName and ConfigString fields in the Scheduler block of the ansoftrsmservice.cfg configuration file must be specified. The contents of these fields are described in the following table:

Field Name	Contents	Examples
SchedulerName	Identifier of Scheduler	IBM Spectrum LSF: ’`lsf`’ PBSPro or Torque: ’`pbs`’ Univa, SGE, etc.: ’`sge`’ SLURM: 'generic'
ConfigString	Directory containing scheduler commands	IBM Spectrum LSF: ’’ (not required) PBSPro or Torque: ’`/opt/pbspro/PBSPro_13.0.0/default/bin`’ Univa, SGE, etc.: ’`/opt/univa/bin/lx-amd64`’ SLURM:'{"Proxy":"slurm"}'

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The environment should be configured for job submission before starting the ansoftrsmservice. The ansoftrsmservice should be run as a non-privileged user; no special privileges are required. It should be run as a user without login privileges, so that only privileged users have access to this process.

Prerequisites for Job Monitoring

For job monitoring, all prerequisites for job submission are required. One additional requirement, described below, is also required for job monitoring.

Network Access from Windows Host to Linux Cluster Hosts

In order to obtain full monitoring information from a job, the Windows host needs access to some of the job processes. That is, the Windows monitoring host requires network access to the processes running on the Linux cluster execution hosts. If this communication is blocked, then only limited monitoring information is available.

Supported Schedulers

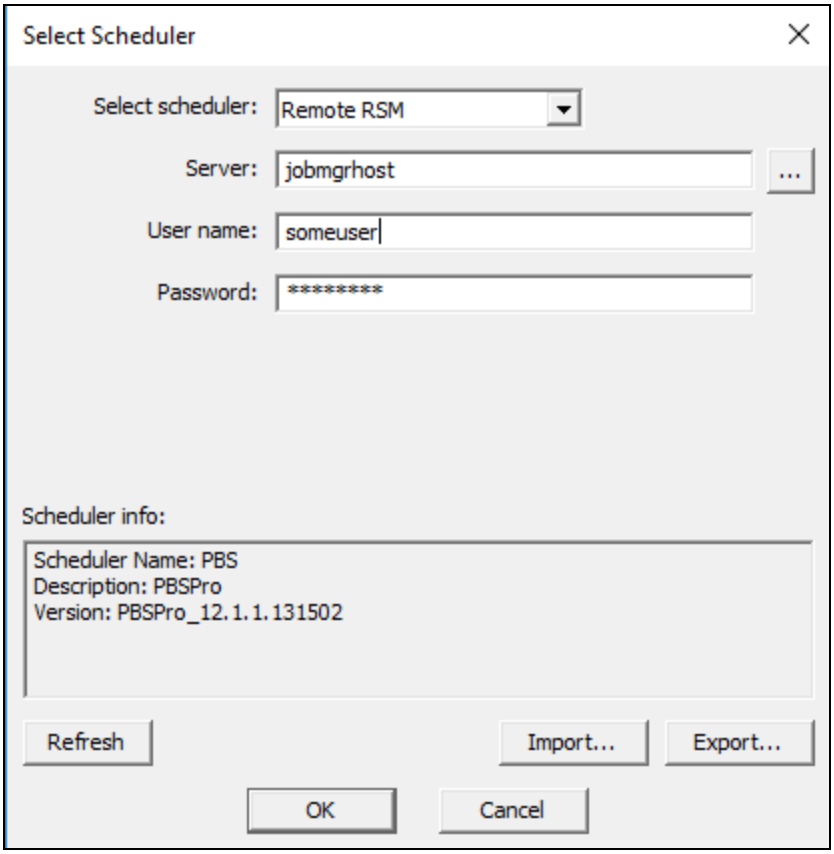
This feature may be used with all Linux schedulers for which job submission from the GUI is supported:

- IBM Spectrum LSF
- Univa Grid Engine (GE)
- PBSPro
- Torque

Select Scheduler Dialog

If you select the **Use a computer on the network** option, you can enter a username and password. This username and password are used when the job is submitted to the Linux scheduler.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

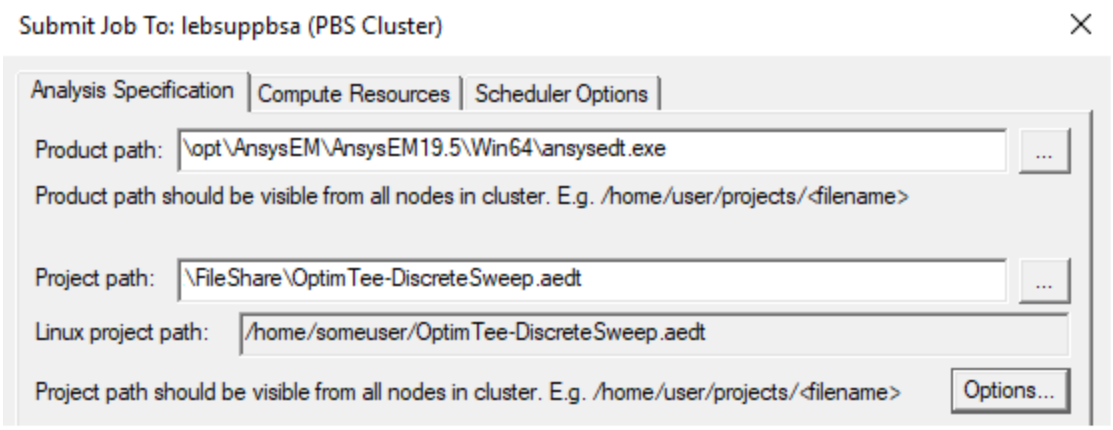


Submit Job

In the ***Submit Job To*** window, you must enter the Linux path to the product in the Product path edit control. The browse button (labeled "...") may not be used to browse for the product. There is no requirement for the product installation directory to be accessible from the Windows GUI host.

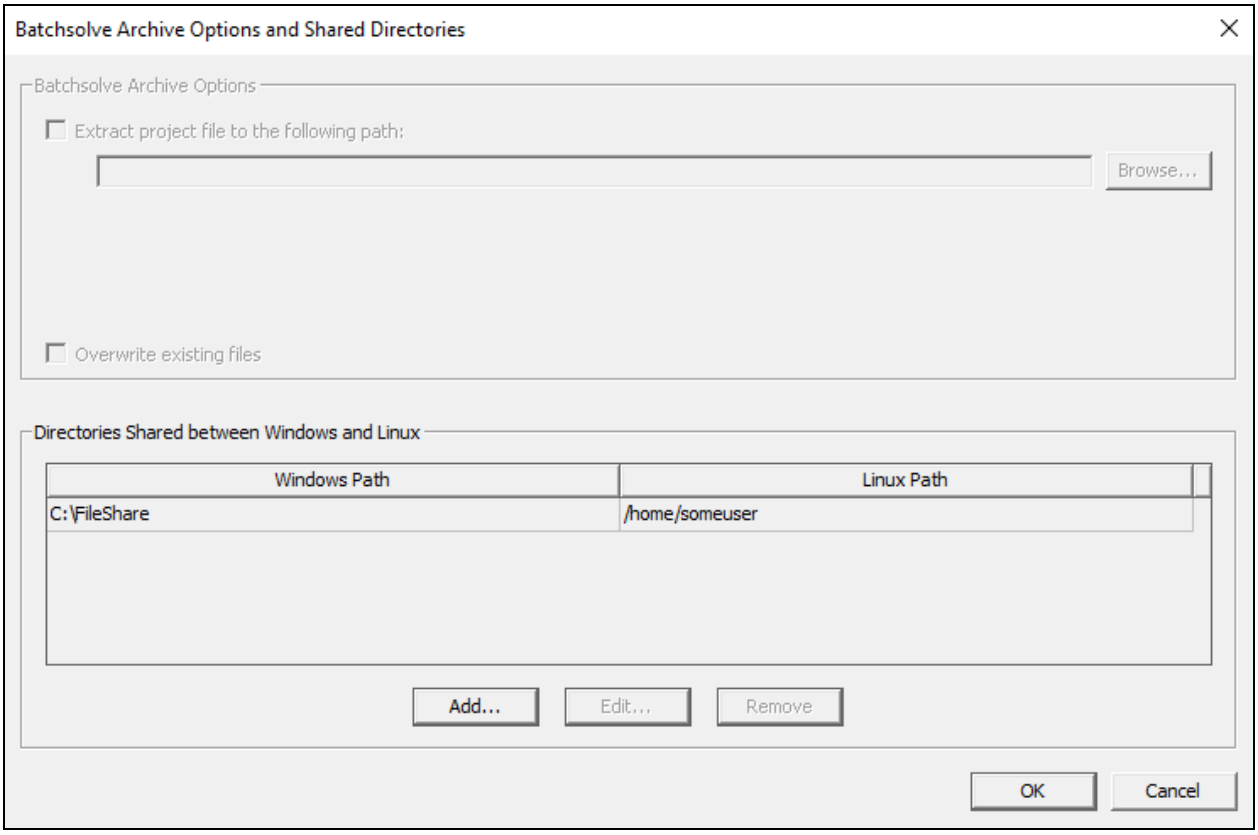
You must enter the Windows path of the project file in the Product path edit control or use the browse button (labeled "...") to select the Windows path of the project file. If the Linux path of the project file can be determined from the specified Windows path and the directories shared between Windows and Linux, then the Linux path of the project file is shown in the Linux project path edit control. This edit control cannot be edited directly.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



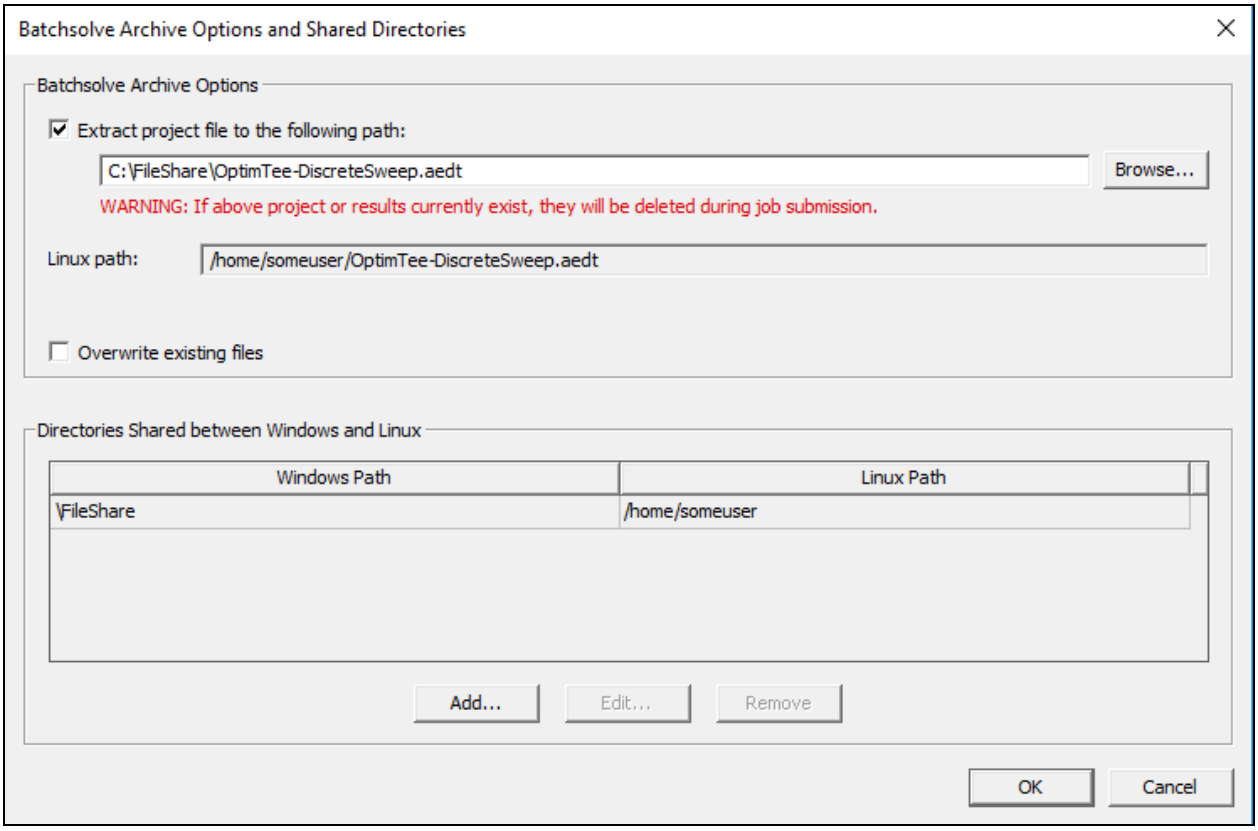
The **Options** button opens a window that you can use to specify archive options for a job. The lower portion of this window allows you to specify one or more directories shared between Windows and Linux. The mapping of directories between Windows and Linux is shown in a grid which displays the Windows path and the Linux path for each shared directory. There are also buttons to add a new shared directory, to edit an existing shared directory, or to delete one or more shared directories. The Windows path or the Linux path of any shared directory may be selected in the grid and directly edited, as well.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



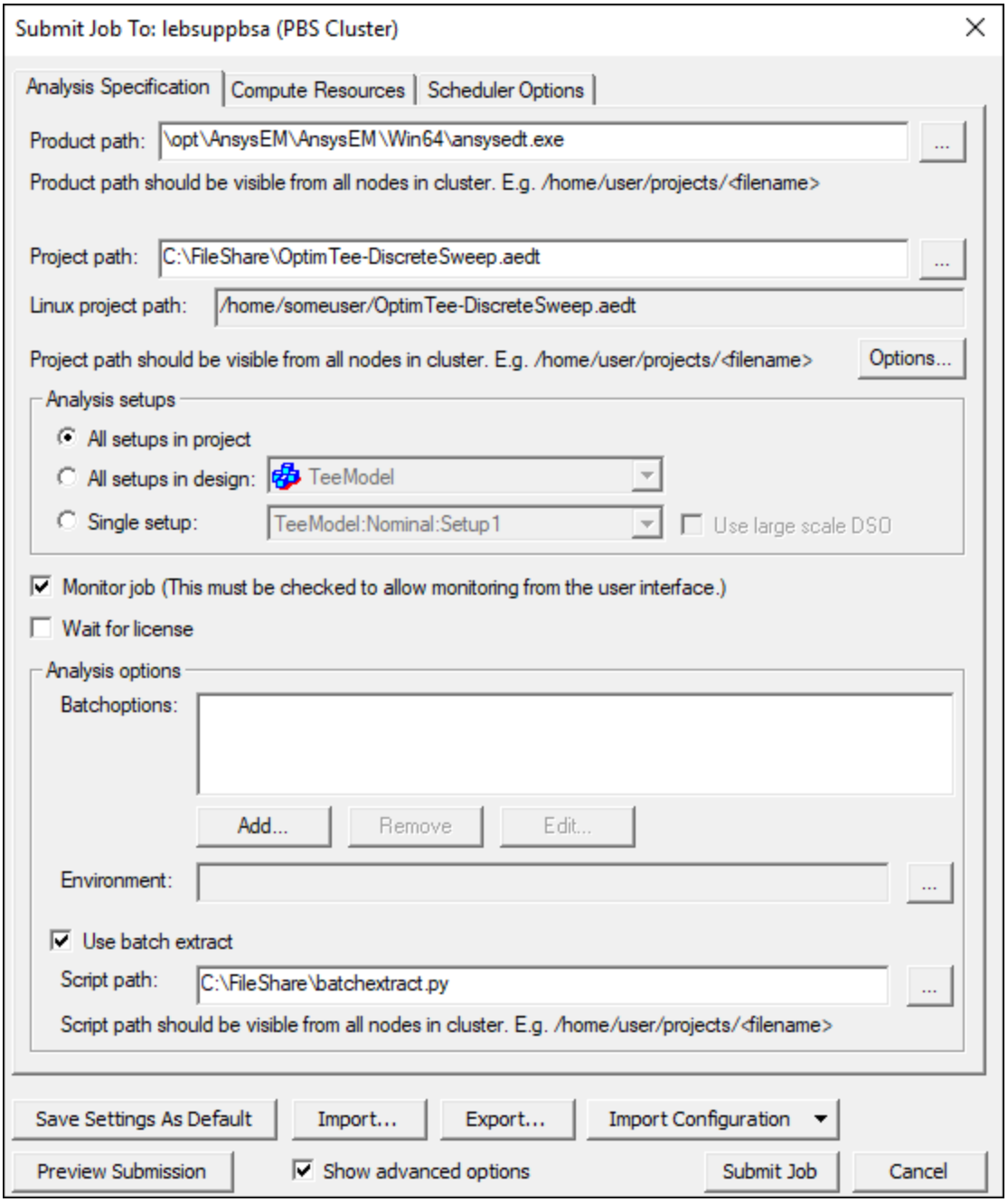
If you specify a project in an archive, the window may be used to specify the Windows pathname of the project to be extracted from the archive. If this is done, the Linux pathname of the target project is determined from the directories shared between Windows and Linux and shown in the upper portion of this window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you specify a batchextract script, the Linux path of the batchextract script is determined from the Windows path of the batchextract script and the directories shared between Windows and Linux.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



User Passwords are Encrypted

Jobs are submitted to the Linux cluster using the user name and password entered in the **Select Scheduler** window. These settings are persistent; in general, these settings need to be entered only if they change. To ensure security, user passwords are stored in an encrypted format. When a job is submitted from a Windows host using the ansoftrsmservice running on a Linux submission host, the user credentials are sent over the network in an encrypted format.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Job Submission Scripting

To help with automation, you can submit batch jobs can through script commands of the **oDesktop** object. The **SubmitJob** script command uses job submission settings that have been exported from the **Submit Job** window to an .areg file. The path to this .areg file is thus the first argument to the SubmitJob command. Additional arguments include the path to the project file, the design name (if restricting the solve to a particular design), and the setup name (if further restricting the solve to a single setup within a design).

For further automation, you can use the **SelectScheduler** scripting command to determine what scheduler to use for submission, to include options for head node, username, and whether to require password entry from the user. (If the username differs from the cached username, or the force password flag is set, then the *Select Scheduler* dialog box appears.) If there are any issues with the scheduler selection (for example, a password is required or the requested host wasn't found), then the *Select Scheduler* window appears. If Ansys Cloud Direct or Ansys Cloud Burst Compute are selected then the "Login" button will open the default browser and prompt for an AnsysID login. This is the only part of job submission scripting that may required user intervention. This same mechanism is used if, from within the SubmitJob command, there is failure to connect to the scheduler. Even though there are allowances for graphical user intervention if something goes wrong, if the password (if required) is cached and all settings are correct, the entire submission process can run non-graphically and fully automated.

Limitations

All settings besides the arguments passed to the **SubmitJob** command must be stored in the .areg file containing settings exported from the job submission window. These include (but aren't limited to) batch options, environment variables, batch extract settings, and compute resource selections. To run many job submission scripts with variation of these settings, there must be multiple .areg files available.

Note that the same project can be submitted multiple times with a single script. Care must be taken in this situation because each time a project is submitted, the state-keeping files used for monitoring are removed so that the job can create them from scratch to ensure consistency. While this ensures proper monitoring for a job that is just being submitted, it could interfere with monitoring (or even correct solving if a lock file is deleted) of a job that is already in progress. Because of this, if the same project is to be resubmitted from within a single script, the job should be monitored (waiting for completion) before trying to submit it again. This monitoring/waiting can be done with a combination of a single **LaunchJobMonitor** command followed by a loop that checks the result of a **RefreshJobMonitor** command.

How to Perform Job Submission Scripting

The typical scenario for job submission scripting would be to do the following:

1. Manually select the scheduler. Use the **Select Scheduler** window to open the **Submit Job To** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Choose a representative project (with the desired design type), and select appropriate analysis settings.
3. Make the required compute resource selections and try to preview the job.
4. If preview is successful, export the dialog settings and record the path to the new .areg file.
5. Create a script containing at a minimum a **SubmitJob** command with the path to the .areg file, and the path to the project file. Note that there must be double backslashes for each backslash of a path because the backslash is an escape character. When the script is run, and all is successful, there should be a message in the message windows stating that the job was submitted, including the job ID(s). There could be multiple job IDs if multi-step submission is used.

See the Scripting help (click **Help** > **HFSSScripting Help**) for details on the **SelectScheduler**, **SubmitJob**, **LaunchJobMonitor** and **RefreshJobMonitor** commands.

Monitoring Jobs

There are a number of tools that allow you to monitor jobs, including the **Monitor Job** window and detailed .log files.

- Monitor Job Window
- Monitoring Ansys Cloud Direct Jobs
- Monitoring Ansys Cloud Burst Jobs
- Monitoring Large Scale DSO Jobs
- Web Client for Batch Solve Monitoring and Reporting

Monitor Job Window

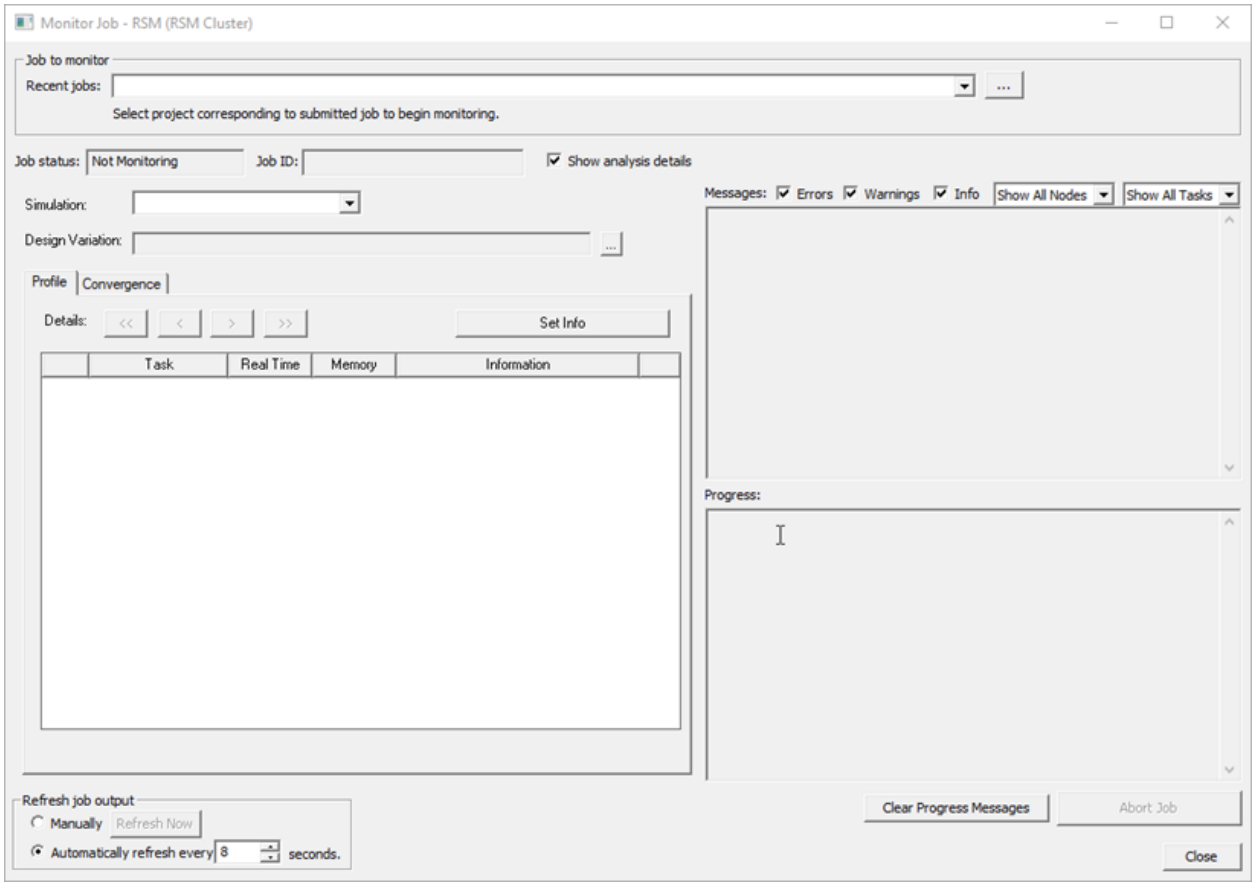
The **Monitor Job** window allows you to monitor the progress and status of jobs, including information on variations solved so far, variations currently solving, and the number of variations remaining.

To monitor jobs:

1. Access the Monitor Job window one of the following ways:
   - Click **Tools** > **Job Management** > **Monitor Jobs**.
   - Select the **Simulation** tab and click the **Monitor** icon.
   - From a Command window, use the -showmonitorjobs command.

   The **Monitor Job** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> The window may look slightly different, depending on your selected scheduler and design type.

The **Monitor Job** window contains the following areas:

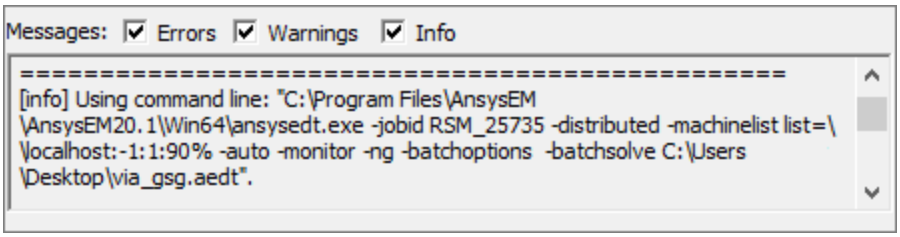
- **Job to Monitor** – Allows you to select either a recent job, or a project containing the job you want to monitor. You can select an archive.
- **Job Status** – The current job status. The normal progression is: Starting Monitoring, Queued, Running, Shutting Down, Completed. A status of Unknown may indicate a connection problem.
- **Job ID** – Identified by the scheduler prefix and job number.
- **Simulation** – Drop-down menu that allows you to select an individual simulation setup.
- **Design Variation** – Click the ellipses button (**...**) to select or deselect design variations.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

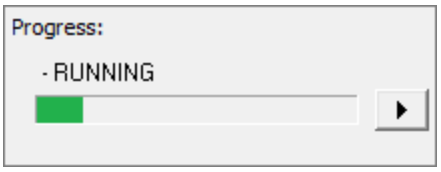
- **Profile Tab** – Displays detailed information about completed tasks, including execution time and memory usage.

Task	Real Time	CPU Time	Memory	Information
Start				Time: 10/16/2019 14:17:38; Host:: Processor: 12;
				Executing from C:\Program Files\AnsysEM\AnsysEM20.1\Win64\Q
				Performing minimal design validations
InitMesh	00:00:08	00:00:07	106 M	70131 triangles
PreProc	00:00:00	00:00:00	106 M	70131 triangles
AdaptMesh_1	00:00:01	00:00:00	107 M	78514 triangles
Solve(1p)_1	00:00:00	00:00:00	126 M	51207 unknowns
ErrorCalc_1	00:00:00	00:00:00	126 M	78514 triangles
PostProc	00:00:01	00:00:00	126 M	995 elements
Solution Process				Elapsed Time: 00:02:15
Stop				Time: 10/16/2019 15:15:30; Status: Normal Completion

- **Convergence Tab** – Displays the completed, maximum, and minimum **Number of Passes**.
- **Messages** – Displays errors, warnings, and job information. Use the check boxes to choose which of these to display.



- **Progress** – Displays progress bars when tasks are currently in progress.



The arrow button opens a menu that allows you to either **Abort** that analysis, or perform a **Clean Stop**.

- **Refresh Job Output** – Allows you to select either manual or automatic refresh for the **Monitor Job** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

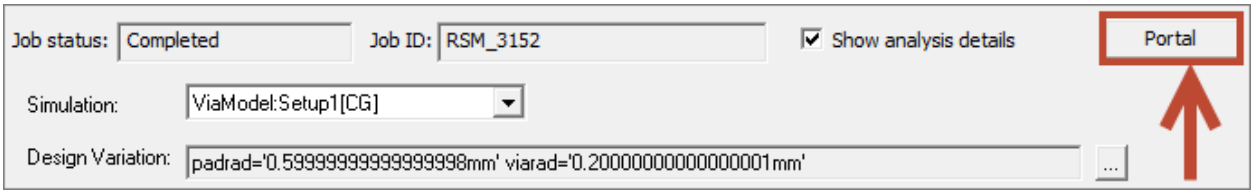
To display less information, deselect **Show analysis details**. This removes the **Simulation**, **Design Variation**, **Profile**, and **Convergence** information, instead displaying only **Messages** and **Progress**.

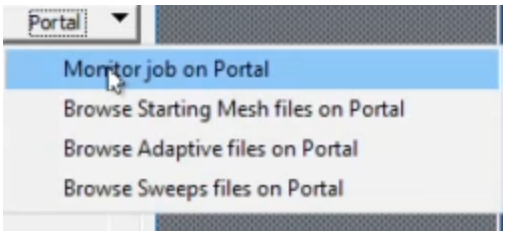
2. Click **Close** when you are done monitoring.

Monitoring Ansys Cloud Direct Jobs

For Ansys Cloud Direct submissions, the **Monitor Job** window includes two additional items:

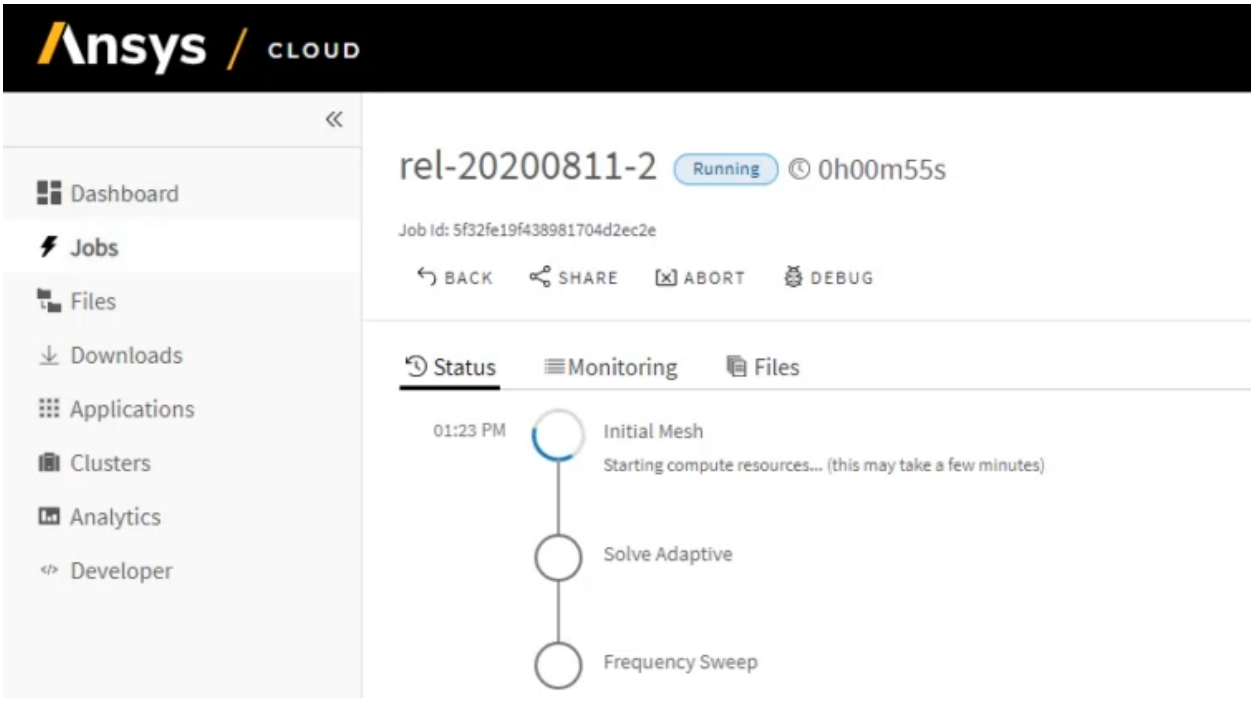
- **Portal** – link to the Ansys Cloud Direct Portal, where job details and additional monitoring information are available.



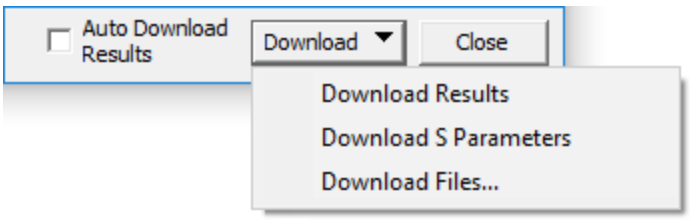


Monitor job on Portal Opens a Portal Web interface that lets you monitor an Ansys Cloud Direct job. The additional menu options for **Portal** are for multi-step submissions, opening the Portal Web interface directly to folders with Starting Mesh Files, Adaptive Files, or Sweeps files.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

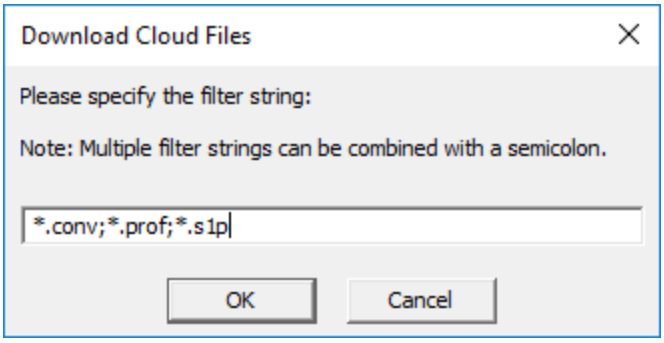


- **Download** – button that allows you to download the results, s-parameters, or specified files. These are saved in a folder with same name as the job ID inside the `/<project-name>.aedtdownload` folder (or `/<project-name>.aedtzdownload` folder, if the submission was based on an archive). You can also select the **Auto Download Results** check box to have the results downloaded to this location automatically.



If you choose to **Download Files**, the **Download Cloud Files** dialog box appears:

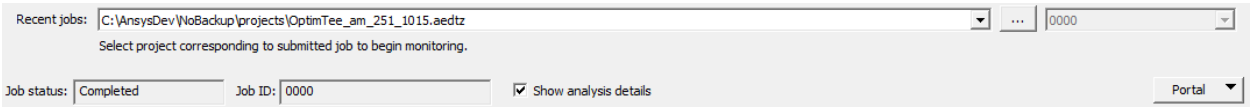
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



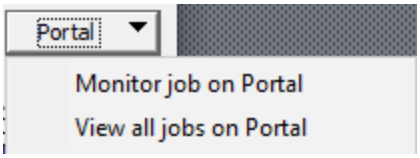
Specify the types of files you want to download (separated by semicolons) and click **OK**.

Monitoring Ansys Cloud Burst Jobs

When you submit a job to Ansys Cloud, the job status is shown in the **Monitor Jobs** window. The Job ID shown here is a short numeric identifier used to distinguish multiple jobs for the same project. Use the drop-down list at the upper right to select a different job for the same project. **The Monitor Job** window includes two additional items, **Portal**, and **Download**.



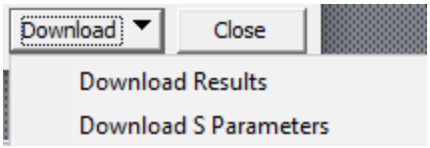
The **Portal** drop-down button provides a link to the Ansys Portal, where job details and additional monitoring information are available. For more information, see the Ansys Portal Guide.



Monitor job on Portal opens a job details view for the current job in Ansys Portal. **View all jobs on Portal** opens a job list view in Ansys Portal, enabling you to select a job to monitor.

Download allows you to download the results or s-parameters. These are saved in a folder with same name as the job ID inside the /<*project-name*>.`aedtdownload` folder (or /<*project-name*>.`aedtzdownload` folder, if the submission was based on an archive). You can also select the **Auto Download Results** check box to have the results downloaded to this location automatically.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

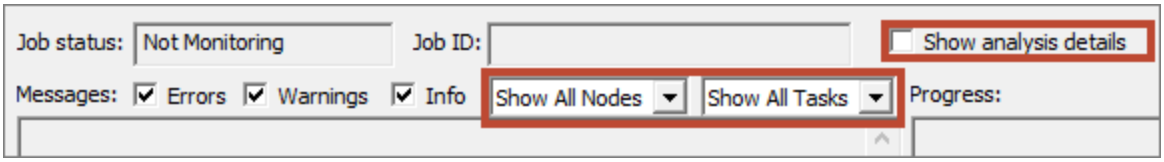


Monitoring Large Scale DSO Jobs

For Large Scale DSO submissions, the Monitor Job window includes the following items:

- **Show Messages in Node Tree** – Instead of viewing job details, you can select this option to view nodes and tasks in a tree format.

- Drop-down boxes allow you to filter which Nodes and Tasks to display in the tree.



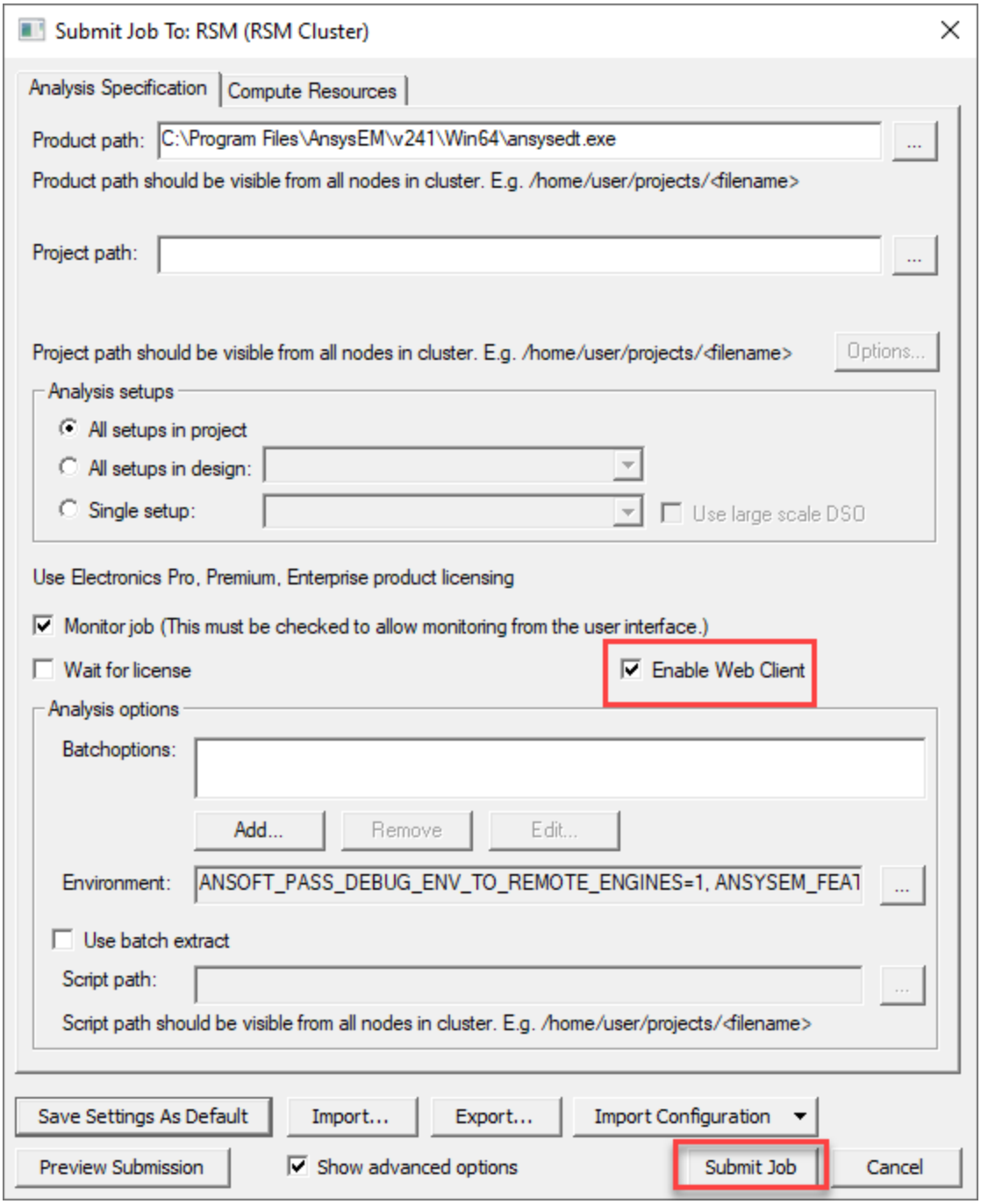
To leave this mode, select **Show analysis details**.

Web Client for Batch Solve Monitoring and Reporting

There is a Beta Option to use a Web client for batch solve monitoring and reporting. If you set the Beta Option f under General for Enable Web Client for Monitoring, you will have access to the feature.

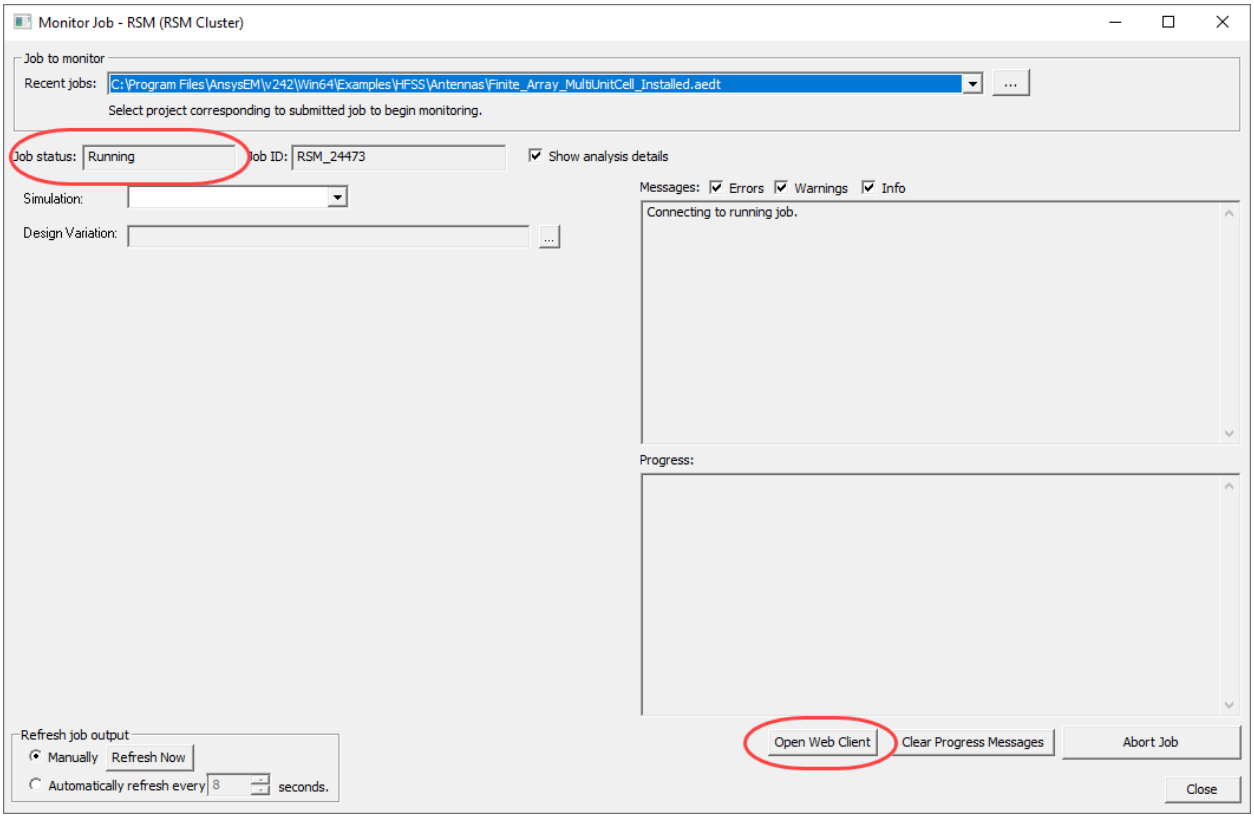
After you have selected the scheduler to use, when you access the **Submit Job To:** dialog, you well see Enable Web Client checked.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



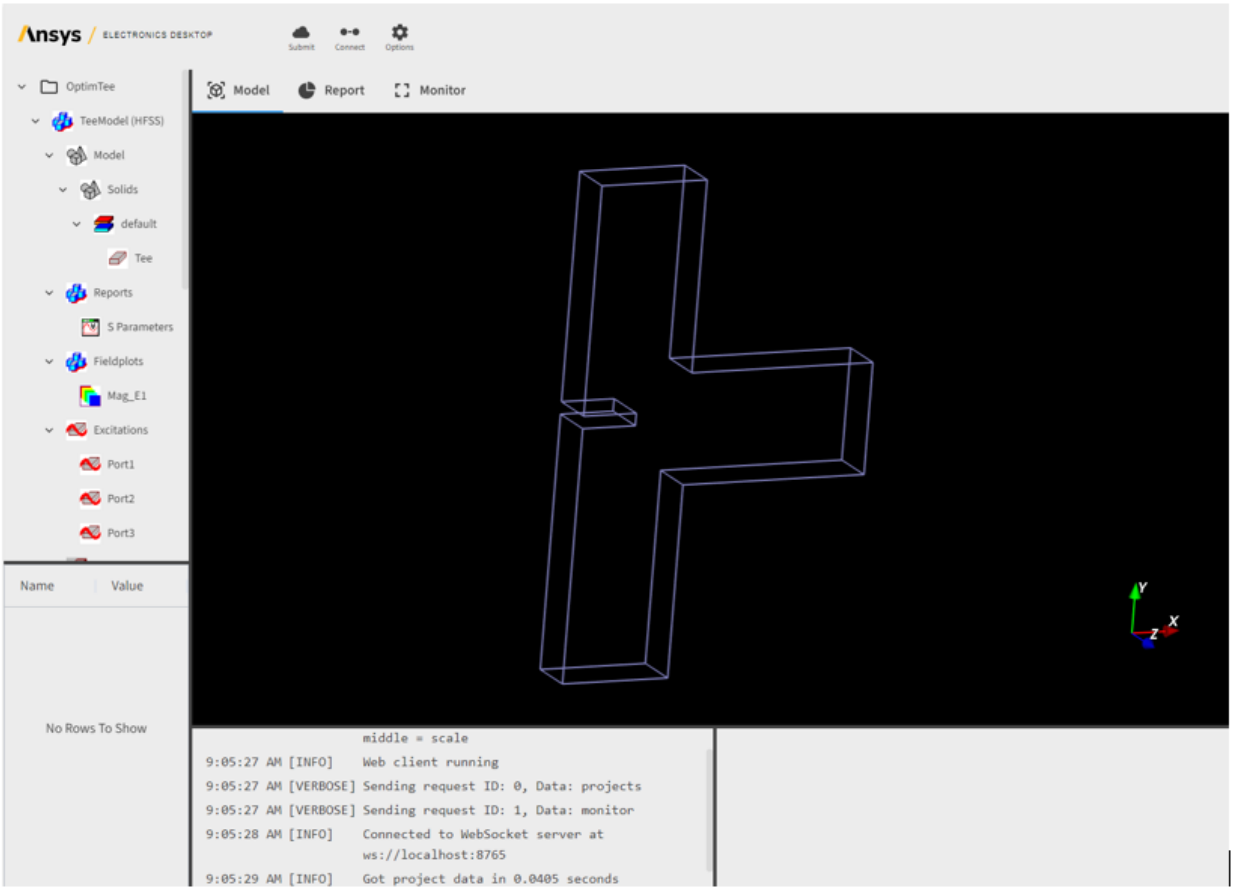
After specifying the project path and setting other parameters, you can then select **Submit Job**. This opens the Monitor Job dialog for your scheduler. Once the **Job Status** changes to Running, press **Open Web Client**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



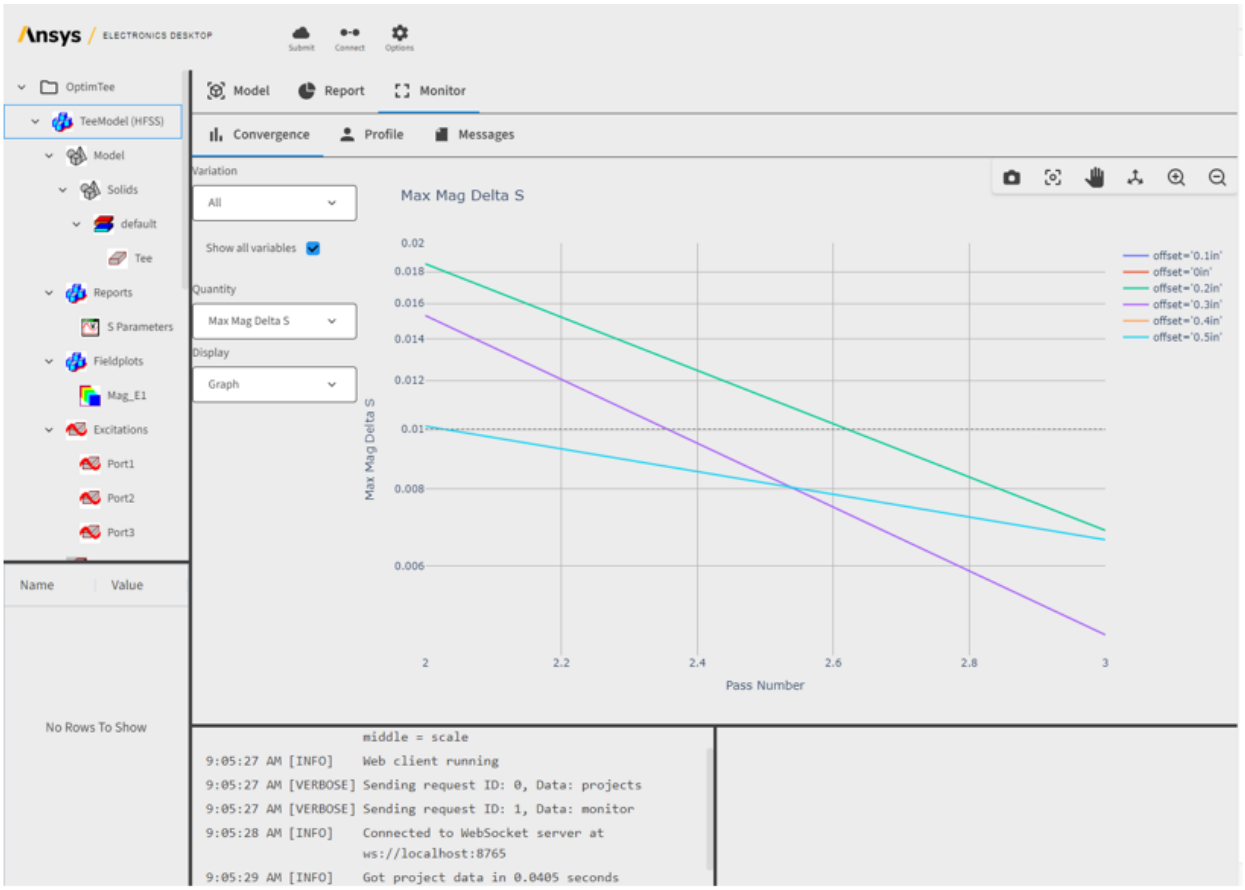
Click **Open Web Client** to open the Web Client in your default browser.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



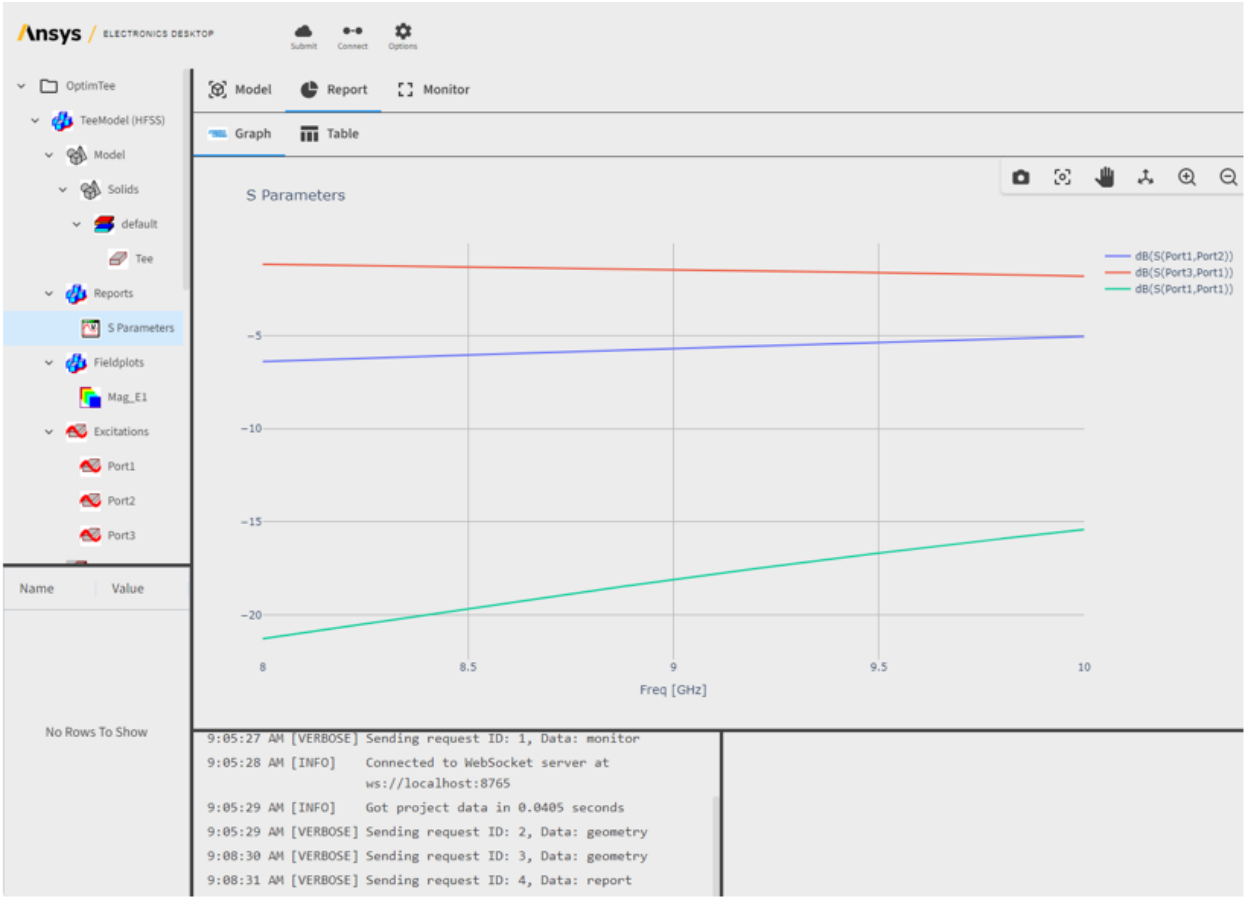
Switch to the **Monitor** tab to gain access to the subtabs for Convergence, Profile and Messages.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



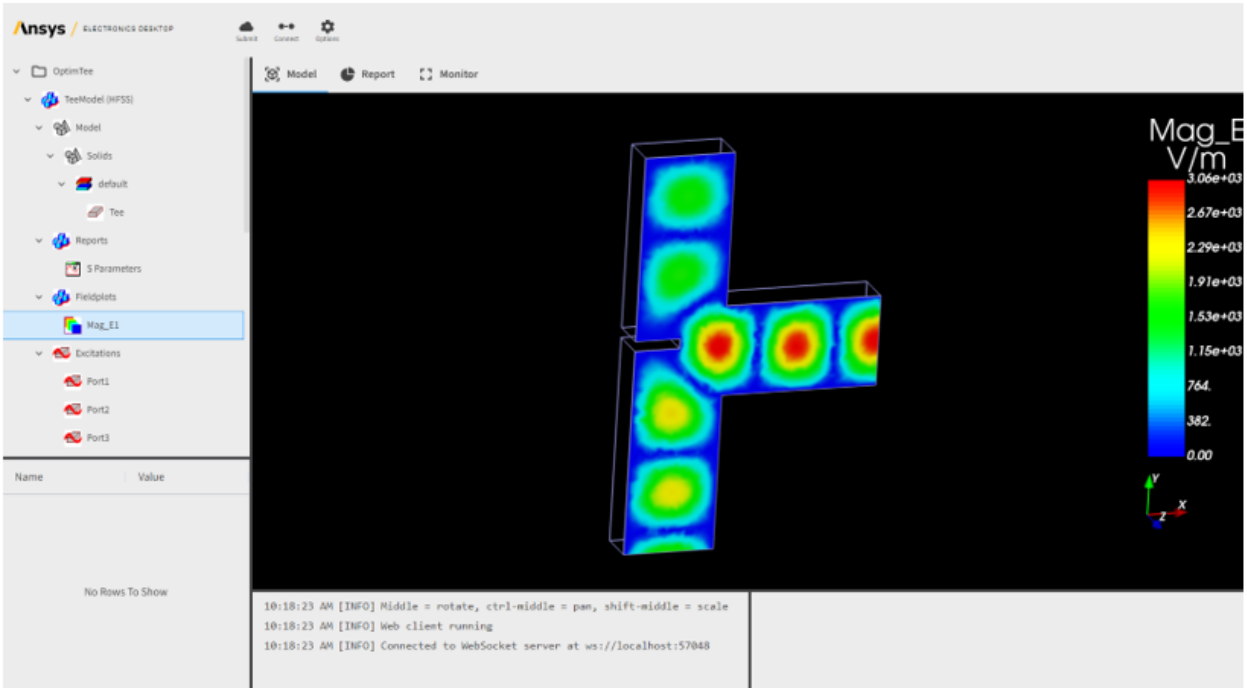
Select a Report in the **Project Tree** to view it.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Select a Field Plot in the **Project Tree** to view it.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Aborting an Analysis

To end the solution process before it is complete:

- Right-click In the **Progress** window and click **Abort** or click on drop-down button next to progress bar(s).

  The solver ends the analysis immediately.

The solver ends the analysis immediately. The data for the currently running adaptive pass, frequency point, time step, or variation is lost. Previously completed solutions may or may not be retained (depending on what process was interrupted by the **Abort** command). If you want to ensure that the results of all completed solutions are saved, use **Clean Stop** instead of **Abort**.

To abort the solution process after the current adaptive pass or solved frequency point is complete:

- Right-click the **Progress** window, or click on drop-down button next to progress bar(s) and click **Clean Stop** on the shortcut menu.

  The analysis ends after the currently running adaptive pass, frequency point, time step, or variation has been solved. Solutions completed before the stop request are retained.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you request a clean stop during the third adaptive pass, the solution for the third pass will be available once the third pass has finished solving, but the fourth pass will not run.

Ansys EM Application as an LSF Job

If you have an Ansys EM application running as an LSF job, you can use the command "bkill -s SIGTERM *jobid*" to terminate that application. Here *jobid* is the LSF job id. The response will be "Job <jobid> is being signaled". The response is the same whether the job is actually being signaled or not.

In cases where the SIGTERM parameter is ignored, the command kills the LSF job, but does not clean the lock files, and other files may not be in a consistent state.

Linux

For Linux, you can use TERM commands. Sigterm handling is done in the Desktop library. You can abort a running batchsolve by sending a TERM signal to hfss.exe.

Distribution Command Line Options

The user should include options specifying how the job should be distributed in the command line. These command line options will override options specified using the **HPC and Analysis Options** dialog box or obtained from the registry. See Running Ansys Electronics Desktop from a command line for more information on command line options.

For both scheduler batch jobs and interactive scheduler jobs, the -MachineList num=<num distributed tasks> format is the most common way to specify the number of tasks for the job. The other formats (-MachineList list=... or -MachineList file=...) allow the user to specify the number of tasks and cores to use on each host. These formats may be useful with clusters of heterogeneous machines, by allowing the user to specify different numbers of tasks or cores for different hosts. If either of the latter two formats is used, the user must ensure that the hosts and cores specified on the product command line are compatible with the hosts and cores allocated to the job.

Batchoptions for Interactive Scheduler Jobs

For interactive scheduler jobs, only a limited set of batchoptions are supported. These batchoptions include the DSO configuration options, the design-type-specific options, and the following additional design-type-specific options that are not currently listed in the user interface or command line help windows:

- **CreateStartingMesh:** Create the starting mesh only. Do not solve any other steps. Often used for multi-step jobs where the mesh creation is solved in a separate step using fewer resources than the rest of the analysis.
- **NumCoresPerDistributedTask:** Specifies the number of cores that are allocated to the job to use for each distributed task, used when running jobs under a scheduler. The

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

scheduler communicates the number of cores allocated to the job on each host. This setting is used to determine how to allocate the cores to each task.

- **RAMLimitPercent:** Specifies the percentage of system RAM that the analysis is expected to use on each host. Running out of memory can cause processes to be killed by the Operating System or other failures. The analysis will be gracefully terminated if it requires more than this limit. Typical limits are about 90%, to allow some RAM for the Operating System and other services. If there are multiple jobs running on a single host, then the RAM limit should be reduced accordingly. For example, if there are two jobs running on a host, then the limit could be 45%, which would allow 10% of the RAM for the Operating System and other services.
- **RAMLimitPerCoreInGB:** This is an alternative way to specify the RAM limit, instead of using the RAMLimitPercent. In this case, the RAM limit is the limit per core multiplied by the number of cores allocated to the job on each host. Typically used for jobs running under a scheduler to ensure that the scheduler memory limit and the ansysedt memory limit are consistent.
- **SolveAdaptiveOnly:** Solve adaptive passes only. Do not solve any other steps. Often used for multi-step jobs where the adaptive passes are solved in a separate step using fewer resources than the rest of the analysis.
- **TotalNumOfCores:** The total number of cores for the job. This option is only used for EKM (Engineering Knowledge Manager).
- **ValidateOnly:** Only validate the specified setup, design, or project. Do not analyze the specified setup, design or project.

Any other batchoptions will result in a warning message and will be ignored.

Example:

```
ansysedt -batchoptions " 'TempDirectory'='C:\\TEMP'
'HFSS/SelectedDSOConfiguration'='Local'
'Desktop/Settings/ProjectOptions/DoAutoSave'=1
'LargeScaleDSO/MaxFolderInMB'=100 "
```

Batchoptions

Batchoptions may be specified in the command line used to launch the product. Any valid batchoptions specified in the command line will override the associated registry settings. Batchoptions also override options specified using **HPC and Analysis Options** window or other dialog boxes used to specify options.

Setting the Number of Cores per Distributed Task

When submitting a job using the Ansys Electromagnetics Desktop job submission GUI, the number of cores per distributed task for a job is specified using the batchoption with pathname '<*DesignType*>/NumCoresPerDistributedTask', where <*DesignType*> is the design type to

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

analyze. The batchoption setting is automatically included in the product command line when the job is submitted to the scheduler.

For interactive scheduler jobs, the user must include the associated batchoption setting or settings in the product command line when the product is launched. Multiple batchoption settings are required if the user analyzes multiple design types using the same product process. Batchoptions are the only way to specify this setting for batch jobs. There is an alternative to using the NumCoresPerDistributedTask batchoption for Interactive Scheduler Jobs, . The user may specify the total number of tasks and the total number of cores for each machine using the **Machines** tab of the **Analysis Configuration** window. You may use **Edit** in the **HPC and Analysis Options** to open the **Analysis Configuration** window for the "Interactive Scheduler Job" configuration.

Setting the Ram Limit Per Core in GB

When cores and RAM per core are requested, the cores could be allocated in an arbitrary/non-uniform way across nodes that themselves could be non-uniform/heterogeneous. For example, nodes could range from 4 cores to 20 cores and from 64 GB to 384 GB. In such environments, the RAM percentage set in the HPC configuration, Job Submission Compute Resources, or batch option may not be appropriate. In such cases, you can use the batchoption for RAMLimitPerCoreInGB. This setting specifies the Maximum amount of RAM used for each core allocated by the scheduler in GB. This batchoption cannot b e combined with RAM limit percent and is only valid when solving in a Linux scheduler environment.

The scheduler GUI automatically passes this new batch option instead of percent limit. From a scheduler GUI, such a request is available only for auto.

You can also use this new batch option for command line submission. The desktop does the computations and passes the percent limit to product/solver.

Setting the Remote Spawn Command Option to Scheduler

The Remote Spawn Command setting is only meaningful when running on the Linux Operating System. The value 'Scheduler' is valid if the job is a scheduler job running under an LSF, SGE or SLURM scheduler, and only if the MPI Vendor is 'Intel'.

When submitting a job using the AnsysEM job submission GUI, the Remote Spawn Command for an analysis may be specified using the batchoption with pathname 'DesignType/RemoteSpawnCommand', where DesignType is the Design Type to analyze. The Remote Spawn Command setting is only meaningful when running on the Linux Operating System. The value 'Scheduler' is valid if the job is a scheduler job running under an LSF, SGE or SLURM scheduler, and only if the MPI Vendor is 'Intel'. To specify the value 'Scheduler' for this option for a scheduler job, the Remote Spawn Command must be specified using the 'DesignType/RemoteSpawnCommand' batchoption in the product command line when the product is launched. In addition, the 'DesignType/MPIVendor' batchoption must be specified with value 'Intel' in the product command line when the product is launched. For interactive

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

scheduler jobs, the Remote Spawn Command and the MPI Vendor may be specified with batchoptions or as design type options in the **HPC and Analysis Options** dialog box.

Setting MPI Version

When submitting a job using the Ansys Electromagnetics job submission GUI, the MPI Version may be specified using the batchoption with pathname DesignType/MPIVersion, where DesignType is the type of design to analyze (e.g. HFSS). It allows selection of which Intel MPI version to use, for both Windows and Linux. Valid values are "Default", "2018", and "2021".

For interactive solves, the MPI version may be specified with the batchoption in the command line used to launch the product, or as a design type option in the HPC and Analysis Options dialog box.

Ansys Cloud Direct Support for HPC Job Management

Ansys Electromagnetics Desktop supports the Ansys Cloud Direct service that allows quick and easy access to cloud-based HPC resources. The service is accessible through both the Simulation Ribbon and **Tools** > **Job Management**.

> **Important:**
>
> For this release, you can submit HFSS, HFSS 3D Layout, Icepak, Maxwell 3D, Maxwell 2D, Mechanical, Q3D Extractor, and 2D Extractor jobs, monitor their progress, and download and post process the results. Other solvers are either in Beta or are not yet supported for Ansys Cloud Direct.

Uploads to and downloads from Ansys Cloud Direct use archives to improve speed. The Project browser shows both .aedt and .aedtz (archive) files. If you submit an unarchived project, an archive will be automatically created for upload. When the job completes, the results are packaged into an archive. When the download is initiated, the repackaged archive file is transferred to the download folder along with the other job-related files, such as the log.

For single setups, you can use multi-step submission. This separates the resource specification and reporting as appropriate for each stage of the solve.

Prerequisites for Using Ansys Cloud Direct with Ansys Electronics Desktop

- You will need an "Ansys Cloud Compute Essentials" subscription associated with your Ansys Account. The Ansys Account is an extension of the Ansys Discovery account that includes those products and services that require sign-in, such as the AppStore, Cloud Compute and others. Contact your account manager or service representative to obtain a

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

subscription invitation.

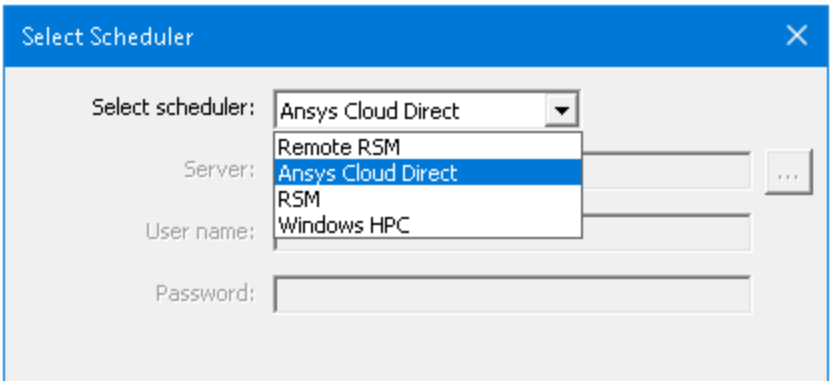
- You must select Ansys Cloud as your Job Management Scheduler, and, if necessary, log in to the Ansys Cloud Service using the Ansys Discovery Account and associated password.

Job Management UI for Ansys Cloud Direct

You can use Ansys Electronics Desktop to submit batch jobs to Ansys Cloud Direct and monitor those jobs.

This involves the following steps:

1. Use **Tools** > **Job Management** > **Select Scheduler** to select Ansys Cloud Direct as the scheduler.



2. Click **Log in** to launch a login window for Ansys Cloud Direct.

   After a successful login, you may not need to log in again for several days if you do not log out.

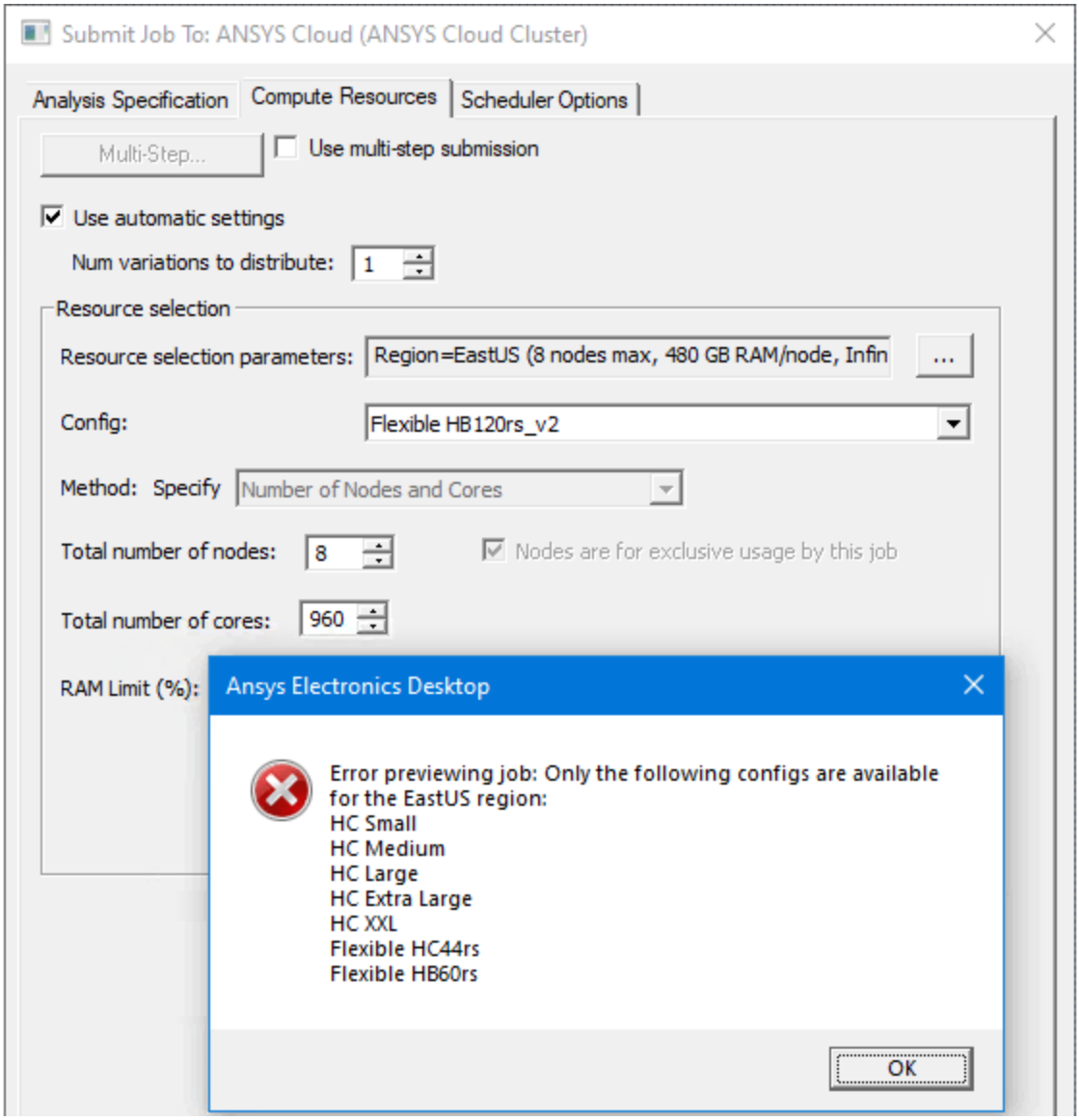
   If you wish to log out, click **Log out** from the **Select Scheduler** window.

3. Use **Tools** > **Job Management** > **Submit Job** to submit a batch job to Ansys Cloud Direct.

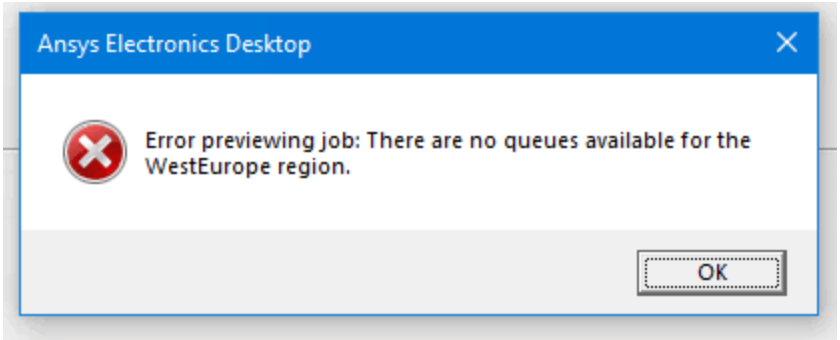
   Ansys Cloud Direct-specific Settings:

   - On the **Compute Resources** tab, **Resource Selection Parameters** will allow you to select a **Region** (for example, EastUS, JapanEast, WestEurope). The **Config** drop-down menu does not display configurations that are not available in any of the selectable regions. That is, if the regions are EastUS, WestEurope, and SouthwestUS, and none of these regions have the smallhc-based queues, then the "HC Small" configuration does not appear. If you select a resource that is not available in the specific region you select, an error messages shows the available choices for that region.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

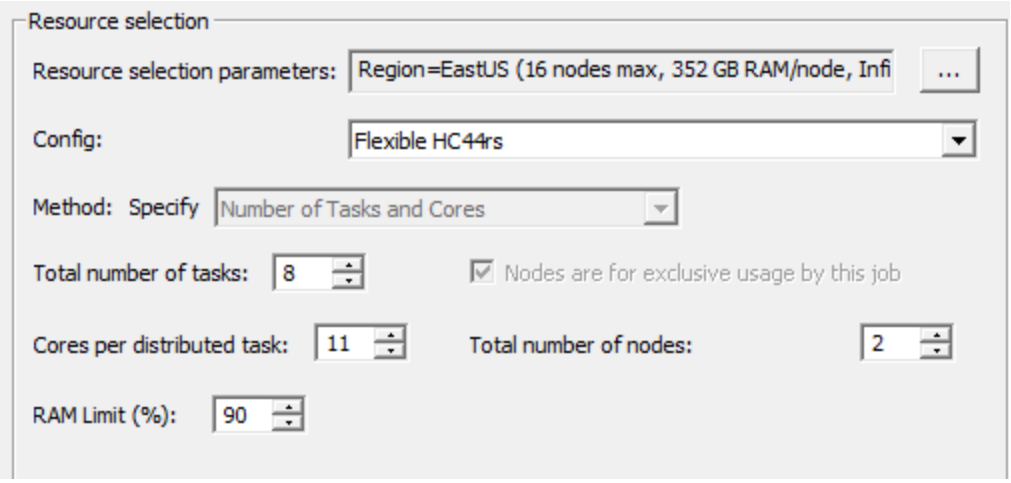


If no queues are available, an error message provides that message:



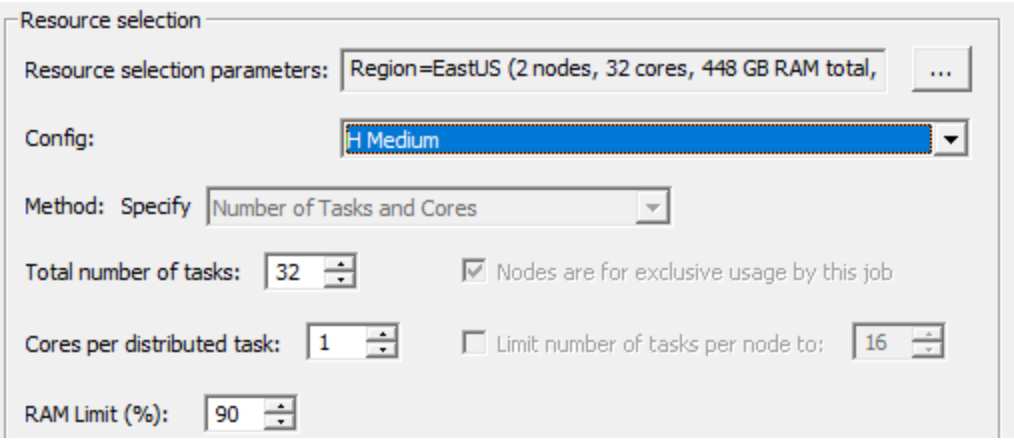
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- On the **Compute Resources** tab, select a **Config** size and style (Flexible or fixed) based on the following:
  - **Small:** 16 cores, 1 node, 224 GB RAM total
  - **Medium:** 32 cores, 2 nodes, 448 GB RAM total, Infiniband
  - **Large:** 128 cores, 8 nodes, 1792 GB RAM total, Infiniband
  - **Extra Large:** 256 cores, 16 nodes, 3584 GB RAM total, Infiniband
  - **XXL:** Even larger.
  - **Flexible**: Flexible queues permit the following resource selection methods: Automatic Number of Nodes and Cores, Manual Number of Tasks and Cores. If you select a Flexible queue for manual Number of Tasks and Cores, you will see a new item in **Resource Selection** for "Total number of nodes" replacing the "Limit number of tasks per node to" option that appears for fixed queues.



In comparison, if you selected a fixed configuration, the **Resource selection** panel appears like this:

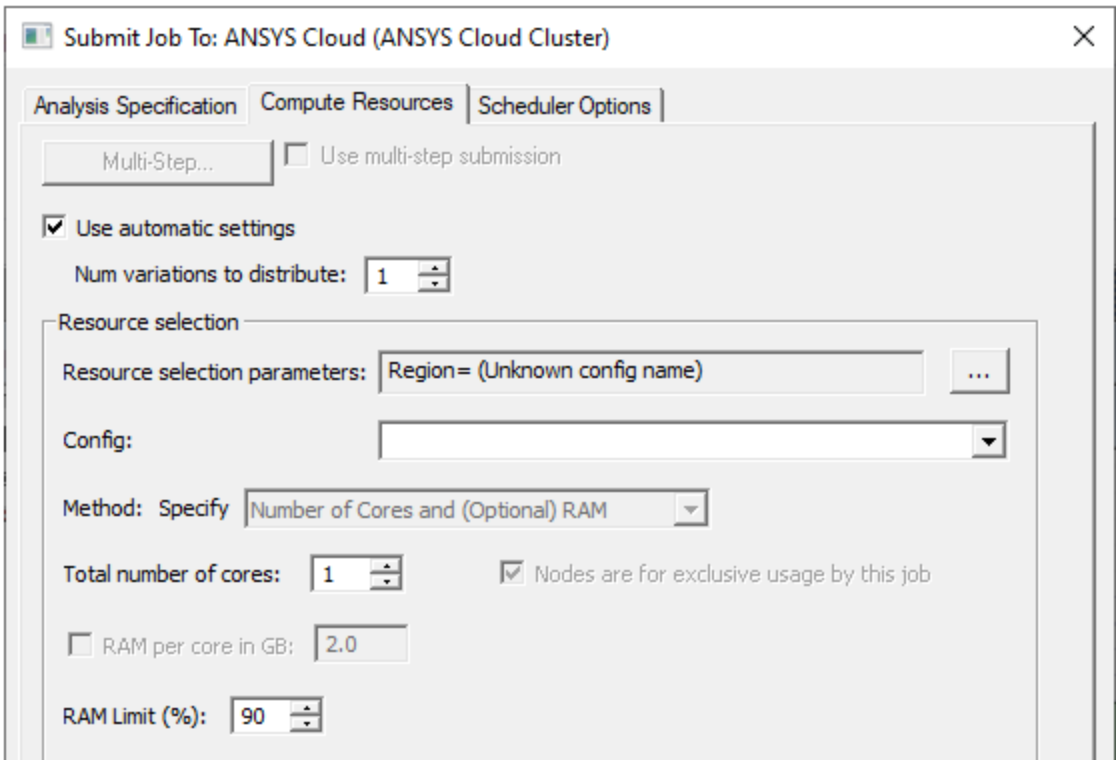
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



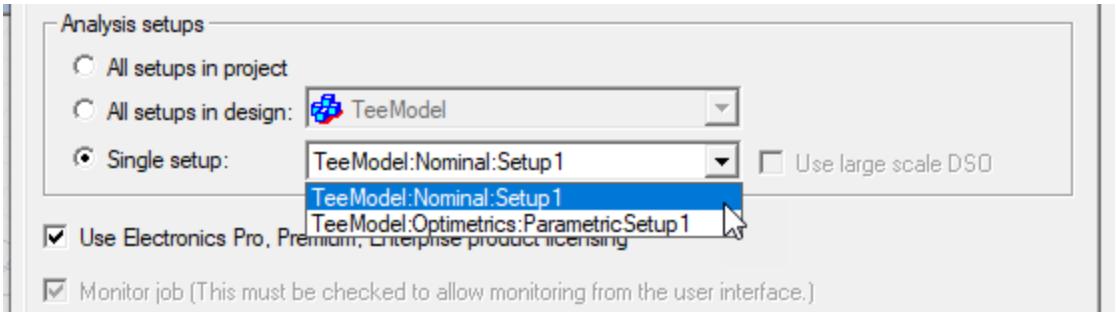
> **Important:**
>
> Attempts to use more for submission will result in error messages.

- On the **Compute Resources** tab, if you select **Use automatic settings** with **Num variations to distribute** set to 1, Optimetrics variations will be solved sequentially. Other distribution types will be distributed automatically. If you set **Num variations to distribute** to 2 or more, Optimetrics variations will be solved in parallel. Other distribution types will be distributed automatically.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

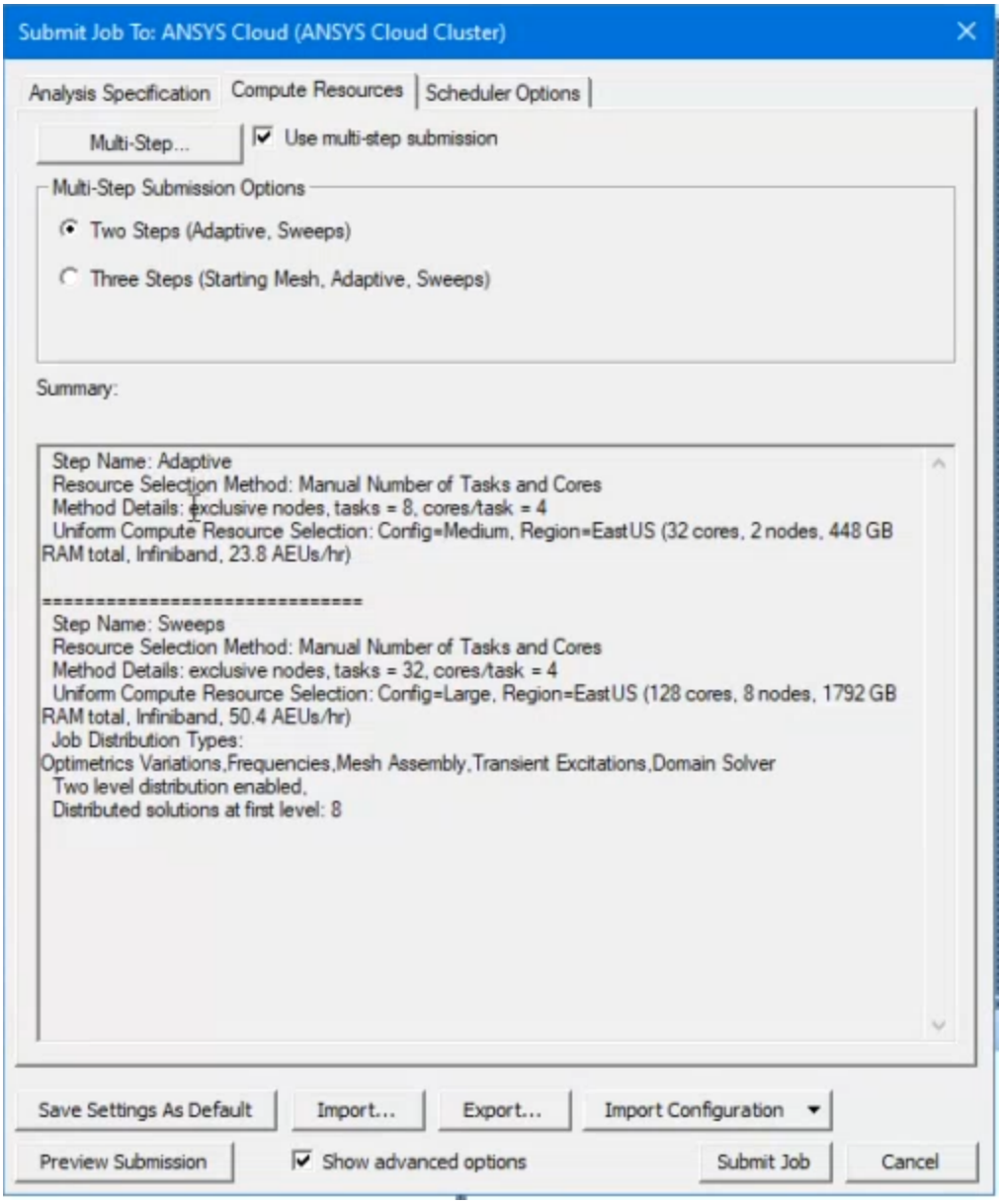


- In response to a set of minimal constraints, Ansys Cloud Direct scheduler may increase the resources assigned beyond the minimal values in order to meet the full set of requirements. For example, if you specify 7 distributed engines with two processors per engine and also limit the number of engines per node to 4, the scheduler may increase the number of cores used in order to meet the limit specified for engines per node. Click **Preview Submission** to see the number of resources assigned, and that the scheduler-generated code includes an MPI specification.
- To use **Multi-Step Submission** on Ansys Cloud Direct, you must specify a single setup on the **Analysis Specification** tab.



This enables **Multi-Step** options on the **Compute Resources** tab.

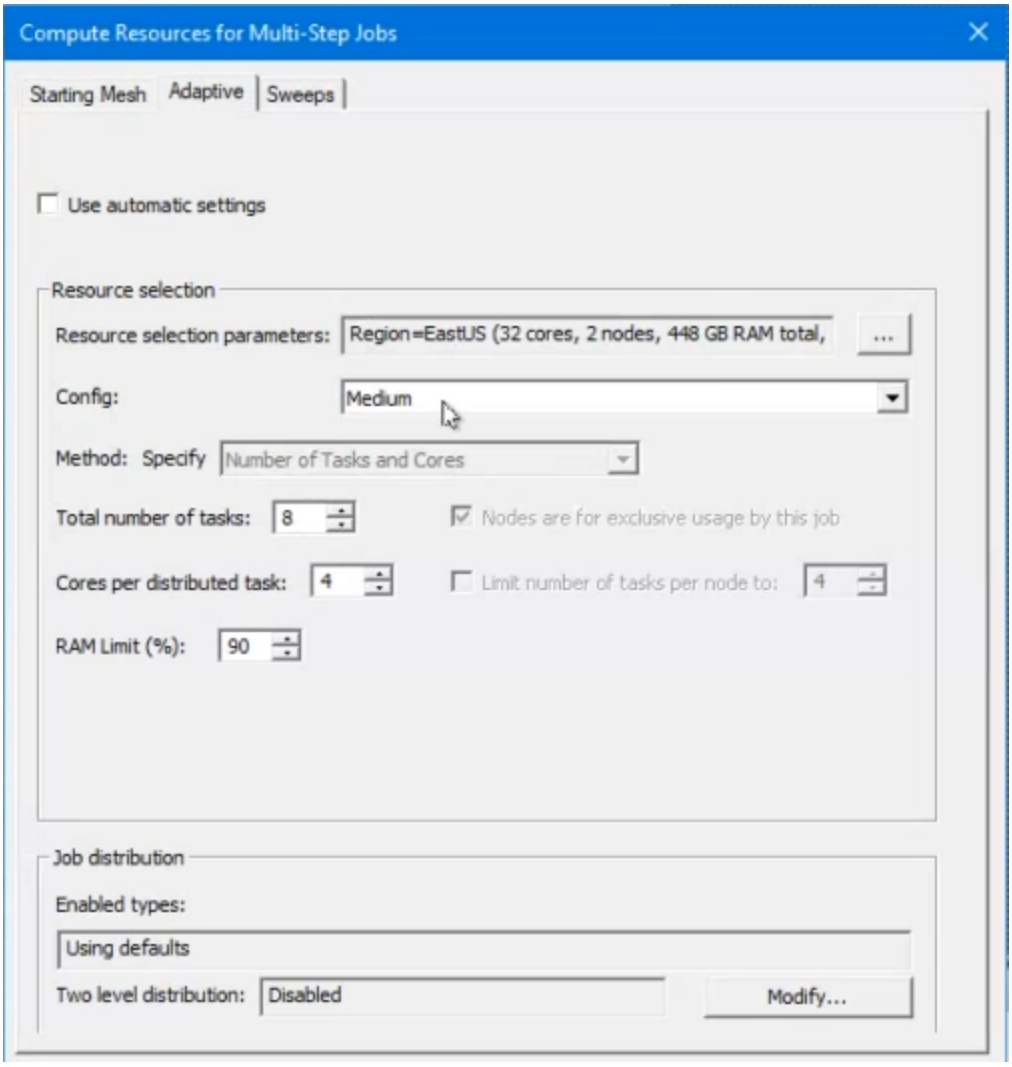
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Click the **Multi-Step** button to open the **Compute Resources for Multi-Step Jobs** dialog box. It has tabs for **Starting Mesh**, **Adaptive**, and **Sweeps**.

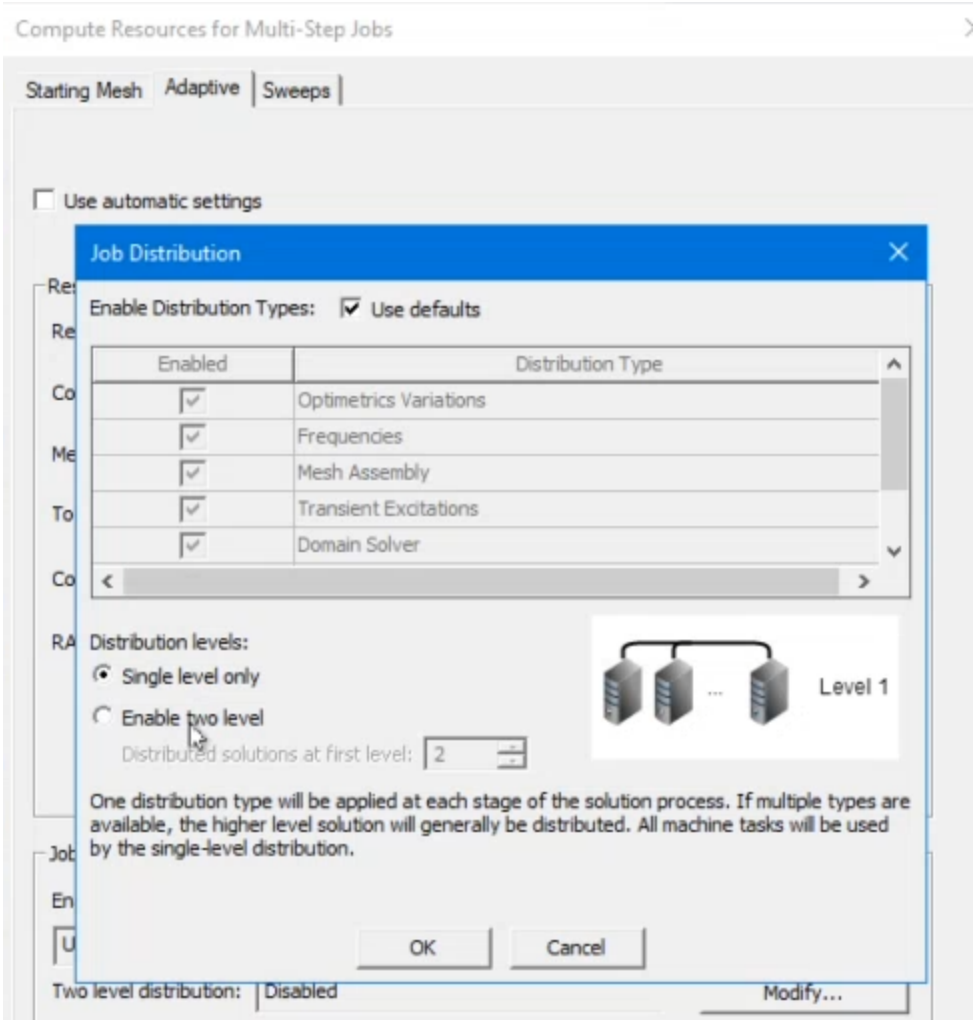
On the **Adaptive** tab, you can **Use automatic settings**, or specify **Resource Selection**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



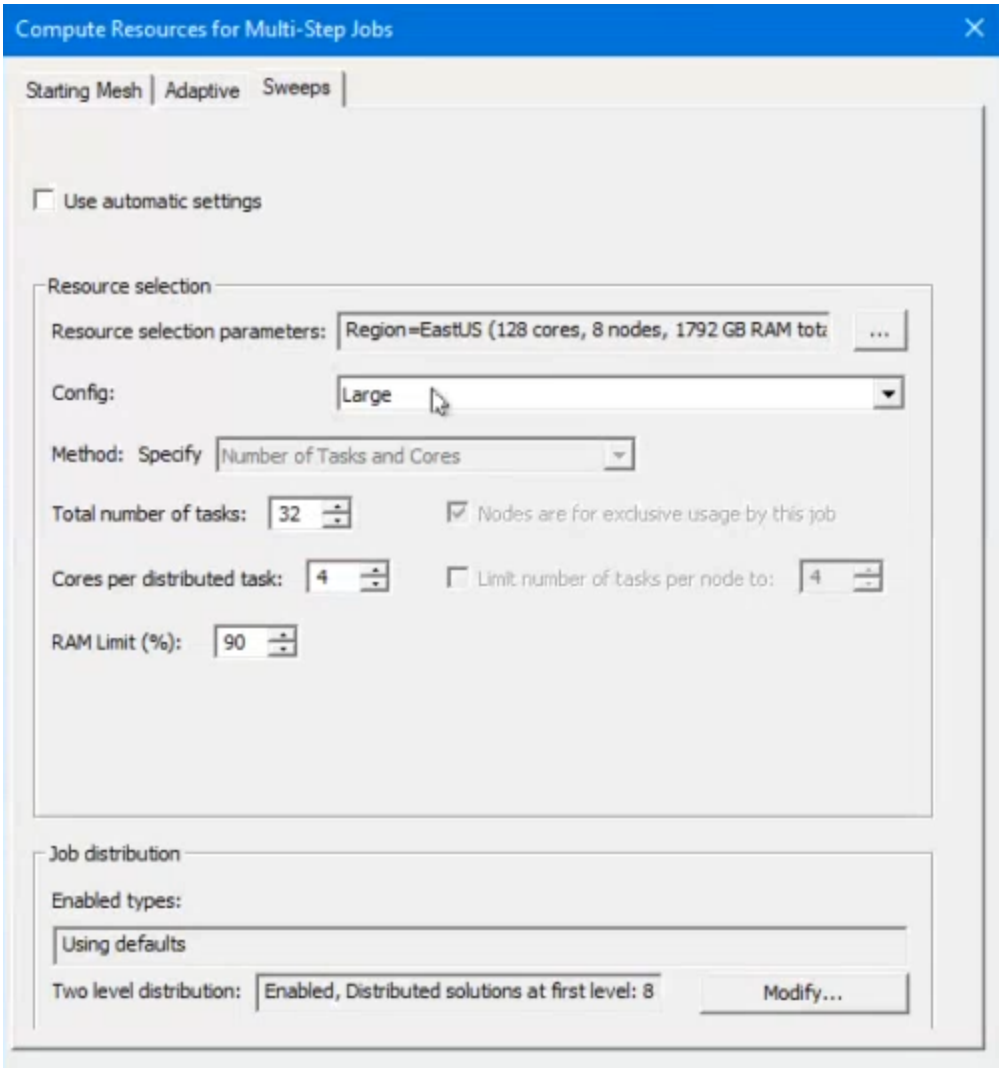
Click the **Modify...** button on the **Adaptive** tab to open the **Job Distribution** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Sweeps** tab lets you specify resources or **Use automatic settings**.

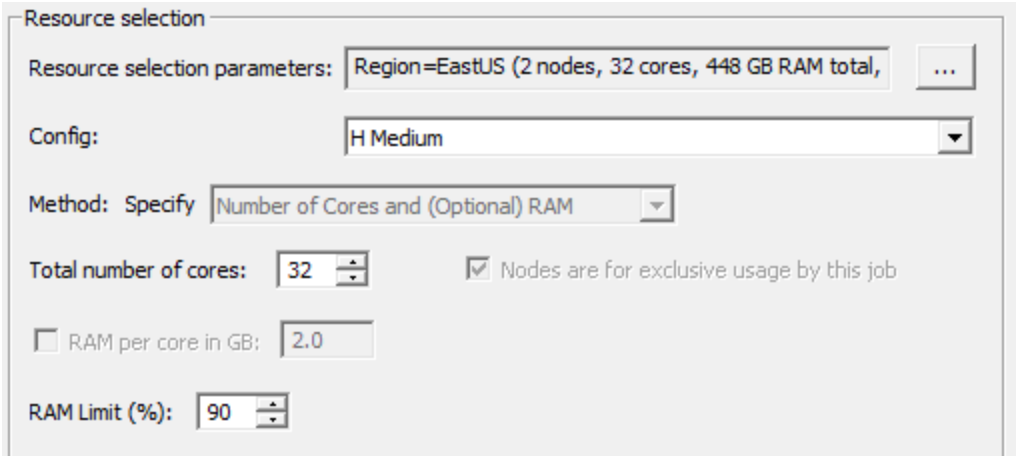
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



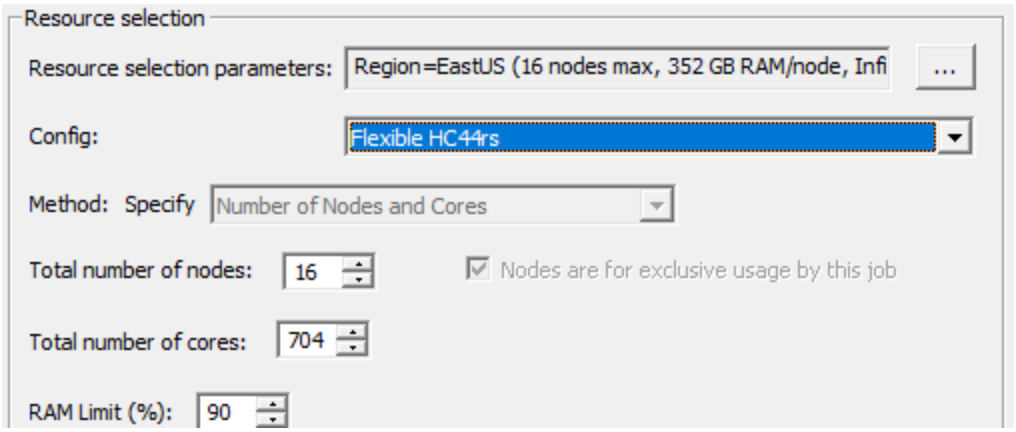
The **Sweeps** tab includes an independent **Modify...** button to specify **Job Distribution**.

Select automatic settings, or specify different resources for each step. When a job is finished, that is, on the last step, reports update and traces are extracted into CSV files. If select fixed queue for automatic sections, the **Resource selection** panel only allows cores and has RAM with RAM disabled:

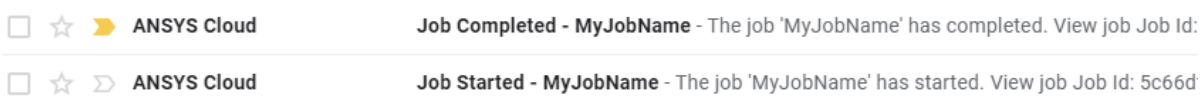
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you select flexible queue for automatic settings, The **Resource selection** panel only allows nodes and cores for a flexible queue:

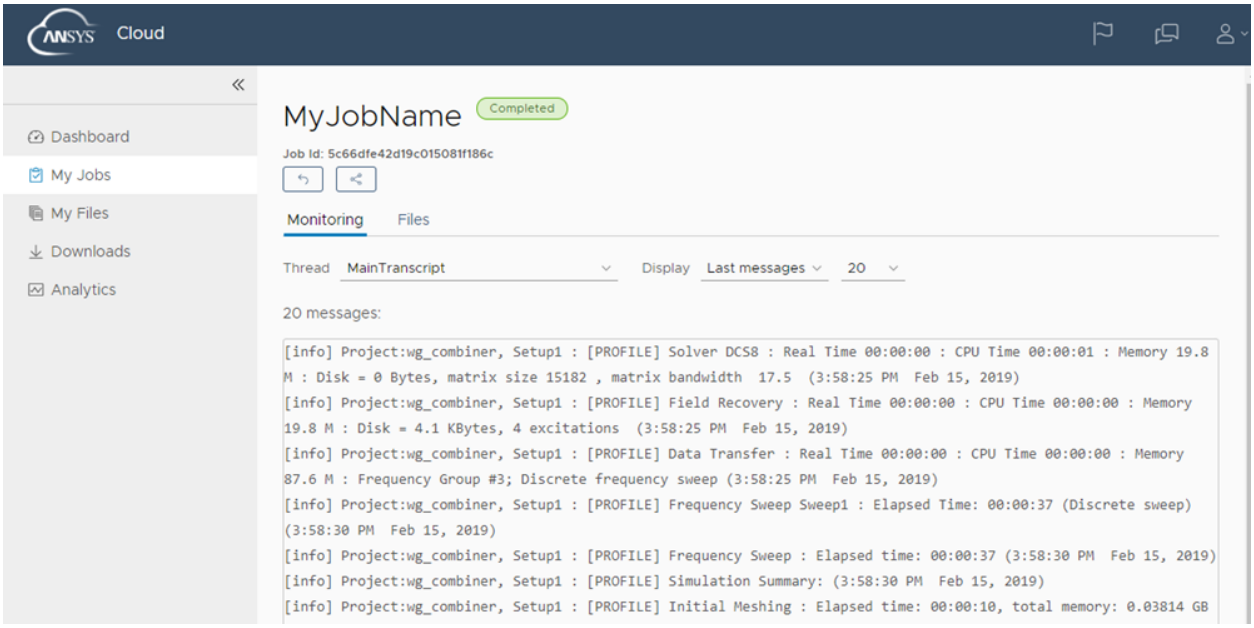


4. Use **Tools** > **Job Management** > **Monitor Job** to monitor the job's progress.
5. You will receive an email when the job has started, and another when it has finished.



6. Use the link in the email (or the **Portal** button on the job monitoring window) to launch the Ansys Cloud Portal and view detailed results:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Environment Variable Settings for Ansys Cloud Direct

When submitting jobs to Ansys Cloud Direct from Electronics Desktop, certain environment variables and batchoption settings must be made to ensure proper operation on the cloud. These settings are done automatically through the job submission window.

These environment variable settings will override any attempts from the user to set the same environment variables:

- ANS_NODEPCHECK=1
- ANSYSEM_FIX_REVERSE_LOOKUP=1
- ANSYSEM_DESKTOP_SUBNET_FROM_COMENGINE_ADDR=1

The environment variable ANSYSEM_ENV_VARS_TO_PASS contains a semicolon-separated list of filters for environment variables to be passed from ansysedt.exe to COM engines. You are allowed to set this environment variable. After being prepared for the job it will have the following filters set at minimum (with the user-requested filters following these):

```
ANSOFT_*;ANS_*;ANSYSEM_*;ANSYSLMD_LICENSE_*;I_MPI*
```

Troubleshooting and Debugging Environment Variables

To keep downloaded files that help with troubleshooting, enable the environment variable ANSYS_EM_PRESERVE_DOWNLOAD_FILES.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Debug logs (and other files generated in the working directory) from remote nodes are automatically collected.

Option Settings for Ansys Cloud Direct

The following option is automatically added for Ansys Cloud Direct jobs:

- -autoextract
- For single step jobs it will include "-autoextract reports".
- For multi-step jobs, the last step will include "-autoextract reports" but the first steps will only include "-autoextract" (no reports). Reports are only extracted on the last step.

See: Running Electronics Desktop from a Command Line.

Automatic Batchoption Settings

The license type batch option ("HPCLicenseType") is forced to be "Pack" because Cloud is configured to only work with pack licensing. The following MPI batchoptions are automatically set as "Intel" because Ansys Cloud Direct is configured to only work with Intel MPI.

- HFSS 3D Layout Design/MPIVendor
- HFSS/MPIVendor
- Icepak/MPIVendor
- Maxwell 2D/MPIVendor
- Maxwell 3D/MPIVendor
- Q3D Extractor/MPIVendor

Ansys Cloud Burst Support for HPC Job Management

Ansys Electromagnetics Desktop supports Ansys Cloud Burst Compute which allows a job to use resources in the cloud for analysis.

Uploads to and downloads from Ansys Cloud use archives to improve speed. The Project browser shows both .aedt and .aedtz (archive) files. If you submit an unarchived project, an archive will be automatically created for upload. When the job completes, the results are packaged into an archive. When the download is initiated, the repackaged archive file is transferred to the download folder along with the other job-related files, such as the log.

For single setups, you can use multi-step submission. This separates the resource specification and reporting as appropriate for each stage of the solve.

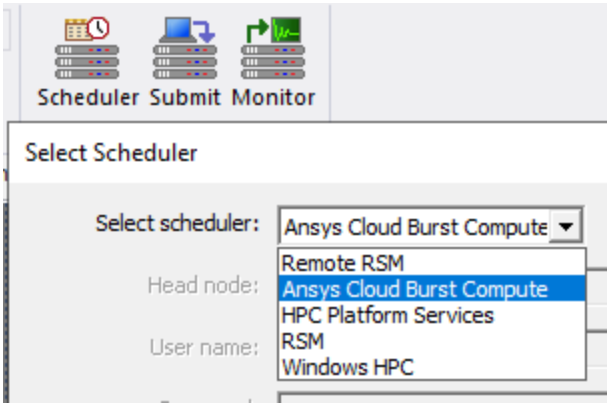
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Prerequisites for Using Ansys Cloud Burst Compute with Ansys Electronics Desktop

To solve remotely in Ansys Cloud, your company must subscribe to an Ansys Cloud plan. Each plan includes a pool of cloud compute credits and provides cloud access to a limited number of users. Your company's Ansys administrator manages cloud user access in the Ansys Admin portal at https://admin.ansys.com/. For more information, refer to the Ansys Admin Guide.

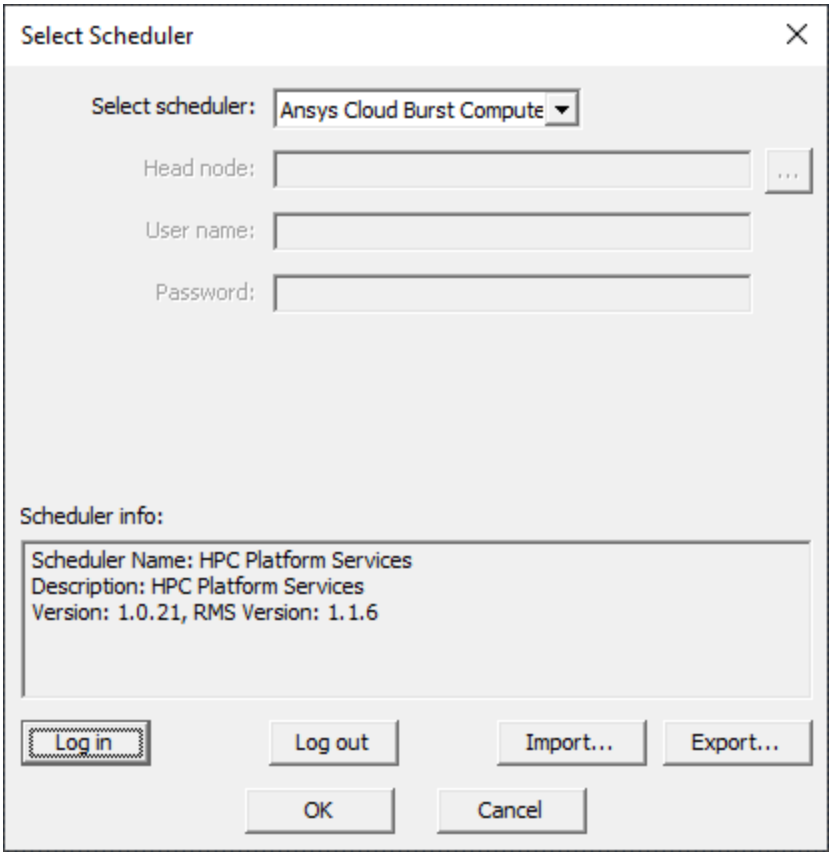
To solve remotely in Ansys Cloud, your company must subscribe to an Ansys Cloud plan and provide you with access to the Ansys Cloud service. For details, see Requirements for Cloud Burst Computing in the Ansys Cloud Subscription User's Guide.

You can select Ansys Cloud Burst Compute through both the Simulation Ribbon (**Scheduler** button) and **Tools > Job Management > Select Scheduler**.



After you select **Ansys Cloud Burst Compute** from the **Select Scheduler** dialog, press **Log in** to open the default browser and prompt a login. After successfully logging in the Scheduler info box will contain a valid HPC Platform Services description.

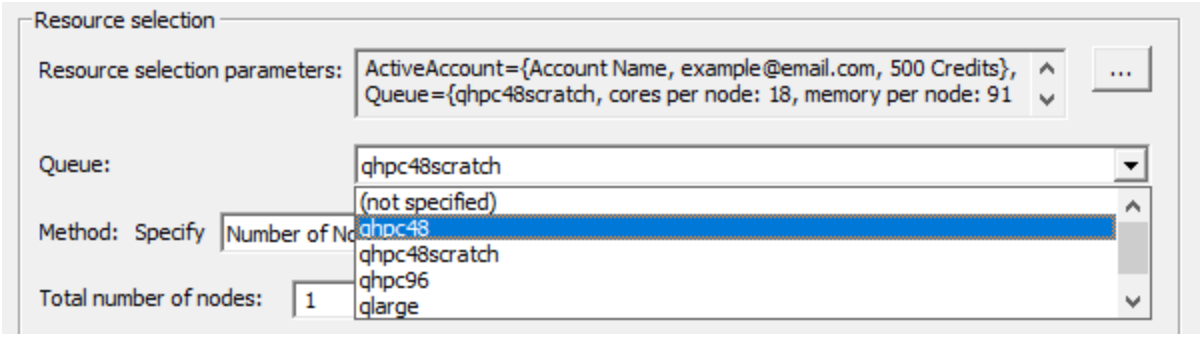
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To log out press the **Log out** button from the **Select Scheduler** dialog.

Ansys Cloud Burst Compute-Specific Settings

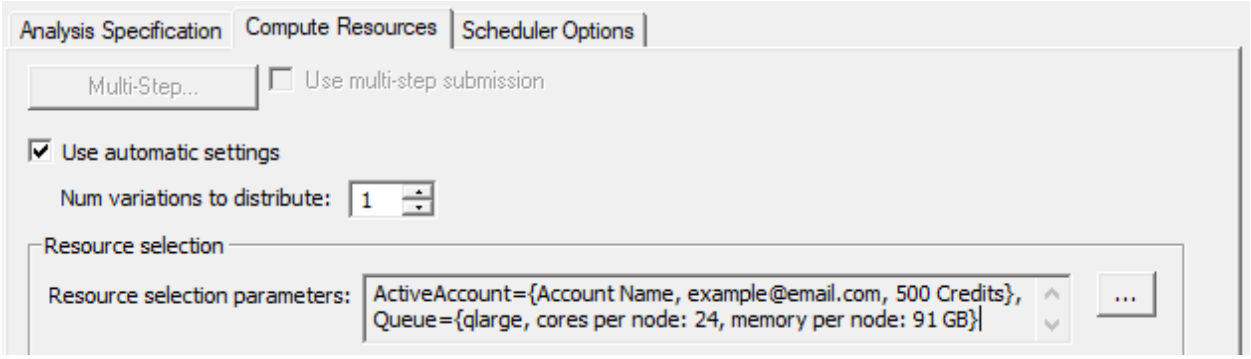
The **Compute Resources** tab allows you to select a **Queue**. The queue can also be left as (not specified) unless the submission method is automatic and **Method** is Number of Nodes and Cores. You can see information for the selected queue in the **Resource selection parameters** box. This includes cores per node and memory per node in GB.



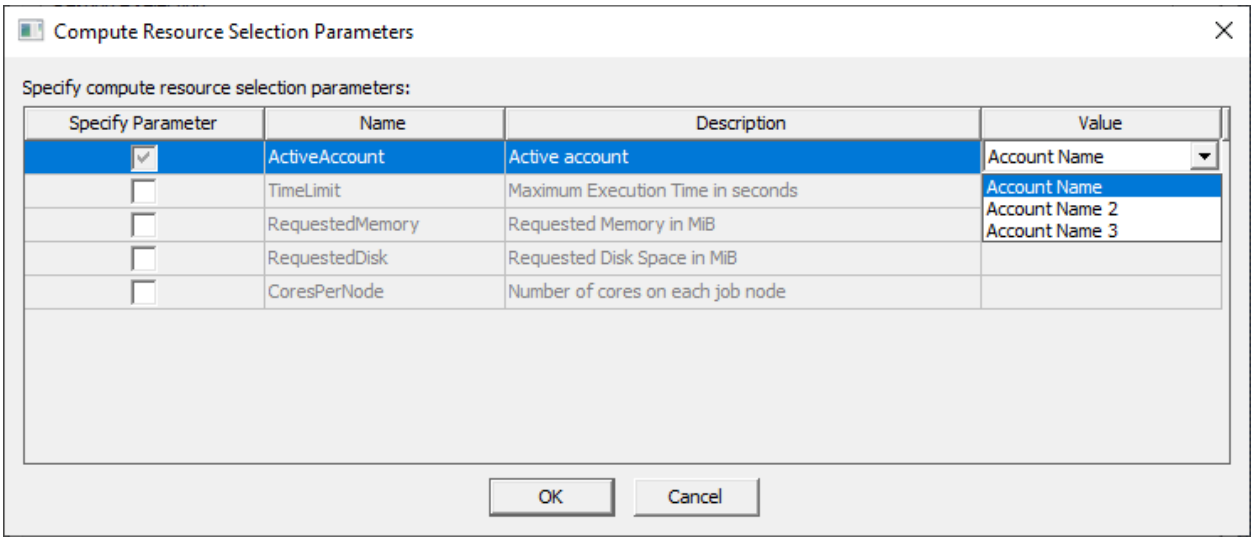
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Cloud Burst Compute uses accounts for job submission, monitoring, and results download. When you submit a job to Ansys Cloud, cloud compute credits in your company's Ansys Cloud plan are consumed. Each credit corresponds to one unit of computation. The rate of credit consumption varies depending on the complexity of your simulation and the size of the model. Individual users can monitor their credit consumption via the Ansys Portal at https://portal.ansys.com.

On the **Compute Resources** tab, the **Resource selection parameters** box will display the currently selected account and its associated email, allocated core hours, and available core hours.



To select a different account, use the ellipsis [**…**] button to the right of the **Resource selection parameters** box and select an account from the Value column of the row with ActiveAccount Name field.

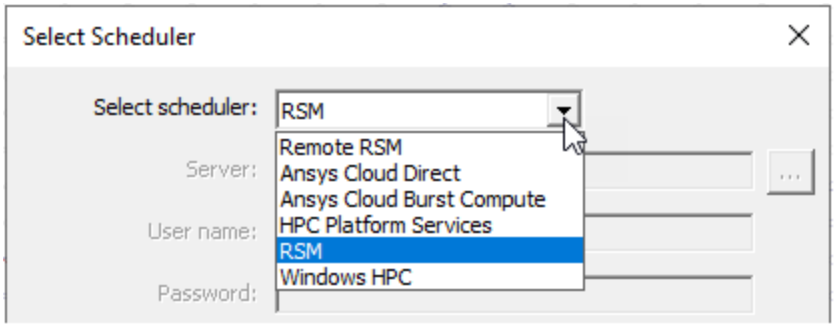


See: Monitoring Cloud Burst Jobs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

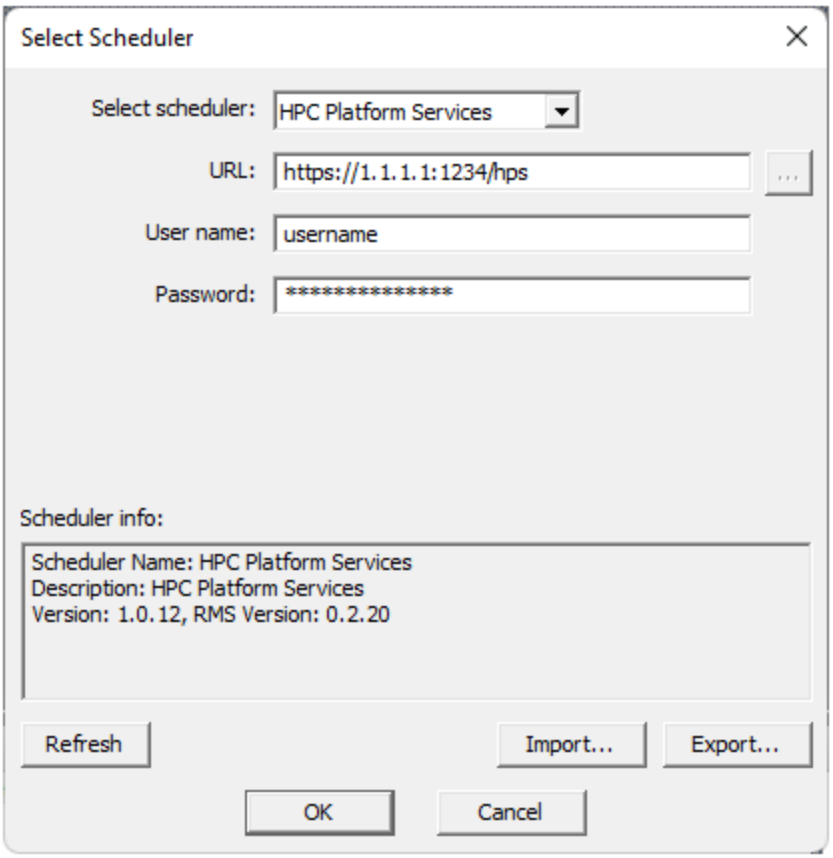
Ansys HPC Platform Services Support for HPC Job Management

Ansys Electromagnetics Desktop supports the Ansys HPC Platform Services mechanism that allows quick and easy access to HPC resources, to include Ansys Cloud Direct Gateway. HPC Platform Services can be selected through both the **Simulation Ribbon (Scheduler** button) and **Tools > Job Management > Select Scheduler**.



This opens the **Select Scheduler** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You must supply the URL for the HPC Platform Services. You must also enter the username and password for the HPC Platform Services account to be used for job submission. If you press the **Refresh** button, the versions of the HPC Platform Services server and the JMS (Job Management Service) are reported in this dialog.

Uploads to and downloads from Ansys HPC Platform Services use archives to improve speed. The Project browser shows both .aedt and .aedtz (archive) files. If you submit an unarchived project, an archive will be automatically created for upload. When the job completes, the results are packaged into an archive. When the download is initiated, the repackaged archive file is transferred to the download folder along with the other job-related files, such as the batch log.

Prerequisites for Using Ansys HPC Platform Services with Ansys Electronics Desktop

You must select HPC Platform Services as your Job Management Scheduler, including an endpoint URL, username, and associated password.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Job Management UI for Ansys HPC Platform Services

You can use Ansys Electronics Desktop to submit batch jobs to Ansys HPC Platform Services and monitor those jobs.

- To select HPC Platform Services as the scheduler, use **Tools** > **Job Management** > **Select Scheduler**.
- To submit a batch job to Ansys HPC Platform Services, use **Tools** > **Job Management** > **Submit Job**.

Ansys HPC Platform Services-Specific Settings:

The main use case for Ansys HPC Platform Services is where a SLURM cluster is used for computing at the endpoint.

Other schedulers could potentially be used in a compute cluster for Ansys HPC Platform Services.

When a job is submitted from AEDT, HPC Platform Services in turn submits a job to the compute cluster using the scheduler running on that cluster.

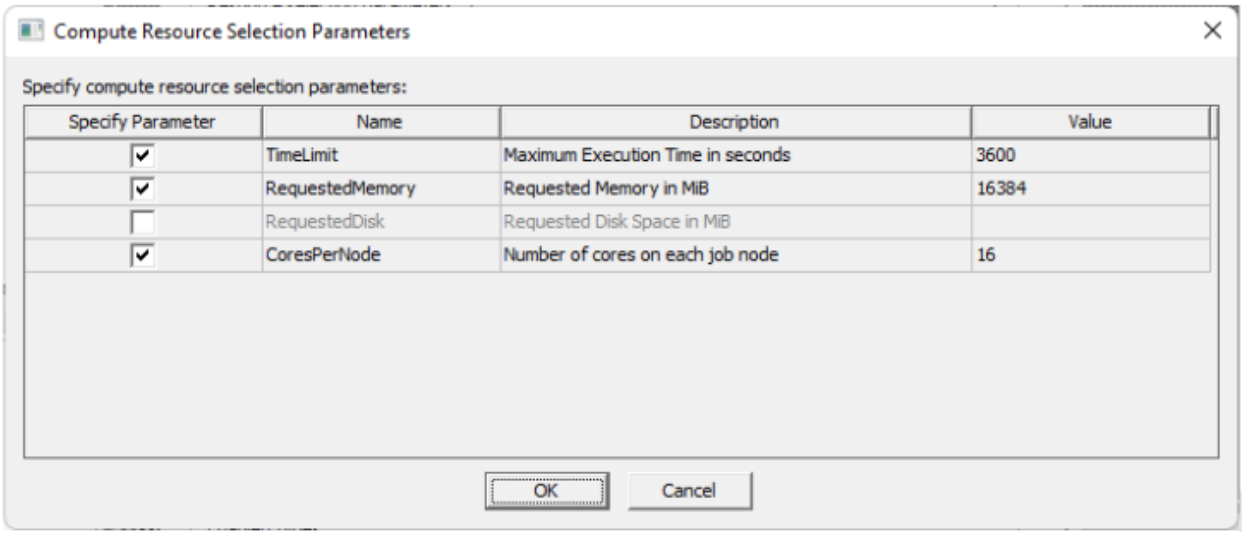
In case there are multiple queues (SLURM partitions) set up in the cluster, the **Compute Resources** tab has a drop-down list box for selecting the queue.

See your cluster administrator for the correct queue/partition name.

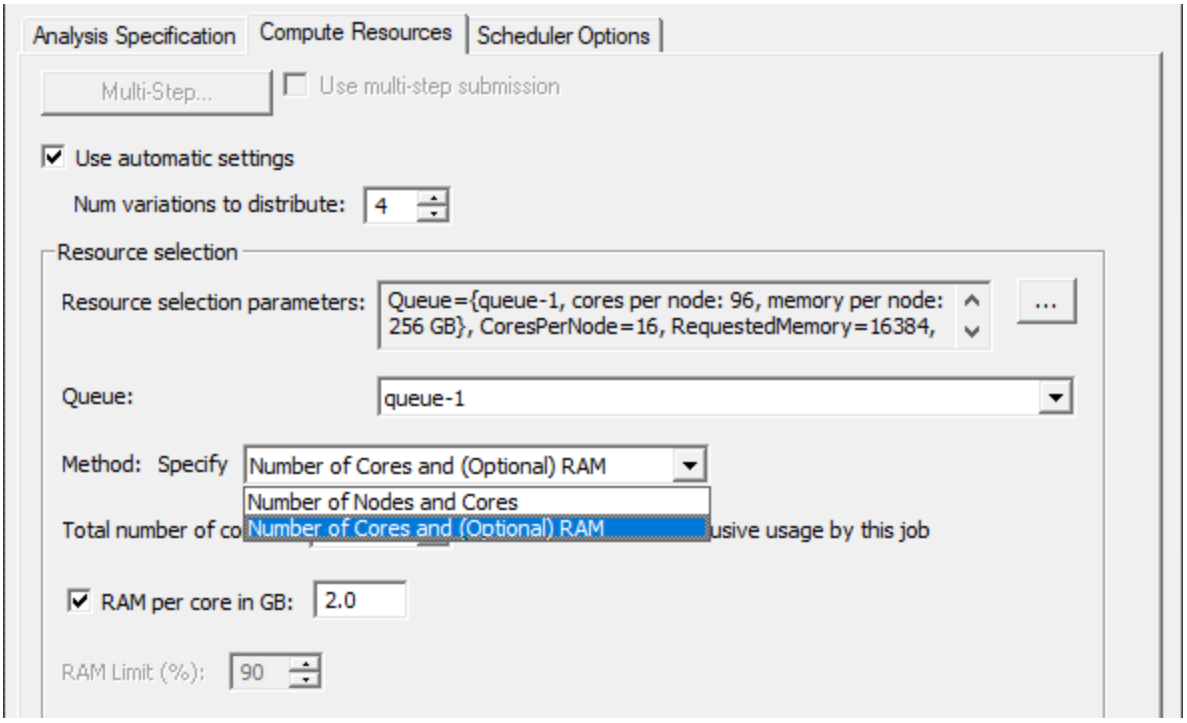
On the **Compute Resources** tab, **Resource Selection Parameters** allows you to select the following:

- **TimeLimit**: Maximum execution time for the job in seconds
- **RequestedMemory**: Requested memory for the job (all nodes combined) in megabytes
- **RequestedDisk**: Requested disk space for the job (all nodes combined) in megabytes
- **CoresPerNode**: The number of cores allocated to the job on each node

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



On the **Compute Resources** tab, if you select **Use automatic settings** with **Num variations to distribute** set to 1, Optimetrics variations will be solved sequentially. Other distribution types will be distributed automatically.



If you check the **Use automatic settings** checkbox, then you may select one of the following methods for specifying job resources:

Specify Number of Cores and (Optional) RAM

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

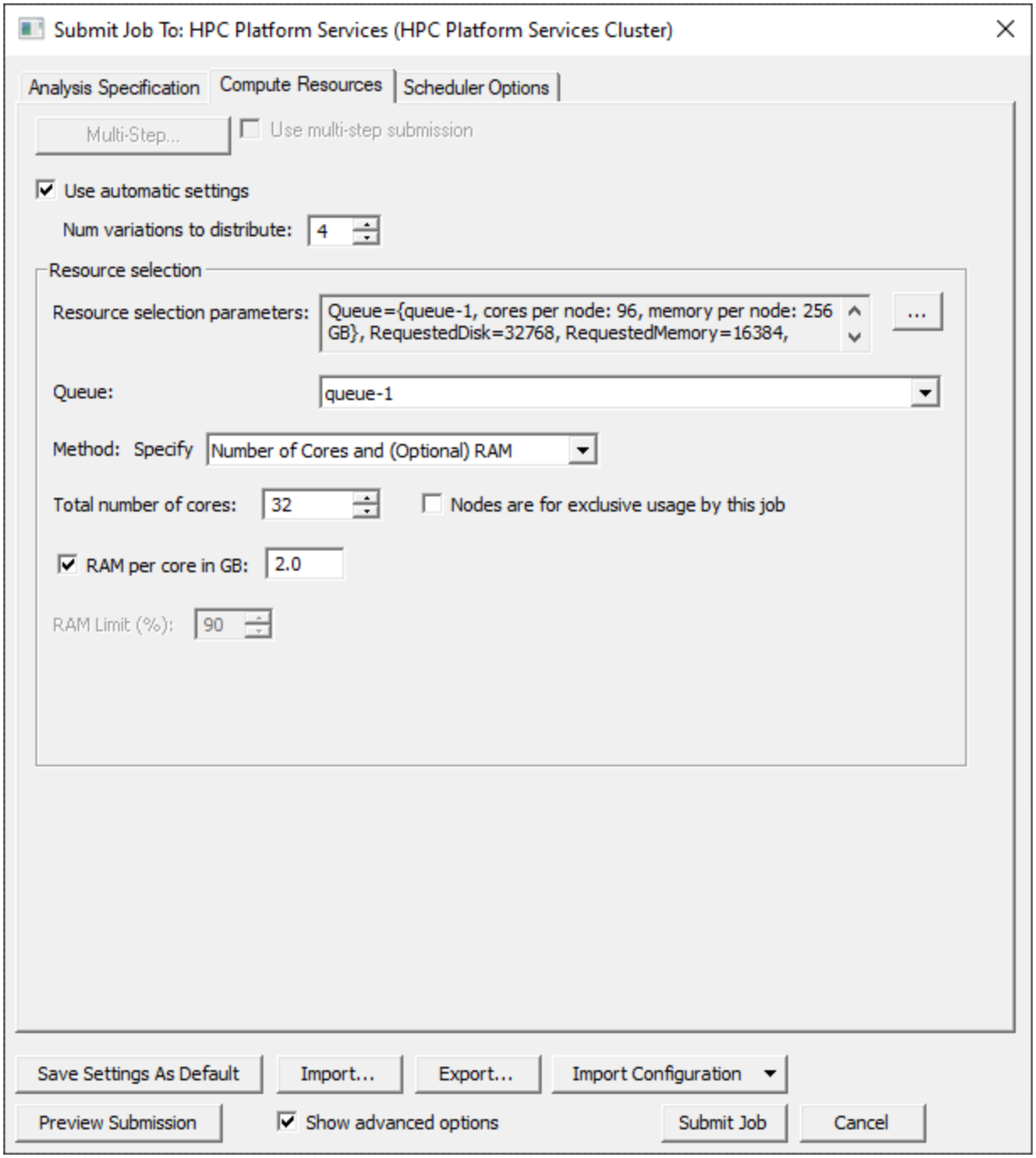
You must select the total number of cores for the job. If you check the associated checkbox is checked, you may also specify the RAM per core for the job. The total RAM requested will be the product of the RAM per core and the number of cores for the job.

Specify Number of Nodes and Cores

The job will request all cores on the specified number of nodes. The analysis will be limited to the specified number of cores; this may be useful to limit the licenses used by the job. This method may only be used if a queue is specified and if all queue execution nodes have the same number of cores. The number of cores specified must be less than or equal to the number of cores available on the specified number of nodes. If these requirements are not met, then the job will not be submitted, and an error message will be displayed.

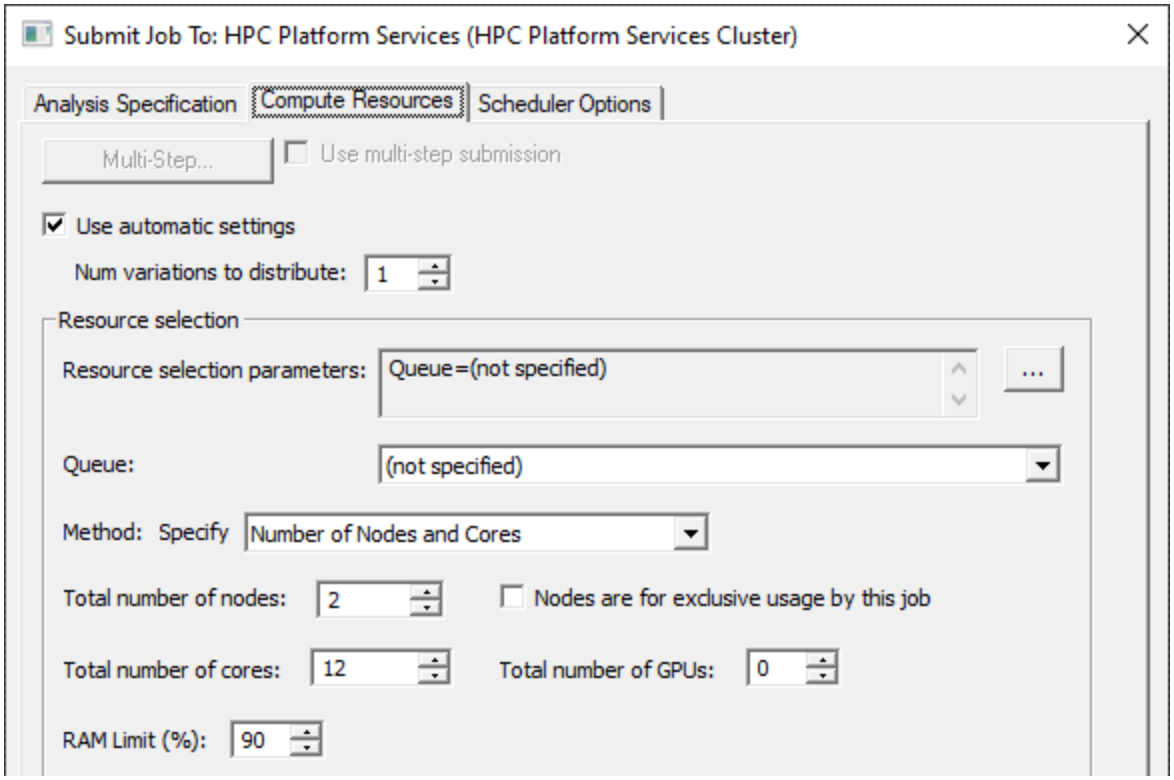
If you set **Num variations to distribute** to 2 or more, Optimetrics variations will be solved in parallel. Other distribution types will be distributed automatically.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you set **Num variations to distribute** to 1, Optimetrics variations will be solved in sequentially. Other distribution types will be distributed automatically.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

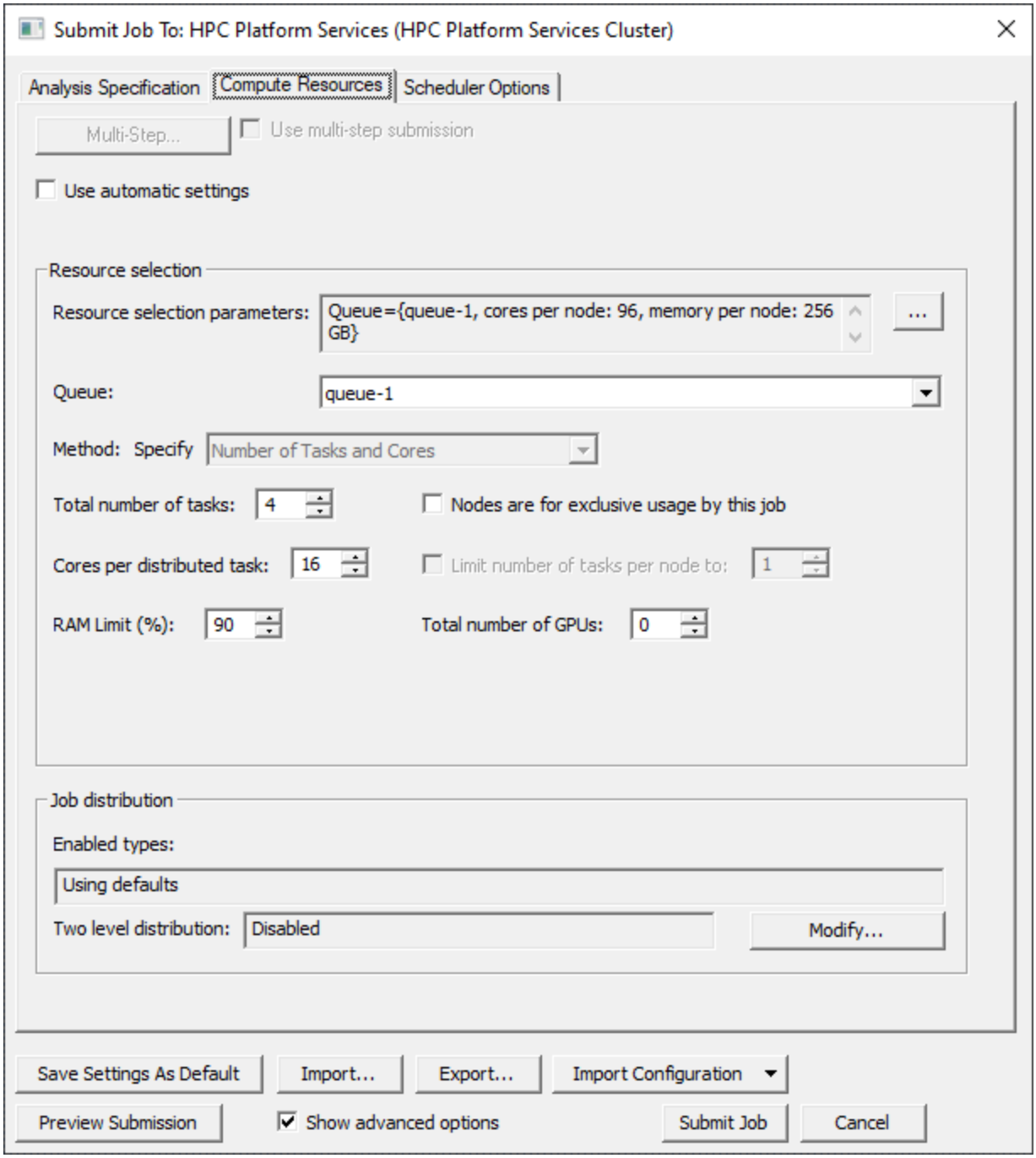


If you do not check **Use automatic settings**, then you may only select the following method for specifying job resources:

Specify Number of Tasks and Cores

You must specify the number of distributed tasks and the number of cores for each distributed task. The number of cores allocated to the job on each host must be an integral multiple of the number of cores per distributed task. If this requirement is not met, then the job will not start, and an error message will be output. In this case, you may specify a different submission method, or set the CoresPerNode resource selection parameter to an integral multiple of the number of cores per task and resubmit the job.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



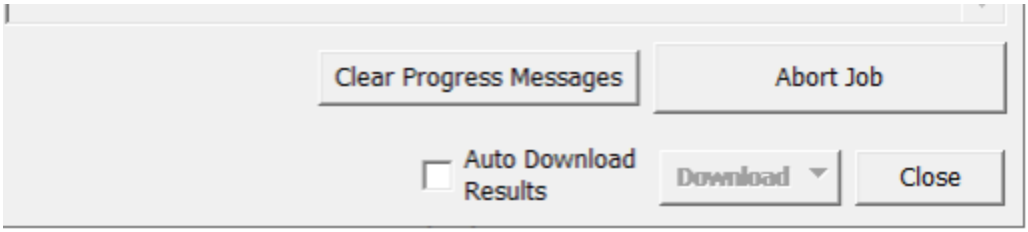
Use **Tools > Job Management > Monitor Job** to monitor the job's progress.

To get more detailed monitoring information, there is a **Job Management Service** that can be accessed from the base endpoint URL (without "/hps" at the end).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Aborting an Analysis

Jobs submitted via Ansys HPC Platform Services from can be aborted through the **Monitor job** dialog. To abort a job in a Queued state, you should press the **Abort Job** button on the **Monitor Job** dialog.



To abort any running job, you can either **Abort** or **Clean-Stop** the job. Using either of these abort types, debug logs can be collected from all compute nodes in the job.

For more information, please see Aborting an Analysis.

Cluster Specific Considerations

Ansys HPC Platform Services

Job Submission Methods

Manual:

- Number of Tasks and Cores: allows specification of Total number of tasks, Cores per distributed task, and RAM Limit %. Also allows manual specification of allowed Job distribution types and two-level distribution.

Automatic:

- Number of Nodes and Cores: allows specification of Total number of nodes, Total number of cores, Total number of GPUs, RAM Limit %, and Num variations to distribute.
- Number of Cores and (Optional) RAM: allows specification of Total number of cores, RAM Limit %, and Num variations to distribute.

Configuration Guidelines

Compute resource selection for Ansys HPC Platform Services involves choice of queue. These will depend on the cluster setup. The queue can also be left as “(not specified)” unless the submission method is automatic and Method is Number of Nodes and Cores. Information for the selected queue can be seen in the Resource selection parameters box. This includes cores per node and memory per node in GB.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option Settings for Ansys HPC Platform Services

The following option is automatically added for Ansys HPC Platform Services jobs:

"-autoextract reports".

See: Running Electronics Desktop from a Command Line.

GPU Acceleration

Several Ansys Electronics Desktop solvers can benefit from GPU acceleration:

- HFSS frequency domain, transient, and SBR+ solvers.
- HFSS 3D Layout for matrix solves.
- Maxwell 3D eddy current matrix solves.
- EMIT

License Options

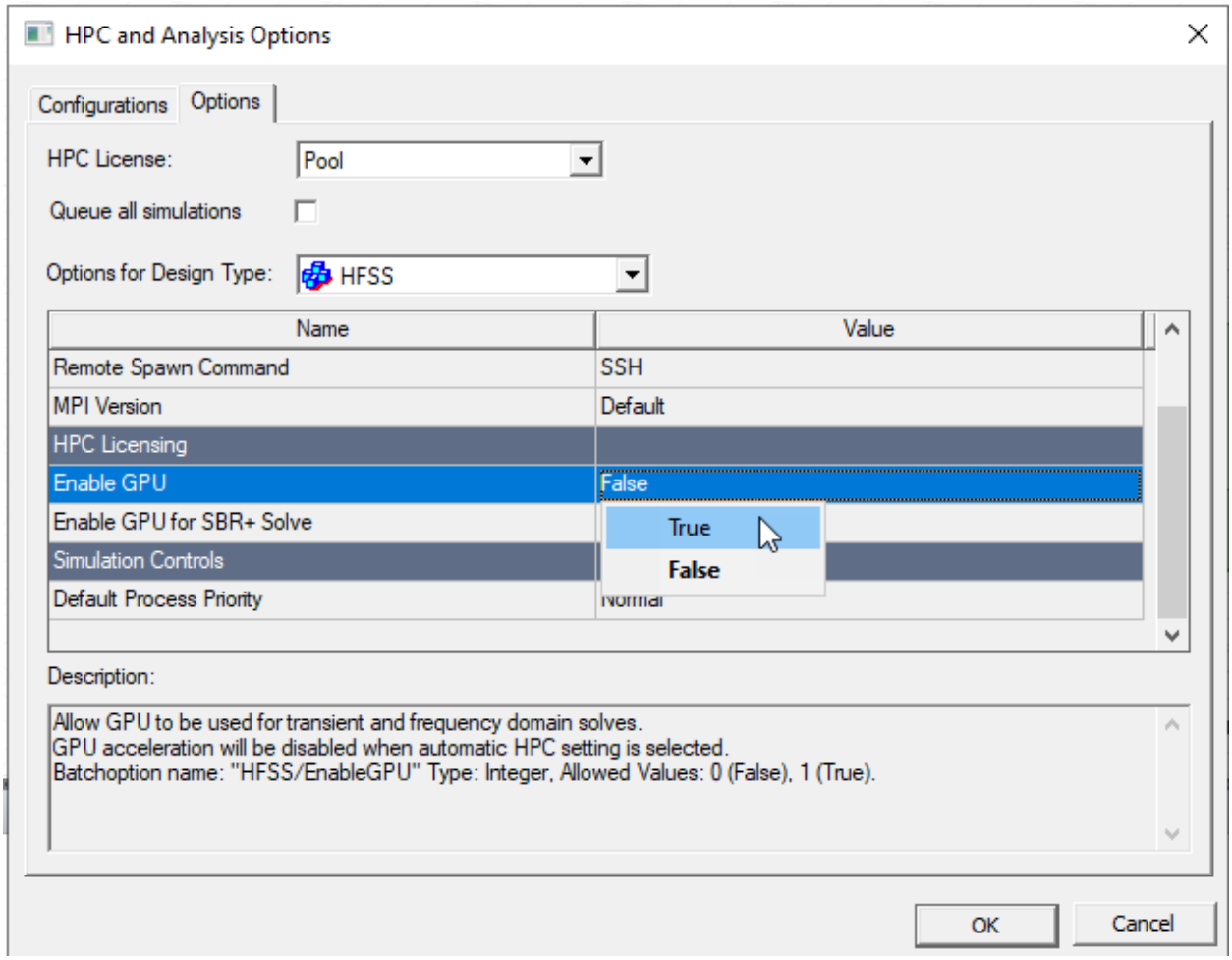
HPC licensing enables the use of cores and GPUs to accelerate simulations. In general, each core requires one unit of HPC, while each GPU requires eight units. The selected HPC license type determines which license is used, and how units of HPC are converted to license counts. For details, see the discussion of licensing under Options in Setting HPC and Analysis Options.

If the number of simulation jobs exceeds the number of GPUs in a system, the excessive jobs will fall back to CPUs and will be accelerated by up to 8 CPU cores for each job.

Enable/Disable GPU Acceleration from the Ansys Electronics Desktop Solver User Interface

To turn on or off GPU acceleration for supported solvers, click **Tools** > **Options** > **HPC and Analysis Options** and select the **Options** tab. Click on the value of Enable GPU and toggle it to either True or False. HFSS includes a separate toggle for Enable GPU for SBR+ Solve.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



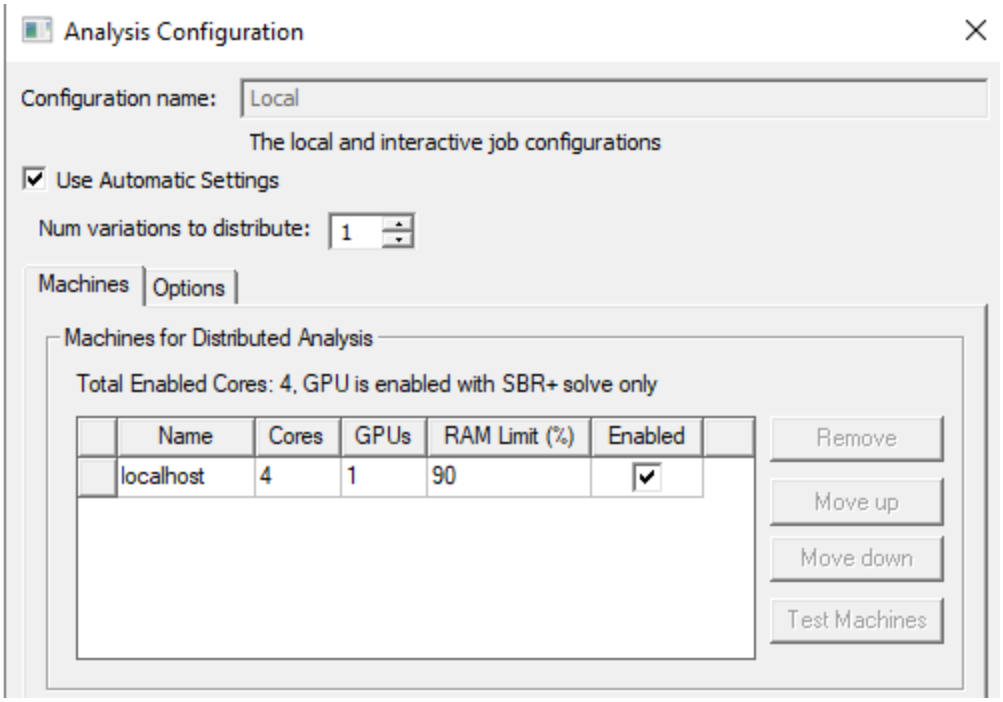
Enabling GPU-Distributed

There are two steps for setting up GPU-distributed.

1. Go to **Tools** > **Options** > **HPC and Analysis Options** and click **Edit** in the dialog box, or select the **Simulation** tab of the ribbon, and click **Analysis Config** to edit Analysis Configurations.

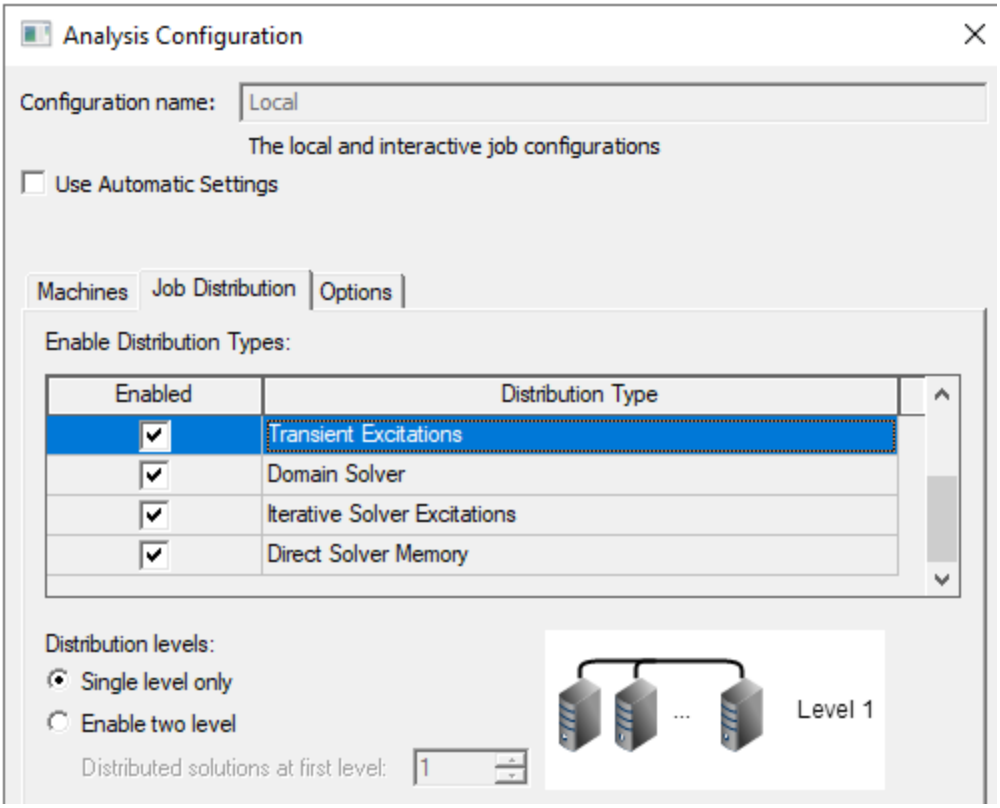
   If Use Automatic Settings is checked and the solver is HFSS, a message notes that only SBR+ has GPU Enabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Uncheck **Use Automatic Settings**, select the **Job DIstribution** tab, and select the appropriate Distribution Types for the solver. For example, GPU-DSO in HFSS Transient can be used either for parametric sweep (Variations) or Transient Network analysis of multiport networks (Transient Excitations).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

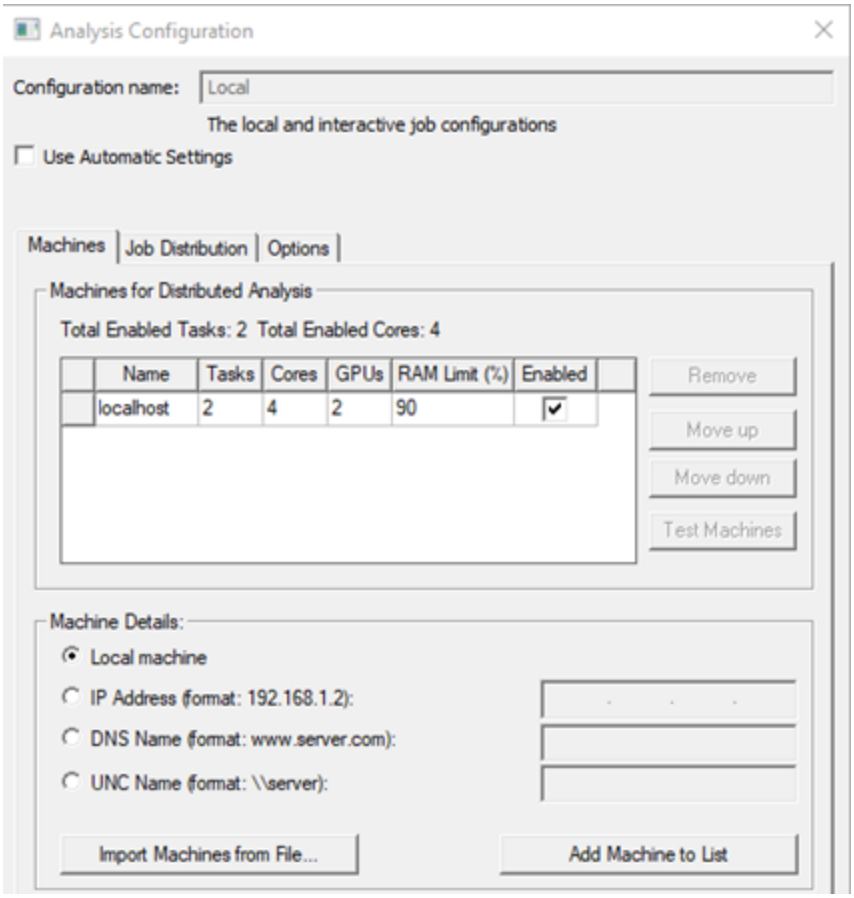


2. Set up the Machines for GPU-Distributed.

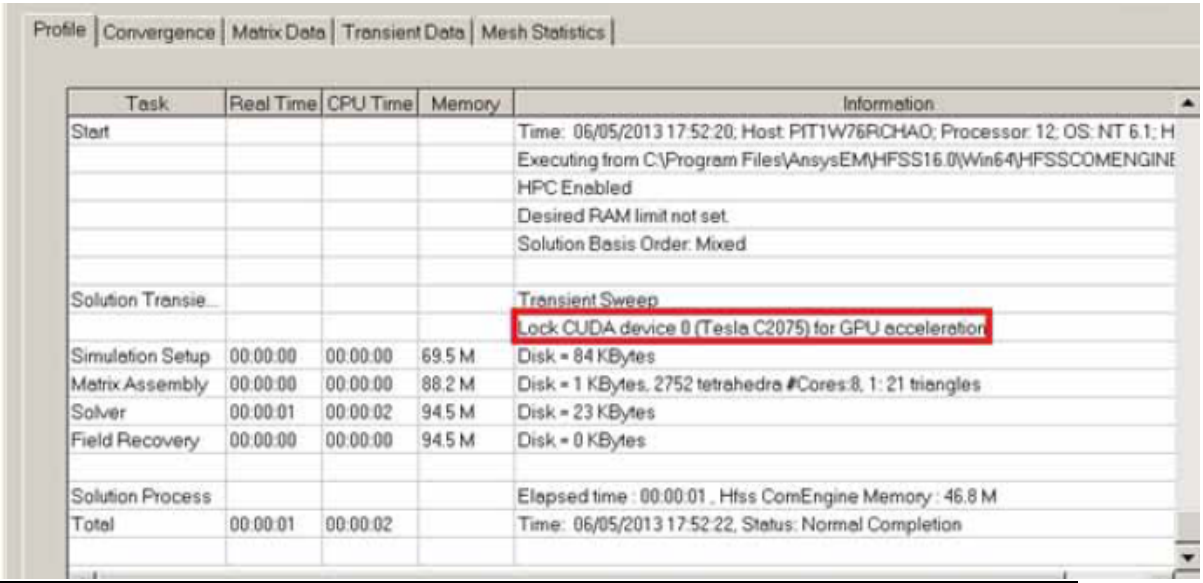
   In the following figure, two simulation jobs are allowed to run on the local host, or up to two GPUs can be used for acceleration depending on the availability of licenses and GPU

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

cards.

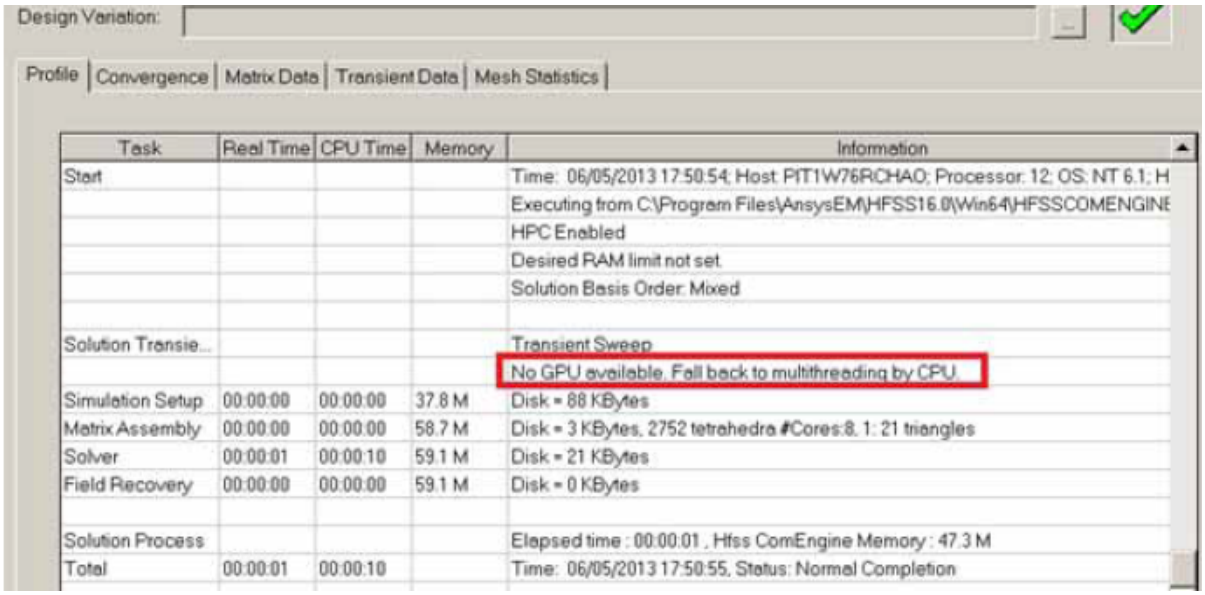


Whether GPU is used for acceleration can be checked by viewing the **Solutions** dialog box, **Profile** tab. If a GPU is successfully locked for the use by an hf3d process, the profile will show the GPU's CUDA device ID and its name.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Otherwise, the profile will indicate the fallback to CPUs. More information about why GPUs are not available for acceleration can be found in the log file. See further discussion under Step 5 of Setting HPC and Analysis Options.



Enable/Disable GPU Acceleration from the Command Line

GPU acceleration can be toggled by the `-batchoptions` command line argument:

`EnableGPU=[0/1].`

For example, the following command turns on GPU acceleration.

```
ansysedt.exe -batchsolve -batchoptions "'EnableGPU'=1"
projectname.aedt
```

Supported GPUs by Solver

GPU support includes Quadro and Tesla cards and GPU generations (Kepler (K), Maxwell (M), Pascal (P), Volta (V), Turing/RTX, Ampere(A)). GPU acceleration has been developed for Nvidia cards and is officially supported with the Tesla series. We highly recommend NVIDIA Tesla cards for the best performance when using several cards on one machine to solve either multiple variations (DSO) or excitations (HPC) in parallel which is referred to in this document collectively as distributed.

- HFSS Frequency-domain and Time-domain solvers support NVIDIA Data Center GPUs of the Ampere series and Tesla GPUs of the Volta, Pascal, and Kepler generations. NVIDIA Workstation RTX and Quadro GPUs for all generations are not supported except for Quadro GV100.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- HFSS SBR+ solver supports NVIDIA Data Center GPUs of the Ampere series and Tesla GPUs of the Volta, Pascal, Maxwell, and Kepler generations. NVIDIA Workstation GPUs of the RTX and Quadro families are supported by the HFSS SBR+ solver.
- Maxwell solvers support NVIDIA Data Center GPUs of the Ampere series and Tesla GPUs of the Volta, Pascal, and Kepler generations. NVIDIA Workstation RTX and Quadro GPUs for all generations are not supported except for Quadro GV100.
- Ansys EMIT supports NVIDIA Data Center GPUs of the Ampere series and Tesla GPUs of the Volta, Pascal, Maxwell, and Kepler generations. NVIDIA Workstation GPUs of the RTX and Quadro families are supported by EMIT.
- Icepak supports NVIDIA's CUDA-enabled Tesla and Quadro series workstation and server cards.

Driver requirements:

NVIDIA GPU minimum driver requirements are dictated by the NVIDIA CUDA version. The current Ansys EM uses CUDA 10.2 which requires a minimum driver version of 441.22 for Windows x86_64 and 440.33 for Linux x86_64.

We recommend downloading standard NVIDIA drivers for the user-specific cards rather than “DCH” drivers on Windows OS.

To contact Ansys technical support staff in your geographical area, please log on to the Ansys corporate website, ansys.com/support.

> **Note:**
>
> nVIDIA Tesla M2090, is a previous generation (code Fermi) GPU card and doesn’t work for Workstation since it has no fan for active cooling but needs a server with GPU cooling solution (passive cooling) similar to the nVIDIA Tesla K80.
>
> To get the best performance, the GPU used for running simulation jobs should not be attached to any display. Only GPU cards with CUDA Compute Compatibility 3.0 (Kepler) and above should be used. To improve the speedup of transient field visualization, you should install GPU cards on a system with PCI-E 3.0 slots. A mixture of interface cards with lower PCI-E versions may result in the data not being transferred from GPU to CPU at the highest speed.

GPU Installation and Setup

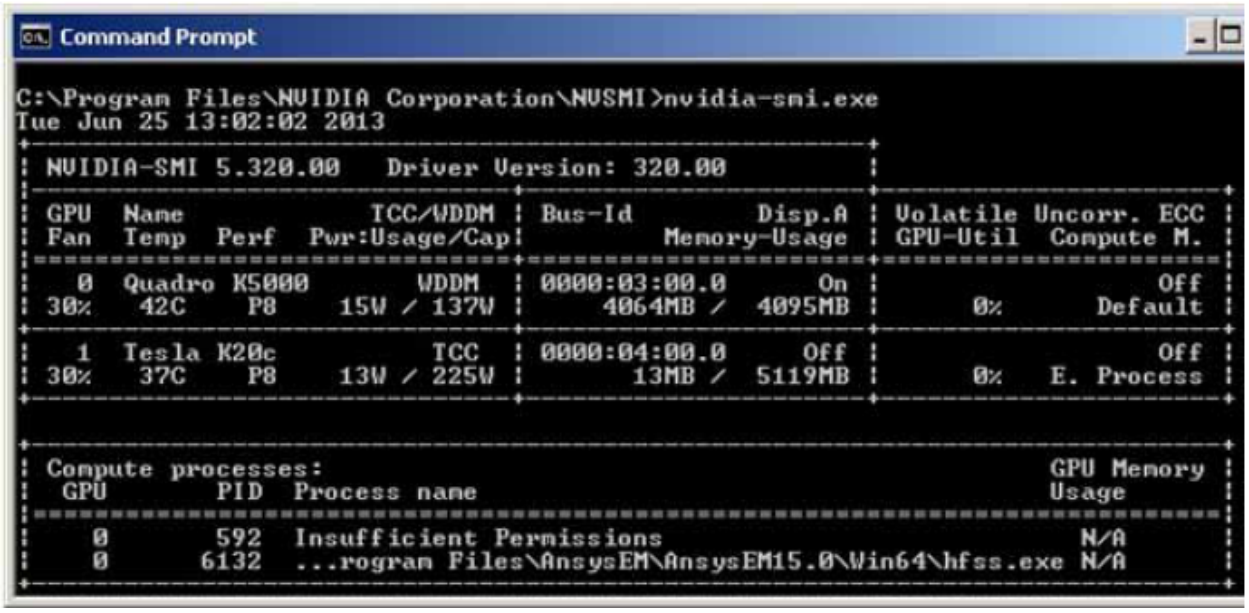
This chapter describes the GPU installation and setup for Windows and Linux.

Setup for Windows

1. After you install GPU cards and Nvidia graphics drivers downloaded directly from the Nvidia website, you should be able to find the cards in Windows Display Manager.

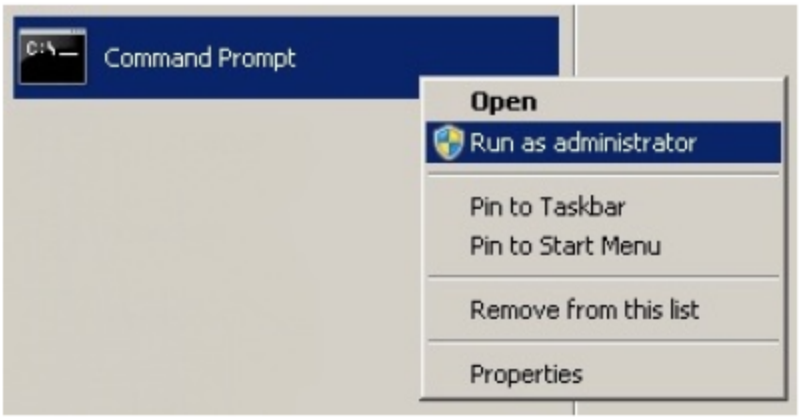
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. You should run nvidia-smi.exe at C:\Program Files\NVIDIA Corporation\NVSMI to check if GPU cards are installed successfully. The executable nvidia-smi.exe should exist after the display driver is installed.

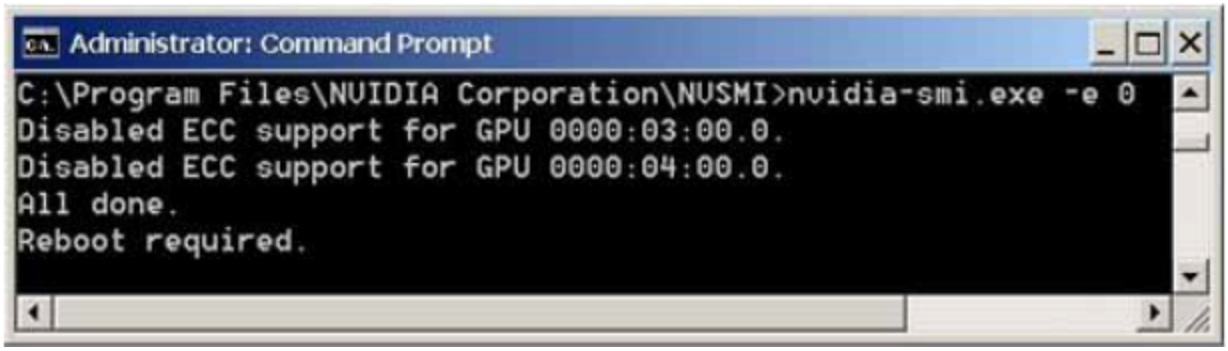


3. To further set up the configuration of GPU cards, open a command window as an administrator.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To improve the performance of GPU acceleration, it is recommended that you turn off the Error Correction Code (ECC) support by the -e 0 option of nvidia-smi. New ECC settings will be effective only after system reboot.

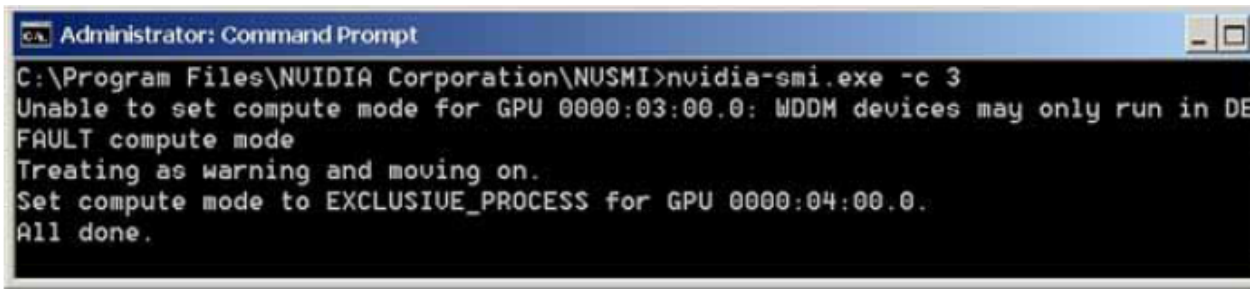


4. (Optional) For remote execution of GPU accelerated jobs (e.g., through Windows Remote Desktop Connection or RSM options), it is necessary to turn on the Tesla Compute Cluster (TCC) mode by the -dm 1 option of nvidia-smi. New TCC settings will be effective only after system reboot. This step is unnecessary if you run GPU acceleration from a local machine.

5. (Optional) For users who want to run multiple GPU-accelerated jobs on one machine through distributed mode, it is required to install Nvidia Tesla cards with EXCLUSIVE_

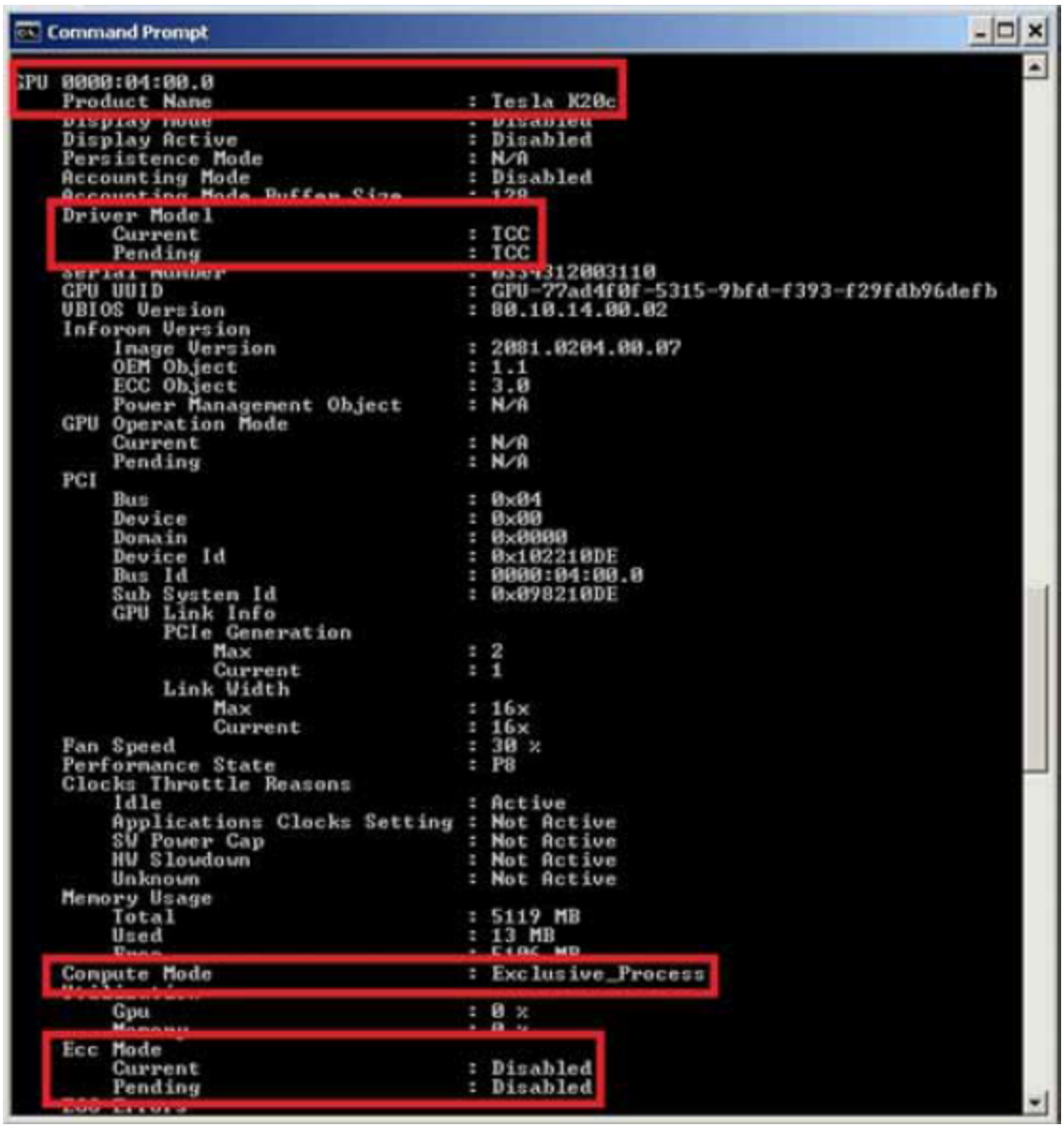
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PROCESS support. Using the -c 3 option of nvidia-smi, one can set GPUs in a system to be Exclusive_Process. HFSS Transient relies on this compute mode to assign each simulation job to a dedicated GPU card. Please note that GeForce cards do not support EXCLUSIVE_PROCESS. Therefore, they should not be used for GPU-acceleration of HFSS Transient in distributed mode.



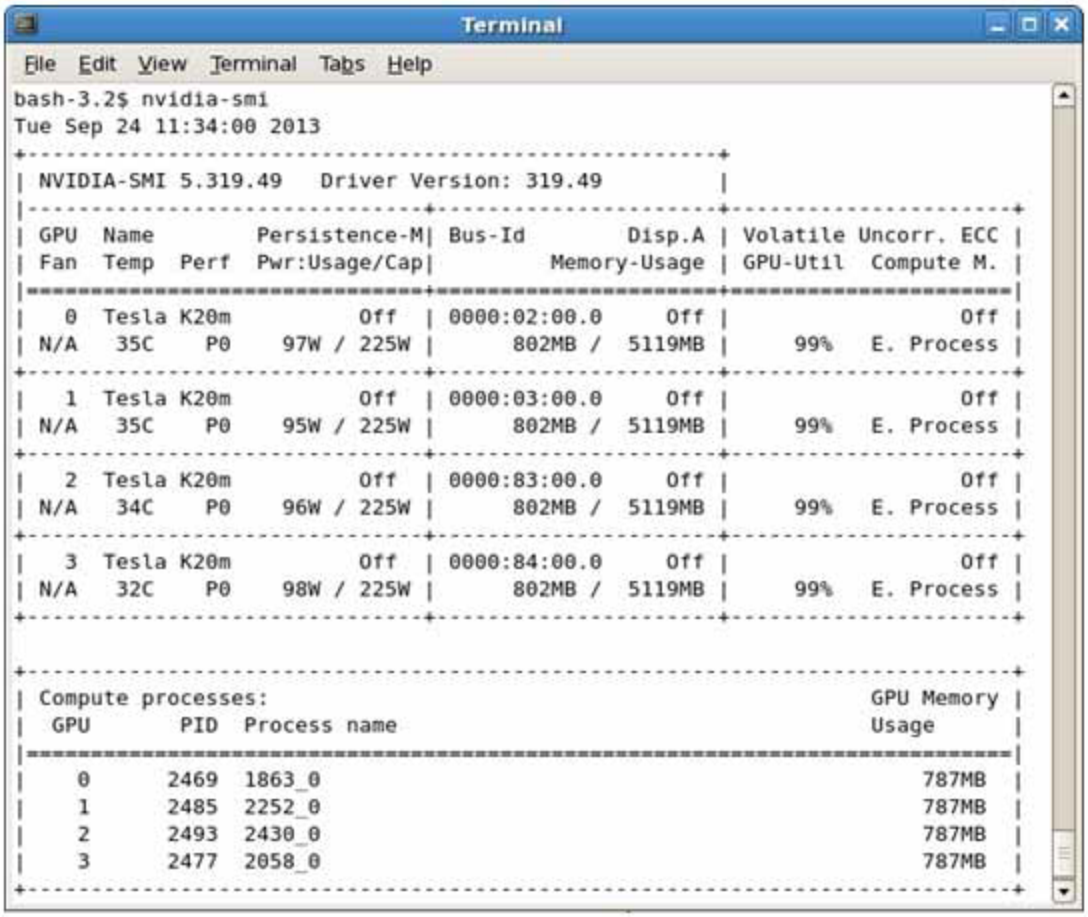
6. (Optional) Using -q option of nvidia-smi, one can check if the compute mode is set properly.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. (Optional) For HFSS Transient to run in GPU-distributed, it is necessary to check if the dynamically linked library nvml.dll exists in its default directory C:\Program Files\NVIDIA Corporation\NVSMI. If not, its path should be added to the Windows environment variable Path.
8. The setup for a Windows GPU system is complete through Steps 1 to 7.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Setup for Linux

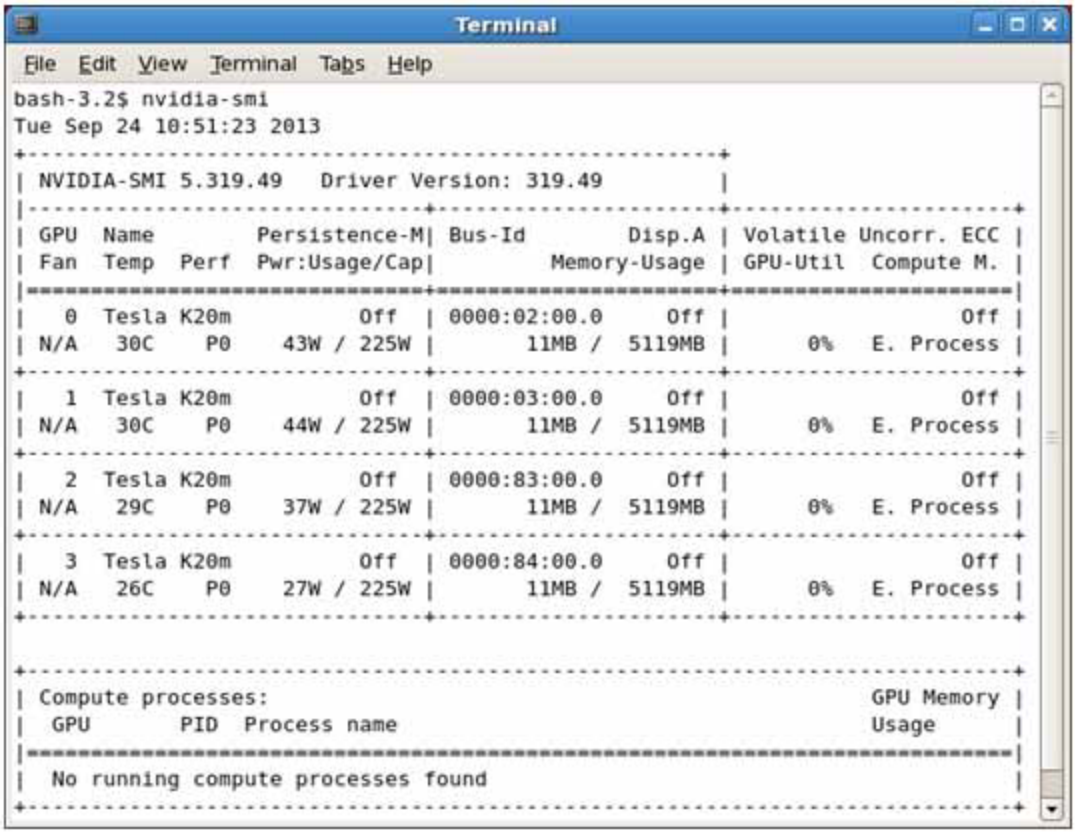
1. After you install Nvidia GPU cards and graphics drivers downloaded directly from the Nvidia website, you should be able to find the cards by the command.

   ```
   /sbin/lspci | grep -i nvidia
   ```

   You can also use the following command to check if GPU cards can be recognized by the system.

   ```
   /usr/bin/nvidia-smi
   ```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



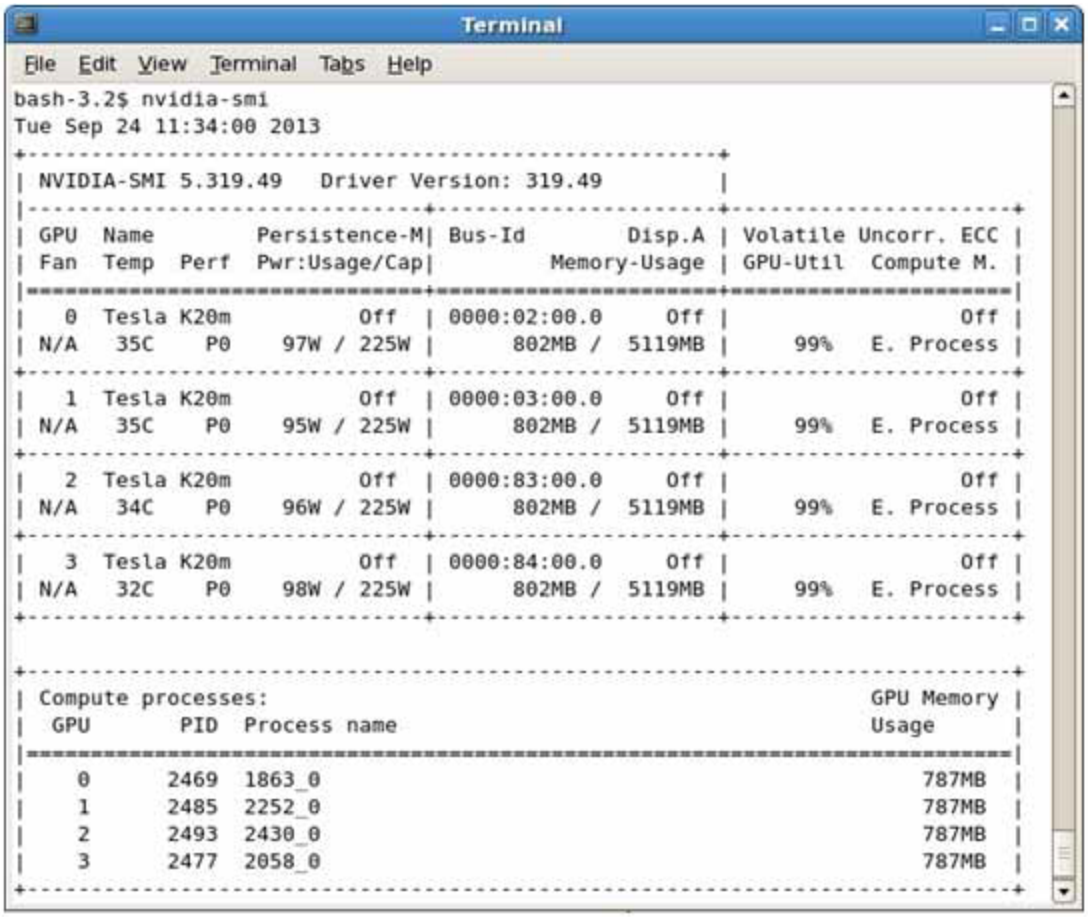
2. (Optional) The setting of GPUs to disable ECC (for performance), enable TCC (for remote execution), and enable Exclusive_Process (for GPU-distributed) are similar to Windows. You need the administrative right to make such changes.

   ```
   sudo nvidia-smi -e 0
   sudo nvidia-smi -dm 1
   sudo nvidia-smi -c 3
   ```

3. (Optional) On Linux Maximus platforms, the environment variable CUDA_VISIBLE_DEVICES can be set in the shell file ~/.bash_profile to toggle the visibility of GPU cards as CUDA devices. If the variable does not exist, all CUDA devices are visible in a system by default.
4. (Optional) In order to let Ansys Electronics Desktop solvers access the dynamically linked library libnvidia-ml.so in GPU-distributed, one must check if the library exists in its default directory /usr/lib64. If not, its path should be added to the Linux environment variable PATH through a shell startup file.

The setup for a Linux GPU system is complete through Steps 1 to 4. As an example, the following figure illustrates four distributed hf3d processes running on four Tesla K20m cards.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Integration with Ansys Remote Simulation Management (RSM)

Ansys Electromagnetics supports its own Remote Simulation Management (RSM) software along with other High Performance Computing (HPC) software management programs (see High Performance Computing (HPC) Integration).

When do you need RSM?

RSM is generally required if you want to run remote or distributed simulations. However, if you have a separate scheduling system that Ansys Electromagnetics supports, and you plan to run batchsolve simulations only, you may not need to install RSM. For details of installation and configuration of RSM, see the *Ansys Electromagnetics Installation Guides*.

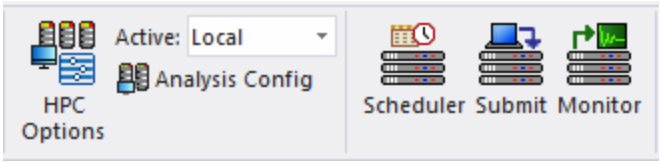
Job Management UI for RSM

You can use the Job Management UI to submit batch jobs to RSM. The Job Management UI is accessed by running Ansys Electronics Desktop on the designated 'Postprocessing node' of the

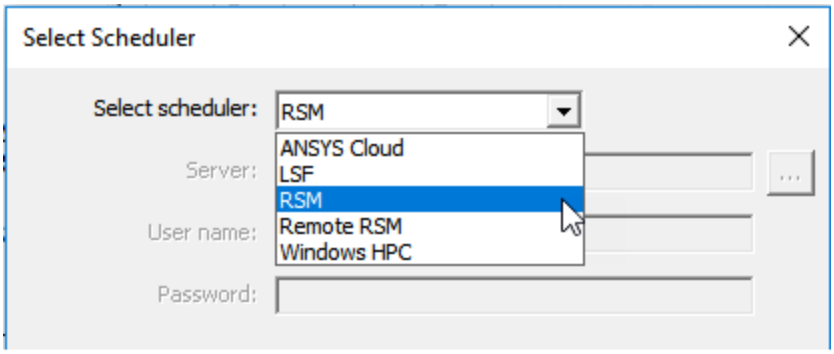
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

cluster. The Desktop provides UI commands for Scheduler selection, Job submission and Job monitoring/control. You access the Scheduler User Interface by clicking **Tools** > **Job Management** > **Select Scheduler...**

You can also select the **Simulation** tab of the ribbon, and click the **Scheduler** icon.



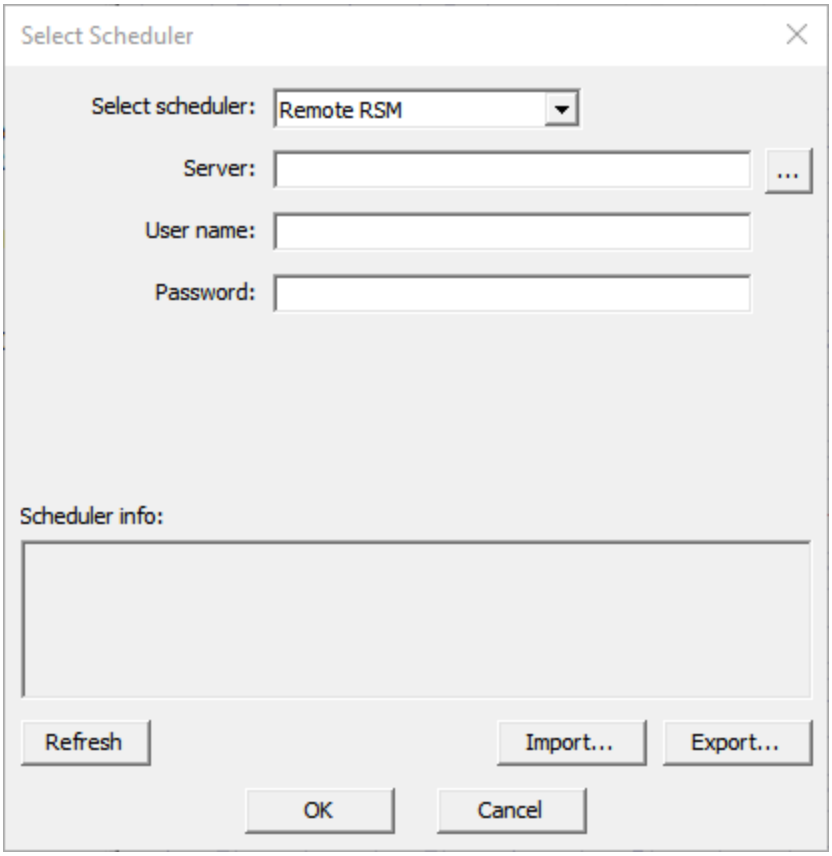
This displays the selection dialog box. The **Select scheduler** drop-down menu lists potential schedulers (which can include RSM, LSF, Windows HPC, or SGE, depending on the environment).



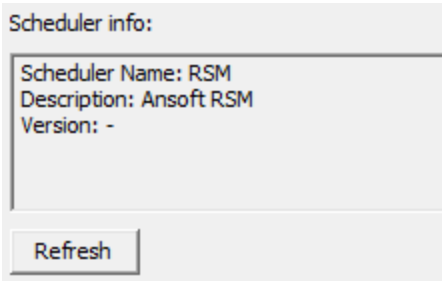
If you select a scheduler that is not supported in your environment, you receive a warning.

If you select Remote RSM and your environment has been configured , you can select a computer, user name, and password.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After selecting a scheduler, click **Refresh** to display information for that scheduler.



Once you have selected a scheduler supported in your environment, you can go through the following steps to submit a batch job.

1. Set up and prepare model on local workstation.
2. Copy the input project (or folder, if the project references external files) from a personal workstation to a shared-drive on cluster (say project is copied to /home/projects/spool/test.adsn).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the RSM environment, you are required to specify a machine-list. See: HPC and Analysis Options. For example, say the machine-list is: 3 cores from 'm1' and 3 cores from 'm2', for a total of 6 engines. You select the list on the **Compute Resources** tab of the **Submit Job to RSM** window, as described below.

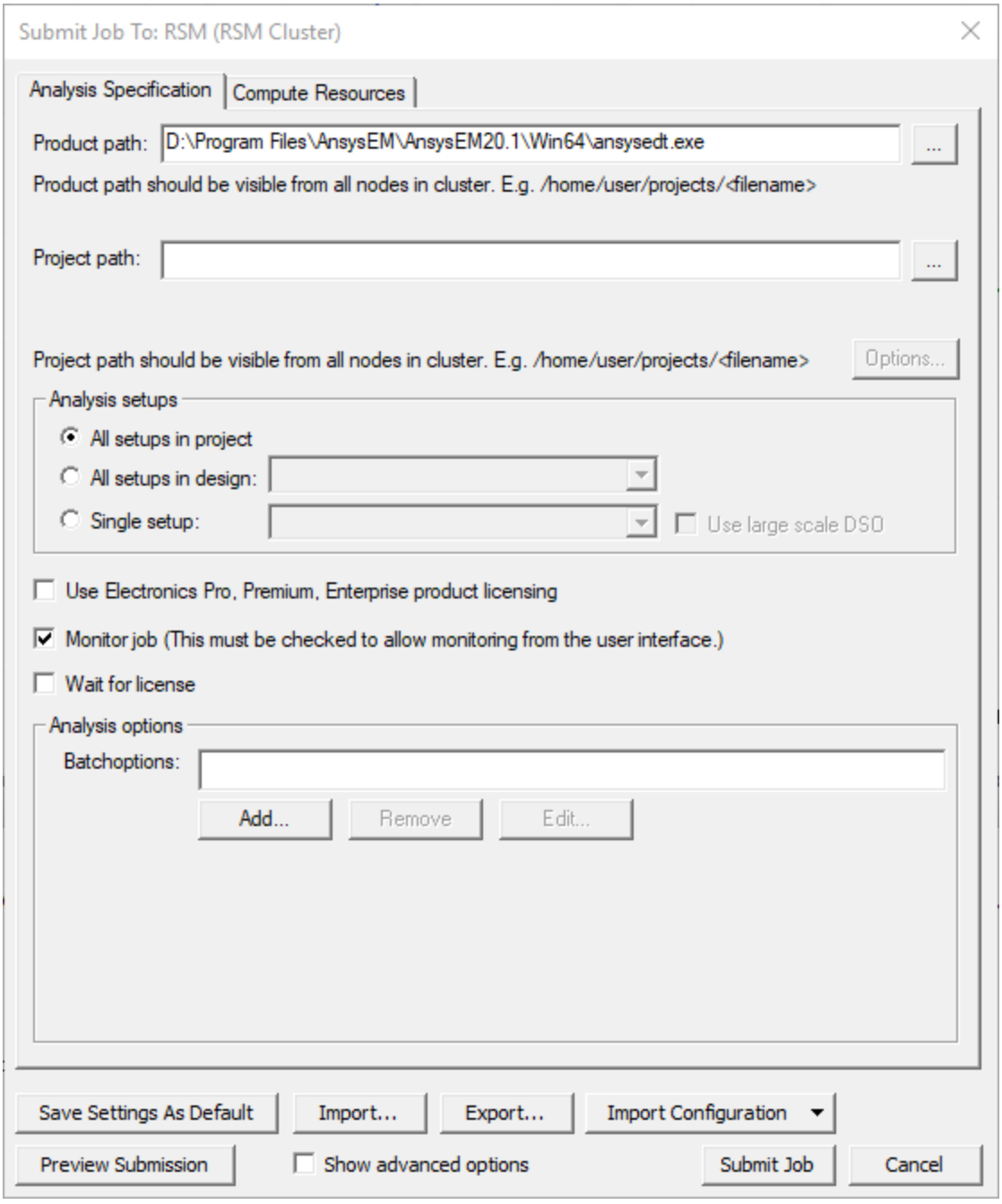
3. Open a remote-desktop session (or equivalent such as vnc session) on the node corresponding to the first machine of job's machine-list, 'm1' in this case. Launch Desktop graphically on 'm1'.
4. After setting the job submission node, select **Tools** > **Job Management** > **Submit Job...** or **Project** > **Submit Job...** or **[ProductName]** > **Submit Job...** to open the **Submit Job To:** window. You can also access **Submit Job** from the shortcut menus for the Project Name, Design name, Analysis Setup, or Optimetrics Setup.

   The **Submit Job To:** window contains two tabs:

   - **Analysis Specification**– specify the Product path, Project name, the setups, and analysis options such as batchoptions, or, for advanced users, Environment variables. If you select the Analysis or Optimetrics setup, the Analysis Specification is pre-populated.
   - **Compute Resources** – specify the amount of compute resources and how to select specific resources from those available, and automatic settings, if supported by the design types.

   The standard Job Submission window displays.

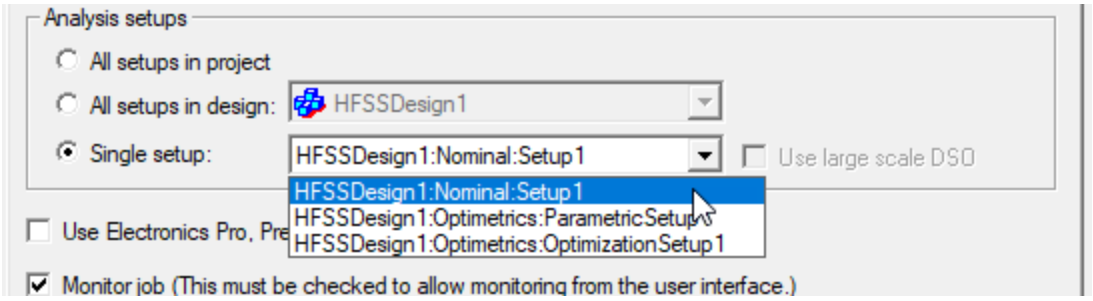
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Use the ellipsis button [**...**] to open a browser to select the project. The project can be an archive.
6. In the **Analysis setups** area, you can select radio buttons for **All setups in the project**, **All setups in the design**, or a **single setup**. For instance, the OptimTee example includes setups for Nominal, Parametric, and Optimization. If you accessed the **Submit Job** window from the right-click menus for Setup or Optimetrics Setup, this field can be

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

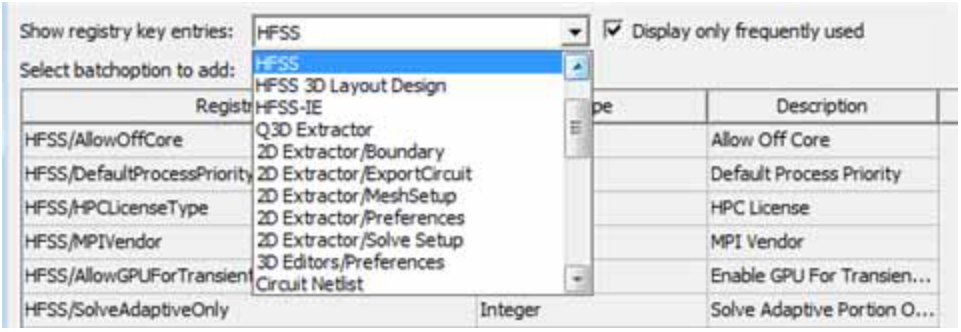
pre-populated.



For Parametric setups, you have the option to select **Use Large Scale DSO**. For details on how and when you use this feature, see Job Management Interface for Large Scale DSO.

7. The Analysis options include check boxes for Monitoring the job, whether to wait for a license, and a field for adding Batchoptions via a graphical interface, or as text.
    - If you intend to monitor the job through a user interface, you must select **Monitor job**. You can then monitor this job through the **Tools** > **Job Management** > **Monitor Jobs...** command or by checking the dialog that opens when you submit the job.
    - The Batchoptions field allows you to add additional -batchoptions parameters, either as text, or by using a dialog with selection menus. Click the **Add** button to view the **Add Batchoption** dialog box.

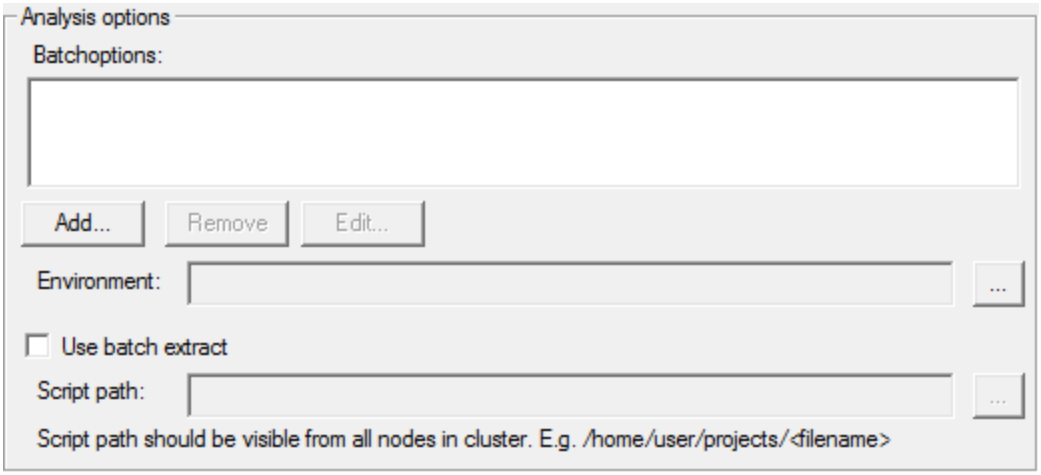
The **Show registry key entries** field lets you filter the entries displayed, by means of drop-down menu selection, and a check box to **Display only frequently used entries**.



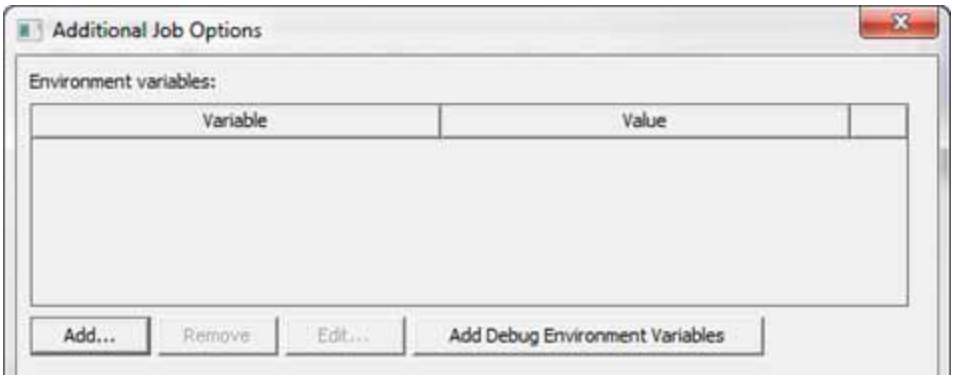
- When you have selected a batchoption, you can type the value in the field, and click the **Add** button to add the option to the batchcommand.
- In the **Submit Job To:** window, you can enable **Show advanced options** to display additional fields for environment variables, and whether to **Use batch**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

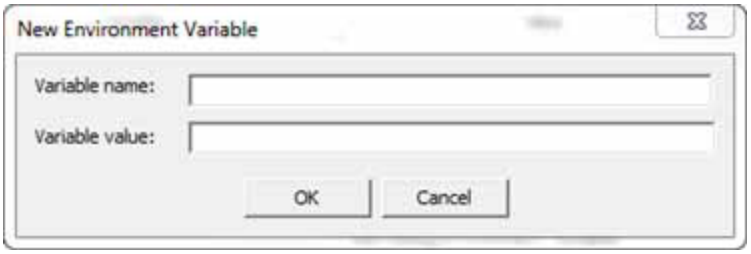
extract.



The Environment field is for environment variables, for instance, for debugging features or other variable controlled features. Click the ellipsis [**...**] button to open a dialog box for **Additional Job Options**.

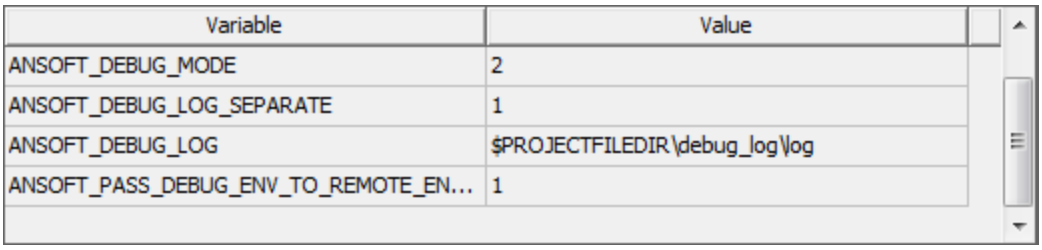


The **Add...** button opens a **New Environment Variable** dialog box in which you can include a variable name and value.



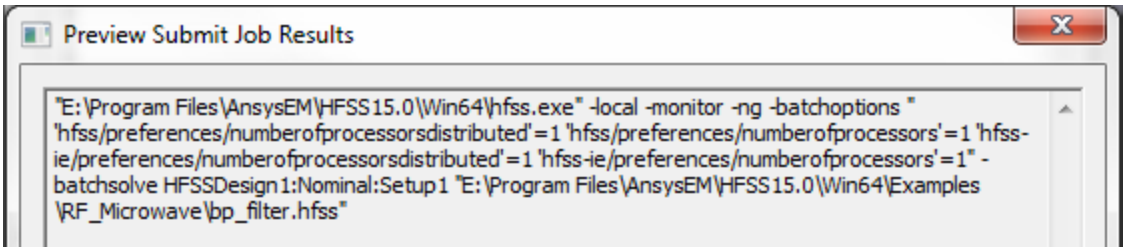
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clicking the **Add Debug Environment Variables** button automatically adds a set of debug variables. This can be useful in working with Ansys Application Engineering support.



Selecting a variable in the dialog enables the **Remove** and **Edit** buttons. The **Edit** button opens a dialog box where you can change the variable and value.

8. To see the command-line to be submitted to the scheduler, click **Preview Submission.** This opens a dialog box showing the command to be sent to the scheduler.

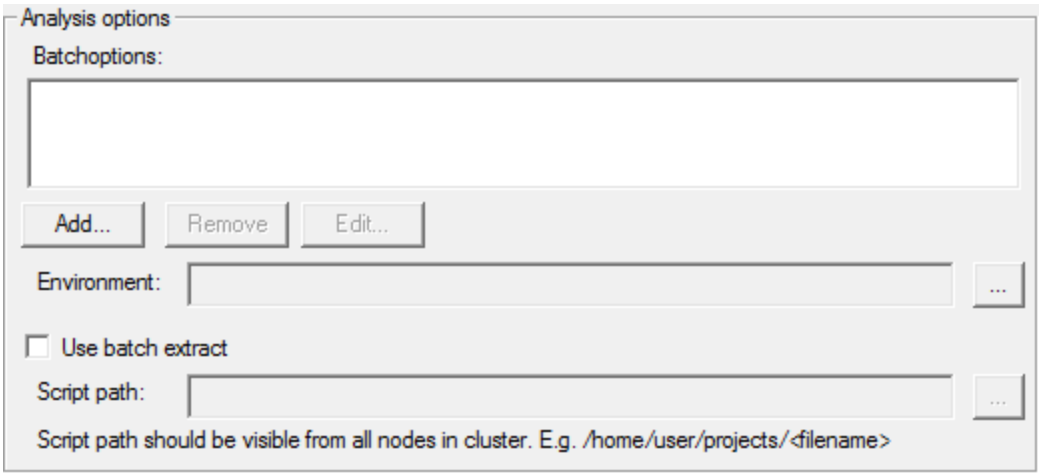


The text can be copied to the clipboard, if desired.

Use Batch Extract for RSM

Selecting **Show advanced options for RSM** also shows the Use batch extract fields.

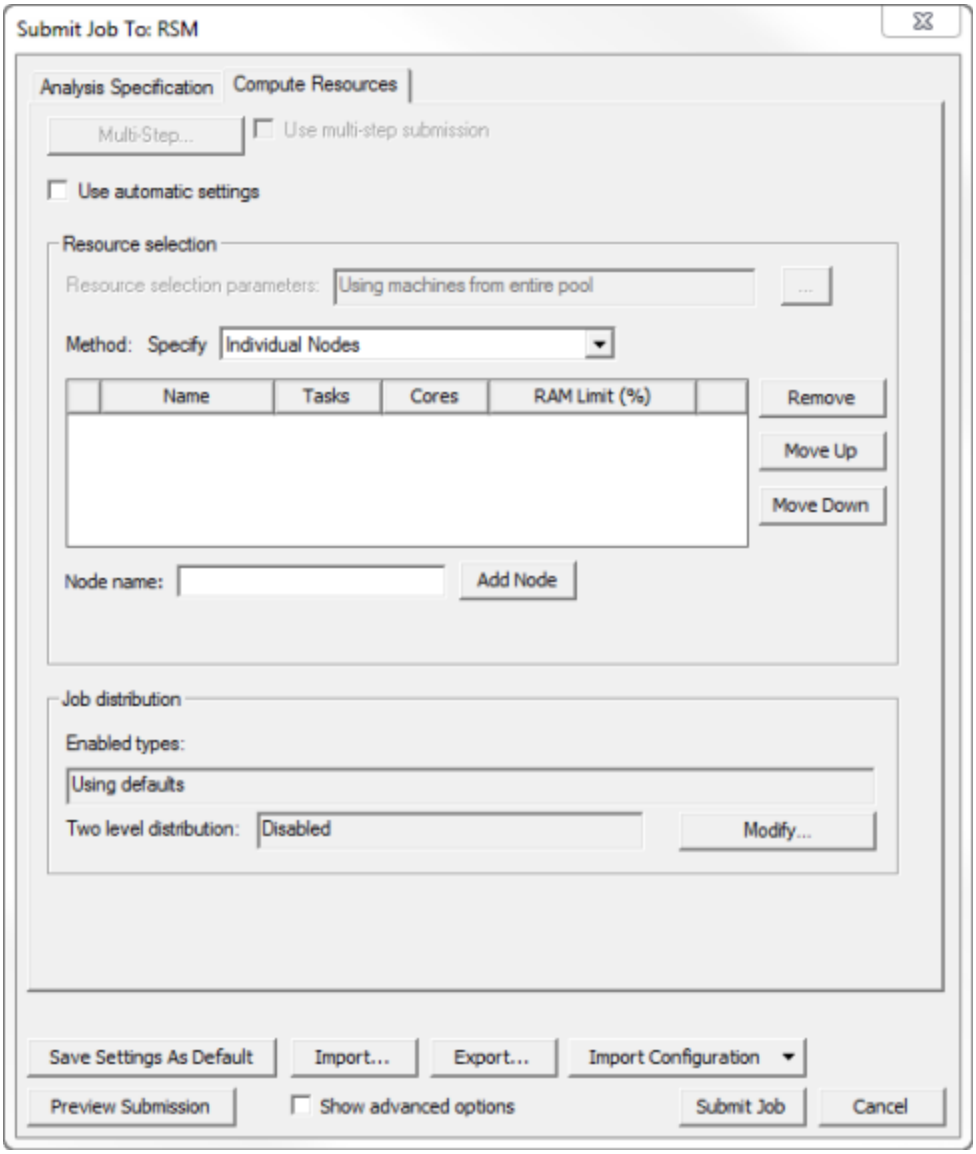
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



See the discussion on Running Ansys Electronics Desktop from a Command line for a discussion of the solve information available through batch extract.

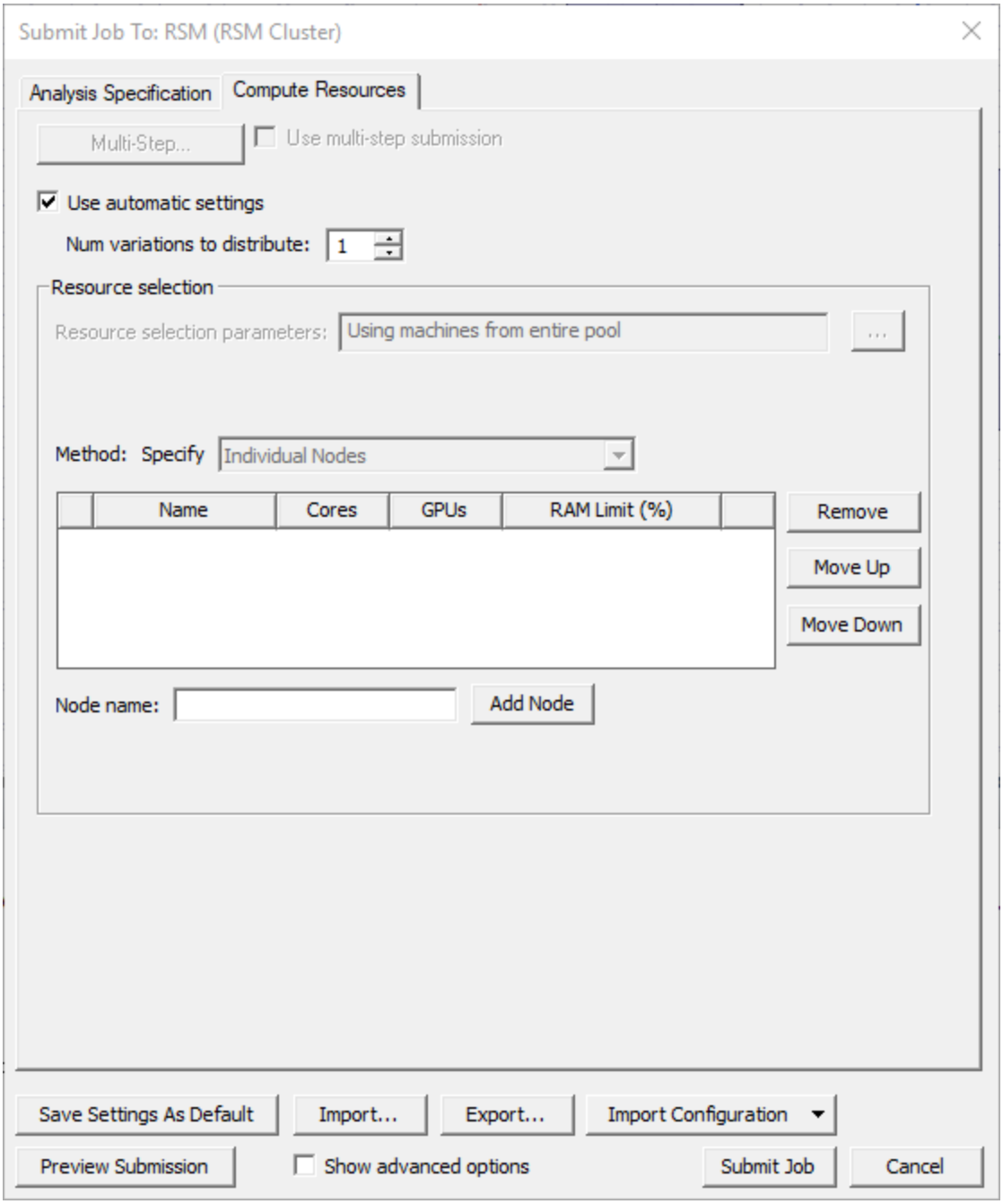
9. The **Compute Resources** tab of the **Submit Job to: RSM** window displays other parameters. Depending on the resources available for a scheduler environment, some of the fields may be disabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With **Use automatic settings** selected, the **Job distribution field** is removed and the **Use automatic settings** check box and **Num variations to distribute** field appear.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



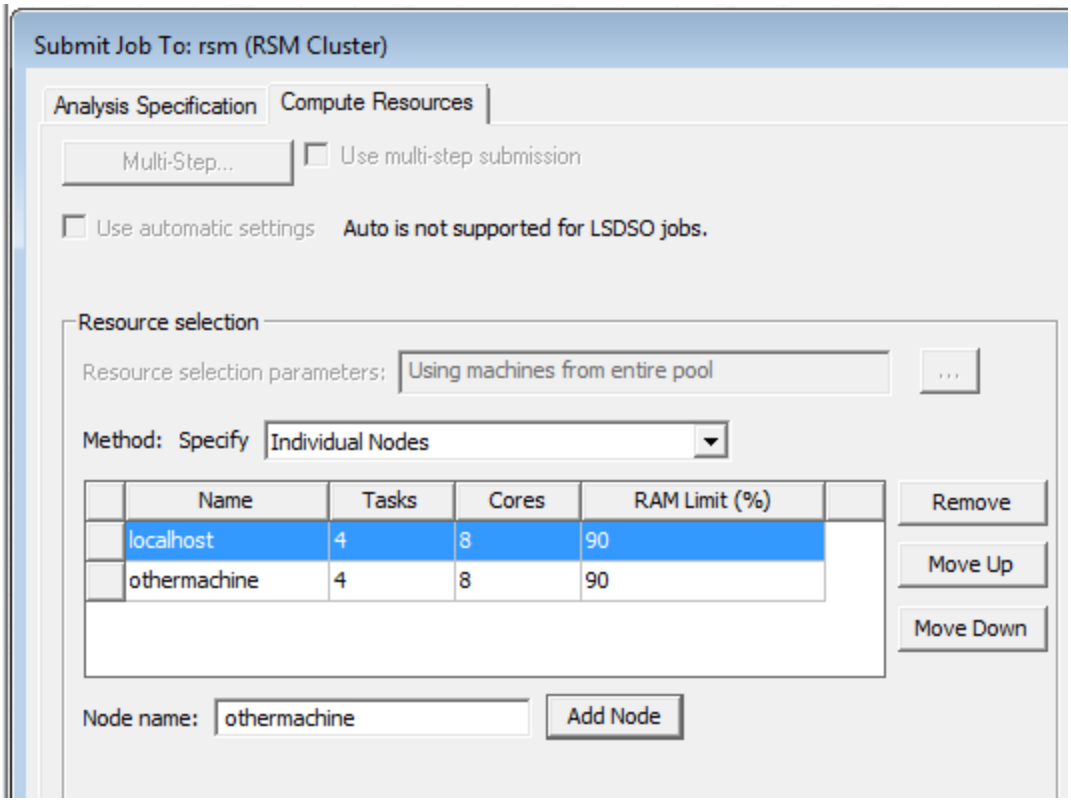
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> If you select **Use automatic settings** with **Num variations to distribute** set to 1, Optimetrics variations will be solved sequentially. Other distribution types will be distributed automatically. It does distribute frequencies, domains, and use of multiple level domains. If you set **Num variations to distribute** to 2 or more, Optimetrics variations will be solved in parallel. Other distribution types will be distributed automatically.

Otherwise:

- Specify node list

  Here you can specify a node list. In a computing environment where the available cores are not uniform, you can use this to control which resources your job will use. For use with Large Scale DSO for RSM, for jobs that are submitted from job submission panel, localhost must be the first node in the resource selection node list, other wise LSDSO solve with RSM will fail.

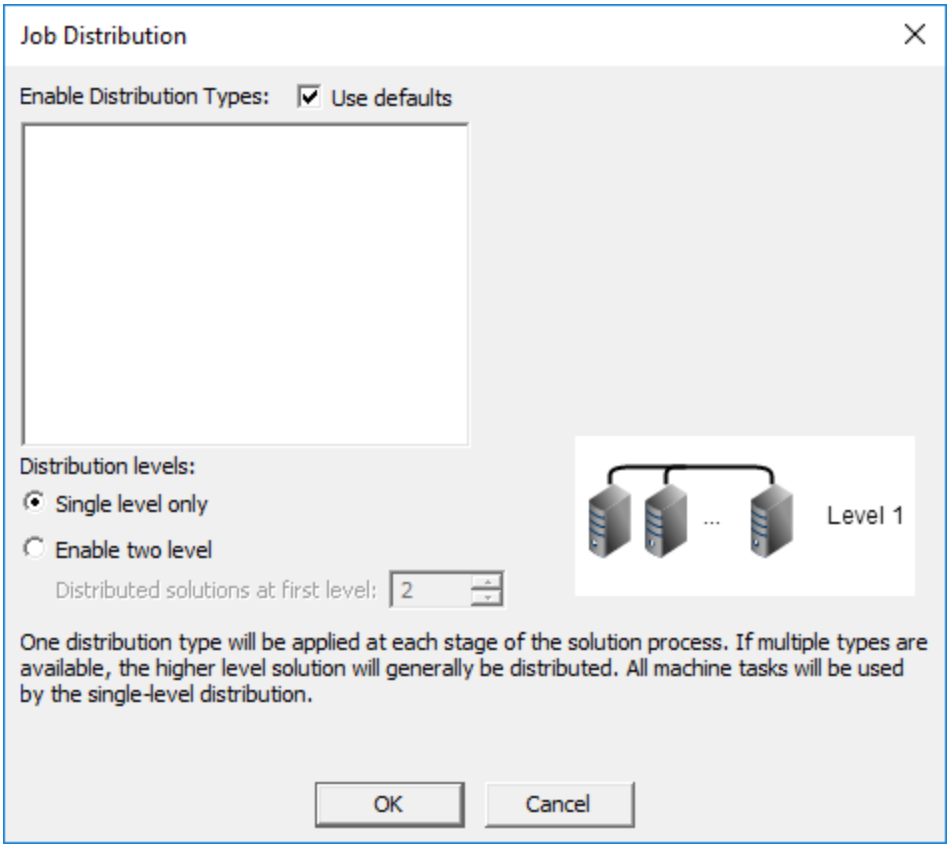


Job Distribution

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you disable User automatic selection, you can modify the Job distribution settings.

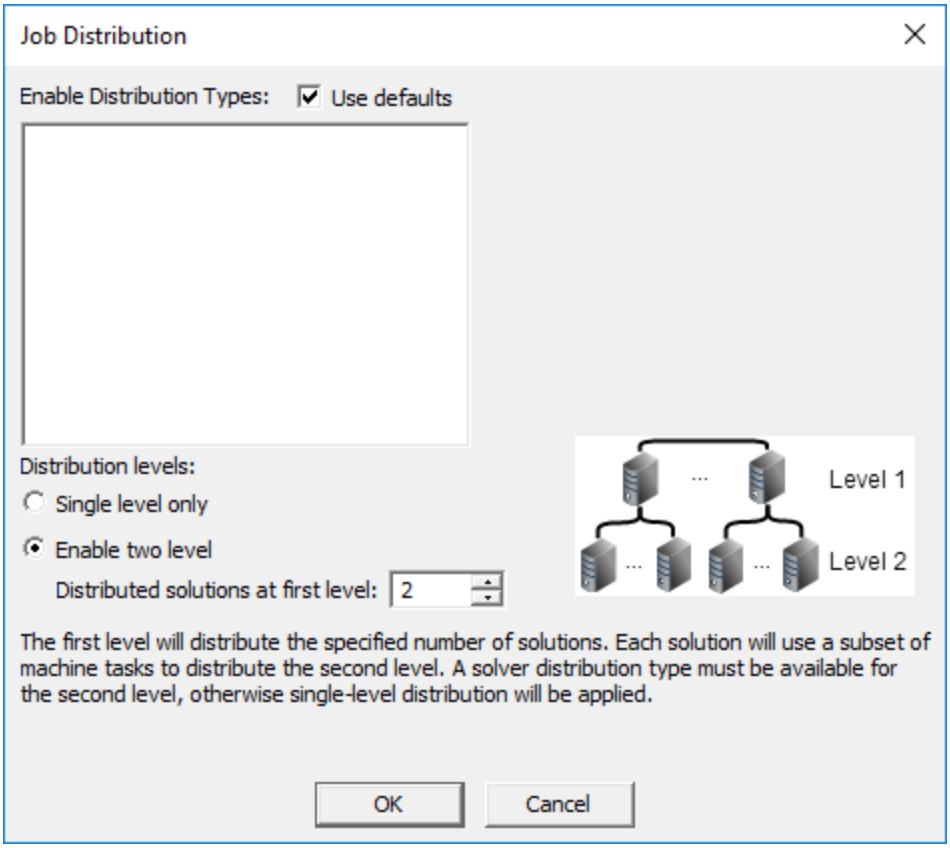
- Single level or two level distribution (*single level* is the default). Click **Modify** to display the *Job Distribution* dialog box and select the **Enable two level** option if appplicable and desired.



Enabled distribution types can be modified here.

Second-level distribution operates within DSO. If available and enabled you can specify a number of engines for level 1.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



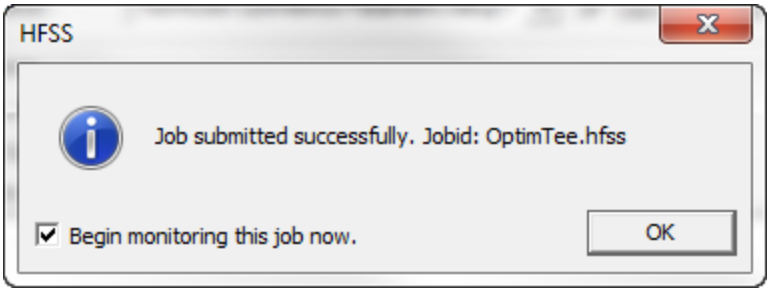
In response to a set of minimal constraints, the Scheduler may increase the resources assigned beyond the minimal values in order to meet the full set of requirements. For example, if you specify 7 distributed engines, with two processors per engine, and also limit the number of engines per node to 4, the scheduler may increase the number of cores used in order to meet the limit specified for engines per node. Notice that a preview of the Submit Job Results shows the number of resources assigned, and that the scheduler generated code includes an MPI specification.

1. To submit the command with the specified parameters, click **Submit Job**.

> **Note:**
>
> The RSM environment does not support for queuing, so **Submit Job** will immediately start running the job.

A dialog box displays. You can select **Begin monitoring this job now** and click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. You can monitor this job either automatically (by selecting the option above) or through the **Tools** > **Job Management** > **Monitor Jobs...** command. For more details, see Monitor Jobs window.

Process for Changing the Listening Port used by AnsoftRSM Service

To change the listening port used by the AnsoftRSMService, you need to change the configuration file, ansoftrsmservice.cfg, as follows:

You must specify the ListenPort within a 'CommDetails' block, which must be within a 'Default:CommDetails' block, which must be within the top level block of the file, the 'AnsoftCOMDaemon' block. The following example shows the listen port changed from 32958 to 32957, with these blocks at the beginning of the file:

```
$begin 'AnsoftCOMDaemon'
  $begin 'Default:CommDetails'
    $begin 'CommDetails'
      ListenPort='32957'
    $end 'CommDetails'
  $end 'Default:CommDetails'
  . . . .
$end 'AnsoftCOMDaemon'
```

For the second level block, ensure that there is a single colon character and no spaces or tabs separating the two parts of the block name 'Default:CommDetails'. The third level block, with name 'CommDetails' is also required. Use caution when editing this file by hand, because any typos in the block or value names may cause the data to be ignored.

Changing the AnsoftRSMService Listening Port

For Remote Analysis or Distributed Analsysis, processes may need to be started on multiple hosts. If the Ansys Electromagnetics Desktop needs to start a process on a remote host, the AnsoftRSMService is used to start these remote processes. By default, the AnsoftRSMService listens for socket connections from the Ansys Electromagnetics Desktop on port 32958.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This section describes how to change the port number used by the AnsoftRSMService.

To change the port number, both the AnsoftRSMService and the Ansys Electromagnetics Desktop must be configured to use the new port number. The same port number must be used for Ansys Electromagnetics Desktop and for the AnsoftRSMService process running on each host used for the analysis.

AnsoftRSMService Configuration

The AnsoftRSMService port number is configured in the ansoftrsmservice.cfg configuration file. This configuration file is located in the platform specific subdirectory of the RSM installation directory.

The default location of this directory is "C:\Program Files\ANSYS Inc\v251\EM_RSM" on Windows.

The default location of this directory is /opt/ansys_inc/v251/EM_RSM on Linux.

To modify the AnsoftRSMService configuration, first stop the ansoftrsmservice, then modify the ansoftrsmservice.cfg configuration file, then restart the ansoftrsmservice.

> **Note:**
>
> When using Linux, root maintains exclusive control over systemd-managed services. Once the startonboot command has been given, only root users can start and stop AnsoftRSMService, using the commands:
>
> ```
> sudo ansoftrsmservice start
>
> sudo ansoftrsmservice stop
> ```

The beginning of the configuration file should appear as follows:

```
$begin 'AnsoftCOMDaemon'
$begin 'Default:CommDetails'
$begin 'CommDetails'
'ListenPort'='32958'
$end 'CommDetails'
$end 'Default:CommDetails'
```

If there are additional lines between the following two lines, then they should not be modified:

```
$begin 'AnsoftCOMDaemon'
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$begin 'Default:CommDetails'

To change the port number, modify the ListenPort setting within the single quotes from 32958 to the desired port number. The single quotes should not be removed or changed.

For previous versions of the software, the ansoftrsmservice.cfg file may not contain the lines:

$begin 'Default:CommDetails'

$begin 'CommDetails'

'ListenPort'='32958'

$end 'CommDetails'

$end 'Default:CommDetails'

If these lines are not present, then add them to the ansoftrsmservice.cfg file after the first line of the file, and change the ListenPort to the desired port number. For example, to change the port number to 32000, the beginning of the ansoftrsmservice.cfg file should look like the following after the changes:

$begin 'AnsoftCOMDaemon'

$begin 'Default:CommDetails'

$begin 'CommDetails'

'ListenPort'='32000'

$end 'CommDetails'

$end 'Default:CommDetails'

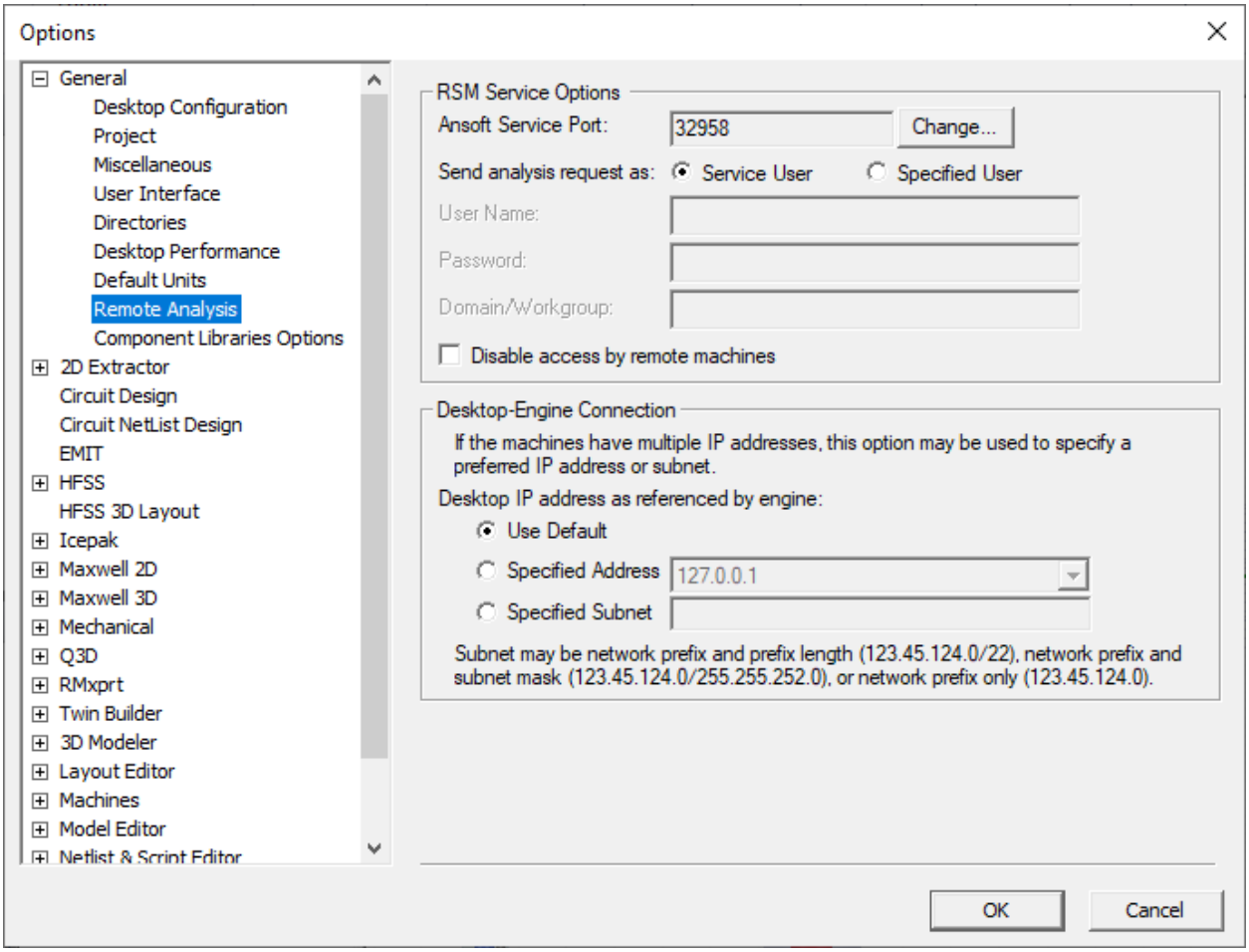
Ansoft Electromagnetics Desktop Configuration

The port number for connecting to the AnsoftRSMService is configured from the **General Options** window.

To access these options:

- Select **Tools** > **Options** > **General Options** to open the **General Options** window.
- If it is not already expanded, expand the **General** tree and click **Remote Analysis**.

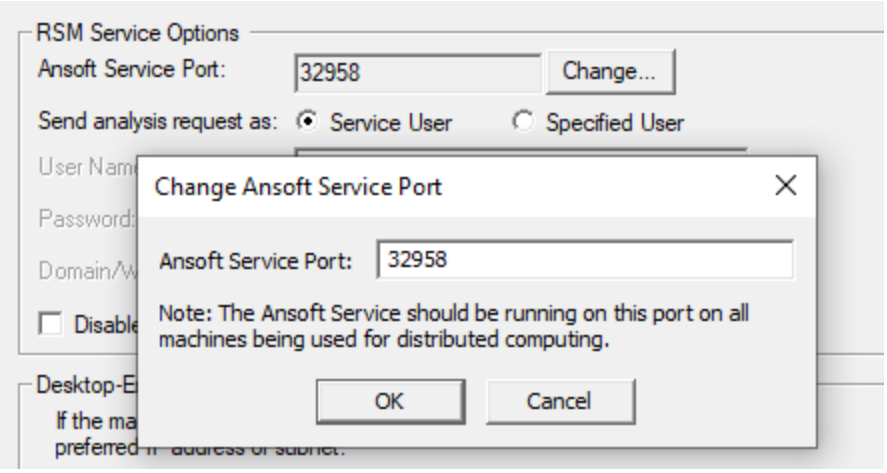
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Under **RSM Service Options**, the **Ansoft Service Port** field shows the current AnsoftRSMService port number.

1. Click **Change** to modify this setting. This opens the **Change Ansoft Service Port** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Enter the new port number in the **Ansoft Service Port** field.
3. Click **OK** to close the dialog box and accept the new port number.
4. Click **OK** to close the **General Options** window and accept the changes.

Running HPC Diagnostics

The Ansys EM HPC diagnostics tool simplifies HPC troubleshooting by automating diagnosis of routine issues. The diagnostics tool is run on the cluster as a scheduler managed job. Using its HTML based diagnostics report, cluster administrator or Ansys support staff can either resolve the issue, or guide the user with steps for further troubleshooting. In some cases, Ansys support staff may request to rerun the diagnostics with additional diagnostics tests. The user may extend the diagnostic scripts to suite their HPC environment.

This note describes how to use the diagnostics tool.

- Supported schedulers
- Running the diagnostics job
- Standard diagnostic job
- Using diagnostics scripts on Linux clusters
- Using Windows HPC job file
- Diagnostic report
- Site-specific diagnostics job
- Environment variables
- ANSYSEM_DIAG_PROD_DIR contents
- ANSYSEM_DIAG_RESULTS_DIR contents
- How does the diagnostic tool work

Supported Schedulers

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The tool supports diagnosis of issues on Linux and windows clusters managed by the following schedulers:

- LSF
- SGE
- PBS/Torque
- Windows HPC

For the above schedulers (see High Performance Computing (HPC) Integration), the tool includes basic diagnostic scripts. Further, if password-ssh has been enabled, it also supports generic Linux clusters using ssh. Please note that currently diagnostics tool does not support PBSPro and LSF/Windows.

Running the Diagnostics Job

The diagnostics are run as a scheduler managed job. Once the job finishes, you locate the resulting HTML file and provide it to the cluster administrator or to Ansys support staff. In case, there are any job or test failures, please also provide the networking*.json files from the Hosts subdirectory as well.

Basic Diagnostic Job

To run the basic diagnostics, submit a diagnostic job to the scheduler using a provided job submission script. Each basic diagnostic job is a 12 core job with 4 cores per host. On Linux, running this script submits a scheduler job to run the diagnostic tool on the cluster. On Windows, you need to submit a job using a job file.

Basic scripts for each supported scheduler are available in diagnostics subdirectory of schedulers directory.

Linux:

```
opt/ansys_inc/v251/AnsysEM/schedulers/diagnostics
```

Windows:

```
"C:\Program Files\ANSYS Inc\v251\AnsysEM\schedulers\diagnostics
```

Using Diagnostics Scripts on Linux Clusters

The following basic scripts are provided in the diagnostics directory (.../ansys_inc/v251/AnsysEM/schedulers/diagnostics):

These job submission scripts are scheduler specific.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scheduler	Basic job submission script	Comment
LSF	test_lsf	Supports both lsrun and blaunch
SGE	test_sge	Supports both qrsh and rsh
PBS/Torque	test_torque	Requires changing the PATH and PBS_BINARY_PATH environment variable
Generic Linux cluster	test_ssh	Supports only ssh. Requires password-less ssh. Requires creating a file with the names of hosts and saving it in ${HOME}/ansysem_hostfile

Using Windows HPC Job File

A sample job file winhpctest.xml is available in the diagnostics directory:

`...\schedulers\diagnostics.`

To submit this diagnostic job, you must change the job description to suite your environment as following:

1. Select a directory for saving the diagnostic results. This directory must be accessible at the same path from all the hosts of the cluster.
2. Locate the directory for Ansys EM installation. This directory also must be accessible at the same path from all the hosts of the cluster.
3. Locate the winhpctest.xml in the diagnostics subdirectory of schedulers directory in Ansys EM installation.
4. Start Windows HPC job manager, and choose "New job from XML File…" action.
5. Select the winhpctest.xml job file.
6. Change the value of both the following environment variables with the directories located in the first two steps:

   ANSYSEM_DIAG_PROD_DIR

   ...

   ANSYSEM_DIAG_RESULTS_DIR
7. Now submit the job.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> After making the above changes, you can also save the resulting XML file using "Submit Job XML File…". Then you can submit the job using the job command as following:
>
> job submit /jobfile: XMLfile name

Diagnostic Report

The diagnostic report is an HTML file that (along with other related diagnostics results) is placed in the following directory

Linux:

```
${HOME}/Ansoft/HPCDiag/Results/JOBID
```

Windows:

```
%ANSYSEM_DIAG_DIR%\Results\JOBID
```

Report file:

```
.../HTML/report.html
```

where JOBID is the job ID assigned by the scheduler. On Windows, the user must specify ANSSEM_DIAG_DIR directory.

Site-Specific Diagnostics Job

To run a diagnostic job with job submission parameters of your choice, you need to create your own job submission script. For example, you may want to specify a different LSF queue, or select a different SGE parallel environment. To run such a job, you need to create your own job submission script starting from the basic diagnostic scripts with the following steps:

1. Locate the relevant basic diagnostic script in the diagnostics subdirectory of schedulers directory in Ansys EM installation.
2. Make a copy of the diagnostics script into a directory that is accessible from a submit host for the cluster.
3. Edit the script file to change the value of ANSYSEM_DIAG_PROD_DIR environment variable to point it to the installation directory (See below).
4. Modify the job submission parameters as needed.
5. Optionally, copy any site-specific diagnostic tests provided by Ansys support staff in the *../Custom* subfolder of the *ANSYSEM_DIAG_RESULTS_DIR* directory.
6. Run the diagnostics script from a submit host for the cluster.

Environment Variables

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following environment variables are applicable for both Linux and Windows environment.

ANSYSEM_DIAG_PROD_DIR

Environment variable	**ANSYSEM_DIAG_PROD_DIR**
Description	Location of the Ansys EM installation. This must be available at the same path from all the hosts of the cluster.
Windows example	"\\filer\AnsyEM\v251\AnsysEM"
Linux example	/shared/ansys_inc/v251/AnsysEM
Comments	Windows: Required. Linux: Optional. Export this environment variable if you make a copy of the diagnostic script.

ANSYSEM_DIAG_RESULTS_DIR

Environment variable	**ANSYSEM_DIAG_RESULTS_DIR**
Description	Location of the diagnostic report and other results on a shared drive. This must be available at the same path from all the hosts of the cluster
Example	\\filer\Home\User\Ansoft\HPCDiag
Linux example	/shared/home/user/Ansoft/HPCDiag
Comments	Windows: Required. Linux: Optional. Export this environment variable if the home directory for the user is not accessible from the cluster.

ANSYSEM_DIAG_CUSTOM_DIR

Environment variable	**ANSYSEM_DIAG_CUSTOM_DIR**
Description	Location of the configuration of product tests and other custom site-specific tests. This location must be on a shared drive that is available at the same path from all the hosts of the cluster
Example	\\filer\Home\User\Ansoft\HPCDiag\Custom
Linux example	/shared/home/user/Ansoft/HPCDiag/Custom

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Environment variable	**ANSYSEM_DIAG_CUSTOM_DIR**
Comments	Windows: Optional. You may want to specify it if the path %ANSYSEM_DIAG_RESULTS_DIR%\..\Custom is not suitable Linux: Optional. Export this environment variable if the home directory for the user is not accessible from the cluster.

How the Diagnostic Tool Works

The diagnostics are run as a scheduler managed job. Running the diagnostic script submits a scheduler job that runs the diagnostic tool on the hosts allocated to the job. Once the diagnostic job starts, the tool executes a set of diagnostic tests. These tests run on each host allocated to the job, and collect diagnostic information relevant for running HPC jobs. The tool combines the diagnostic information to produce an HTML report. The tool saves HTML diagnostic report and other results in a shared drive, which must be available at the same path from all the hosts of the cluster. On Linux, the default is Ansoft/HPCDiag subdirectory under user's home directory. On Windows, the user must specify this location using ANSYSEM_DIAG_RESULTS_DIR environment variable.

Changing Solution Priority for System Resources

You can modify the priority of Ansys Electronics Desktop simulations so that system resources are allocated to other computer processes before the solver. If you reduce the priority of Ansys Electronics Desktop simulations, your other software tools will respond as they normally would, but Ansys Electronics Desktop simulations may take longer.

> **Note:**
>
> The Windows Task Manager does not indicate a reduced priority for the Ansys Electronics Desktop solvers. It only lists the priority of the engine manager, which appears normal, not the actual engine. The actual engine is in a separate thread, whose priority is not visible in the Windows Task Manager.

To change the priority of simulations for the system's resources:

1. While a solution is running, right-click the **Progress** window, and click **Change Priority** on the shortcut menu.
    - To affect priority for future simulation runs, click **Tools** > **Options** > **HPC and Analysis** to access the **HPC and Analysis** dialog box, and click the **Options** tab.
2. From the **Change Priority** menu (or the **Default Process Priority** drop-down menu), select one of the following priorities:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lowest Priority	
Below Normal	
Normal	The default.
Above Normal	
Highest	

3. Click **OK**.

desktopproxy Utility to Abort or Cleanstop a BatchSolve

The desktopproxy Utility to abort or cleanstop a batch solve is available in the installation directory: ("...\ANSYS Inc\v251/AnsysEM" on Windows, ".../ansys_inc/v251/AnsysEM" on Linux).

To abort a job, run it on a host which has network access to the first execution host of the job, as well as has access to the directory of the project file. This utility does not abort Large scale DSO jobs for which you need to use desktopjob utility itself.

Usage:

```
desktopproxy -abort <projectpath>
-cleanstop <projectpath>
```

Where the <projectpath> is the full path to the project file.

The behavior (e.g., saving the results computed so far) should be same as if you interactively aborted AnsysEDT solve. if AnsysEDT is 'busy' (e.g., doing Optimetrics calculation, computing a report), abort (using desktopproxy) might take a long time.

The utility exits silently after it requests ansysedt process to abort the job cleanly. After the job ends, the batch log file may contain a message such as:

```
[error] Project:TestProject, Design:HFSSDesign1 (DrivenModal),
Simulation was aborted by user on server: sjohpc-lsf-0.ansys.com.
(3:52:30 PM Oct 11, 2013)
```

The lock file will be automatically deleted.

Example (Linux):

```
desktopproxy -abort /home/user/test/Diff_Via.aedt
```

Example (Windows):

```
desktopproxy -abort \\server\share\projectfolder\Diff_Via.aedt
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Limitations:

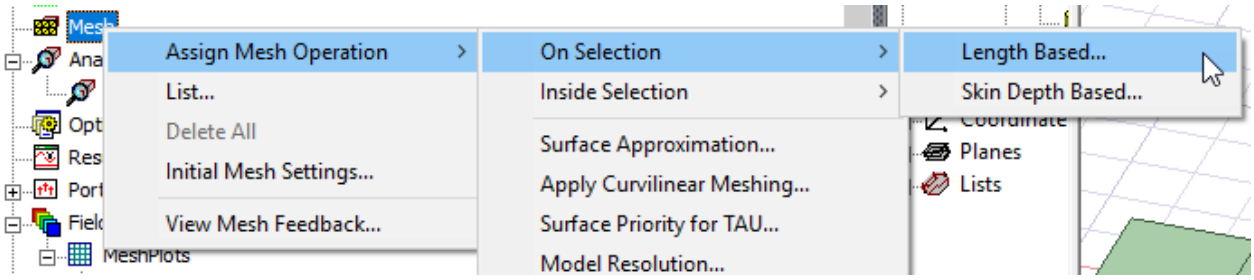
- Aborting a job makes a best effort to delete all the temp directories on all the hosts
- For a hung job, a clean shutdown may not be possible. In that case, the user must use OS/scheduler's facilities to kill the job.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

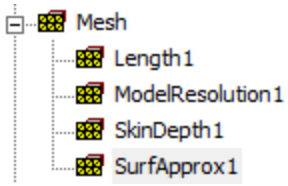
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10 - Defining Mesh Operations

In Project Manager, mesh operations are optional mesh refinement settings that provide the solvers with mesh construction guidance. This technique of guiding mesh construction is referred to as "seeding" the mesh. Seeding is performed using the **Assign Mesh Operation** commands on the respective solver menu.



Each mesh operation you define appears in the Project tree.



Note:

There are no restrictions against assigning multiple mesh operations to the same entities. The *type* of mesh operation can differ between overlapping assignments. For example, length-based refinement could be assigned inside a body, and skin-depth seeding on one or more of its faces. Additionally, overlapping mesh operations of the *same type* can have differing, or even redundant, parameter values (such as element length) without causing an error condition.

Typically, the operation that dictates the finest local meshing results will control the outcome in overlapping assignment areas or volumes.

You can instruct the solvers to reduce the maximum size of tetrahedral elements on a surface or within a volume until they are below a certain value (length-based mesh refinement) or you can instruct solvers to refine the surface triangle length of all tetrahedral elements on a surface or volume to within a specified value (skin depth-based mesh refinement). These types of mesh operations are performed on the *current mesh*, that is, the most recently generated mesh.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In a few circumstances, you may also want to create a mesh operation that modifies a solver's surface approximation settings for one or more faces. Surface approximation settings are only applied to the *initial mesh*, that is, the mesh that is generated the first time a design variation is solved.

You can also specify Initial Mesh Settings to apply to all objects; however, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

You can also choose to override automatic choice of which mesher a product uses, by using the [*solverName*] > Mesh > Initial Mesh Settings command.

See the technical notes for more details about Ansys Electronics Desktop's application of mesh operations.

What do you want to do?

- Length Based Mesh Refinement
- Perform length-based mesh refinement on object faces
- Perform length-based mesh refinement inside objects
- Perform skin depth-based mesh refinement on object faces
- Modify surface approximation settings for one or more faces
- Apply Curvilinear Elements
- SBR+ Curvature Extraction for Faceted Surfaces
- Specify automatic or specified model resolution for a selection
- Specify the initial mesh settings
- TAU Flex Meshing

Length Based Mesh Refinement

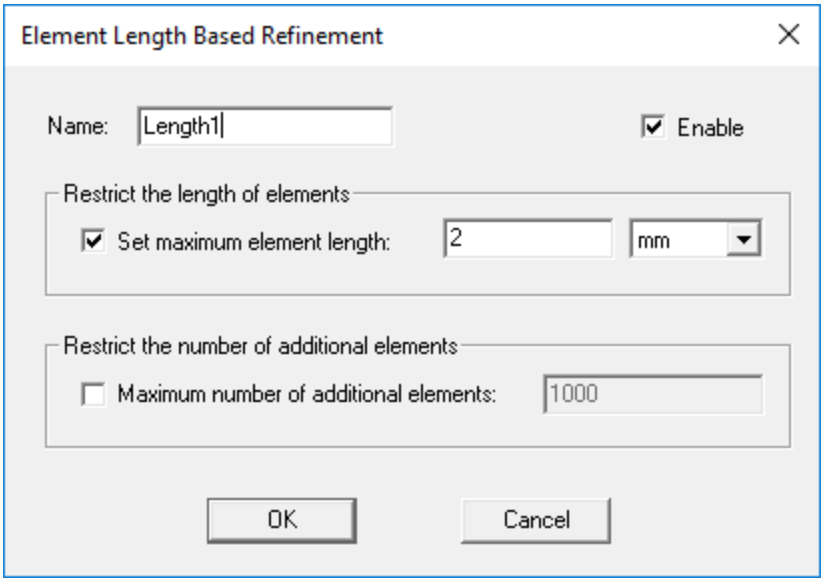
Length-based mesh refinement settings help control the length of the tetrahedral elements and the growth of the whole mesh. If you are not sure how big a refinement is needed, by way of mesh refinement you can set a ceiling on the mesh growth with the option **Restrict the Number of Additional Elements**.

To apply a length based mesh operation either to an object face or inside an object:

1. Select the object in the modeling workspace.
2. In the Project Manager, right-click **Mesh** > **Assign Mesh Operation** > **[On Selection / Inside Selection]** > **Length Based**.

   The **Element Length Based Refinement** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Enter a **Name** for the mesh operation, or leave the default.
4. Select the **Enable** check box to set the operation as active. You can later deselect this to deactivate it.
5. To restrict the length of tetrahedra edges, select the **Set maximum element length** check box and specify values and units using the drop-down menus.
6. To restrict the number of elements added during refinement, select the **Maximum number of additional elements** check box and specify a value.
7. Click **OK**.

When you set a mesh operation on a selection of objects (more than one) and set a maximum length, it applies to the whole mesh. Even if you selected just one object, when you refine its boundary, elements are added both inside and outside of that object.

When there are no more elements to refine or when no more elements can be added, the refinement process stops because it has either met the growth limit or the element length target. However, there are more quality improvement swaps, smoothing, matching boundary issues performed at the finishing stage that might slightly alter the mesh size.

If you restrict the number of elements to say 10000, it will add 10000 elements. If you have multiple mesh operations each for 10000 elements, each mesh operation will add 10000 elements (assuming there was no element length targets).

If you set a restriction of say 2000 elements on an object which creates 1100 elements after lambda refinement without any mesh operation, the starting mesh will contain more than 3000 elements.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning Length-Based Mesh Refinement on Object Faces

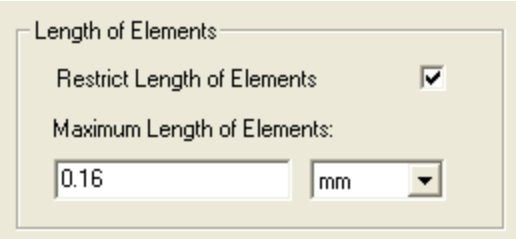
1. Select the faces you want to refine.

   Alternatively, select an object if you want to refine every face on the object.

2. Click **HFSS** > **Mesh** > **Assign Mesh Operations** > **On Selection** > **Length-Based** or right-click **Mesh** in the Project Manager and select **Assign Mesh Operation** > **On Selection** > **Length-Based.**

   The *Element Length-Based Refinement* dialog box appears.

3. Type a name for the mesh operation in the **Name** text box or accept the default name.
4. To restrict the length of tetrahedra edges touching the faces:



   a. Select **Restrict Length of Elements**.
   b. Type the maximum length of the tetrahedral edges touching the faces in the **Maximum Length of Elements** text box.

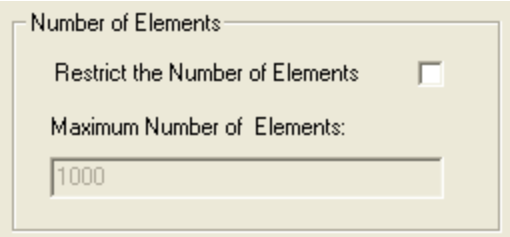
   The solver will refine the element edges touching the selected faces until their lengths are equal to or less than this value.

   The default value is set to 20% of the maximum edge lengths of the bounding boxes of each selected face.

   For first order basis functions, a maximum length of $\frac{\sqrt{2}}{10}\lambda$ is recommended for radiation boundary surfaces.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. To restrict the number of elements added during refinement of the faces:



   a. Select **Restrict the Number of Elements**.
   b. Enter the **Maximum Number of Elements** to be added.
   c. Click **OK**.

When the mesh is generated, the refinement criterion you specified is used. If the maximum number of elements is reached, some elements may exceed the requested maximum element length.

You can also specify Initial Mesh Settings to apply to all objects; however, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

Assigning Length-Based Mesh Refinement Inside Objects

To instruct a solver to refine every face of an object and its interior:

1. Select the object you want to refine.
2. Click **HFSS** > **Mesh> Assign Mesh Operation** > **Inside Selection** > **Length-Based**, or right-click **Mesh** in the Project Manager, and select **Assign Mesh Operation** > **Inside Selection** > **Length-Based...**.

   The *Element Length-Based Refinement* dialog box appears.

3. Type a name for the mesh operation in the **Name** text box or accept the default name.
4. To restrict the length of the tetrahedral element edges inside the object:
   a. Select **Restrict Length of Elements**.
   b. Type the maximum length of the edges inside the object in the **Maximum Length of Elements** text box.

   The default value is set to 20% of the maximum edge lengths of the bounding boxes of each selected object's faces.

   The solver will refine the element edges inside the object until they are equal to or less than this value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

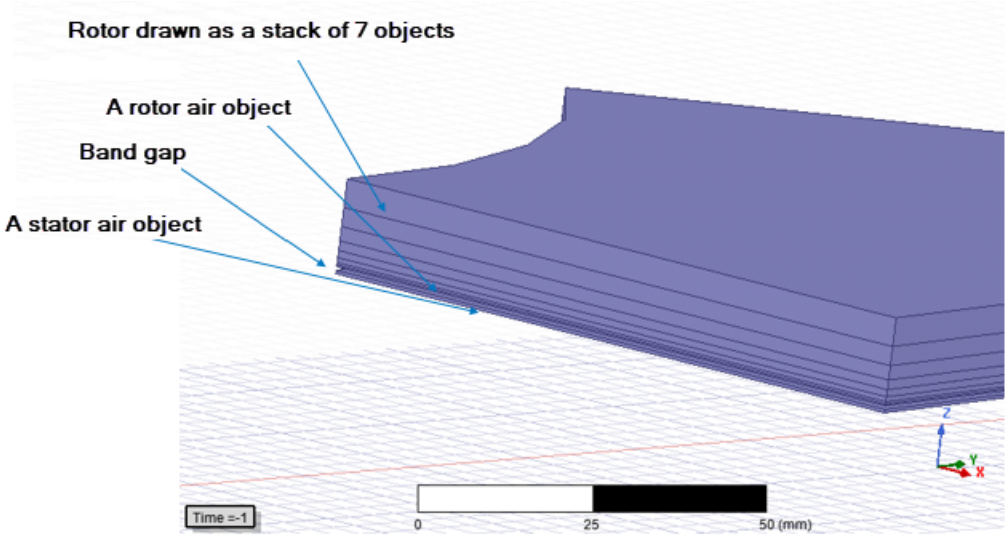
5. To restrict the number of elements added during the refinement inside the object:
    a. Select **Restrict the Number of Elements**.
    b. Enter the **Maximum Number of Elements** to be added.
    c. Click **OK**.

When the mesh is generated, the refinement criterion you specified will be used. If the maximum number of elements are reached, it may result in some elements exceeding the requested maximum element length.

You can also specify Initial Mesh Settings to apply to all objects; however, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

Assigning Skin Depth-Based Mesh Refinement on Object Faces

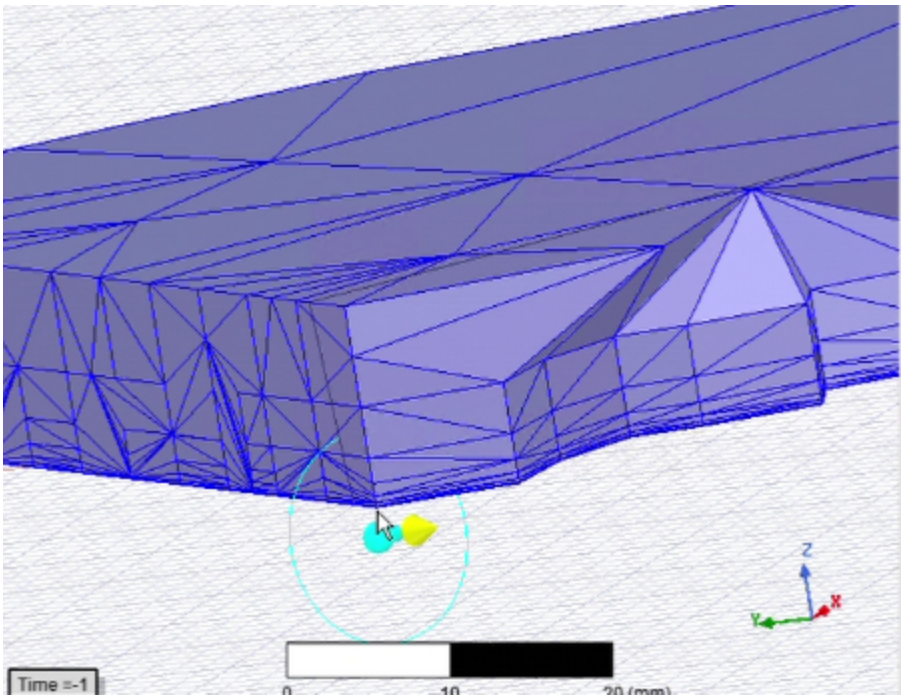
The Skin Depth-based mesh refinement lets you calculate or specify a skin-depth for mesh refinement. You can also specify the number of layers of elements for refinement where the skin depth is the total depth of all layers combined. These layers provide an easy, alternative approach than by creating physical models of each layer, pseudo-sheet bodies. Whereas creating and adjusting a complex, layered physical model is difficult, changing the skin depth parameters and number of layers is very easy. For example, consider a Maxwell rotor, drawn as a stack of seven objects:



To test various stacking models with pseudo bodies involves recreating the model with different thicknesses and remaking the entire initial mesh. The Tau mesher may not be available for all layer heights. You can achieve comparable results by using skin depth based refinement by reverting to the initial mesh, changing settings and applying mesh operations. The following

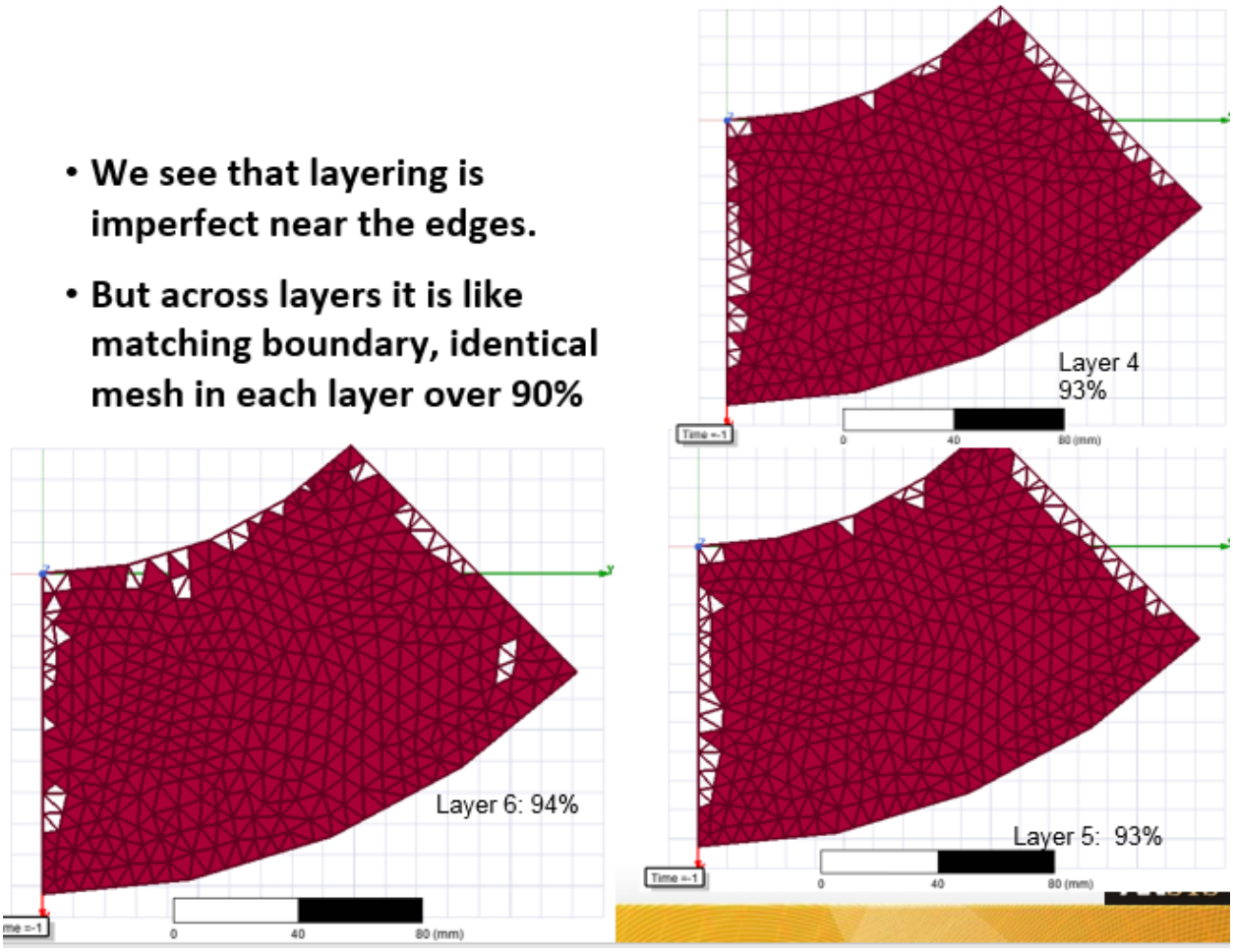
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

figure shows a cut-plane image of the interior of a model that uses skin-depth based refinement and layers of elements. The Skin Depth is the total depth of all layers combined. Layered elements apply to the selected faces of solid bodies. The elements are stretched parallel to face, and are compressed in the normal direction.



While in some places the edges have imperfect layering, the interior is very good. You can also view the results layer by layer by using the Model Analysis dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The layered skin depth mesh operation provides a tool toward finding a good solution. You can try different settings to improve results toward a good solution. For instance, for this example, these different settings were tried. For comparison, a model built with seven layers required 85K tets.

Tri length target mm	Total skin depth mm	Number of layers	Mesh Size element count	Perfect layering by tri count	Perfect layering by area
15.0	6.3	6	63K	43%	65%
10.0	6.3	6	68K	95%	94%
8.0	6.3	6	76K	96%	93%
6	6.3	6	97K	97%	94%

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The goal is not to have perfect layering, but to have a good solution that is much easier to achieve.

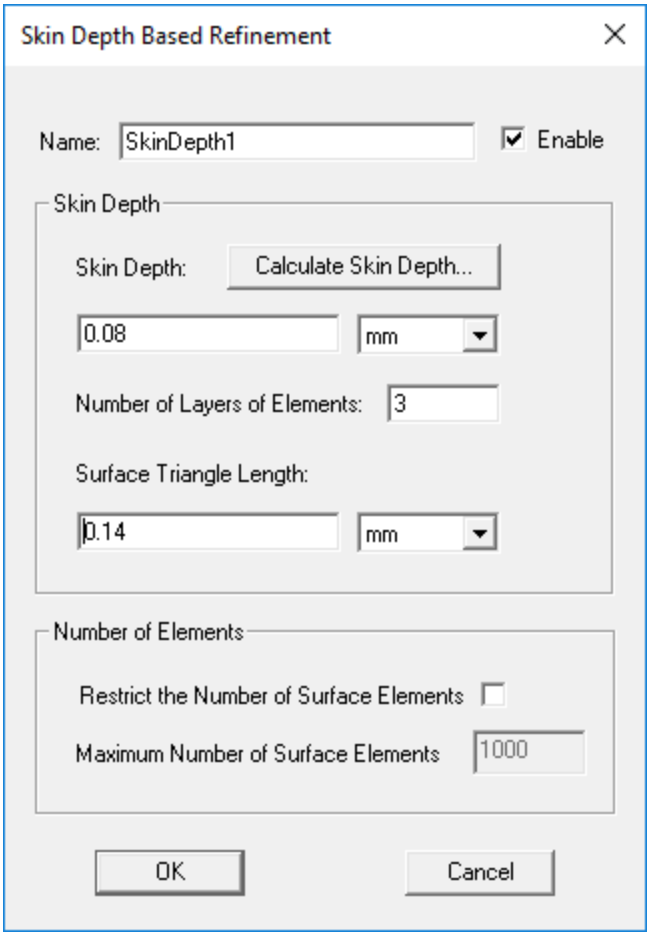
To use Skin Depth-Based Refinement:

1. Select the faces you want to be refined.

   > **Note:**
   >
   > It is possible to select a body and convert it to selecting all faces of the body. The user can use this method to select all faces and toggle a few faces out of selection. Selecting the whole body might select very large regions for refinement and increase the element count a lot.

2. Click **HFSS** > **Assign Mesh Operation** > **On Selection** > **Skin Depth Based**.

   The *Skin Depth Based Refinement* dialog box appears.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. To calculate the skin depth based on the object's material permeability and conductivity and the frequency at which the mesh will be refined, click **Calculate Skin Depth**.

   The **Calculate Skin Depth** dialog box appears with values based on the selected object's material properties and the solution setup frequency.

   Accept or edit these values. When you click **OK** the solver calculates the skin depth and enters its value in the Skin Depth text box. You can accept the calculated values or provide your own.

4. In the **Number of Layers of Elements** text box, type the number of layers to add perpendicular to the object's surface. The skin depth is the total depth of all layers combined.

   The solver will add an equivalent number of mesh points to each layer. For example, if HFSS added 10 points to satisfy the **Surface Triangle Length**, it will add 10 points to each layer.

5. Optionally, provide the maximum edge length of the surface mesh in the **Surface Triangle Length** text box. The default value is set to 20% of the maximum edge lengths of the bounding boxes of each selected face.

   The solver will refine the surface triangle mesh (the faces of the tetrahedra touching the surface) until their edge lengths are less than or equal to the specified value.

6. By default, the Restrict Number of Surface Elements setting is unchecked. This allows the mesher to use symmetry more effectively. If you Restrict the number of surface elements, this may affect symmetry but can be used to protect against runaway refinement in specific cases. To restrict the number of elements added during refinement on the faces uncheck the box to enable the field for Maximum Number of Surface Elements. With the box unchecked, the field, unavailable, the number in the box provides an estimate of mesh growth.

When the mesh is generated, the refinement criteria you specified will be used. This operation will be approximately the same as having slabs of tetrahedra, but it is not guaranteed to prevent tetrahedra from crossing slab interfaces. Caution should be used with this mesh operation, as very thin layers may cause a reduction in mesh quality or unnecessarily cause the generation of a very large mesh. Further regions refined under this operation and its close neighbors do not participate in solution adaptive refinement. This is another reason to use this seeding operation with caution.

You can also specify Initial Mesh Settings to apply to all objects; however, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

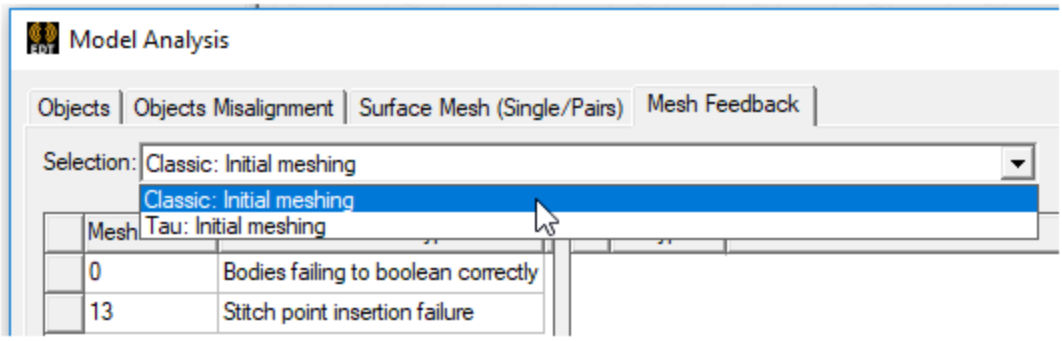
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Visualization for Skin Depth Refinement Mesh Results

You can visualize the mesh to assess results either by using the clip-plane feature on a model with a mesh plot, or by using the Model Analysis dialog box as follows:

1. Select the object on which you have generated a meshplot, and click **Modeler** > **Model Analysis** > **Show Analysis Dialog** > **View Mesh Feedback**.

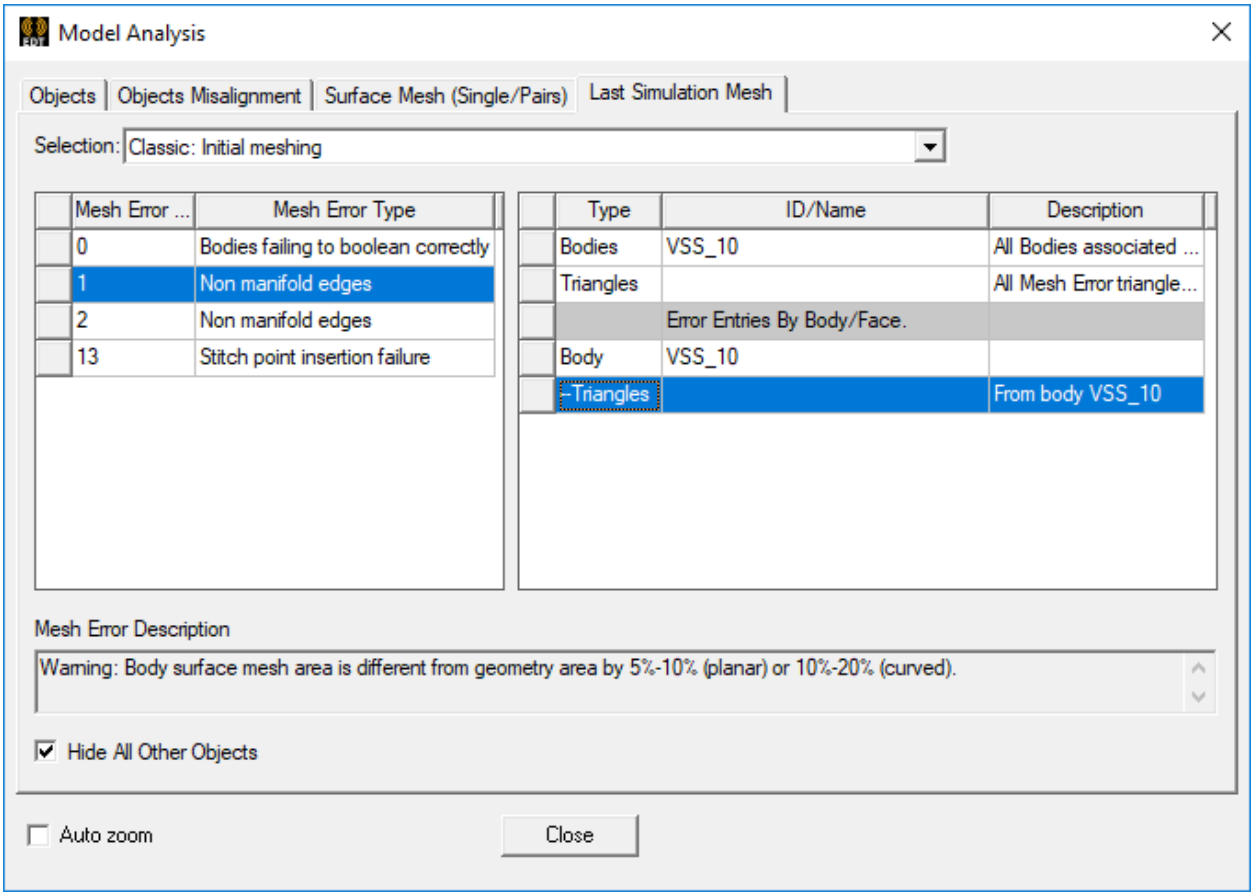
   The **Model Analysis** dialog box appears. If the project has solutions by different solvers, the **Selection** drop-down menu lets you select which mesh to view.



2. On the upper left table, under Mesh Error, select 0 through n , where 0 is the selected body, 1 is the layer closest to the face, 2 is the next, up to layer "n" as farthest.

   Selecting one of these rows causes a **Mesh Error Description** to display for the selected layer or body. The message gives the Total Expected count for triangles, as compared with the Success percentage by count and by area.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. On the upper right table, select the body, face, or triangle desired, scrolling the list to find any object, face, triangle, or area of interest.

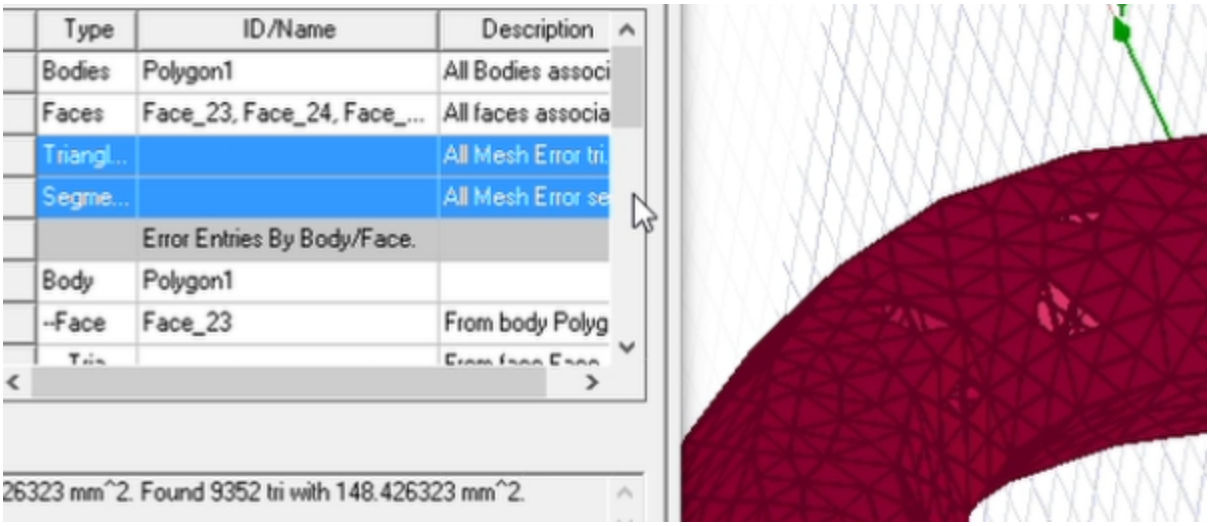
   Click Face, to see the selected face.

   Select Triangles to see the actual triangles created.

   Select Segment to show the triangles attempted.

   The following figure shows Triangle and Segment both selected (using Shift+click). Visually, the unfilled triangles represent places where the triangle is not on the desired layer. However the mesh is still very good and supports an accurate solution.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Check boxes allow you to **Hide All Other Objects** checked by default, and **Auto zoom to selection**, unchecked by default.

Modifying Surface Approximation Settings

Each solver applies surface approximation settings for selected objects when it generates the initial mesh. If you modify a solver's default settings after the initial mesh has been generated, they will not affect the mesh for that design variation. Surface approximation makes sense for curved surfaces, for which the mesh will not exactly reproduce the surface shape. It also can be used to restrict the aspect ratio of triangles on planar surfaces. For a given surface approximation slider setting, the approximation is more refined for IE and SBR+ solvers than for FEM solvers.

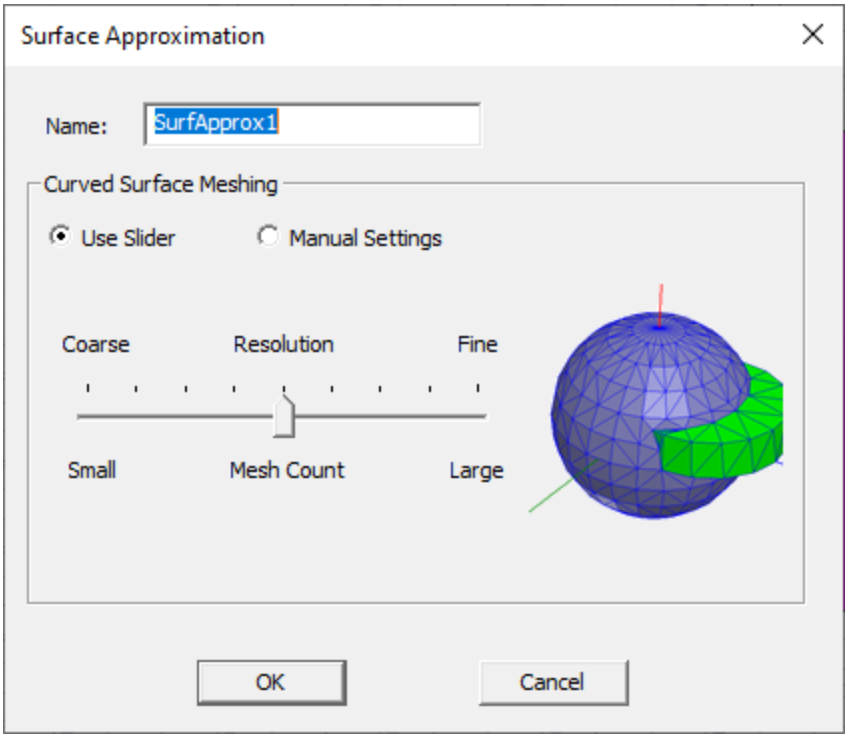
1. Select the faces for which you want to modify the surface approximation settings.

   Alternatively, select an object if you want to modify the surface approximation settings of every face on the object.

2. Click **HFSS** > **Mesh** > **Assign Mesh Operation** > **Surface Approximation** or right click on **Mesh** in the Project Manager and select **Assign Mesh Operation** > **Surface Approximation**.

   The *Surface Approximation* dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

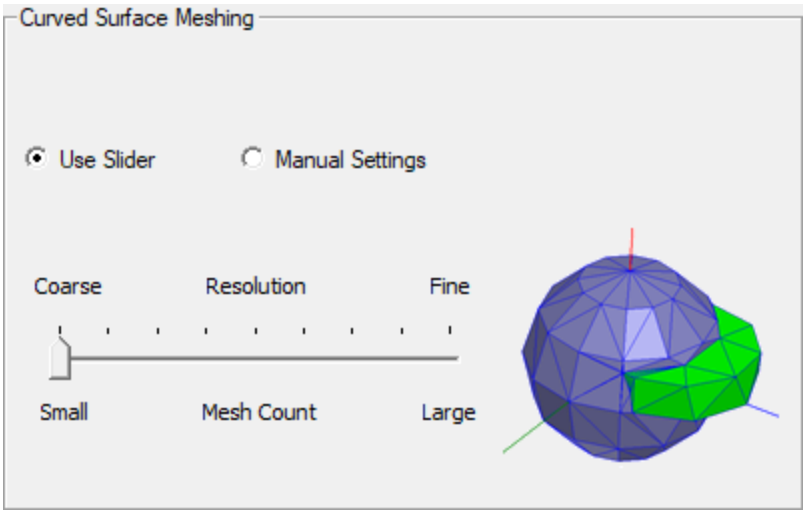


3. Type a name for the group of settings in the **Name** text box or accept the default name.
4. Under **Curved Surface Meshing**, you can select **Use Slider** or specify **Manual Settings**.

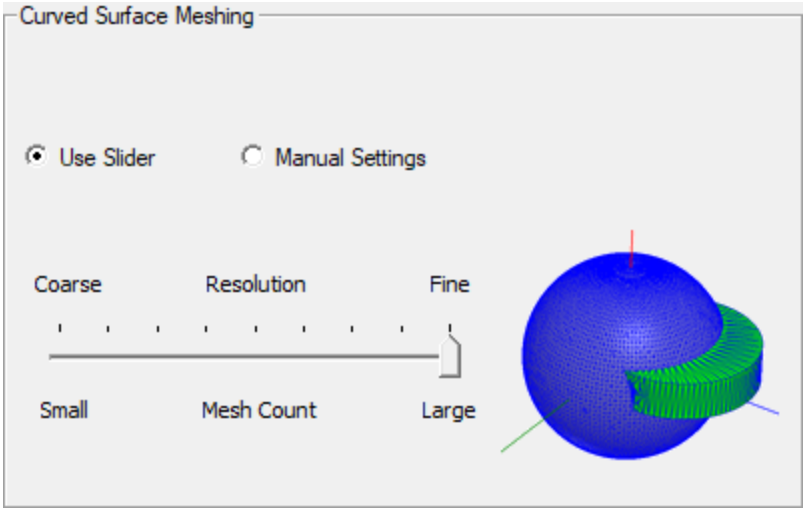
   The slider includes a visual representation of your choice, ranging from a ***Coarse*** *Resolution* (with a ***Small*** *Mesh Count*) through a nine position scale to a ***Fine*** *Resolution* (with a ***Large*** *Mesh Count*). Here, "*Mesh Count*" refers to the number of elements comprising the mesh.

   Coarse Resolution, Small Mesh Count:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

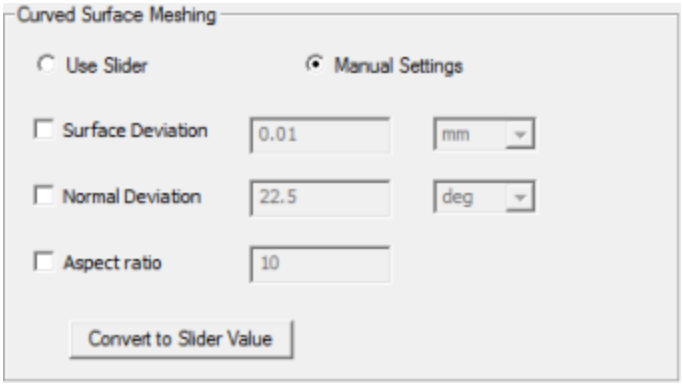


Fine Resolution, Large Mesh Count:



5. If you choose **Manual Settings**, the dialog box changes to show text fields.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> If you selected **Use dynamic surface resolution** in Initial Mesh Settings, use of **Manual Settings** is not permitted. You must use slider bar to specify one of three levels of surface representation: coarse (1-3), normal (4-6), and fine (7-9).

6. Select **Surface Deviation**. Use the drop-down menu to select a unit of measure and then enter the distance between the true surfaces of the selected faces and the meshed faces.
7. Select **Normal Deviation**. Use the drop-down menu to select a unit of measure and then enter the angular distance between the normal of the true surface and the corresponding mesh surface.
8. Select **Aspect Ratio** and then type a value in the text box. This value determines the shape of the triangles. The higher the value, the thinner the triangles. Values close to 1 will result in well-formed, wide triangles.
9. Click **OK**.

   The settings are applied to the initial mesh generated on the selected surface. The group of settings is listed in the project tree under **Mesh**.

   You can also specify Initial Mesh Settings to apply to all objects. However, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Selecting **Use dynamic surface resolution** in the Initial Mesh Settings specifies the best-practice mesh operations over the geometric models. The default mesh operations or user-defined mesh operations may or may not be replaced by optimized mesh operations with model analysis. In general, large curve faces, curve faces with small gaps, or skewed cables get smoother curvature representation while small curve faces, such as fillets and small curve objects, get relatively coarse triangulations to reduce overall mesh size.

This dynamic surface resolution mesh operation supports 3D volume mesh and surface mesh in all products.

Important:

The consolidation of surfaces into a conformal mesh is skipped for dynamic and tolerant meshing, including for light weight geometries. This can lead to overlapping surfaces in SBR+ simulations. The user should carefully avoid overlapping surfaces or objects as SBR+ can produce unexpected results.

Applying Curvilinear Elements

For **HFSS**, **Maxwell**, and **Mechanical** designs, Curvilinear elements provide an accurate representation of the CAD model geometry by improving the conformity of element nodes to curved CAD surfaces. You can apply curvilinear elements as a Mesh Operation on selected curved model faces see procedure below or as a global option in the Initial Mesh Settings dialog box.

For **HFSS** designs:

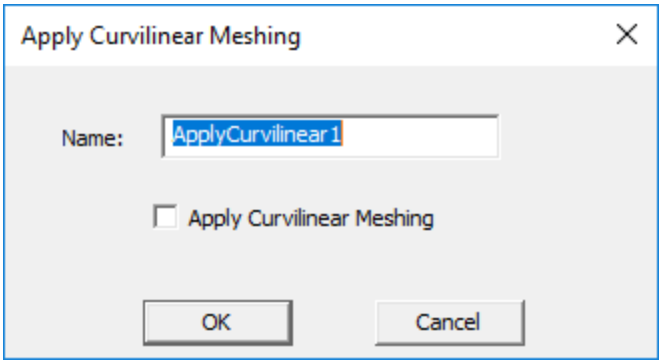
When curvilinear elements are applied globally or to specific faces, the meshing process attempts to pull all nodes associated with a curved face mesh (both corner nodes and midside nodes) to the CAD faces. Ten-node tetrahedral elements are employed, with four corner nodes and six midside nodes. Therefore, with the midside nodes pulled to curved CAD faces, the element edges can be non-straight. See Apply Curvilinear Elements Discussion.

How to Apply Curvilinear Elements to Specific Curved Faces:

1. Select the object or face to which you want to apply curvilinear elements.
2. Select **Mesh** in the Project Manager, right-click for the short cut menu and select **Assign Mesh Operation** > **Apply Curvilinear Meshing**.

   The **Apply Curvilinear Meshing** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Either accept the default name or provide one of your own.
4. Select **Apply Curvilinear Meshing**.
5. Click **OK**.

The defined Mesh Operation appears in the Project Manager.

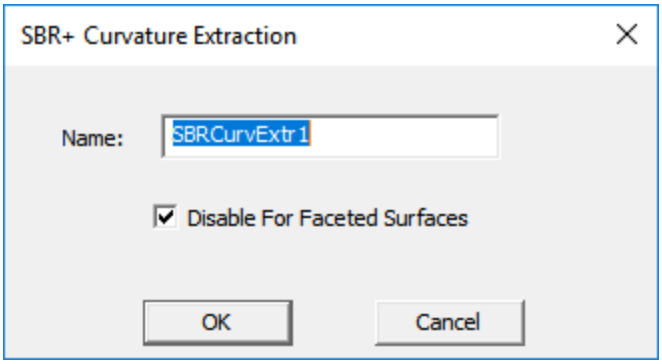
SBR+ Curvature Extraction for Faceted Surfaces

In SBR+, the ray-tube divergence correction and deployment of creeping rays depend on the surface curvature of the geometries. However, faceted CAD models such as STL lack intrinsic curvature information. With these models, a curvature extraction algorithm is used wherever possible to interpolate smooth curved surfaces based on the orientations of neighboring facets. Curvature extraction is enabled by default, and users may disable it on selected faceted faces or objects by using this mesh operation. When disabled, the affected surfaces are assumed to be perfectly flat. This mesh operation can only be applied to faceted or flat surfaces and is ignored on curved surfaces.

1. Select the faceted object or face for which you want to disable SBR+ Curvature Extraction for Faceted Surfaces.
2. Select **HFSS** > **Mesh** > **Assign Mesh Operation** > **SBR+ Curvature Extraction for Faceted Surfaces**, or right-click in the Project Tree and select **Assign Mesh Operation** > **SBR+ Curvature Extraction for Faceted Surfaces**.

   This displays the **SBR+ Curvature Extraction** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. You can accept the default name or provide one of your own.
4. By default, the box is checked to disable curvature extraction on the selected surface. Click OK to apply a mesh operation.
5. Creeping waves will not propagate over faceted surfaces that have curvature extraction disabled.

   The defined Mesh Operation appears in the **Project** tree. Deleting this Mesh operation will invalidate a solution. If applied to a curved surface, it will be ignored.

Specifying Surface Priority for TAU Mesher

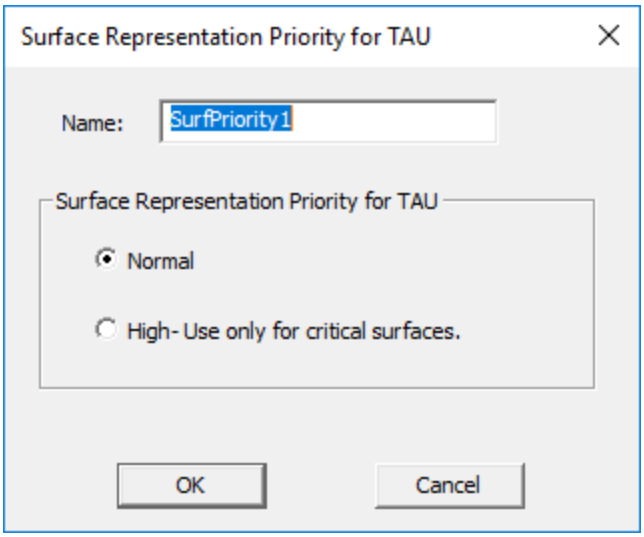
You can choose to specify the surface representation priority for the TAU mesher. For most designs, you can let the solver automatically choose which of two meshing approaches to take. The solver predicts which one gives the best results, balancing mesh reliability, speed, quality, size and design characteristics. In most cases, the solver uses TAU mesh, rather than the classic mesh. In general, it looks for specific features (for example, stacks of large planar parallel facets with small gaps) and situations where the initial TAU mesh is 4 times larger than the Classic. Experienced users may choose to designate a high surface representation priority for the critical surfaces with the TAU mesh is used.

> **Important:**
>
> Use this mesh operation with caution. Use this option only if the model has tiny elements or critical boundary conditions since the solver performs several additional strict mesh verification steps on the selected objects.

1. Select the surface of interest.
2. If desired, specify a Name for the Mesh operation.
3. Right-click the **Mesh** icon in the Project tree to view the shortcut menu and select **Assign Mesh Operation** > **Surface Priority for Tau**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Surface Representation Priority for Tau** dialog box appears.



4. Select **High** for the critical surface.
5. Click **OK**.

   The Mesh operation appears in the Project Manager.

Specifying the Model Resolution

The model resolution parameter is used by the meshmaker to distinguish large features from small features in the model. This setting controls how large a feature must be to be resolved by the meshmaker. For example, if you set the model resolution length to 20 mm, any model features smaller than 20 mm are not represented in the mesh. Neither the model nor the model files are changed. The resolution only controls how the mesh for the model is represented.

While removing small details, if the meshmaker finds that the representation of the model is not accurate enough, it returns an error condition. The meshmaker then starts with the most accurate representation and prunes away the details smaller than the model resolution length. It returns an error if the specified model resolution length forces the final representation to deviate too greatly from the model. You can set Model Resolution on one or more objects to remove unnecessary details from the mesh representation. This can be used to reduce the mesh complexity of the selected objects.

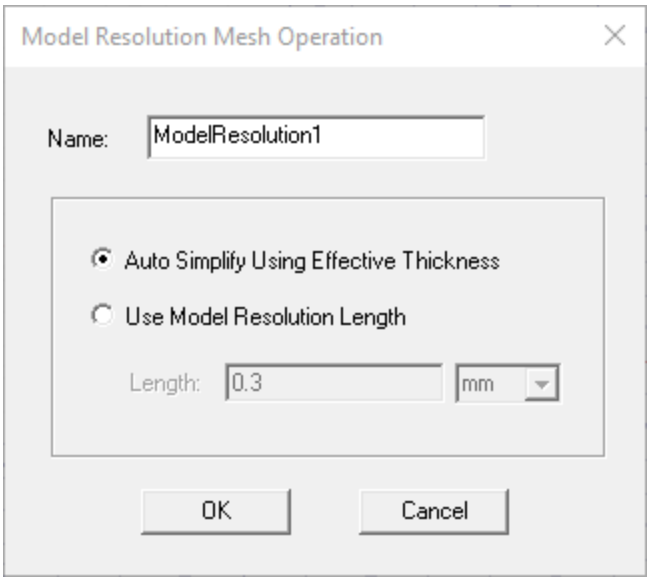
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

A cautionary note is needed concerning the use of model resolution. It can sometimes make model faces fail to be represented in the mesh, which can effectively remove some of your boundary conditions. This can only happen if faces are closer together than the model resolution distance you specify.

1. Select the object or objects on which to specify a Model Resolution operation.
2. Click on **HFSS** > **Mesh** > **Assign Mesh Operation** > **Model Resolution** or right click on the **Mesh** icon in the **Project** tree and select **Assign Mesh Operation** > **Model Resolution**.

   The **Model Resolution Mesh Operation** dialog box displays.



   Alternatively, you can display the same dialog box if you:

   a. Right-click either **Mesh** in the Project Tree, or right-click in the **3D Modeler** window to display the respective shortcut menu.
   b. Click on **Assign > Model Resolution** in the **Project Tree** menu or click on **Assign Mesh Operation > Model Resolution** on the shortcut menu.

   The **Model Resolution Mesh Operation** dialog box contains text fields for the mesh operation *Name* and radio buttons with choices for the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

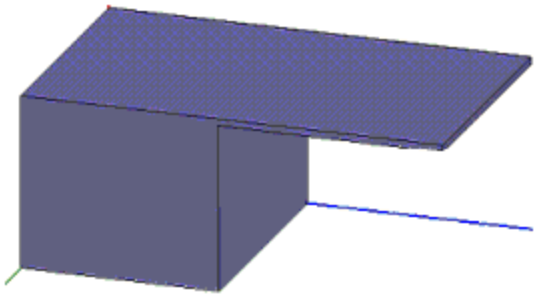
- **Auto Simplify Using Effective Thickness**

  The mesher calculates the resolution length based on each object's effective thickness. One mesh operation can be assigned to many objects, and each will be simplified based on its own dimensions. Use the **Auto Simplify** selection:

- To remove many details while retaining an object's overall shape and size.
- For objects of generally uniform thickness.
- To assign one mesh operation to many objects.
- **Use Model Resolution length**

This enables fields for you to specify the resolution value and units. Use this selection for:

- Tighter control of mesh accuracy.
- Objects of non-uniform thickness. For example, the thin section of the object shown below might be lost with **Auto Simplify**:



3. After defining the operation, click **OK**.

   This adds the named Model Resolution operation under the **Mesh** icon in the Project Manager.

> **Note:**
>
> Setting Model Resolution will invalidate any existing solutions.
>
> When two objects in contact have different model resolution lengths, the smaller length will apply for the common regions.

Reverting to the Initial Mesh

The initial mesh is the mesh that is generated the first time a design variation is solved. It includes surface approximation settings, but does not include lambda refinement or defined mesh operations.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you have modified the design setup, and do not want to use the existing current mesh, revert to the initial mesh prior to solving.

- On the **HFSS** > **Analysis Setup**> **Revert to Initial Mesh**.

Reverting to the initial mesh is useful when you want to evaluate how a different solution frequency affects the mesh generated during an adaptive analysis. You lose all solution data for a solve setup and all of its sweeps when you revert to the initial mesh for that setup. You can do this for all solve setups at once by selecting the command through Analysis in the menu system or project tree, or for a specific solve setup via its right mouse click menu in the project tree.

Generating Mesh without Solving

If you want to refine the mesh on a face or volume but do not want to generate a solution, define mesh operations. Then do one of the following:

- Select [**HFSS**] > **Analysis Setup** > **Generate Mesh**.
- Right-click the **Analysis** or **Setup** icon in the **Project** window and select **Generate Mesh**.

The same solve machine rules that apply to solving any other setup also apply here. The mesh operation will be sent to the default solve machine. If you specified "Prompt for analysis machine when launching analysis" under **Tools** > **Options** > **General Options** > **Analysis Options**, the **Server Setup** dialog box appears and you can specify a solve machine.

- If a current mesh has been generated, the solver will refine it using defined mesh operations.
- If a current mesh has not been generated, the solver will apply the mesh operations to the initial mesh.
- If an initial mesh has not been generated, the solver will generate it and apply the mesh operations to the initial mesh.
- If the defined mesh operations have been applied to the selected face or object, the current mesh will not be altered.

> **Tip:**
>
> Define a new mesh operation rather than modify an existing mesh operation. The solver will not re-apply a modified mesh operation.

Applying mesh operations without solving enables you to experiment with mesh refinement in specific problem regions without losing design solutions. You cannot undo the applied mesh operations, but you can discard them by closing the project without saving them or by reverting to the initial mesh.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specifying Initial Mesh Settings

You can specify the initial mesh settings, including the surface approximation and the meshing approach. Initial Mesh Settings apply to all objects; however, if you apply separate surface approximation mesh operations to specific objects, the object settings take precedence over the general setting.

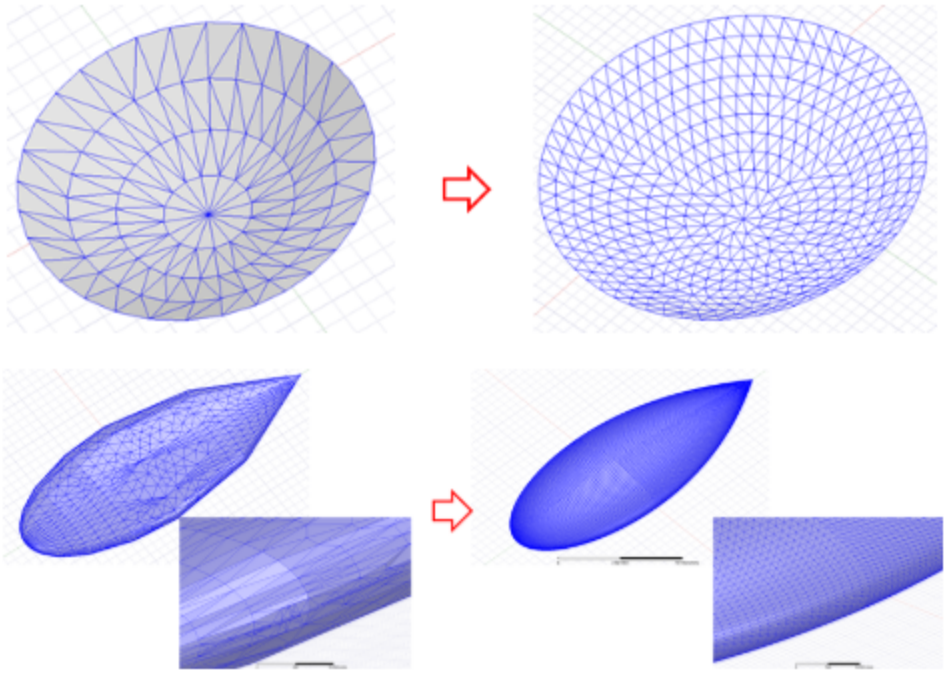
> **Note:**
>
> When you have defined mesh linking between a source and target design, all parameters within the *Initial Mesh Settings* dialog box are ignored for the target design. The mesh is copied from the source design without modification.

For most designs, you can let the solver automatically choose which of two meshing approaches to take. Each solver predicts which one gives the best results, balancing mesh reliability, speed, quality, size and design characteristics.

In most cases, HFSS uses TAU mesh, rather than the classic mesh. In general, it looks for specific features (for example, stacks of large planar parallel facets with small gaps) and situations where the initial Tau mesh is 4 times larger than the Classic.

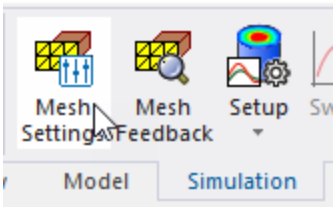
If the model contains IE Regions, the TAU mesher uses a surface mesh routine to detect which curve faces should be remeshed, rather than using the faceting triangles. This produces a high quality mesh, uniform, including the top, sides and bottom. (See the following figure for examples of the benefit.) Other faces that cannot be remeshed still get faceting triangles from the modeling kernel. In the final output, the remeshed surface mesh and faceting surface mesh are coupled to ensure the mesh is watertight and conformal.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



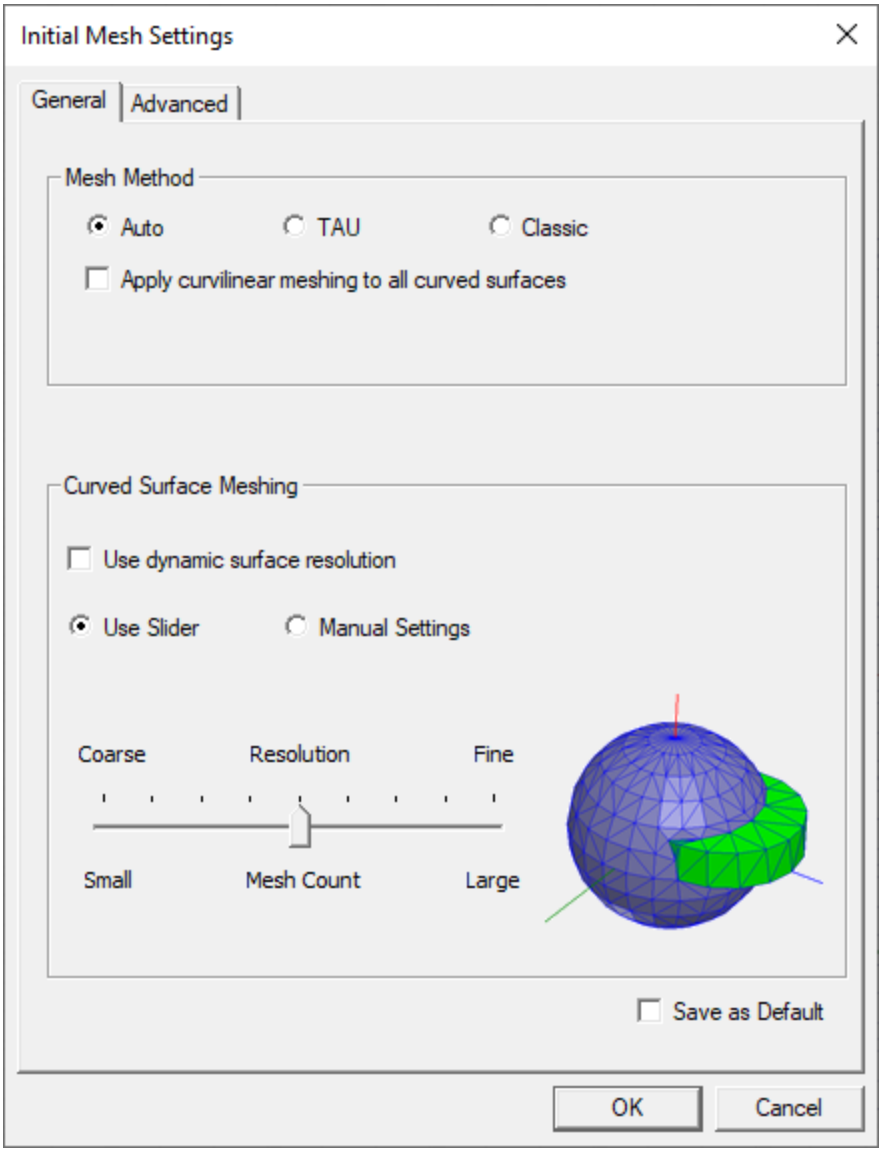
In a few cases, you may decide to override the automatic choice and designate the mesher to use. To do so:

1. Access the **Initial Mesh Settings** one of two ways:
   - Select **HFSS** > **Mesh** > **Initial Mesh Settings...**
   - In the Project Manager, right-click **Mesh**, and select **Initial Mesh Settings** from the shortcut menu.
   - On the **Simulation** tab of the ribbon, select **Mesh Settings**.



The **Initial Mesh Settings** window appears, with the **General** tab selected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



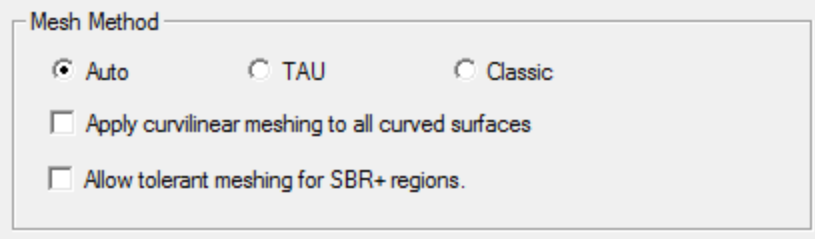
2. The **General** tab contains **Mesh Method** options for:
   - **Auto** – the solver automatically selects the mesher. This is the default setting. In most cases, HFSS uses TAU and HFSS-IE uses Classic mesh.
   - **TAU** – only specific curve faces will be remeshed (for example, equation-based axisymmetric faces). If a curve face connects to non-remeshed curve faces, the curve face will not be remeshed. If a curve face cannot be remeshed, faceting triangles will be used as the surface mesh. Mesh quality depends on how the faceting triangles are made. If the surface mesh is generated by TAU, it shows in the profile as Mesh TAU (Surface). This can fall back to Mesh TAU (Wrapper), which can fall back to the classic Mesh (Stitch).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Classic** – Classic mesh uses a Bowyer algorithm to create compact meshes for the model with very large length scales. It represents the model very accurately. Depending on the model, Classic mesh may converge using fewer resources whereas TAU or Phi Plus may converge faster.

3. For HFSS models with curved surfaces, you can select or deselect **Apply curvilinear meshing to all curved surfaces**. This increases accuracy, but costs more memory. See Rectilinear Elements and Curvilinear Elements and Modifying SAR Settings.

   If the design contains an SBR+ Region, the **Initial Mesh Settings** window includes another option to **Allow tolerant meshing for SBR+ regions**. This is because the SBR+ ray-tracing solver has relaxed surface meshing requirements relative to other HFSS solvers. If an imported scattering geometry for the SBR+ region contains anomalies that make it too time-consuming or impossible to use Auto (default) meshing, the tolerant meshing option should be tried, as it will often yield a successful mesh in cases where Auto won't. On the other hand, if the imported geometry is clean and Auto meshing works well, tolerant meshing can sometimes result in an excessive number of surface mesh triangles that increase SBR+ solution time. The best meshing option for SBR+ is very dependent on the quality of the input geometry, so users need to consider both approaches and choose the one that works best for a particular scattering geometry.



   Dynamic Meshing eliminates mesh file generation during SBR+ simulations, and it makes the simulations faster. Importing a lightweight geometry into an SBR+ design disables editing for the **Use Dynamic Meshing** option for the Initial Mesh Settings, and Use Dynamic Meshing is always On. If the design does not include Lightweight geometries, you may select Dynamic Meshing which skips generating mesh file during simulations. Dynamic meshing either uses the geometry triangulation in the case of the light weight geometry, as the mesh for the SBR+ solver.

> **Important:**
>
> The consolidation of surfaces into a conformal mesh is skipped for dynamic and tolerant meshing, including for light weight geometries. This can lead to overlapping surfaces in SBR+ simulations. The user should carefully avoid overlapping surfaces or objects as SBR+ can produce unexpected results.

4. Under **Curved Surface Meshing**, you can choose to **Use dynamic surface resolution**, select **Use Slider** or specify **Manual Settings**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

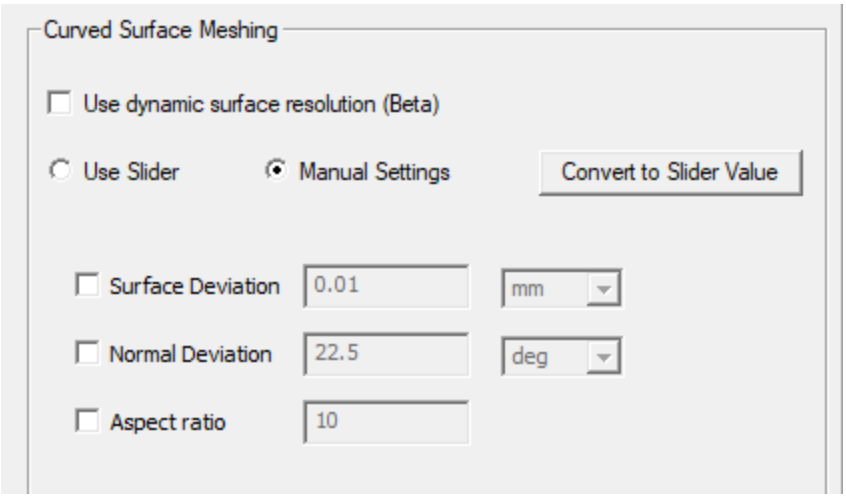
If you select **Use dynamic surface resolution**, the **Manual Settings** option is disabled.

The slider includes a visual representation of your choice, ranging from a ***Coarse*** *Resolution* (with a ***Small*** *Mesh Count*) through a nine position scale to a ***Fine*** *Resolution* (with a ***Large*** *Mesh Count*. Here, "*Mesh Count*" refers to the number of elements comprising the mesh.

Selecting **Use dynamic surface resolution** in the *Curved Surface Meshing* section specifies the best-practice mesh operations over the geometric models. This mesh operation will support 3D volume mesh and surface mesh in all products. The default mesh operations or user-defined mesh operations may or may not be replaced by the optimized mesh operations with model analysis. In general, large curve faces, curve faces with small gaps, or skewed cables get more smooth curvature representation while small curve faces, such as fillets, small curve objects, and so forth, get relatively coarse triangulations so that the overall mesh count will be reduced.

If you select dynamic surface resolution, you can optionally use the slider bar to adjust surface representation levels as: coarse (1-3), normal (4-6), and fine (7-9). You can also adjust the surface representation level for a specific group of faces by specifying slider bar, as described in Modifying Surface Approximation Settings.

5. If you choose **Manual Settings**, the window changes to show text fields. See: Modifying Surface Approximation Settings.



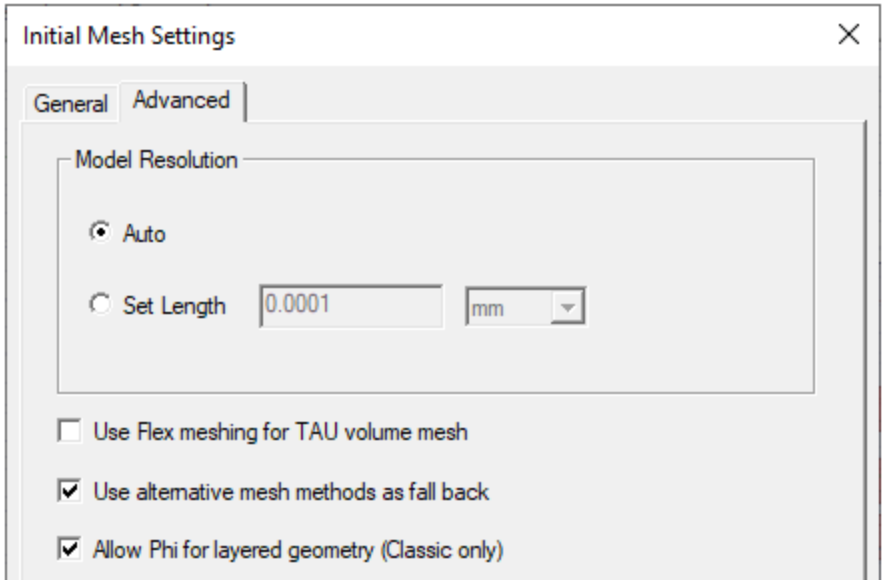
Use the check boxes to enable the fields and specify values:

- **Surface Deviation** – the distance between the true surfaces of the selected faces and the meshed faces.
- **Normal Deviation** – the angular distance between the normal of the true surface and the corresponding mesh surface.
- **Aspect Ratio** – determines the shape of the triangles. The higher the value, the thinner the triangles. Values close to 1 will result in well-formed, wide triangles.

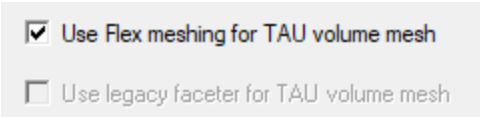
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Clicking **Convert to Slider Value** converts the manually entered values to an equivalent slider setting, and returns the panel to the slider view.

6. To make your choices the default, use the **Save as default** check box.
7. The **Advanced** tab lets you optionally specify a **Set Length** for **Model Resolution**. This is for experienced users who have a good understanding of how particular values will affect their models. In general, the **Auto** setting provides good results.



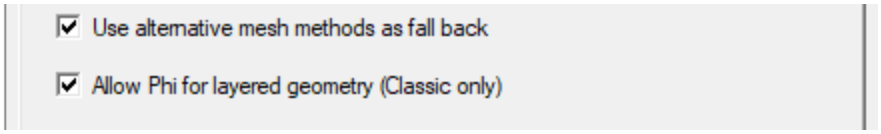
8. **Use Flex meshing for TAU volume mesh** enables a version of the TAU mesher that will rarely fail to generate a mesh. In most cases, the TAU Flex mesh is as accurate as traditional TAU or Classic meshes. However, for some complex models with bad translation or poorly defined surfaces that would fail to produce a strict mesh on all objects, relaxed tolerances will be applied. In these cases the user should review the mesh to evaluate whether it is acceptable for simulation. See TAU Flex meshing for more details.



9. **Use legacy faceter for TAU volume mesh** – By default, TAU uses the latest faceter to generate geometric model for meshing. Occasionally, the mesher may fail, and TAU will automatically fall back to use the legacy faceter to make a second attempt. In such cases, directly using the legacy faceter can get the mesh in the first attempt so total meshing time will be saved. The legacy faceter is not actively maintained by Spatial and will be retired soon. This option should be used only as a last resort.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. If your selected mesh method (Classic/TAU) failed to generate initial mesh, the solver can either stop or fall back on alternative mesh methods if you select **Use alternative methods as fall back**. You can also select **Phi for layered geometry (Classic only)**. For example, specifically, the mesher might go like this: When mesh method is TAU: TAU (>Phi)>Classic. When mesh method is Classic: Classic(>Phi)>TAU. Note: the options do not affect the Auto mesh setups.



11. Click **OK** to apply your choices.

The settings will be applied to the initial mesh generated.

TAU Flex Meshing

TAU Flex is designed for guaranteed mesh success, and may also reduce meshing time. It provides a mesh even for complex and “dirty” geometry model without any preliminary geometric healing or repair.

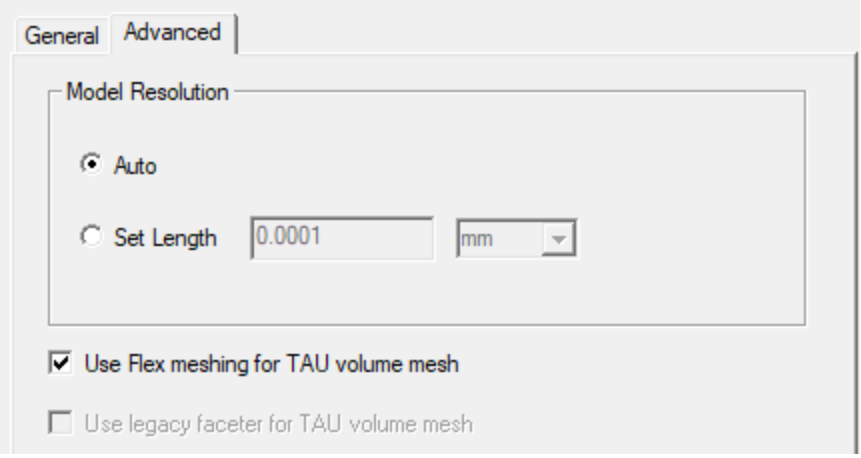
TAU Flex is a multi-domain, priority mesher which provides more efficient and reliable meshing. The priority mesh generation is driven by solver requirements for materials (metal/non-metal), boundary conditions (ports, radiation,...), conduction paths, and so forth. The constraint-tolerant meshing method gives the TAU Flex mesher flexibility to excuse mesh difficulties in lower-priority regions so that the resulting mesh can still provide acceptable simulation results.

> **Important:**
>
> The consolidation of surfaces into a conformal mesh is skipped for dynamic and tolerant meshing, including for light weight geometries. This can lead to overlapping surfaces in SBR+ simulations. The user should carefully avoid overlapping surfaces or objects as SBR+ can produce unexpected results.

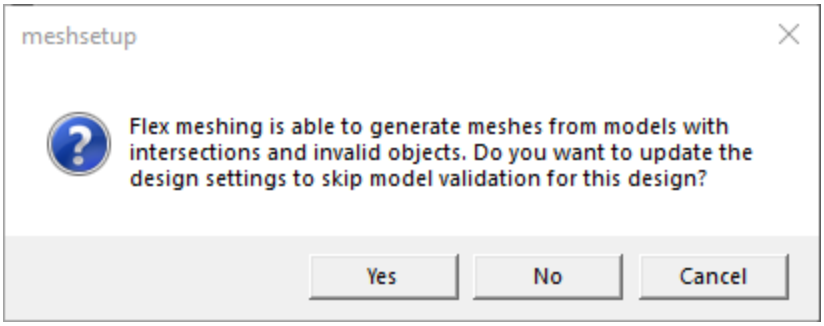
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Steps for Using TAU Flex

1. Navigate to the **Initial Mesh Settings** window.
2. Click the **Advanced** tab and select **Use Flex meshing for TAU volume mesh**.

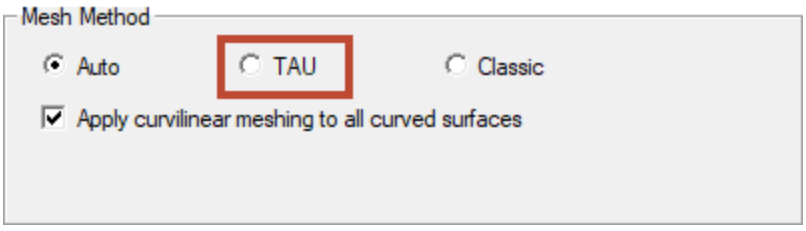


Selecting this option opens a dialog box. Select **Yes** to continue.



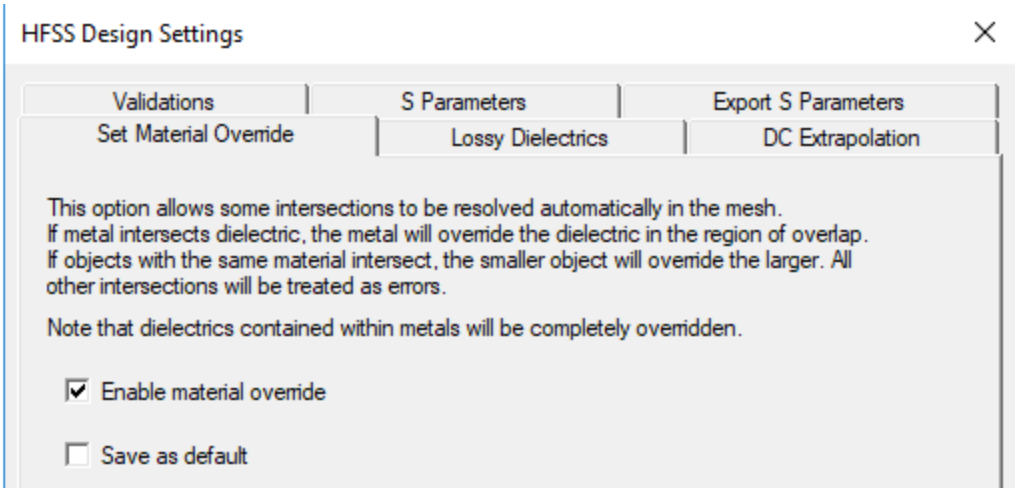
To get the full benefits of TAU Flex, especially for complex or "dirty" models, follow these recommended steps:

1. On the **General** tab of the **Initial Mesh Settings** window, select **TAU**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

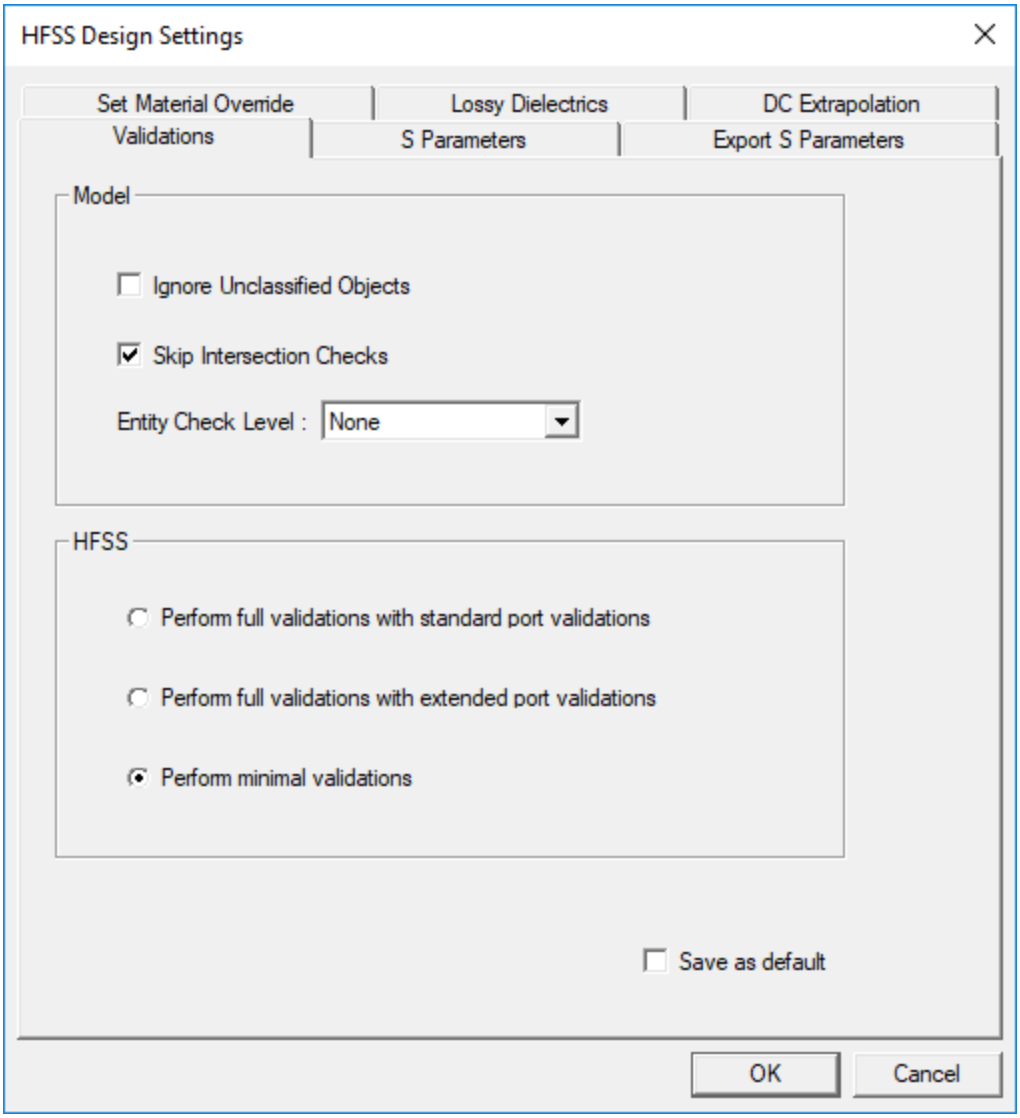
2. Click **HFSS** > **Design Settings** to open the **Design Settings** window.
3. On the Set Material Override tab, select **Enable material override**.



This activates TAU's priority meshing method based on metal/non-metal materials.

4. Still in the Design Settings window, click the **Validations** tab and select **Perform minimal validations**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

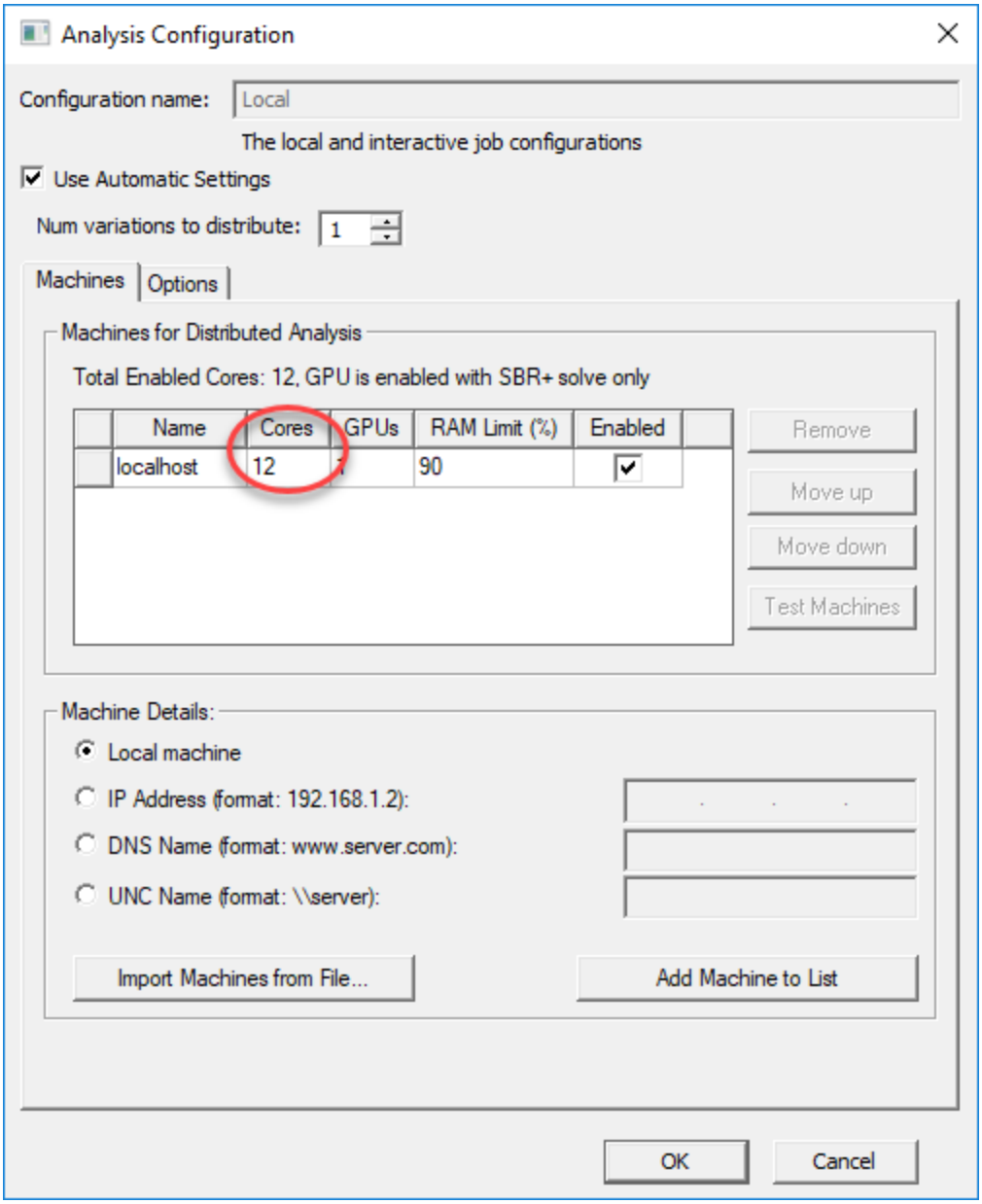


This reduces preprocessing time for complex models, and allows simulation of "dirty" geometry for which TAU Flex can provide a mesh.

5. Because TAU Flex is a parallel mesher for many processes, you should specify multiple cores in **HPC and Analysis Options** to speed up the meshing process. Access the Analysis Configuration window one of two ways:
   - Select the Simulation tab on the ribbon and click **Analysis Config**.
   - Use **Tools** > **Options** > **HPC and Analysis Options** or select the **Simulation** tab on the ribbon and click **HPC Options**. Then double-click a configuration.

   The **Analysis Configuration** window appears.

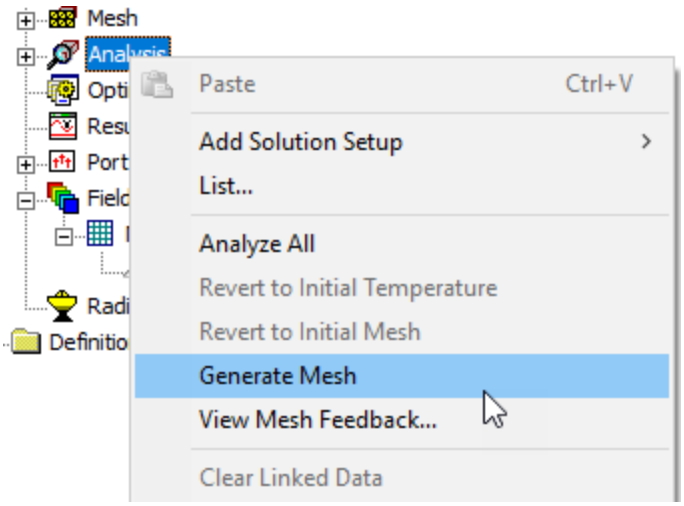
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Specify the number of cores to use.

6. In most cases, TAU Flex can provide mesh as accurate as TAU or Classic, but not always. Therefore, we recommend that you **Generate Mesh** without solving, and that you review the mesh before proceeding the simulation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



TAU Flex Warning Messages

In cases where TAU Flex has difficulty making a strict mesh on some objects, mesh warnings are given for those objects. You should review the mesh on those objects before proceeding to the simulation. You can exercise discretion in deciding whether a given mesh issue needs to be addressed (for example, critical antenna geometry) or can be ignored (for example, a small "leak" in a dielectric substrate far from the source).

There are three types of warnings:

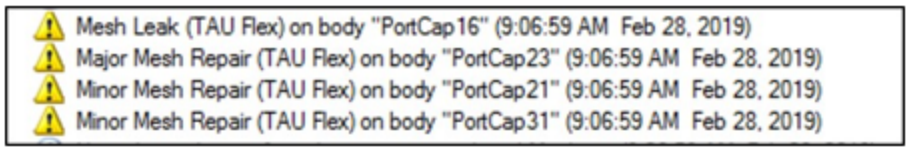
- **"Mesh Leak"** – *The most severe warning*. A body does not have any mesh element, or it has mesh elements with a significant mismatch of surface area between the mesh and geometry – larger than 10% of planar area, or 20% of curved area.
- **"Major Mesh Repair"** – A body has major surface area mismatch between the mesh and geometry – larger than 5% of planar area, or 10% of curved area.
- **"Minor Mesh Repair"** – *The least severe warning*. A body has minor surface area mismatch between the mesh and geometry – less than 5% for planar area, or 10% for curved area.

Reviewing TAU Flex Mesh Warnings

TAU offers three ways to view mesh warnings.

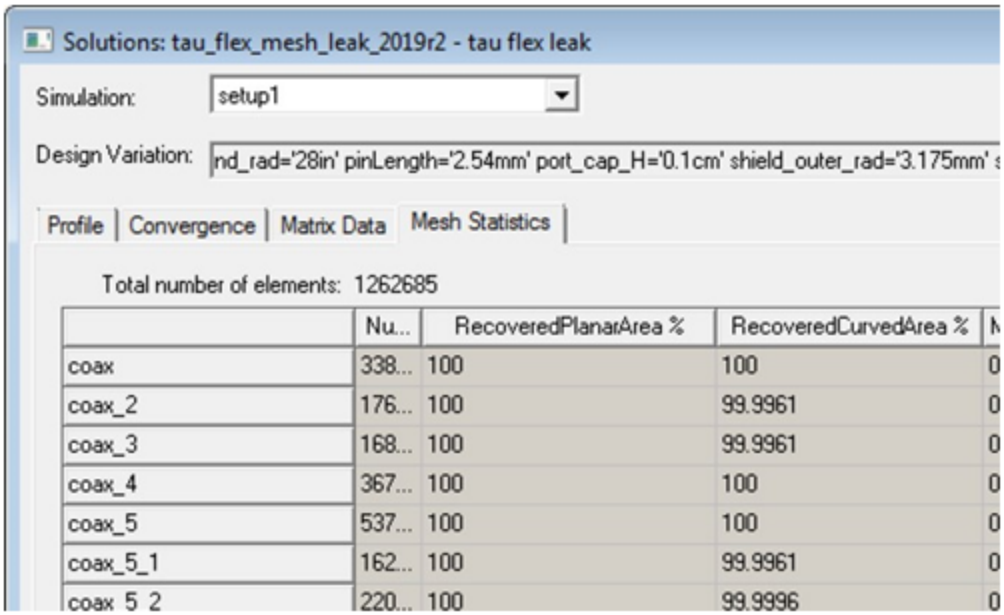
1. **Message Manager** – Immediately after the mesh generation is done, if a tolerant mesh is given, warnings display in the Message Manager. If the window is hidden, click **Show Messages**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. **Mesh Statistics** – On the **Results** tab of the ribbon, click **Solution Data** to open a **Solutions** window.

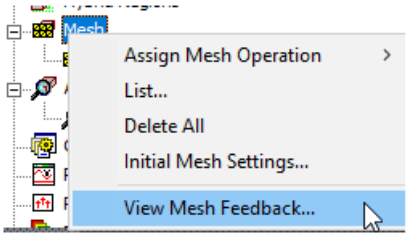
   On the **Mesh Statistics** tab of the **Solutions** window, if a tolerant mesh is generated, the surface area difference between the mesh and geometric model at body level is listed as "RecoveredPlanarArea" and "RecoveredCurveArea". 100% means that the body has a strict mesh. You can click the surface area tab to sort the body list and see the tolerant mesh bodies first.

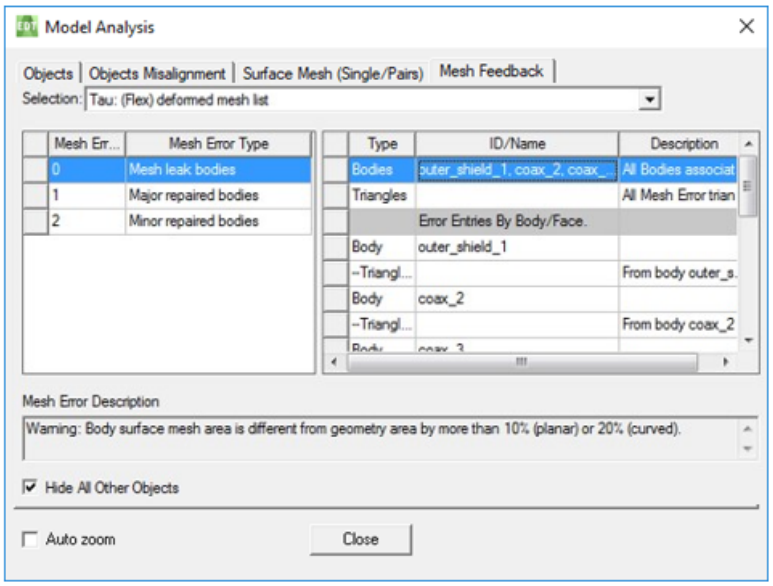


	Nu...	RecoveredPlanarArea %	RecoveredCurvedArea %	
coax	338...	100	100	0
coax_2	176...	100	99.9961	0
coax_3	168...	100	99.9961	0
coax_4	367...	100	100	0
coax_5	537...	100	100	0
coax_5_1	162...	100	99.9961	0
coax_5_2	220...	100	99.9996	0

3. **Model Analysis** – To view the details of the tolerant mesh, click **Modeler** > **Model Analysis** > **Show Analysis Dialog** > **View Mesh Feedback** to launch the **Model Analysis** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Click on the top list to display objects that have tolerant mesh.

Click the "Triangles" to display the mesh on the tolerant objects.

Individual objects are listed under "Error Entities By Body/Face". You can select one or multiple bodies to view the mesh at a more detailed level.

Addressing TAU Flex Mesh Issues

In cases where TAU Flex has difficulty making a strict mesh on some objects, you can address the problem using one of the following techniques:

1. Redraw the geometry in question in the native modeler.
2. Heal the geometry, either with integrated healers or perhaps in Discovery.
3. If the object has curved or true surfaces, consider applying a localized surface mesh operation which creates a finer than default mesh.
4. Remove difficult objects that are not critical to the simulation.

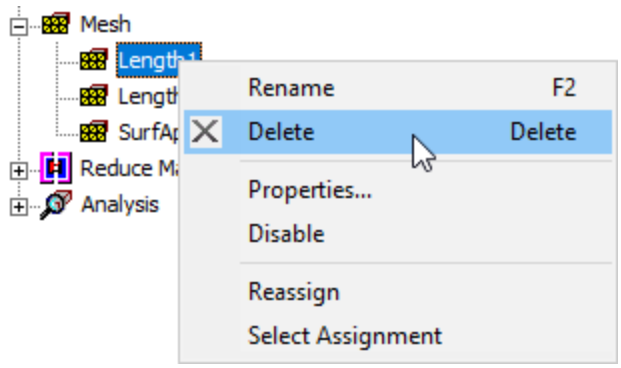
Deleting Previously Assigned Mesh Refinements

You can delete an individual mesh refinement, or delete all previously assigned mesh refinements.

To delete a single refinement:

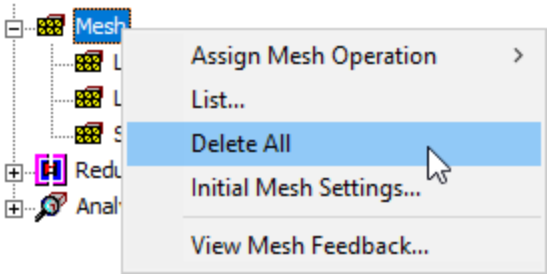
- In the Project Manager, right-click the mesh refinement and click **Delete**, or press the **Delete** key.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To delete all previously assigned mesh refinement data:

- Click **HFSS** > **Mesh** > **Delete All**.
- In the Project Manager, right-click **Mesh** and select **Delete All**.



> **Note:**
>
> This will not alter any existing meshes that have already made use of one or more mesh refinement operations. However, deleted mesh operations will not be used again after reverting to the initial mesh.

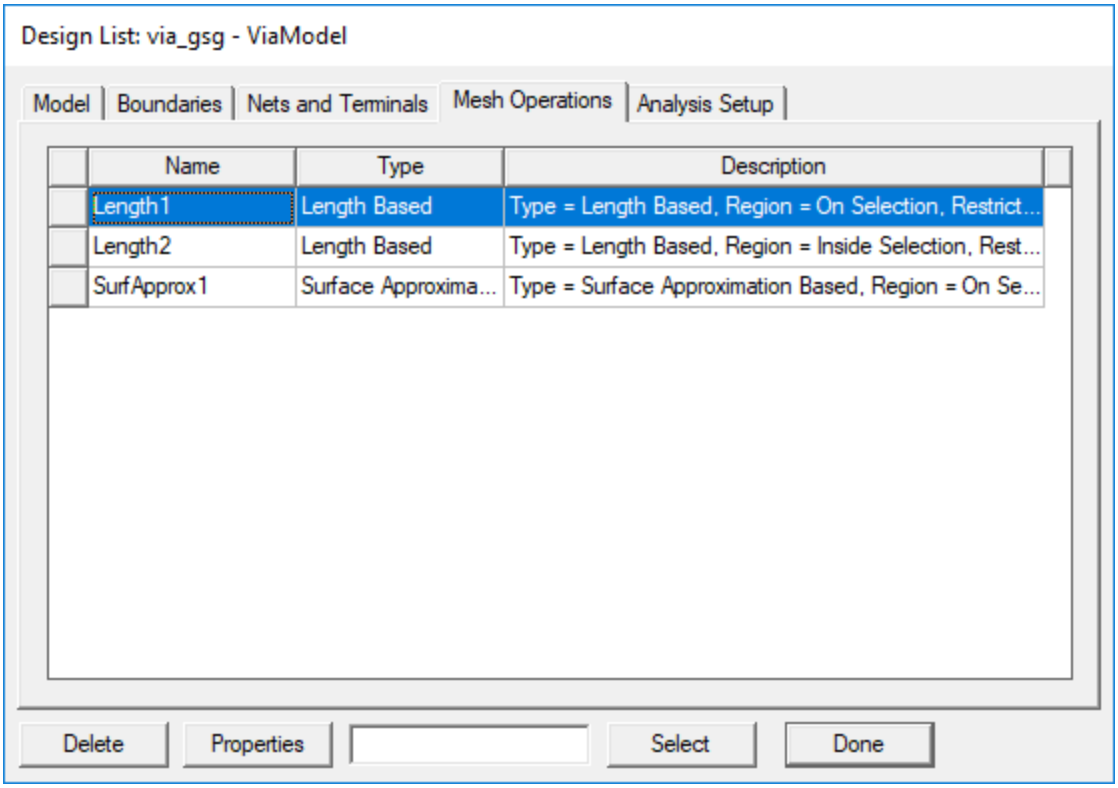
Viewing List of Mesh Operations

To list all mesh operations for the project:

1. In the Project Tree, right-click **Mesh** and select **List**.

   The **Design List** window appears, on the **Mesh Operations** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

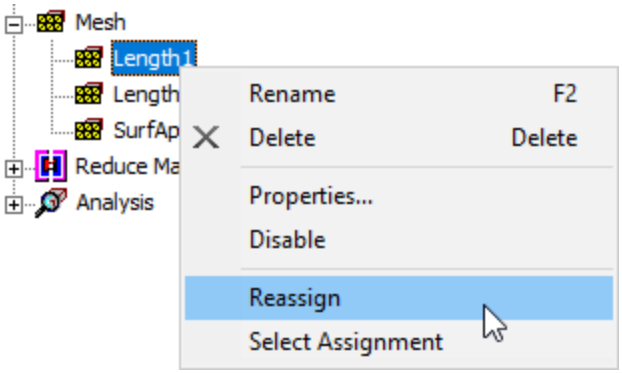


2. Click **Done** to close the window.

Reassigning a Mesh Operation

To reassign a previously assigned mesh setup:

1. Select the object(s) to which you want to reassign the mesh operation.
2. In the Project Tree, under **Mesh**, right-click the mesh refinement you had previously defined and select **Reassign**.



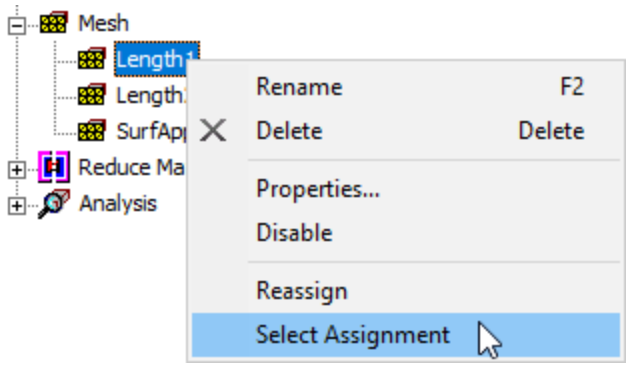
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The mesh operation is reassigned to the currently selected objects.

> **Note:**
>
> This pertains to mesh setup operations only. If you want to see information about a mesh that has been generated, see Viewing Mesh Statistics.

To view the objects to which a mesh operation is assigned:

1. In the Project Tree, select the mesh operation.
2. Right-click the previously completed mesh refinement and select **Select Assignment**.

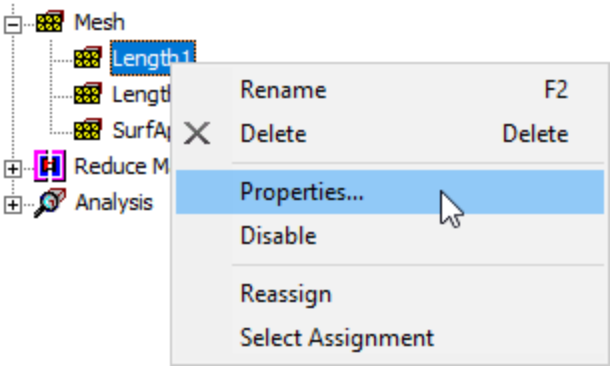


The objects to which the operation is assigned are selected in the modeling workspace.

Viewing Mesh Properties

To view mesh properties:

- In the Project Tree under **Mesh**, right-click a previously completed mesh refinement and select **Properties**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A dialog box opens with properties specific to that mesh (length-based, surface approximation, etc.). You can edit the mesh settings as appropriate.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11 - Example Toolkits

The Toolkit command in the HFSS menu provides access to a design type specific IronPython script, such as the Hearing Aid Compliance Test, MIMO calculation or a module specific task, such as for Boundary, Excitation, or Solve Setup.

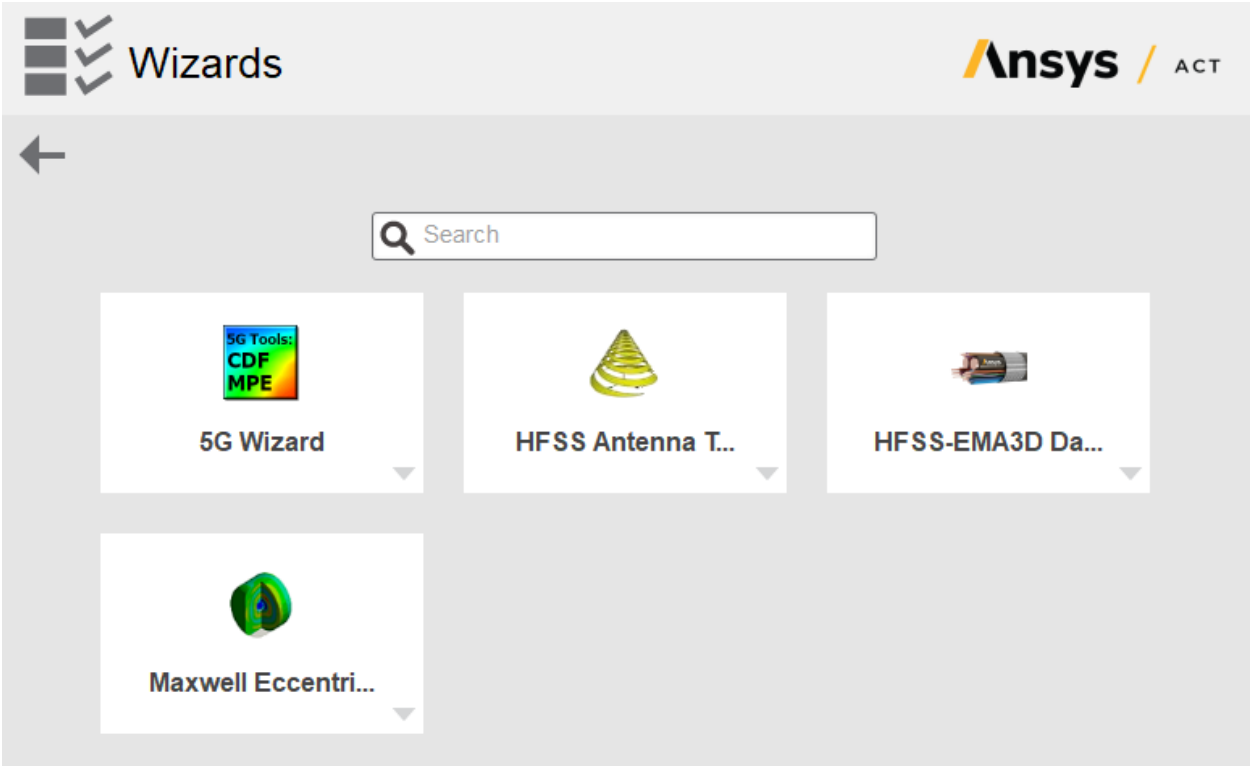
- The Cable modeling toolkits serve automotive cable applications and oil and gas applications.
- Finite Array Beam Angle Calculator toolkit for calculating phase shifts in x and y directions given the scan angles for array elements.
- The Hearing Aid Compliance Test toolkit example involves multiple modules.
- The MIMO example includes a Toolkit script and a User Defined Solution Script.
- The Wavelength Calculator provides a means to calculate wavelengths given frequency or visa versa. It also lists materials in the current model.

Toolkits implemented in Ansys ACT can be accessed through the View>ACT Extensions command.

- HFSS Antenna Design toolkit
- HFSS - EMA3D Datalink toolkit.
- HFSS RADARpre ACT Toolkit
- HFSS RADARpost ACT Toolkit
- HFSS 5G ACT Toolkit

> **Note:**
>
> The **ACT Extensions** window and the design wizards it contains (*5G Wizard, HFSS Antenna Design Toolkit, HFSS-EMA3D Link,* and *Maxwell Eccentricities*) are only available for the Windows version of the Ansys Electronics Desktop software. These items are not available when the software is installed on a Linux platform.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

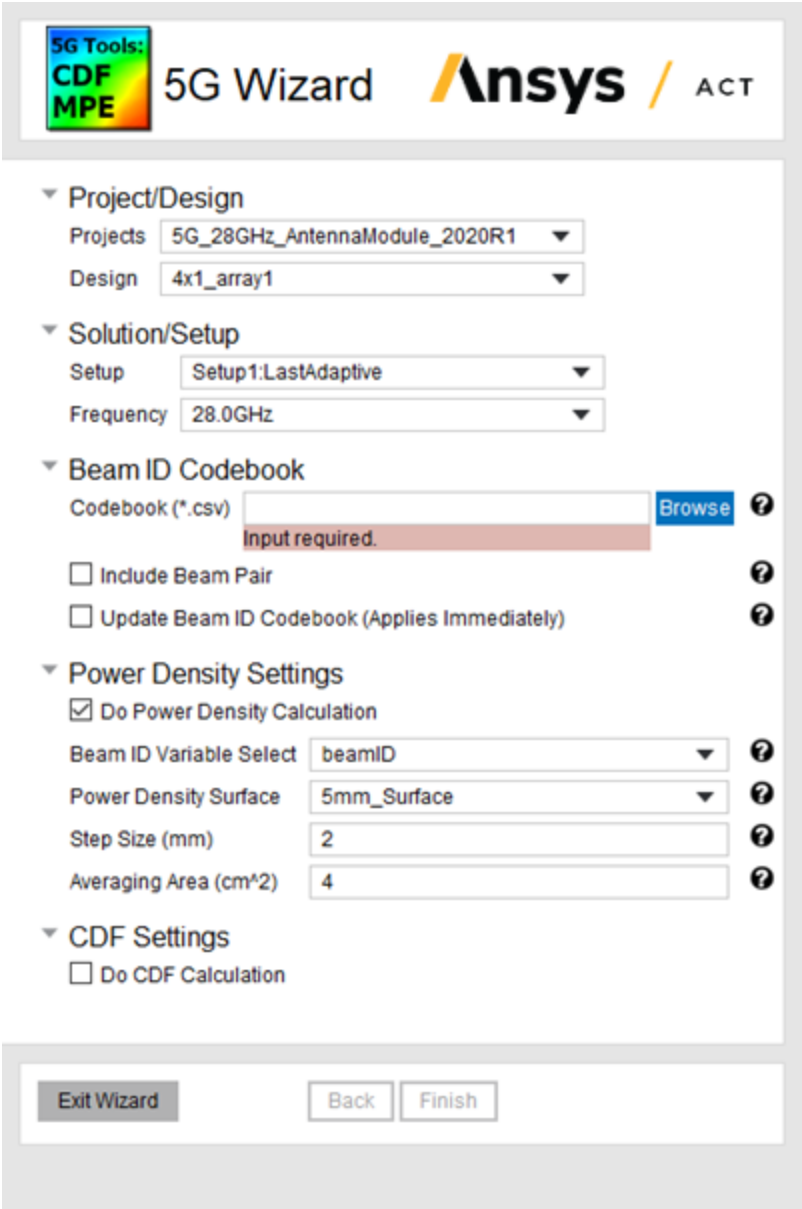


HFSS ACT 5G Wizard

The HFSS 5G ACT Wizard operates on a 5G antenna project with a solution. Given input for a Beam ID Codebook, it provides a set of calculations and reports:

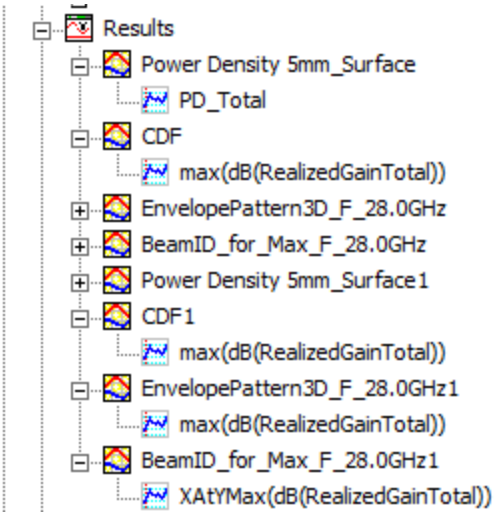
- Calculated maximum permissible exposure (MPE) and Cumulative Distribution Function (CDF) for 5G antenna modules
- Radiation Characteristics Controlled by Codebook
  - Magnitude and phase of antenna ports is defined through external “codebook” csv file
  - Beam ID defines mag and phase on each port
- MPE
  - Power Density (PD, ie. W/m) evaluated on user defined surface
  - Peak and average around peak calculated for each beam ID defined in codebook
- CDF
  - Statistical distribution calculated across beam ID’s for Gain (Total or Realized)
  - Envelope antenna pattern and corresponding beam ID

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

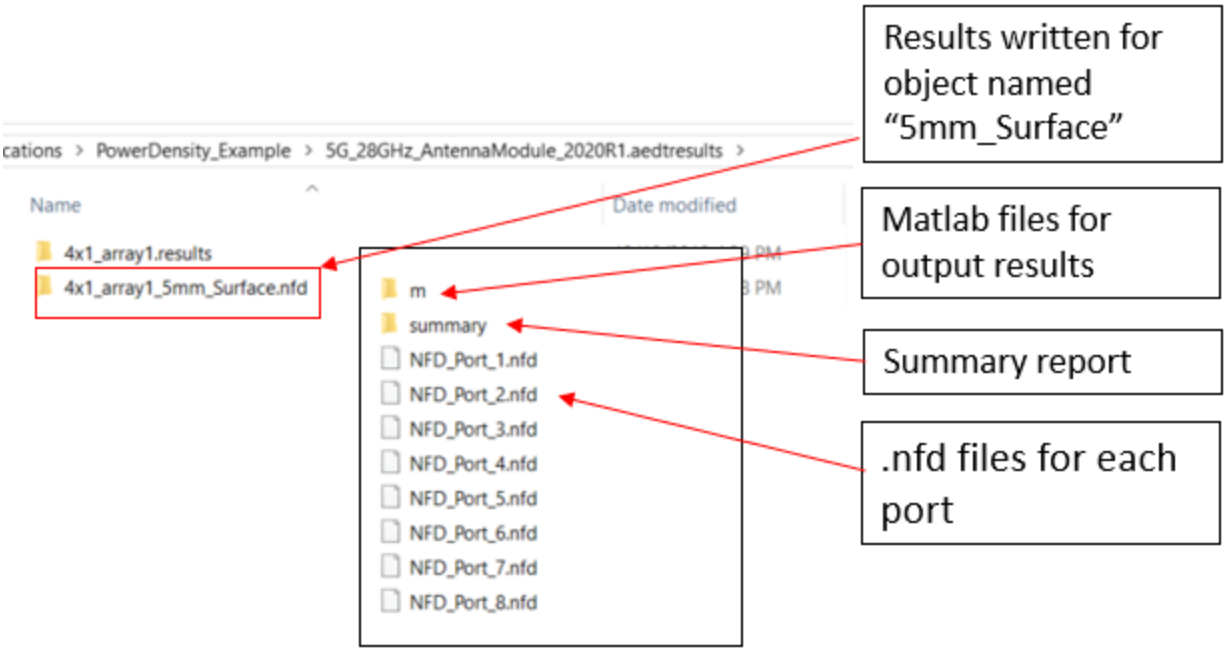


The Wizard generates several predefined plots:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



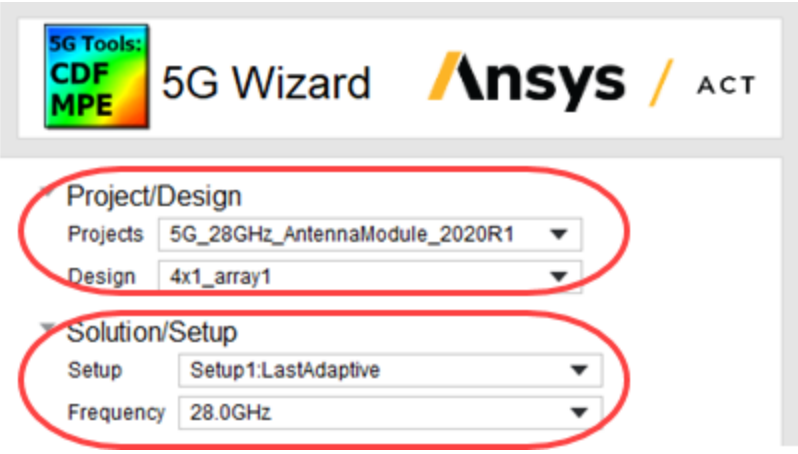
It also creates a set of outputs in the results file:



5G Wizard: Setup

For Project and Design, the HFSS 5G ACT Wizard automatically selects from currently open projects and designs. If necessary, you can use drop down menus to select from listed options.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



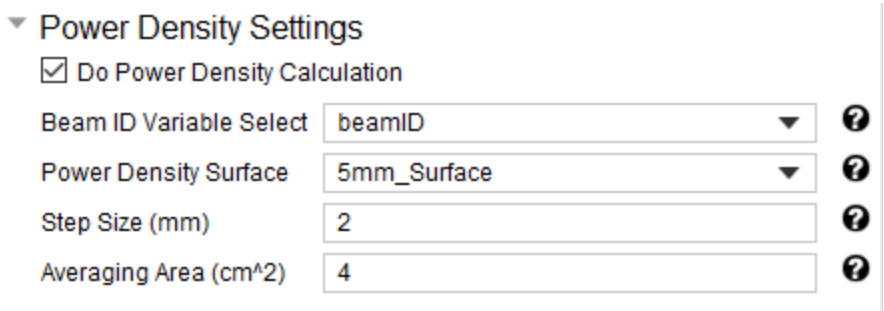
For Solution Setup, the Wizard selects from currently solved solutions. Only solved points with saved fields are selectable.

Note: Menu items will not refresh dynamically, exit wizard and reopen if something has changed in your desktop environments. For example, if you open a new project, complete a solution, or add a geometry, the menu items will not refresh until the 5G Wizard toolkit is reopened

It is recommended that no reports, or field plots exists when calculating PD or CDF. Existing reports may need to be updated many times and will slow down calculation

5G Wizard: Power Density Settings

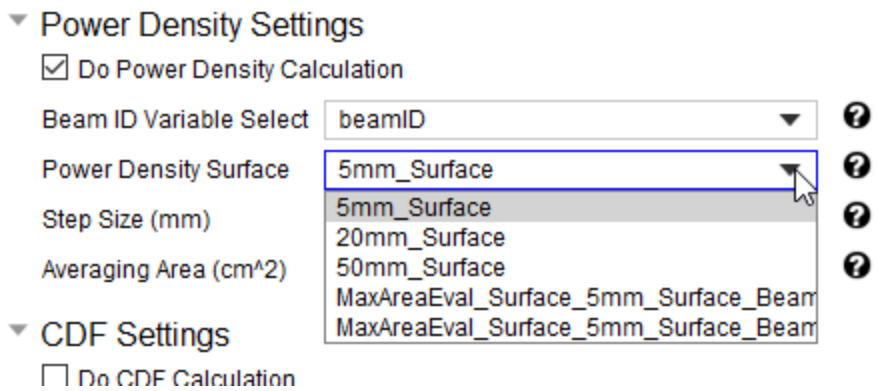
The next section of the 5G Wizard lets you specify whether to **Do Power Density Calculation**, and the applicable Power Density Settings.



Power Density Calculation Overview

1. You generate rectangular surfaces in which Power Density is to be evaluated. In the figure above, the selection is listed as 5mm surface. Any predefined surfaces that exist in the selected Project are listed in the dropdown.

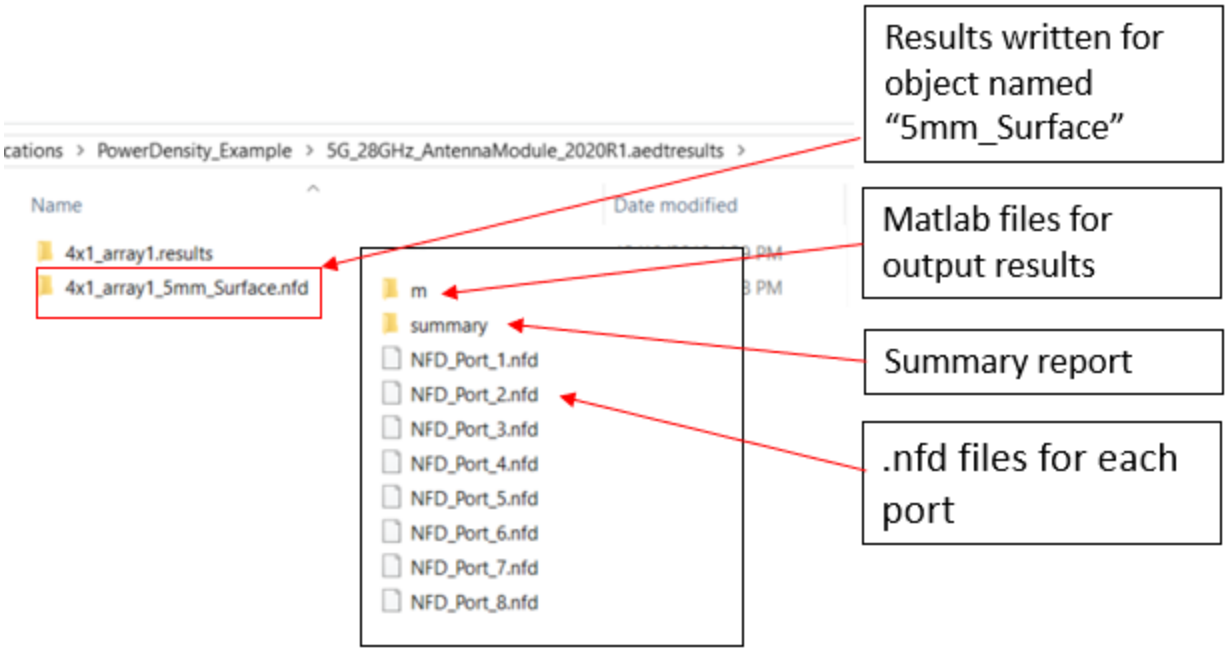
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Import a Codebook to a solved HFSS project.
3. Define sampling size and averaging area to calculate the Power Density over (discussed below).

Fields are exported for each port in the design (one on, all the rest off).

Fields are saved a .nfd file located in the results directory.



4. Superposition of fields is performed for each beam ID

Using .nfd file the appropriate mag and phase is applied to each .nfd file.

5. Peak value and locations are detected for each beam ID

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Average is determined by sum(PD)/#samples over a user-defined averaging area.

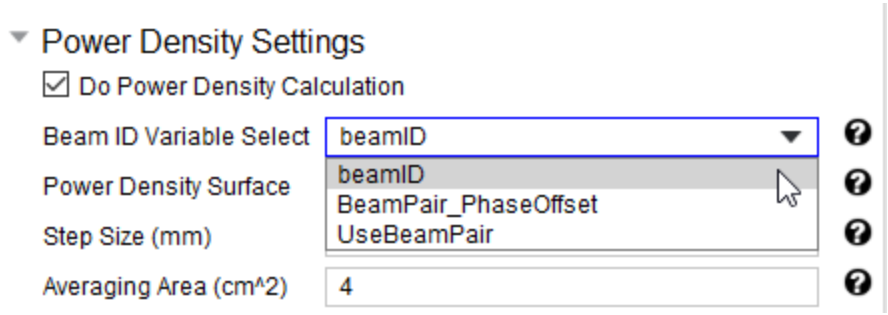
Average area is centered at detected peak location.

6. Summary is output to results directory.

```
Frequency, Beam ID, Total Power (W), Peak Power Density (mW/cm^2), Average Power Density (mW/cm^2),M
28000000000.0,5,6.32247835692,4652.39281647,1456.06119089, [0.0075,-0.0009782609,0.0056] 180.0
28000000000.0,4,6.46935088574,5037.16623029,1500.64199800, [0.0115,0.0009782609,0.0056] 85.0
28000000000.0,7,6.47109324872,4924.64531759,1499.63434421, [0.0035,0.0009782609,0.0056] 80.0
28000000000.0,6,6.45485619160,4626.16505653,1417.81578679, [0.0035,-0.0009782609,0.0056] 60.0
28000000000.0,1,6.45485587488,4626.16501056,1417.81597165, [0.0035,-0.0009782609,0.0056] 300.0
28000000000.0,0,6.32247835006,4652.39283766,1456.06118637, [0.0075,-0.0009782609,0.0056] 180.0
28000000000.0,3,6.45929667120,4658.30215730,1415.13176049, [0.0115,-0.0009782609,0.0056] 60.0
28000000000.0,10,3.20501100428,2225.74886920,731.475188244, [0.0075,-0.0009782609,0.0056] 0.0
28000000000.0,2,6.47109279576,4924.64534844,1499.63447762, [0.0035,0.0009782609,0.0056] 280.0
28000000000.0,9,6.46935036913,5037.16623612,1500.64211761, [0.0115,0.0009782609,0.0056] 275.0
28000000000.0,8,6.45929631334,4658.30209401,1415.13196699, [0.0115,-0.0009782609,0.0056] 300.0

Peak Search Results:
Peak Power Density: 4658.30209401(mW/cm^2)
Average Power Density: 1415.13196699(mW/cm^2)
@Frequency: 28000000000.0Hz
@Location: [0.0115, -0.0009782608999999998, 0.005599999999999999]
@BeamID: 8
@Beam Pair Phase: 300.0deg

Input Power Summary for Peak PD:
Averaging Area: 4.0cm^2
Total Accepted Power: 6.84786625787W
Total Incident Power: 8.0W
Total Radiated Power: 6.91905635326W
```

Beam ID Variable Select

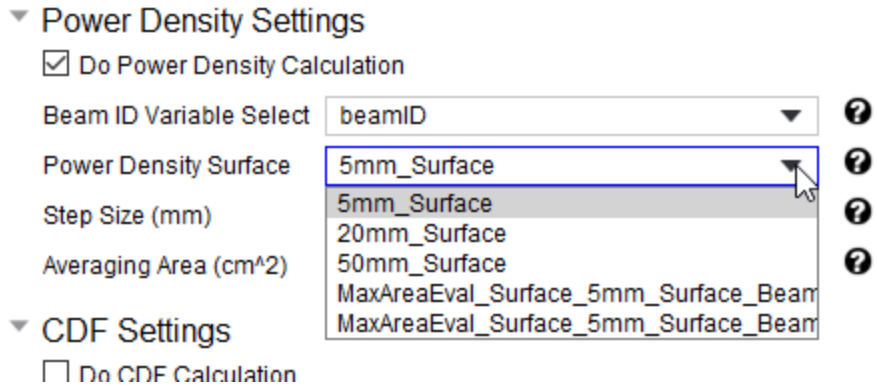
This is the post processing variable used to control the mag/phase of all ports. It corresponds to codebook and is generated when BeamID Codebook is updated. It should always be “beamID” unless custom import of edit sources was defined. The dropdown lists available post processing variables.



Power Density Surface

This is a user defined, rectangular surface. It can be a model or non-model 2D surface. It can exist inside or outside of solved solution domain and should be completely inside or completely outside of the solution domain. The drop down will list all defined surfaces.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Step Size (mm)

This is a field sampling step size defined on a grid. It defines a step size for searching the entire face for maximum field value locations. It defines the grid spacing of the fields when searching for max power density and when performing the average over the averaging area.

Averaging Area (cm^2)

Once a maximum location is determined across all beams, the average is calculated as the sum of all points withing this rectangle divided by the total number of point. A new rectangle is created at this location for visualization.

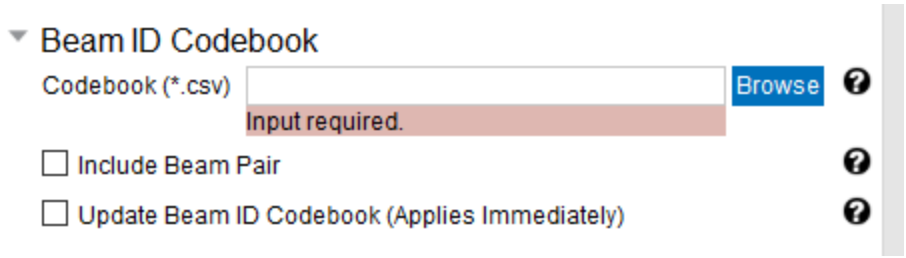
Power Density Worst Case

When determining peak Power Density, the beam pair phase is swept from 0-360 deg, searching for worst possible combination of both beams. The summary report will indicate relative phase difference between beam pairs that results in maximum Power Density. You can select the post processing Beam ID variable “BeamPair_PhaseOffset”to control this relative phase difference.

5G Wizard: Codebook

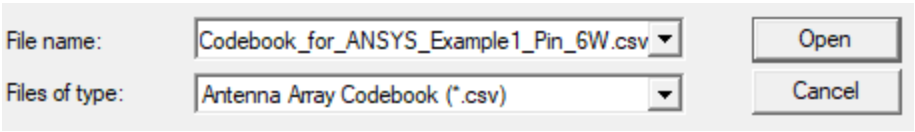
The HFSS 5G ACT Wizard requires that you input the Beam ID Codebook.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Beam ID Codebook

The Codebook is a CSV file that defines magnitude and phase of each port for a corresponding beam ID variable. Clicking the **Browse** button opens a window that lets you navigate and select a file.



Codebook defined as CSV file

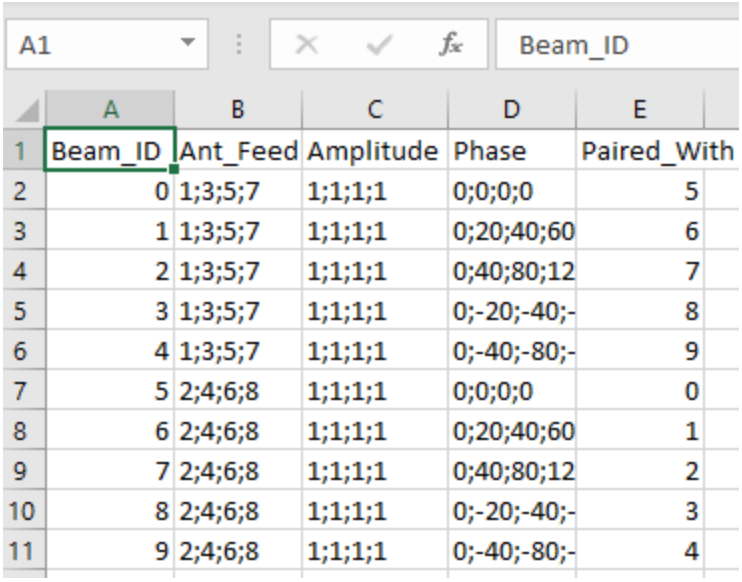
- Each row entry is separated by comma
- Multiple entries for a single column are separated by semi-colon ";"

An example is shown below

```
1  Beam_ID,Ant_Feed,Amplitude,Phase,Paired_With
2  0,1;3;5;7,1;1;1;1,0;0;0;0,5
3  1,1;3;5;7,1;1;1;1,0;20;40;60,6
4  2,1;3;5;7,1;1;1;1,0;40;80;120,7
5  3,1;3;5;7,1;1;1;1,0;-20;-40;-60,8
6  4,1;3;5;7,1;1;1;1,0;-40;-80;-120,9
7  5,2;4;6;8,1;1;1;1,0;0;0;0,0
8  6,2;4;6;8,1;1;1;1,0;20;40;60,1
9  7,2;4;6;8,1;1;1;1,0;40;80;120,2
10 8,2;4;6;8,1;1;1;1,0;-20;-40;-60,3
11 9,2;4;6;8,1;1;1;1,0;-40;-80;-120,4
12 10,2;4;6;8,1;1;1;1,0;0;0;0,-1
```

The same information can be prepared and provided in a csv spreadsheet:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



	A	B	C	D	E
1	Beam_ID	Ant_Feed	Amplitude	Phase	Paired_With
2	0	1;3;5;7	1;1;1;1	0;0;0;0	5
3	1	1;3;5;7	1;1;1;1	0;20;40;60	6
4	2	1;3;5;7	1;1;1;1	0;40;80;12	7
5	3	1;3;5;7	1;1;1;1	0;-20;-40;-	8
6	4	1;3;5;7	1;1;1;1	0;-40;-80;-	9
7	5	2;4;6;8	1;1;1;1	0;0;0;0	0
8	6	2;4;6;8	1;1;1;1	0;20;40;60	1
9	7	2;4;6;8	1;1;1;1	0;40;80;12	2
10	8	2;4;6;8	1;1;1;1	0;-20;-40;-	3
11	9	2;4;6;8	1;1;1;1	0;-40;-80;-	4

The file is organized as columns:

BeamID

Identification number

Ant_Feed (Port Name)

Port name as defined in your HFSS design

Ports not listed are assumed to be not excited for that beam ID

Amplitude and Phase correspond to location in adjacent columns

Name_Port1;Name_Port2,Mag_Port1;Mag_Port2,Phase_Port1;Phase_Port2

Amplitude

Units in Watts if using Driven Modal and Volts if using Driven Terminal

Phase

Phase of corresponding port name in degrees

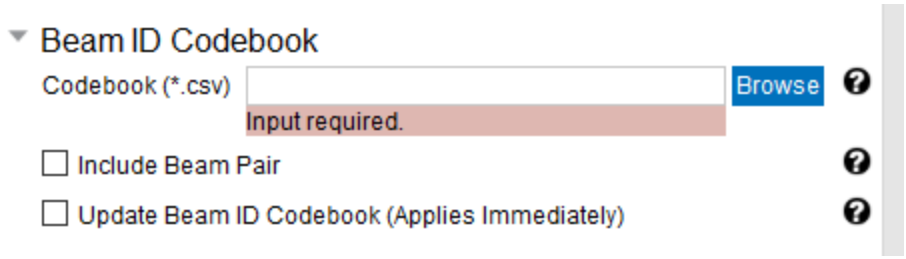
Paired With

Which beam ID to activate when beam pair is used

Use “-1” if no beam pair exist

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Include Beam Pair



Select you want the worst case power density to be evaluated with beam and beam pair excited at the same time. If you select CDF calculation, it will also use the paired beam excited. For power density worst case, the power density with corresponding beam pair is evaluated by sweep a phase offset of the beam pair. This helps determine the worst possible case when the corresponding beam pair may have an unknown phase offset. A variable is created that toggles between true and false. If no beam pair exists, the single beam is used.

For PD worst case calculation, beam pair is automatically used.

Update Beam ID Codebook (Applies immediately)

If a codebook has previously been imported, you do not have to import it again. If a codebook has never been defined, this will import the codebook from a csv file and create 3 post processing variables: beamID, BeamPair_PhaseOffset, and UseBeamPair. These post processing variables will control the sources. If no variable that controls excitations is available, this also needs to be updated.

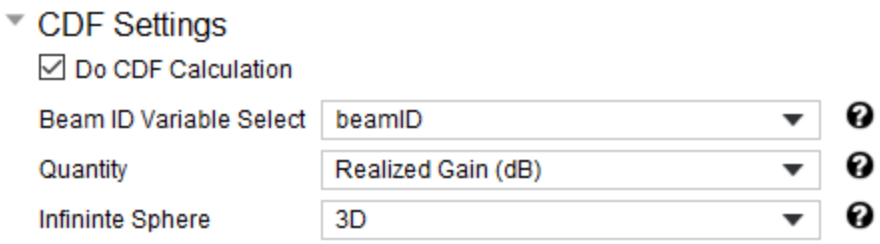
For example, beamID can be set to any defined beamID and all ports will be updated

UseBeamPair can be toggled between true and false to indicate that the beam pair should be also excited along with the primary beam

5G Wizard: Cumulative Distribution Function

The next section of the 5G Wizard lets you specify Cumulative Distribution Function (CDF) settings. Output is for nominal design variation; reports will update if design variables are changed. Also, you can modify reports by selecting Modify Report and editing the Families tab to select additional variations.

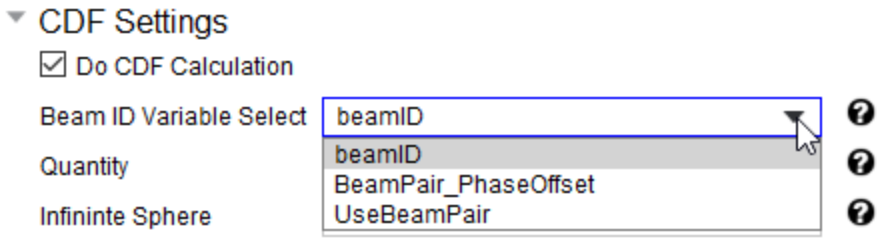
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Checking **Do CDF Calculation** causes the parameters to display.

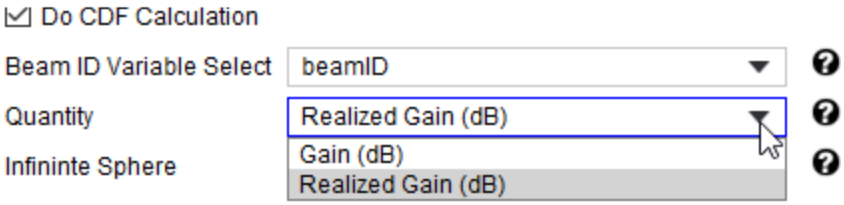
Beam ID Variable Select

This lets you select the post processing variable that controls beam configuration.



Quantity

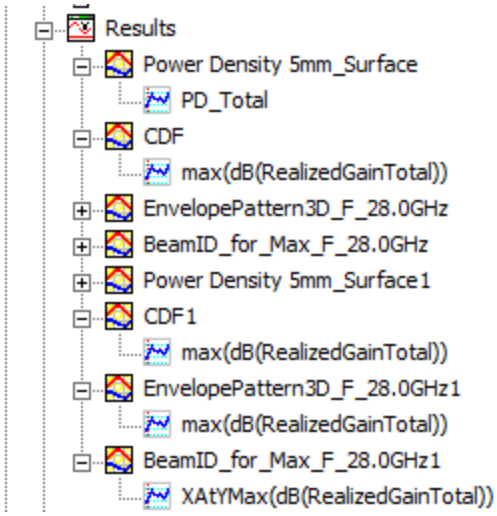
This lets you specify the Quantity to evaluate. Gain function evaluates CDF over each case. With each case, the CDF is plotted showing a distribution of gain values.

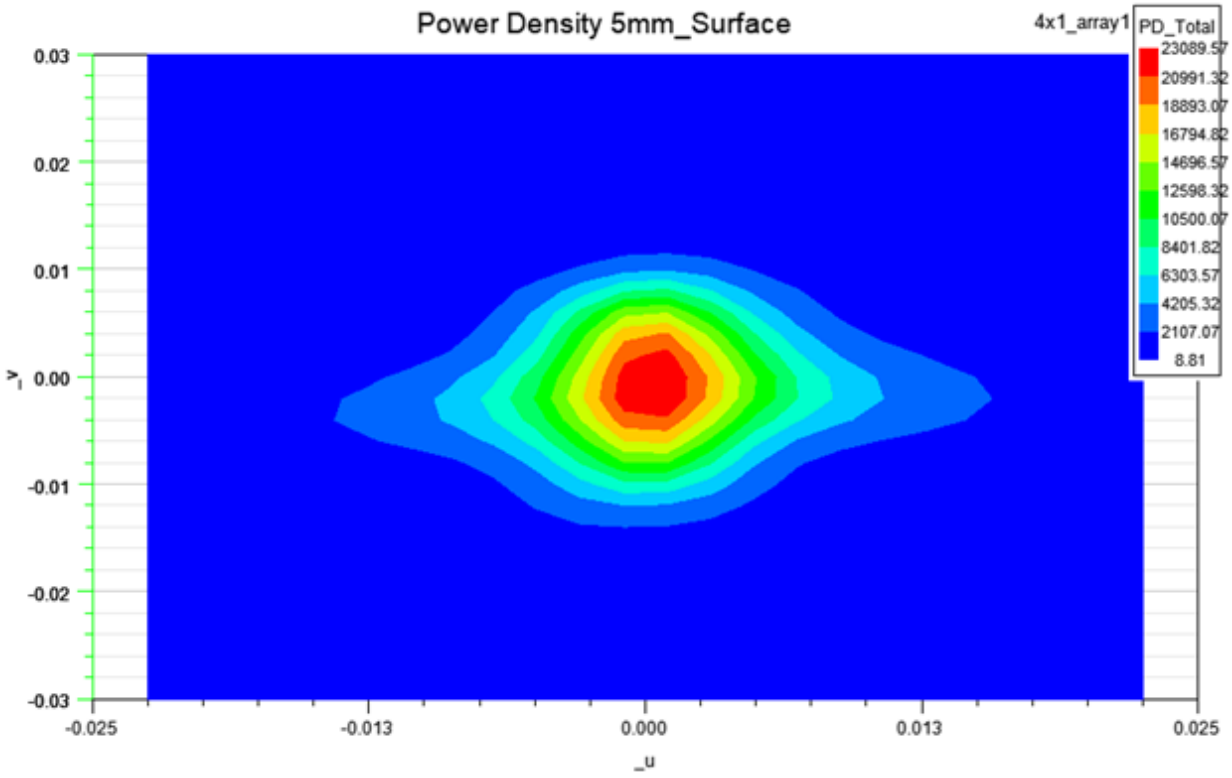


5G Wizard: Example Reports

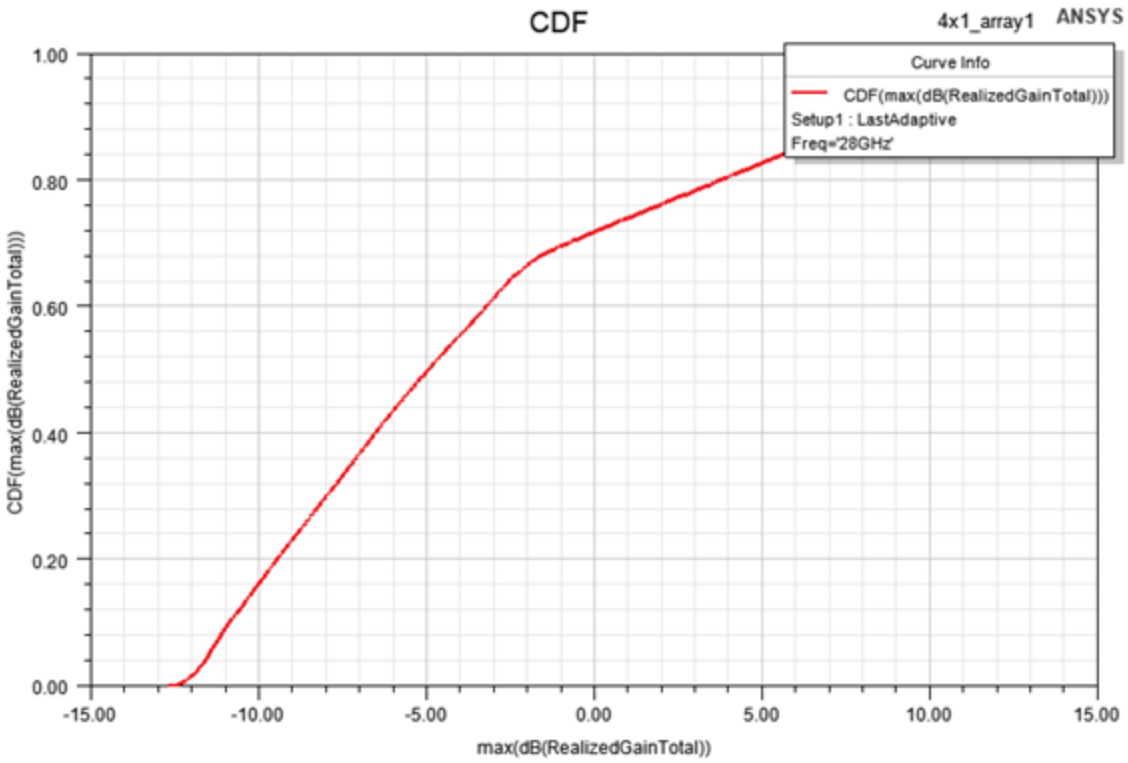
The 5G Wizard creates a set of predefined plots.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

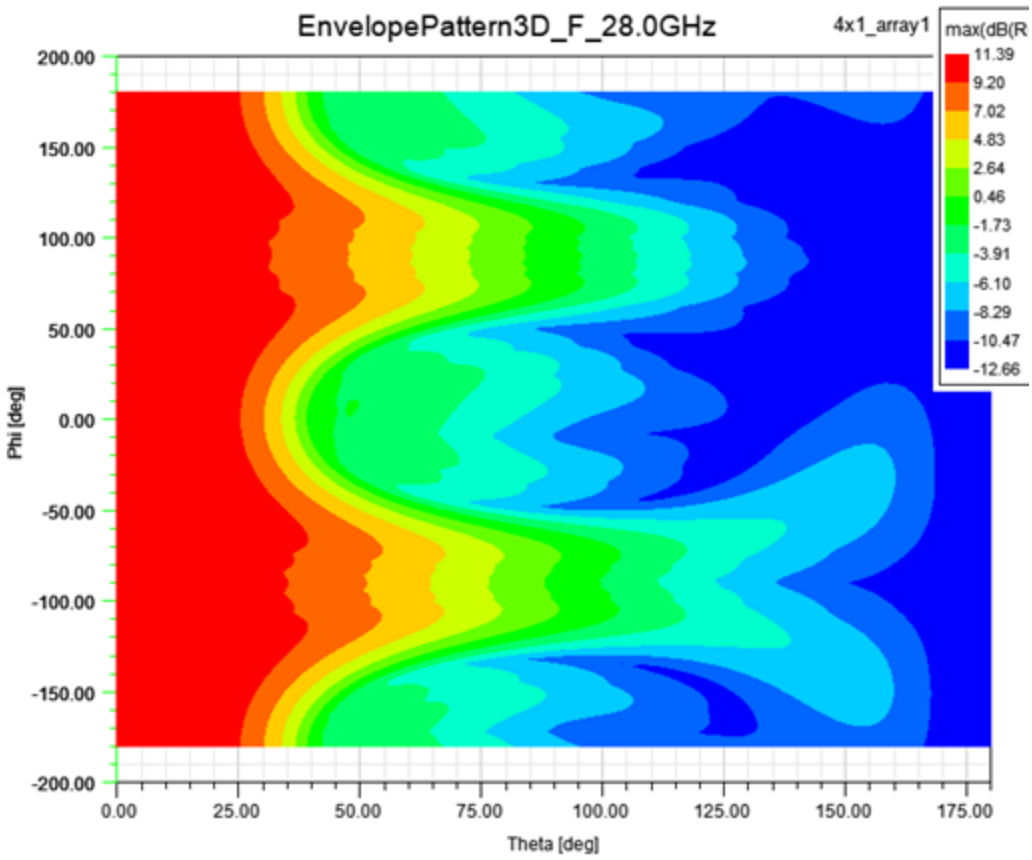




Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



HFSS Antenna Design Toolkit

The HFSS Antenna Design Kit is a GUI-based utility which automates the geometry creation, solution setup, and post-processing reports for over 60 antenna elements. This tool allows antenna designers to efficiently analyze common antenna types using HFSS and also assists new users in learning to use HFSS for antenna design. All antenna models created by the design kit are ready to simulate in HFSS. (Note that on opening an HFSS-IE project, the project will automatically be converted to an equivalent HFSS project using IE Regions.) Simulations, sweeps, and 2D and 3D post-processing reports are automatically defined.

Parametric Antenna geometry:

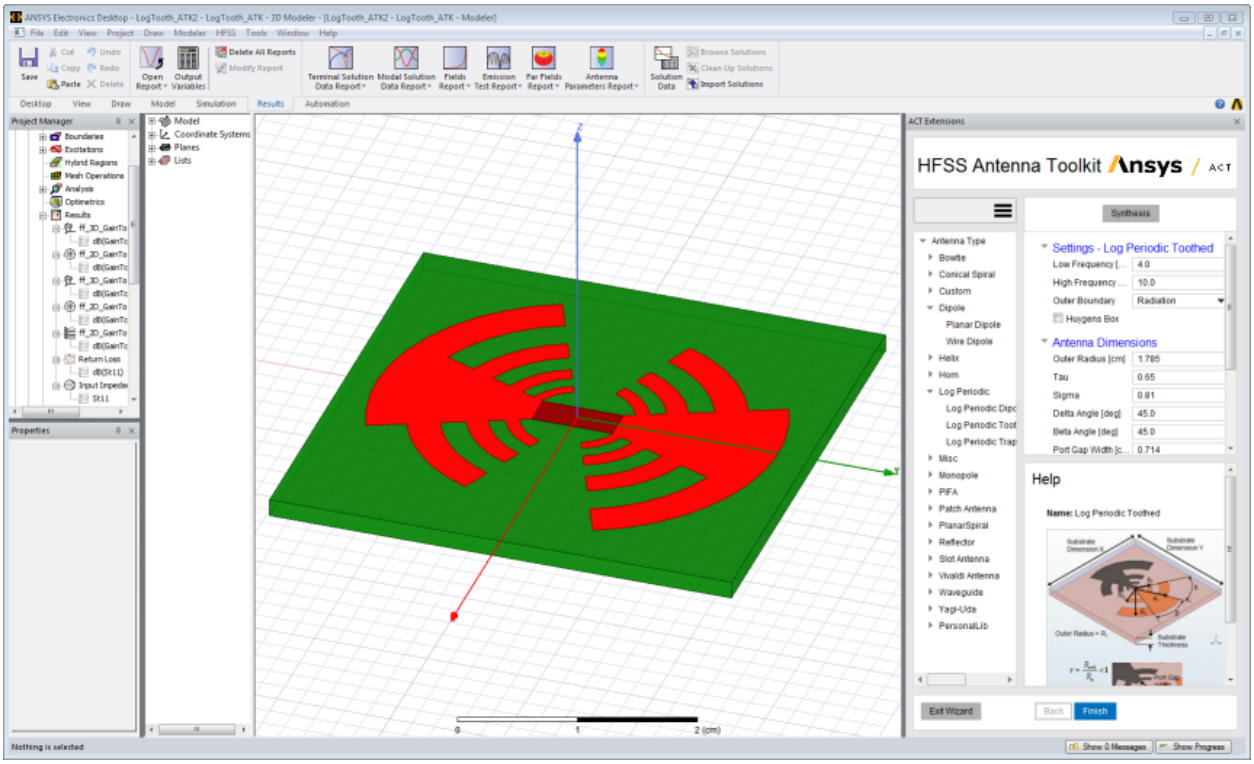
- Easily modify parameters in HFSS after generating initial model.
- Facilitates parametric sweeps and optimizations.

Synthesis features for each antenna:

- Automatically generates physical dimensions for desired frequency when you use the Synthesis button.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

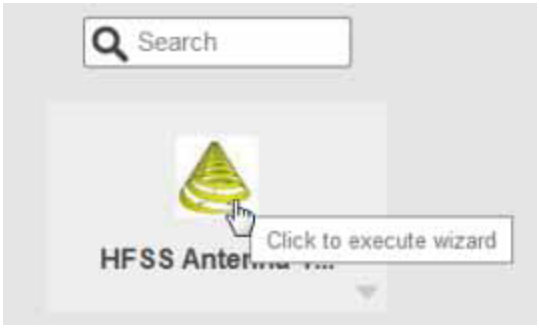
- Provides starting point for new designs.



The HFSS Antenna Design toolkit is implemented in Ansys ACT.

To launch the Wizard:

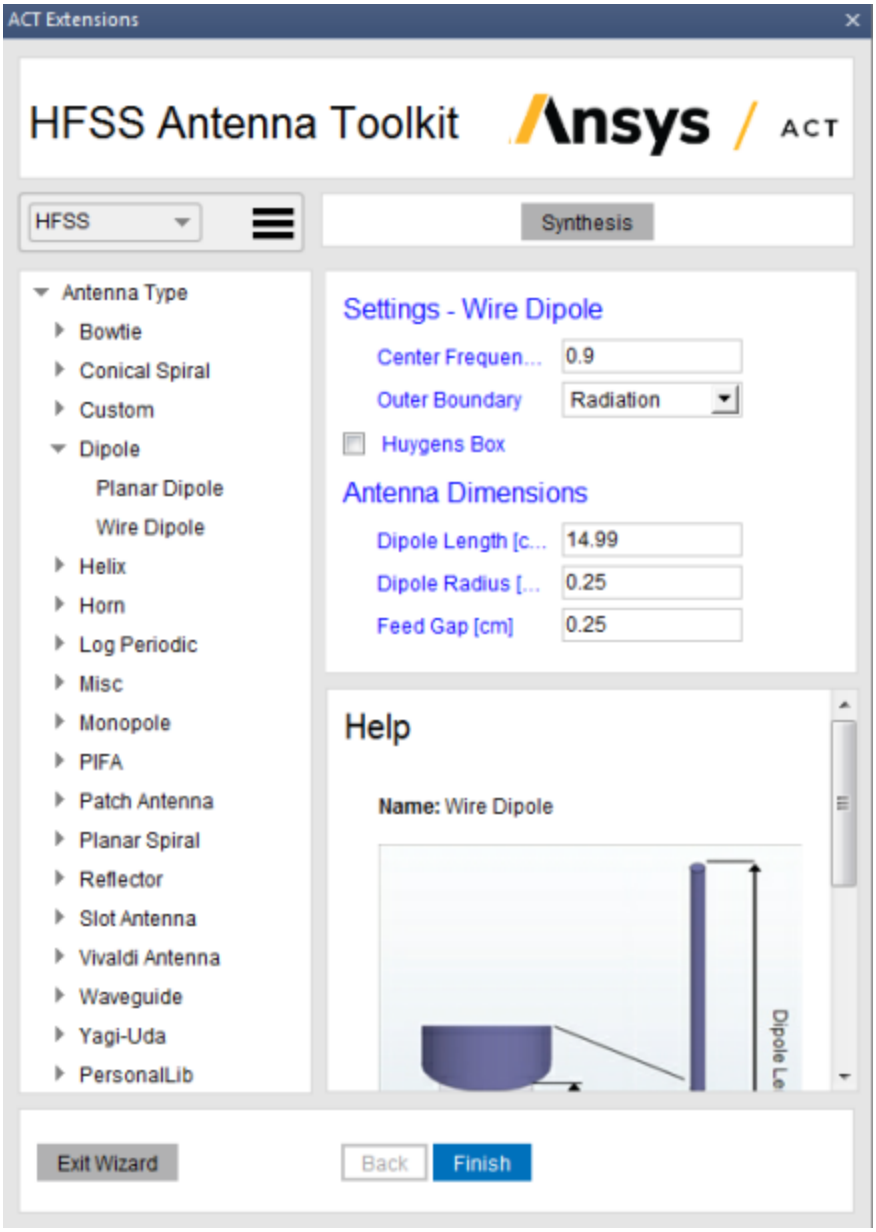
1. Click View>Extensions command to open the Extensions window.
2. Click Wizard to open the ACT Start Page.



3. Click Execute Wizards to change the show HFSS Antenna.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The HFSS Antenna wizard displays.

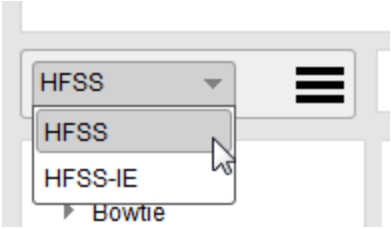


Using the HFSS Antenna Wizard

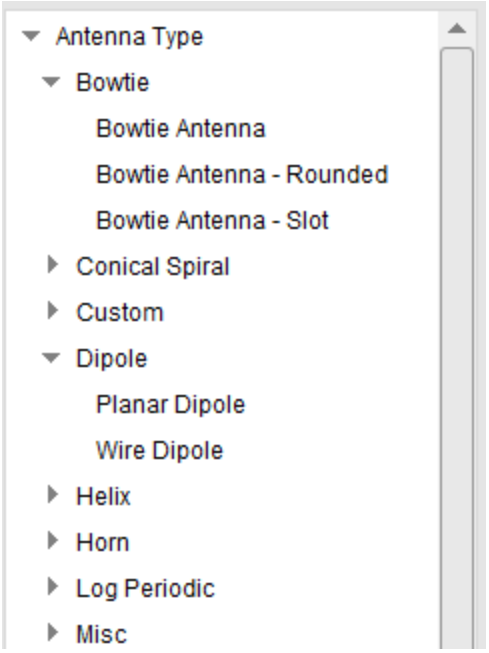
The HFSS Antenna Wizard contains areas for selecting the design type, selecting the category and type of antenna, and a help section that illustrates the selected antenna.

1. Select the type of antenna as HFSS or HFSS-IE from the drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



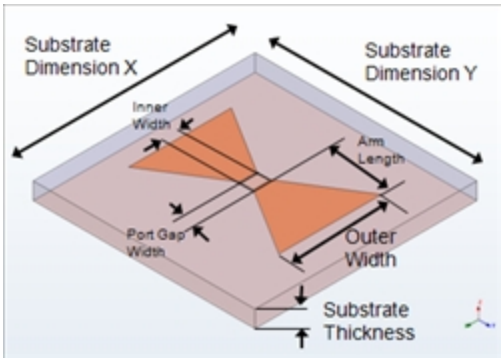
2. You then select the Antenna type and category from the library list. You can show or hide the categories under each antenna type by clicking the arrow icons.



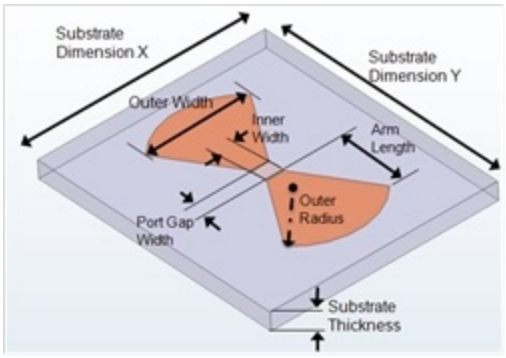
The choices are:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

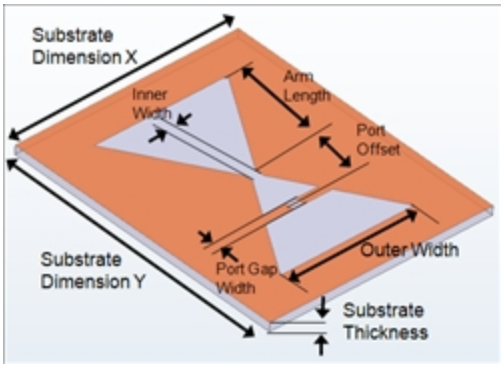
- **Bowtie,** with three types:



Bowtie



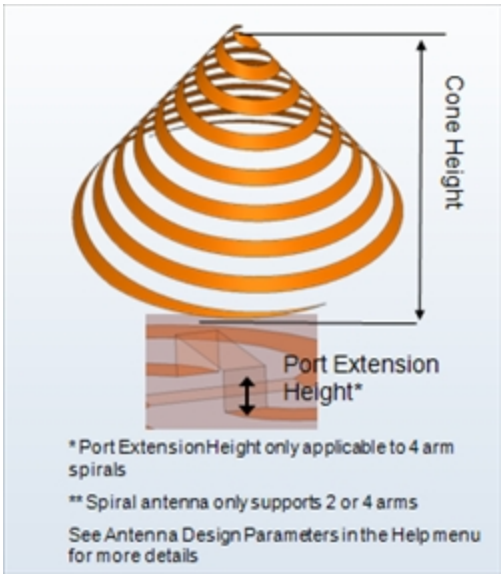
Bowtie Rounded



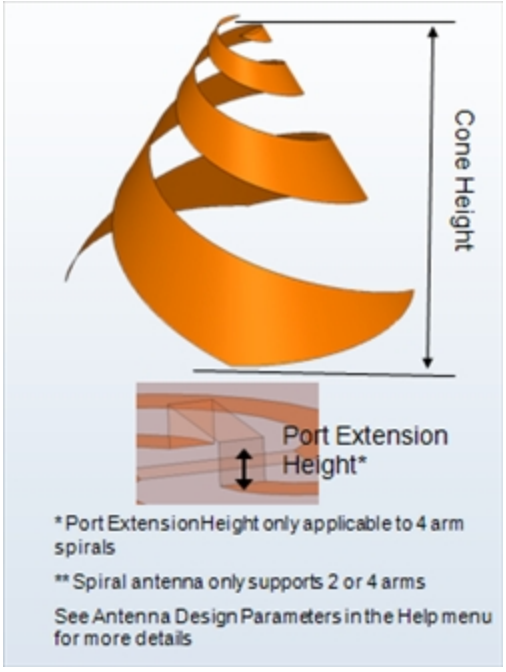
Bowtie Slot

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

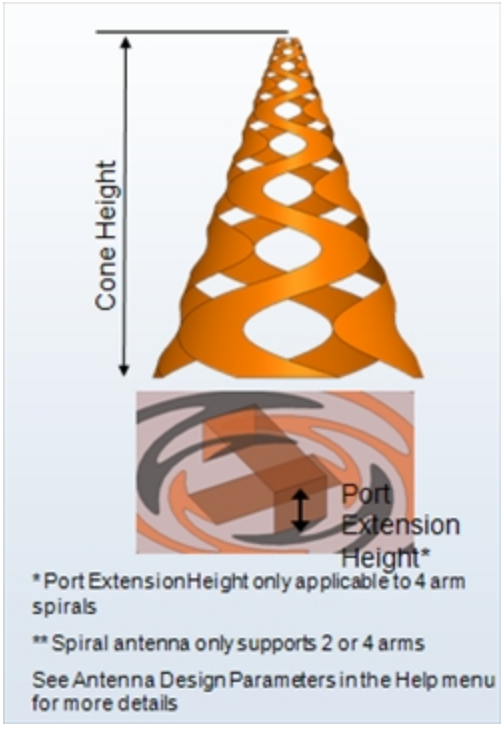
- **Conical Spiral** with three types:



Archimedean Spiral



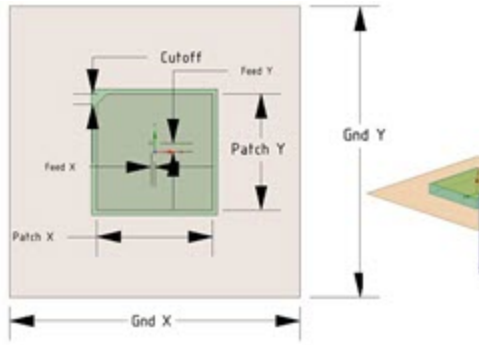
Log Spiral



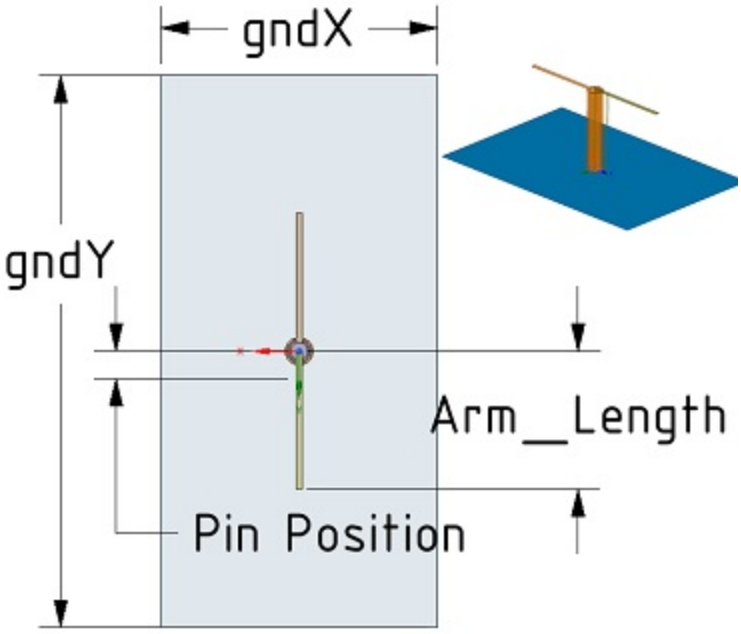
Sinuous Spiral

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Custom** with two types:

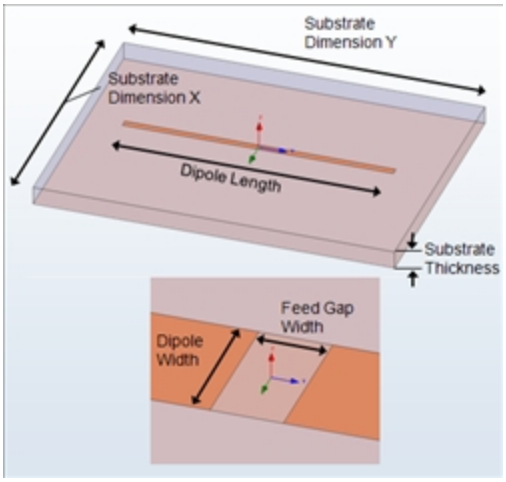


GPS - Ceramic

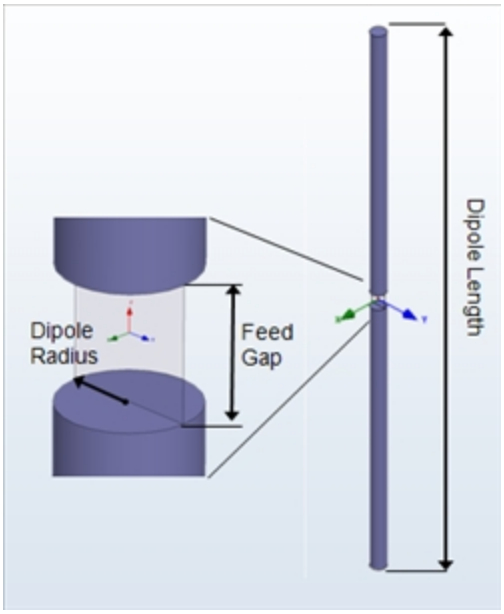


UHF Probe

- **Dipole** with two types:



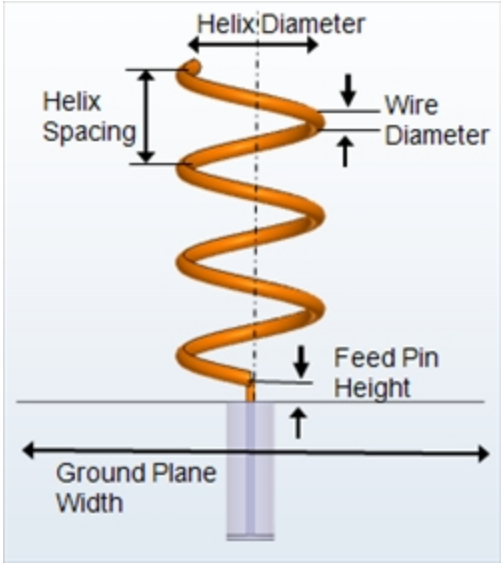
Planar



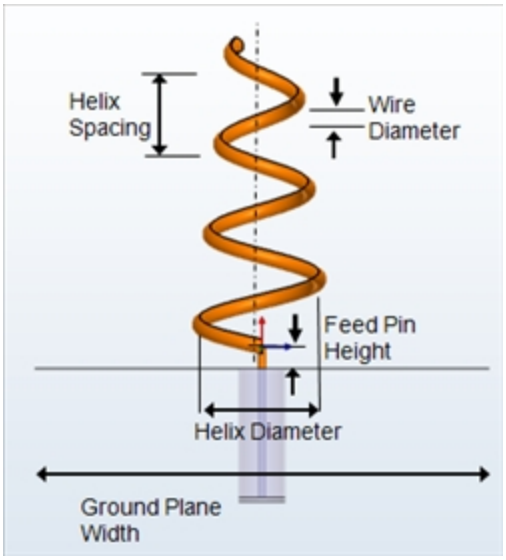
Wire

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

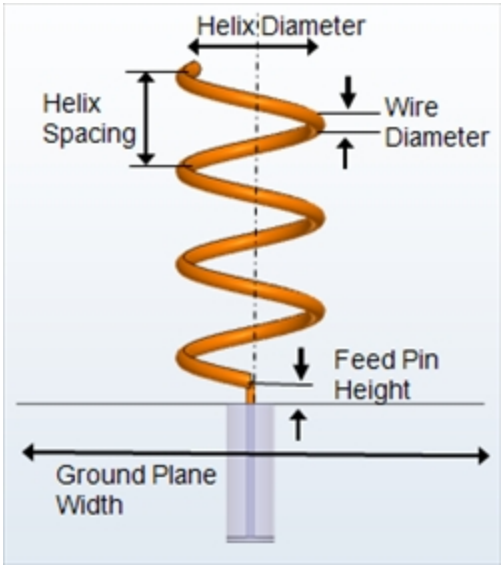
- **Helix**, with five types:



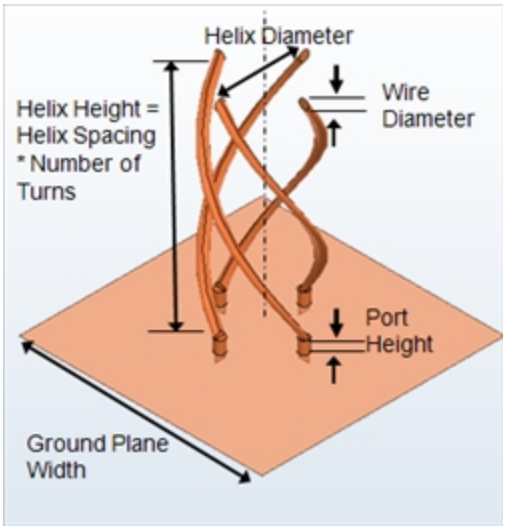
Axial Mode



Continuous Taper

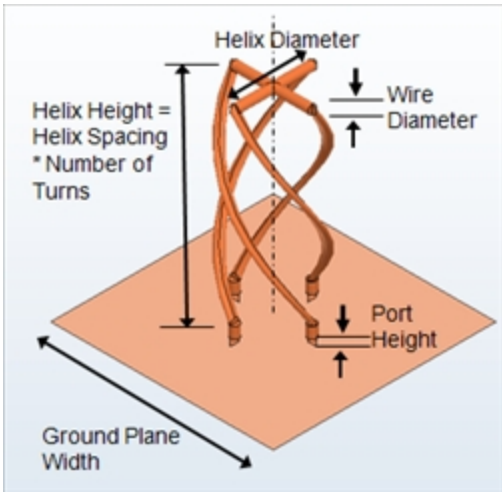


Normal Mode



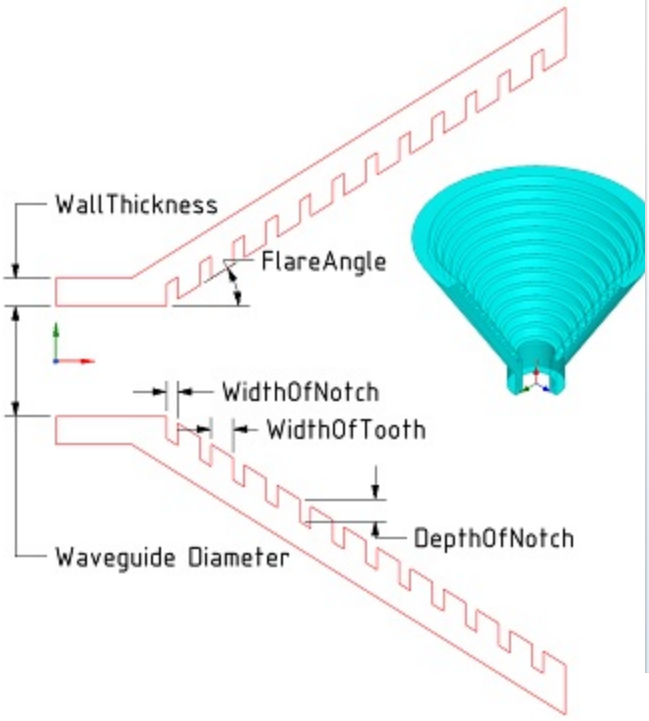
Quadrifilar - Open

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

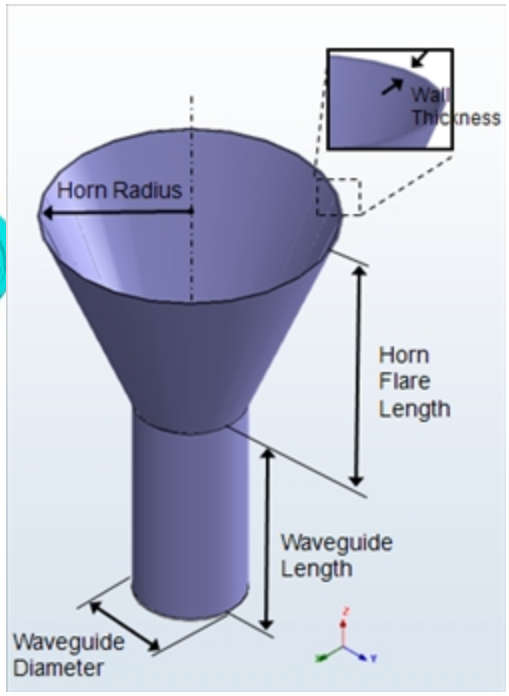


Quadrifilar - Short

- **Horn**, with eight types:

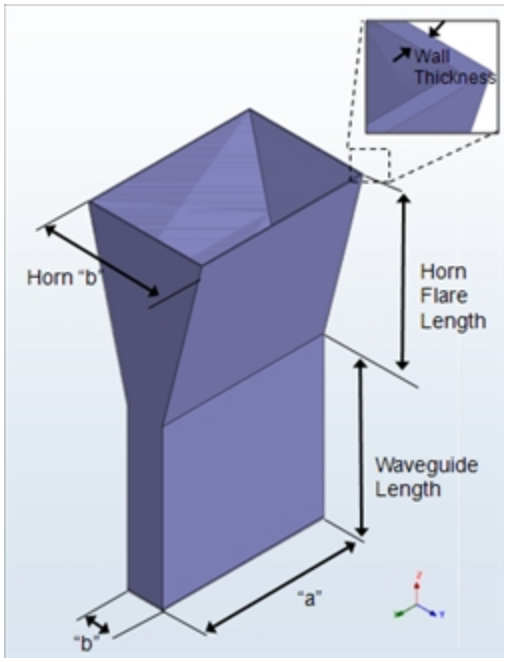


Conical Corrugated Horn

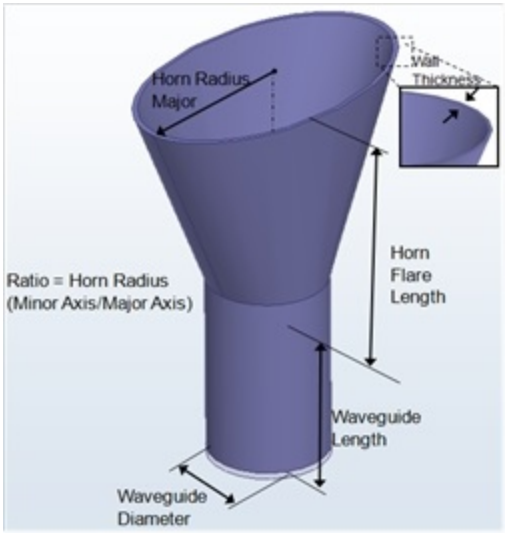


Conical

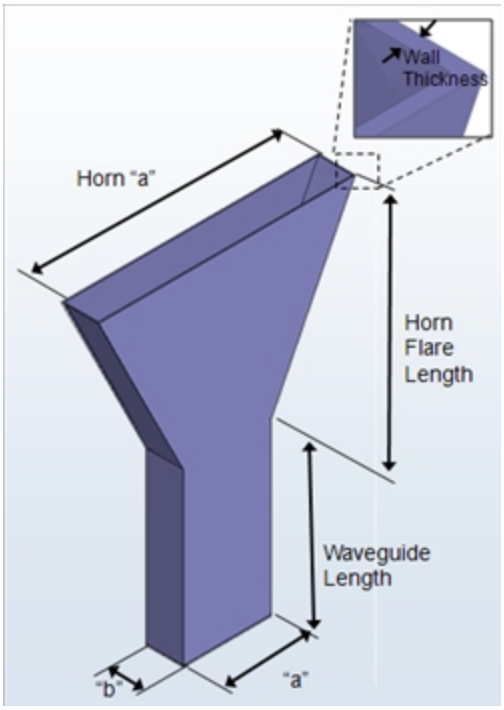
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



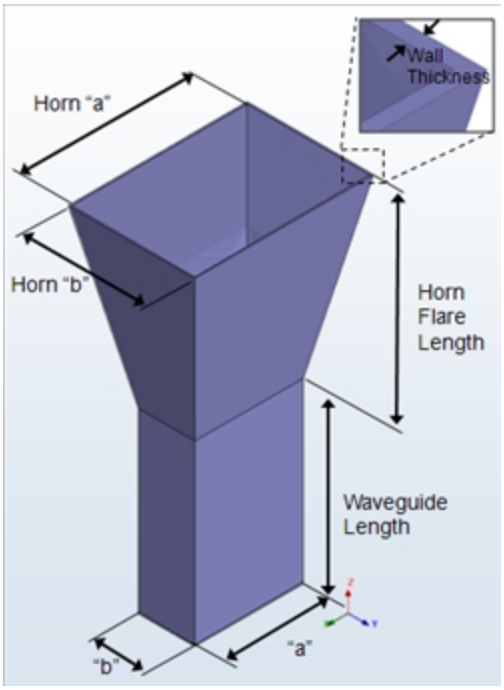
E-Plane Sectoral



Elliptical

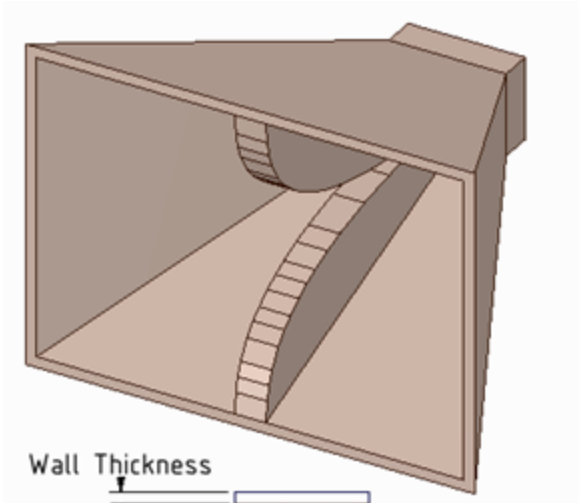


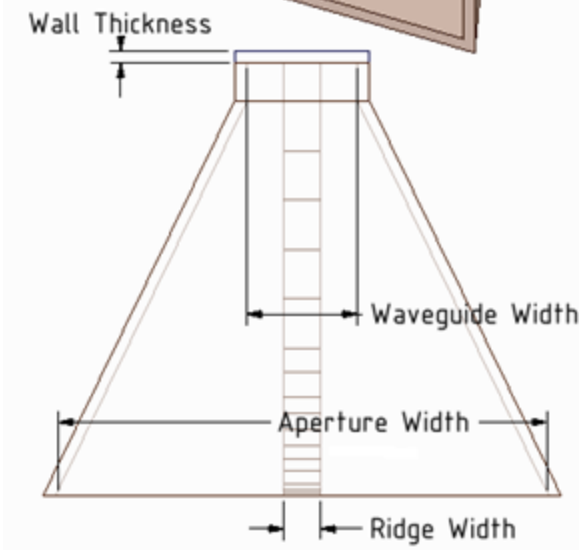
H-Plane Sectoral

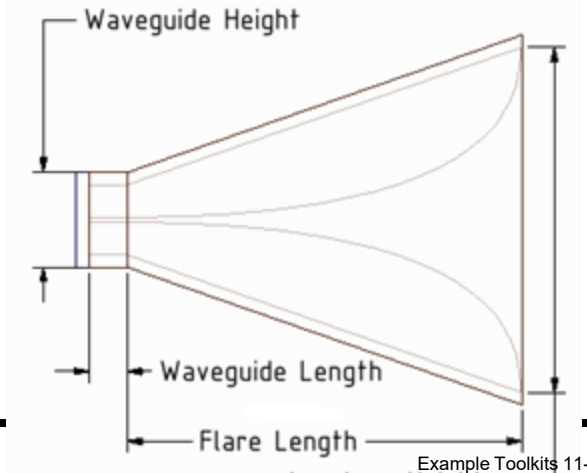


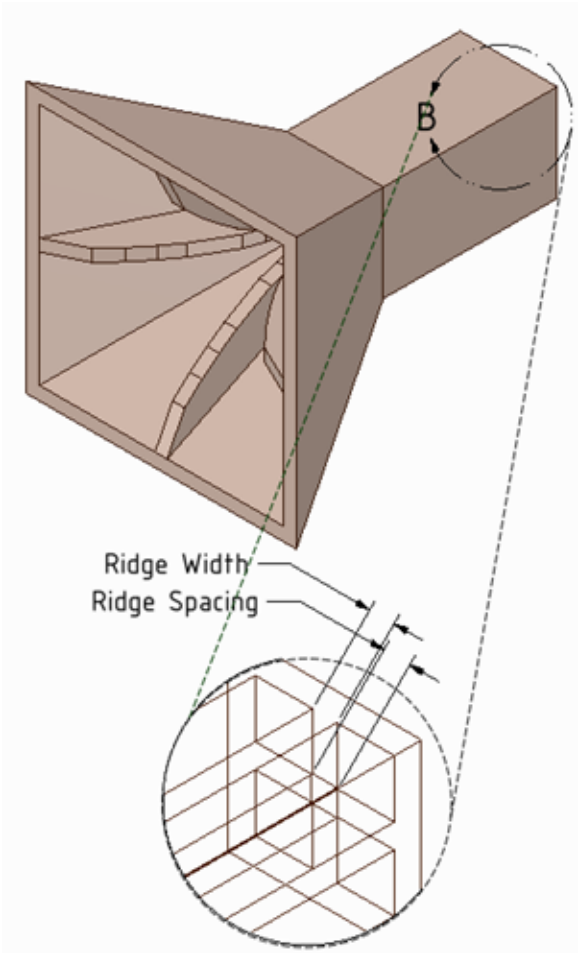
Pyramidal

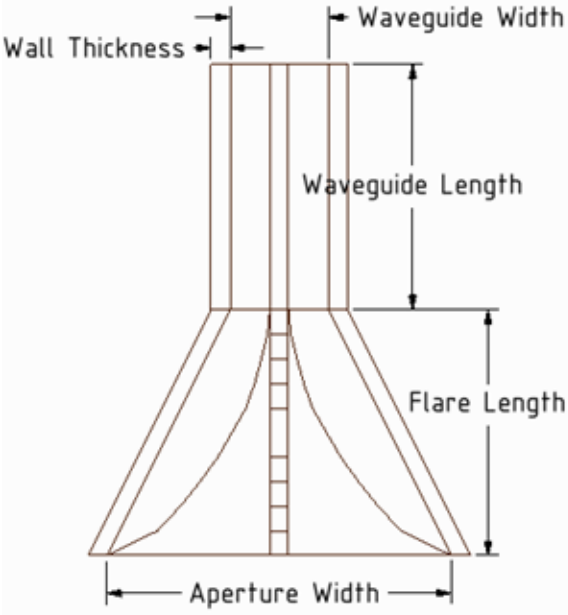
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

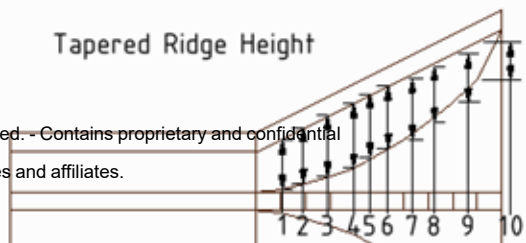








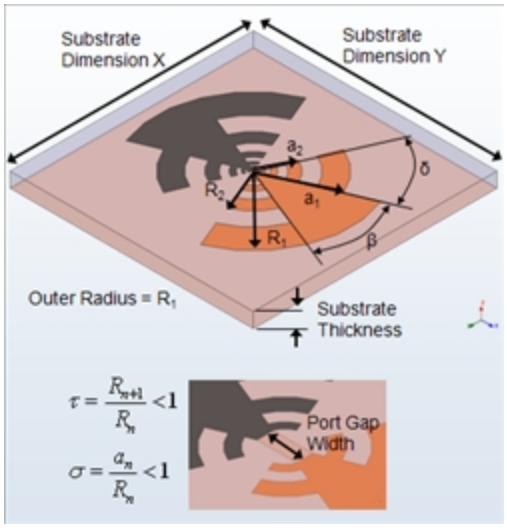


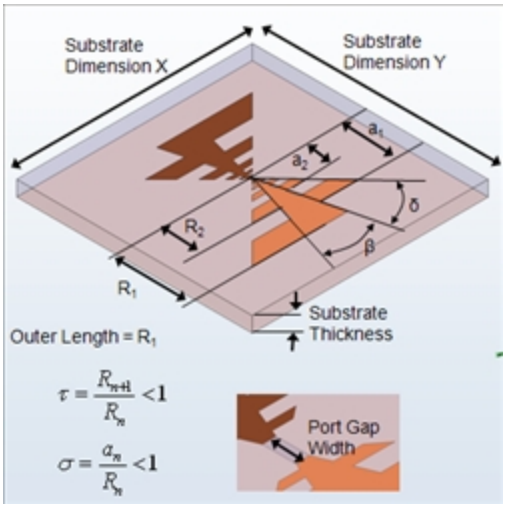


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pyramidal Ridged **Quad Ridged**

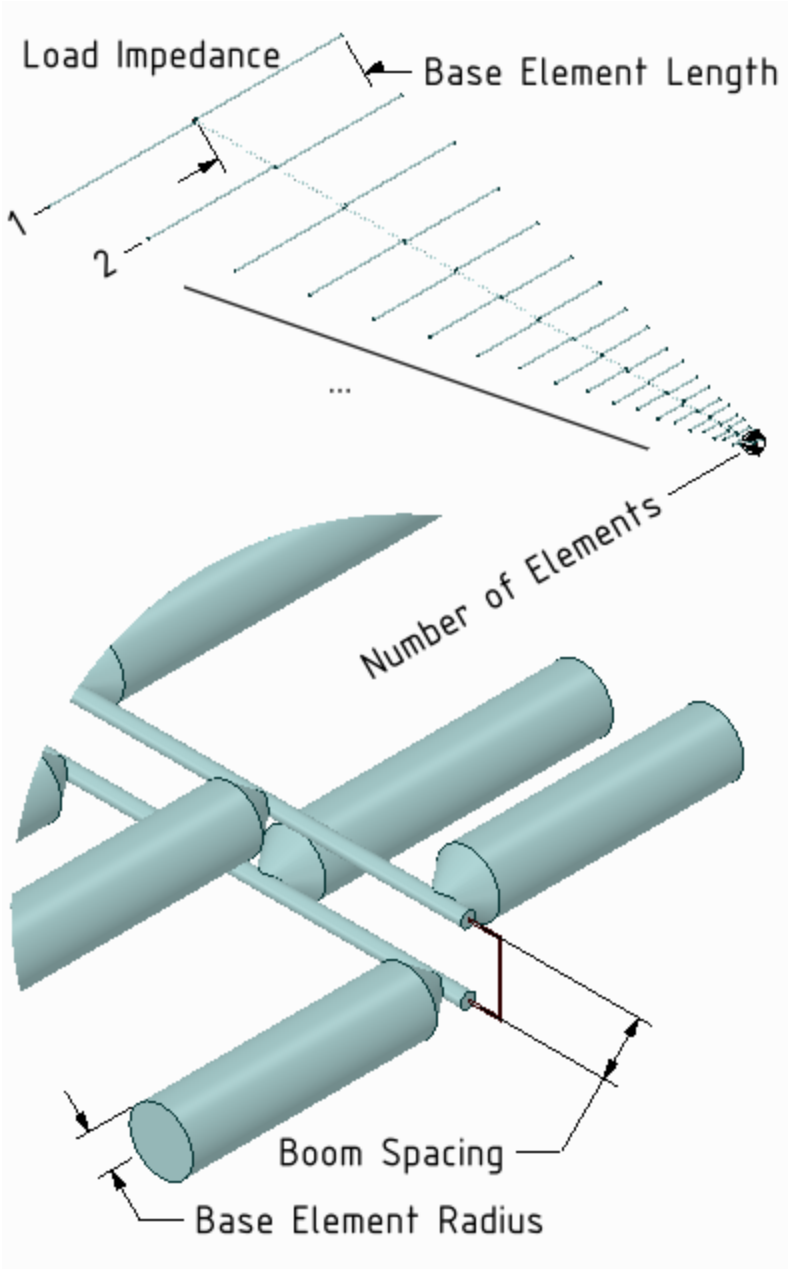
- **Log Periodic**, with three types:





Toothed **Trapezoidal**

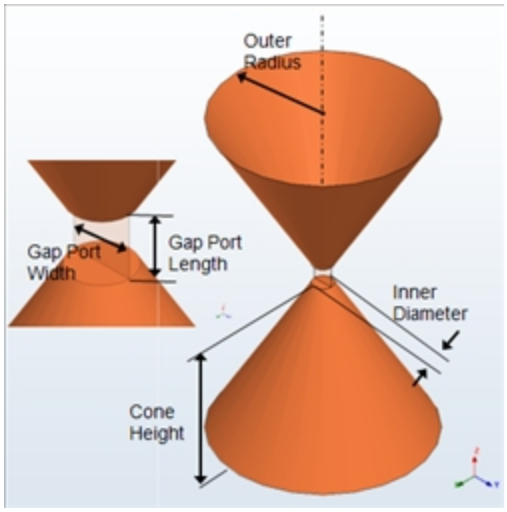
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



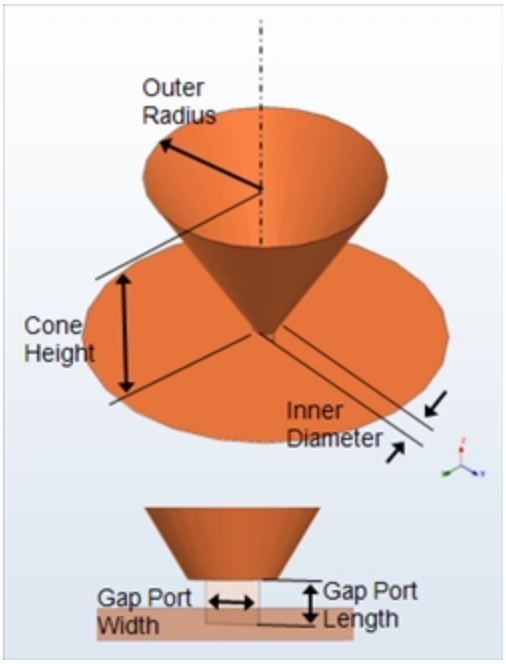
Log Periodic Dipole Array

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Misc**, with two types:



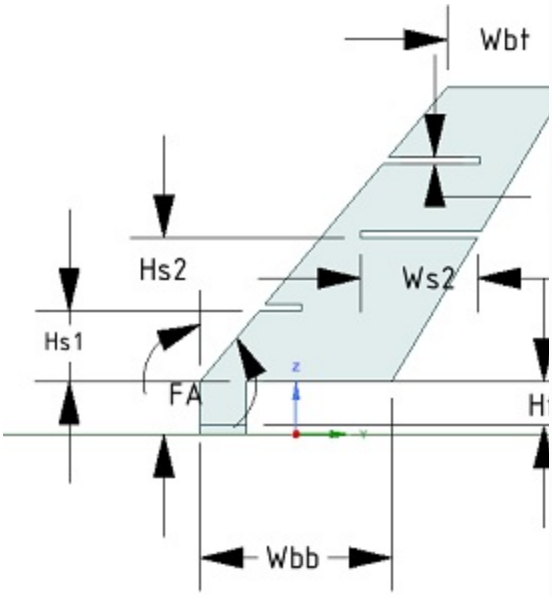
Bicone



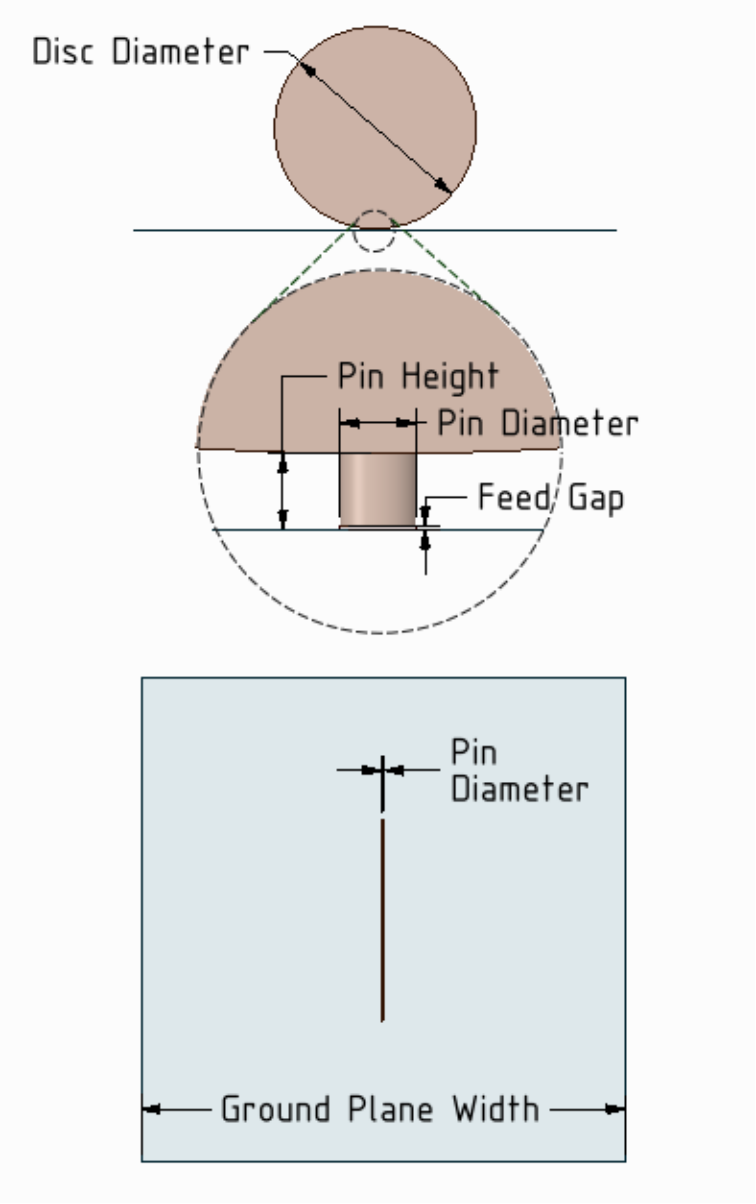
Discone

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Monopole**, with six types:

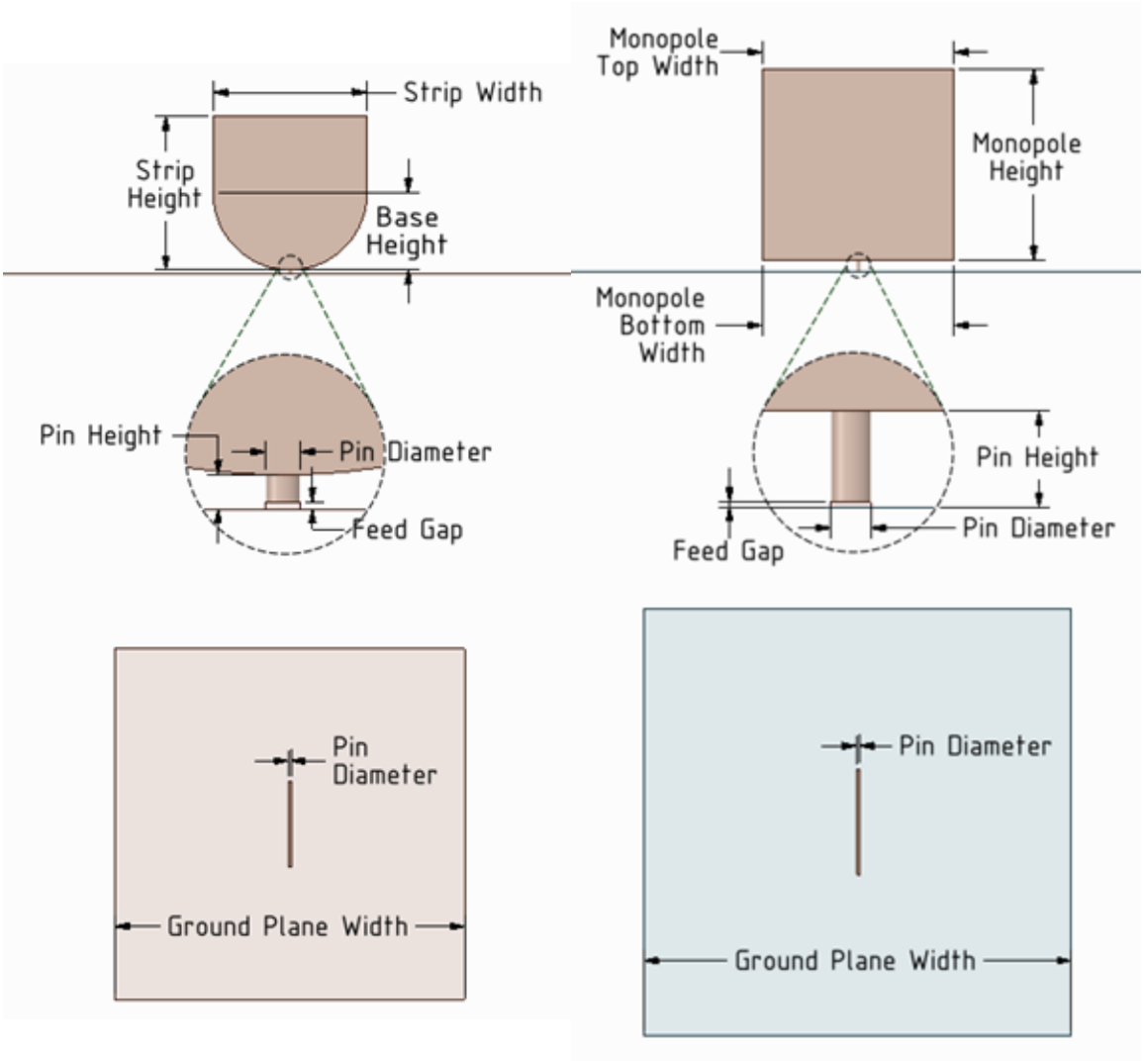


Blade

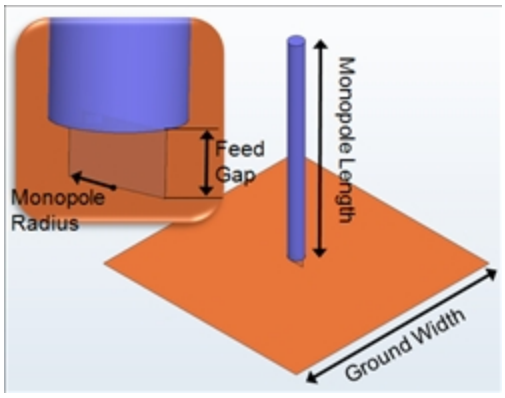


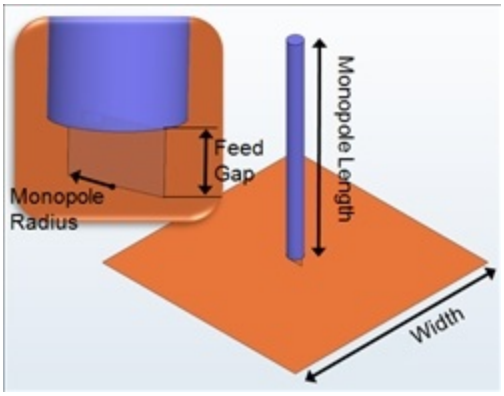
Circular Disk

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Elliptical-Base Strip **Vertical Trapezoidal**

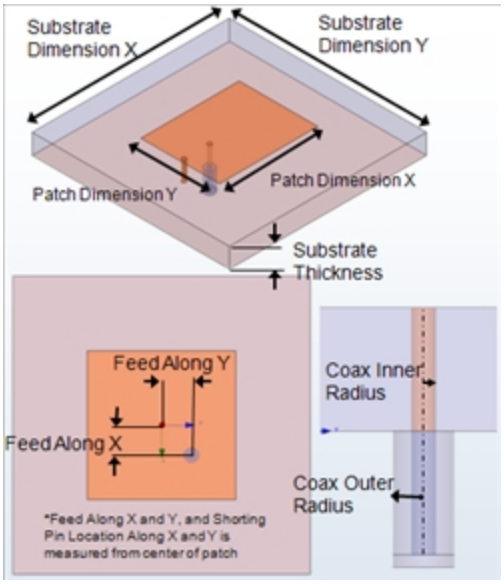




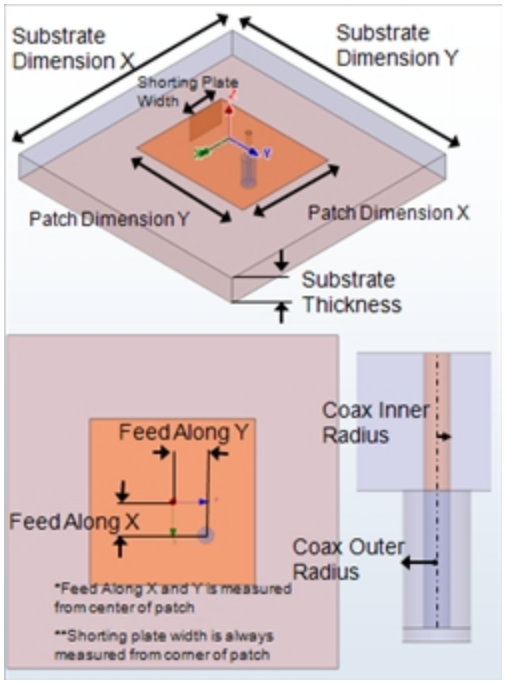
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wire **Infinite Ground**

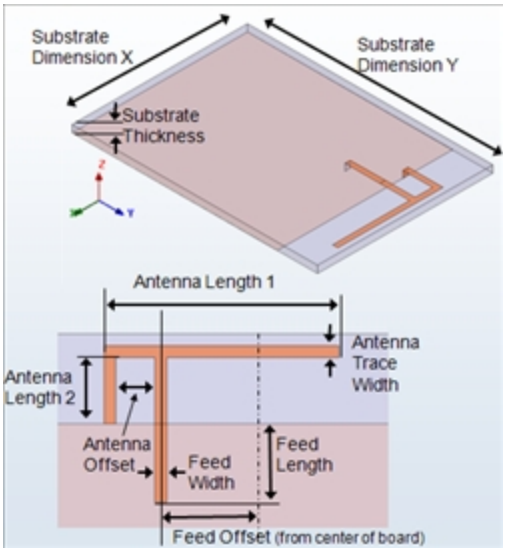
- **PIFA**, with three types:



Shorting Pin



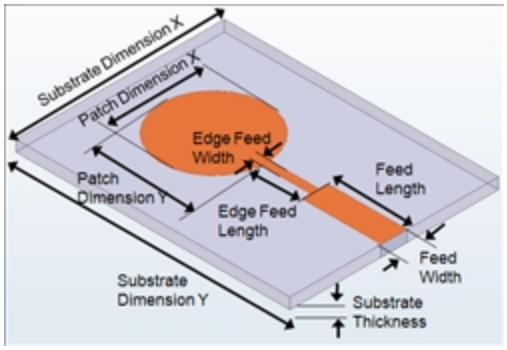
Shorting Plate



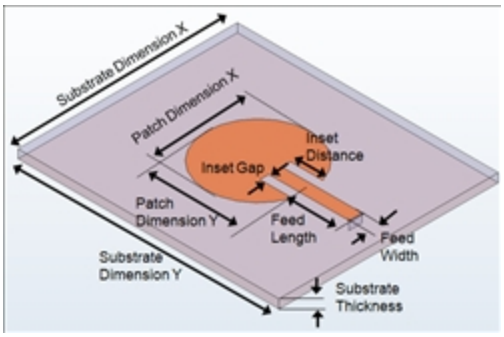
Planar Inverted-F

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

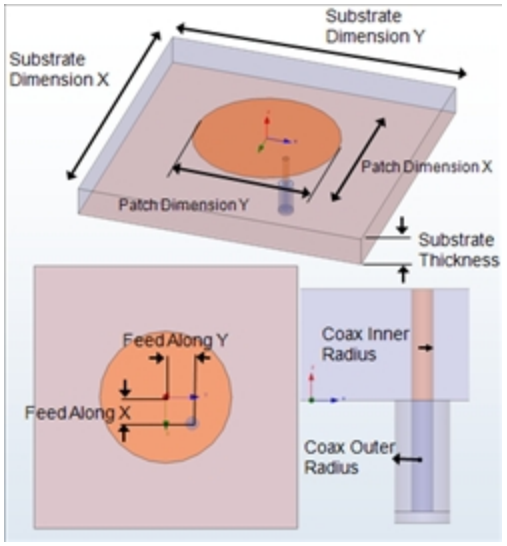
- **Patch Antenna**, with eight types:



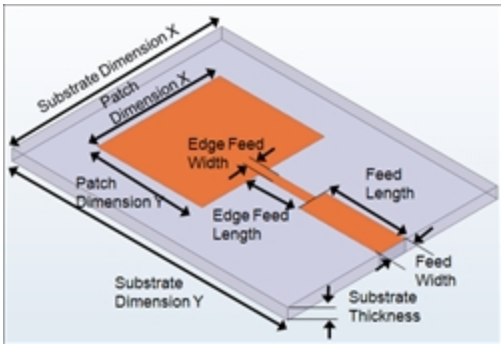
Elliptical-Edge Fed



Elliptical-Inset Fed

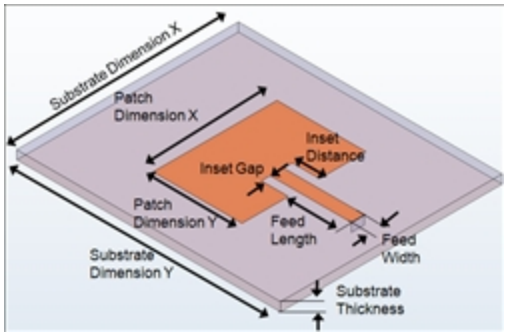


Elliptical-Probe Fed

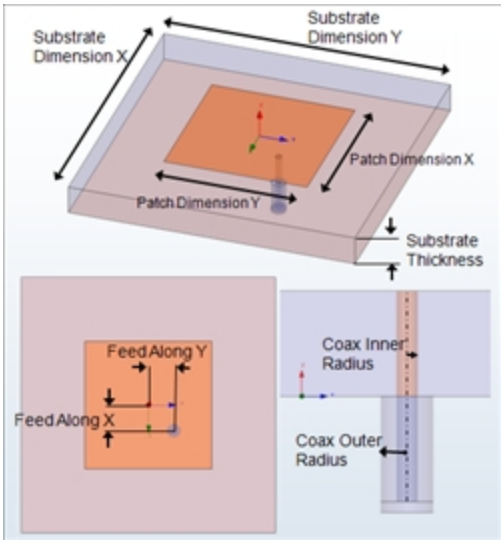


Rectangular-Edge Fed

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

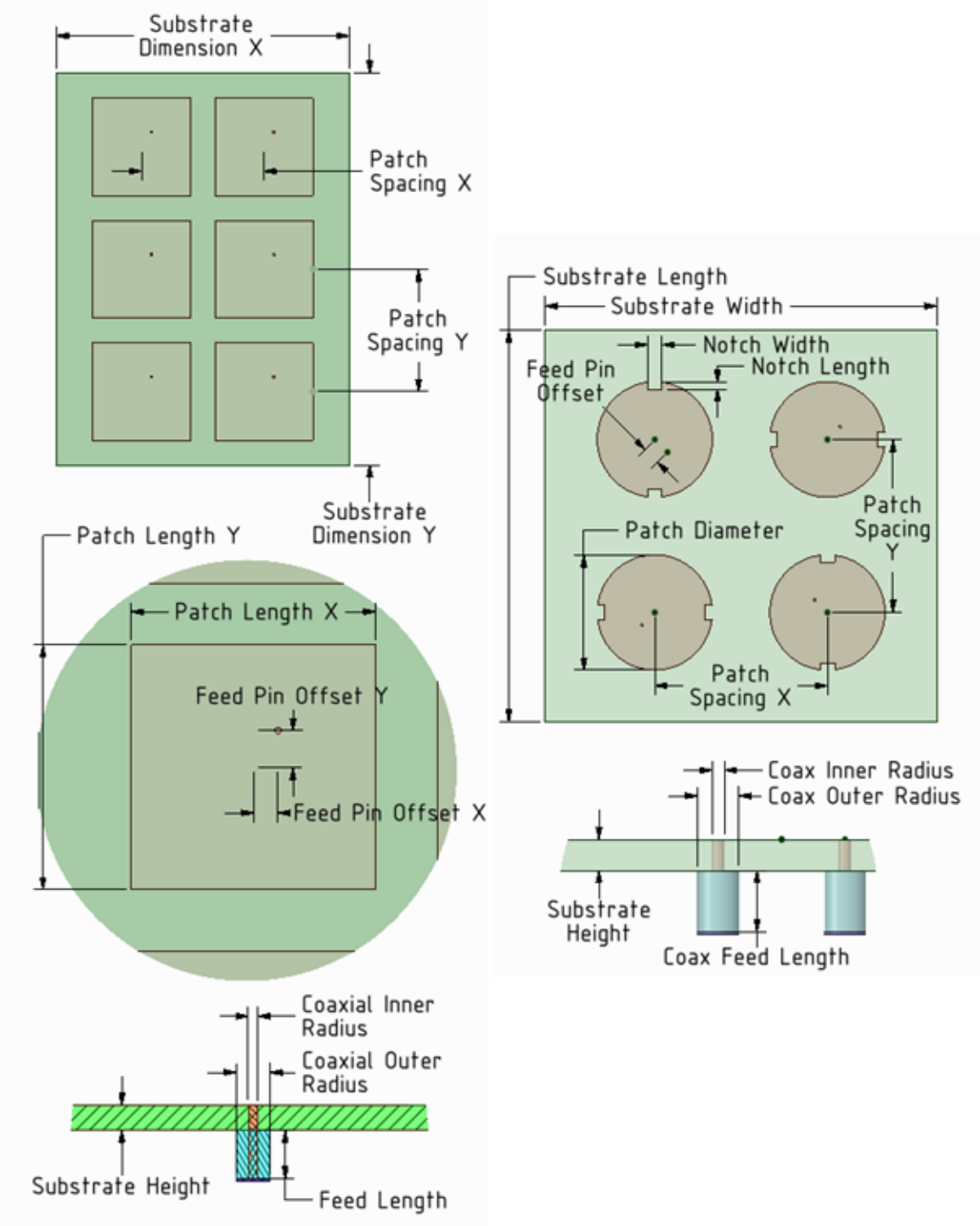


Rectangular-Inset Fed



Rectangular-Probe Fed

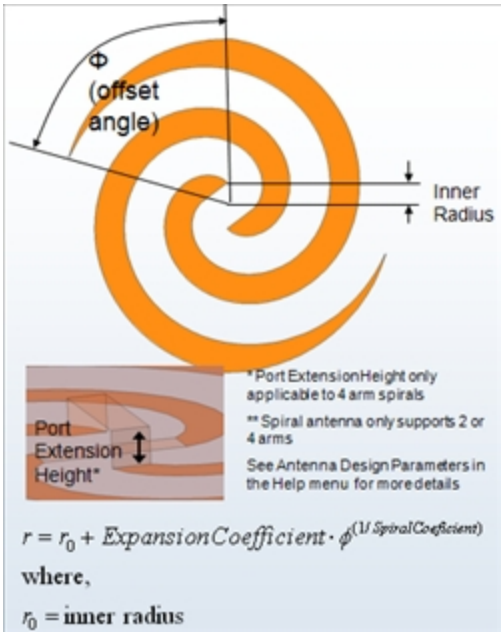
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



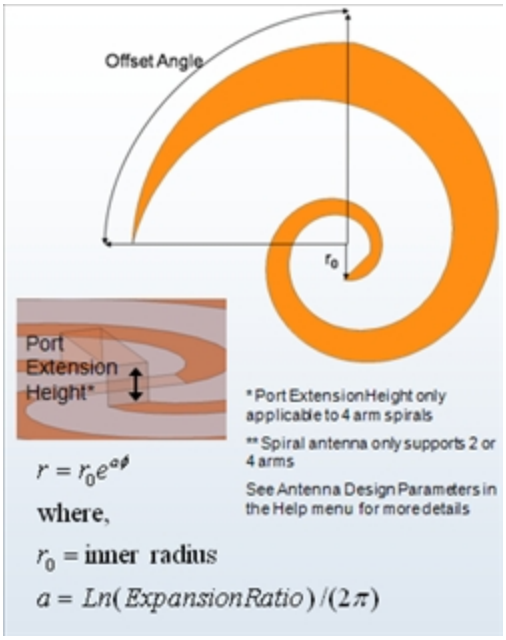
Rectangular-Probe Fed MxN Patch Array Sequentially Rotated 2 by 2 Patch Array

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Planar Spiral**, with six types:

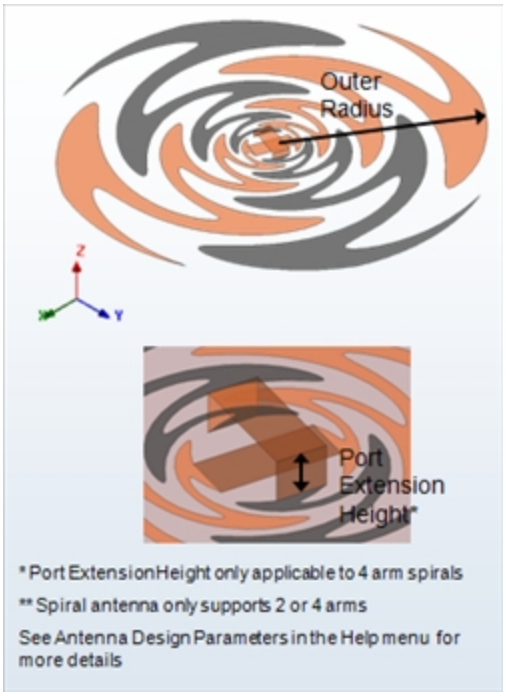


Archimedean

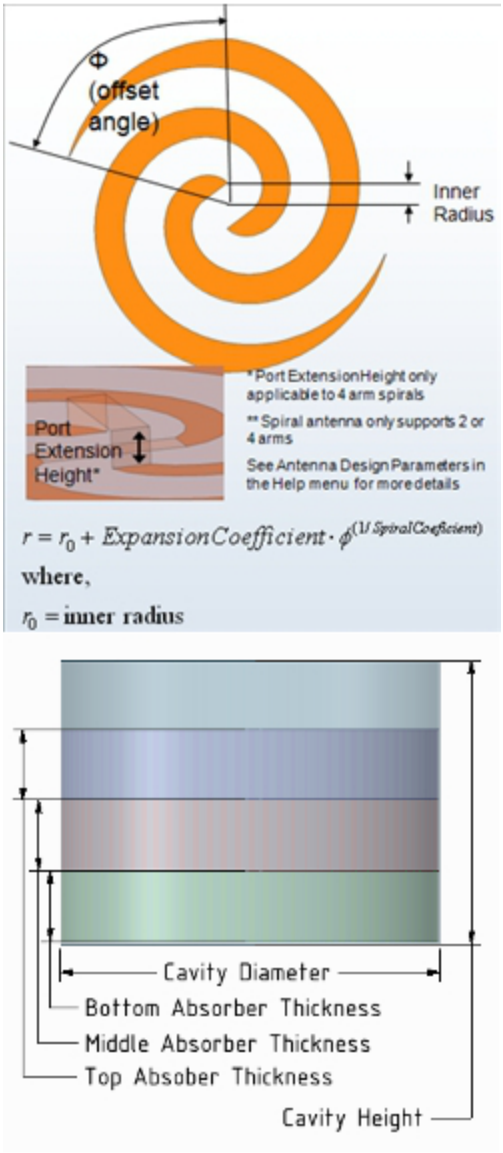


Log

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

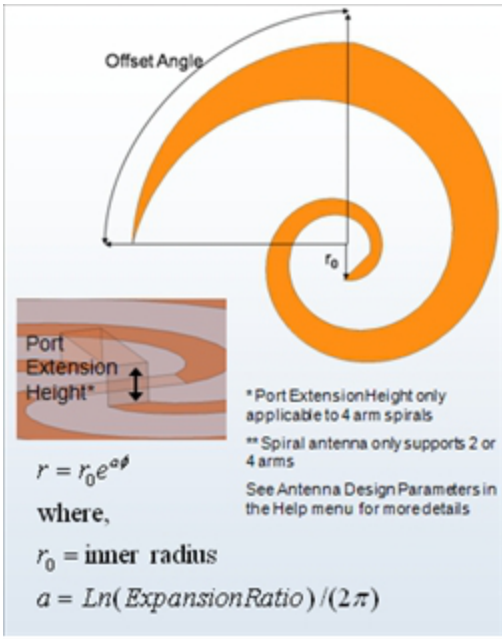


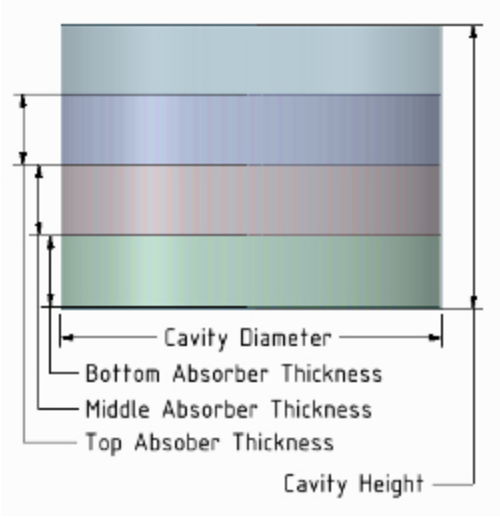
Sinuous



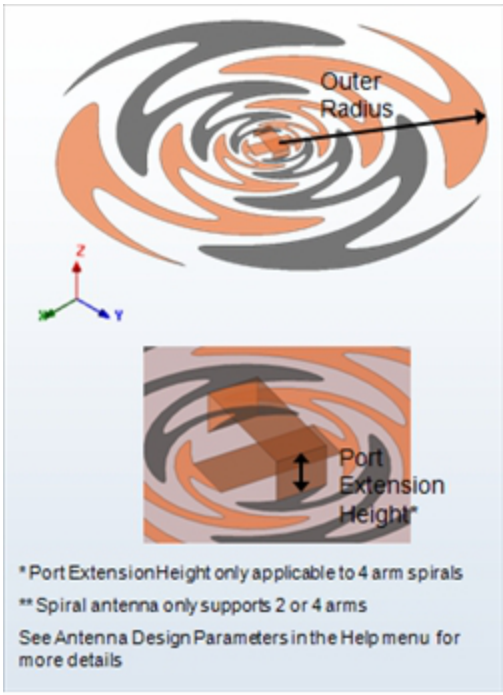
Archimedean with Absorber-Lined Cavity Backing

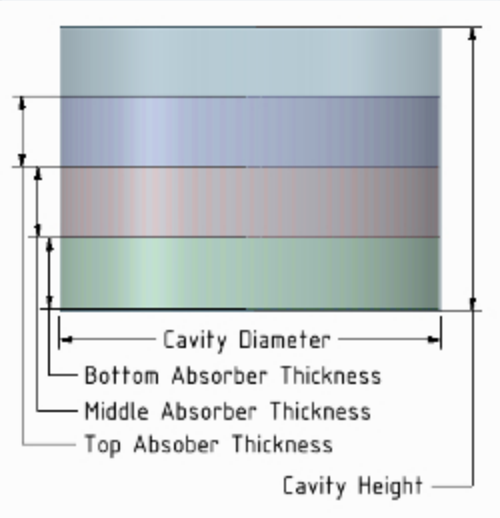
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Log with Absorber-Lined Cavity Backing

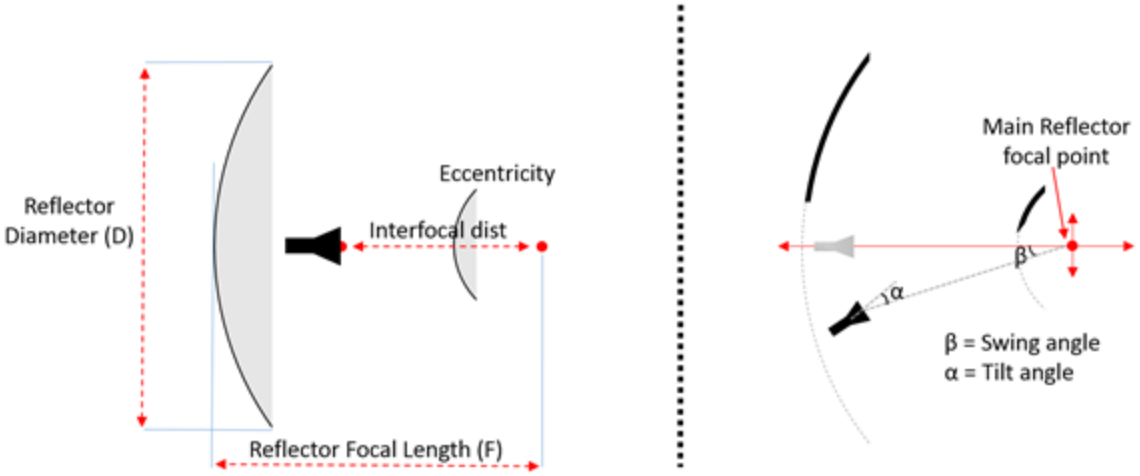




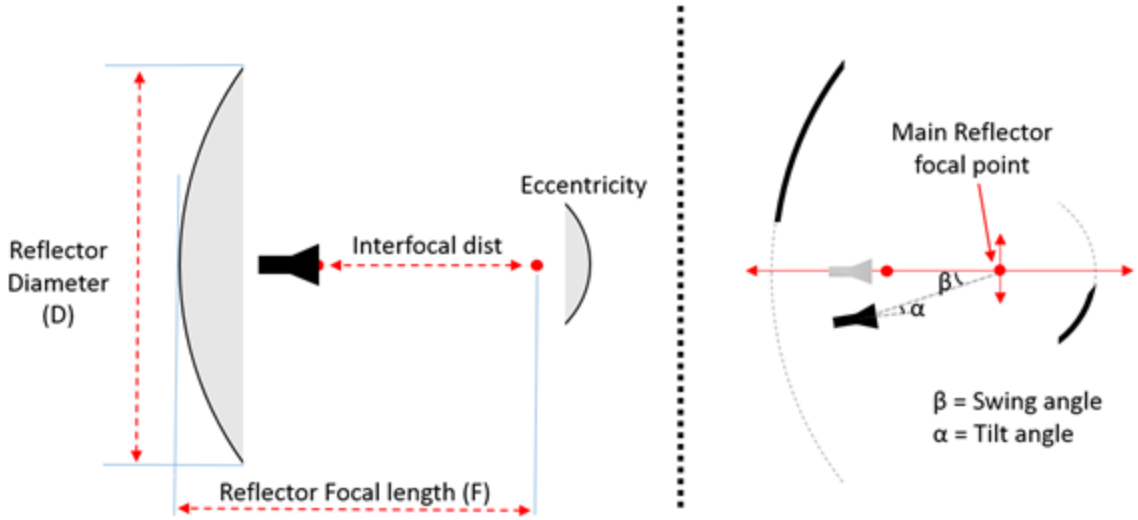
Sinuous with Absorber-Lined Cavity Backing

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Reflector**, with four types:

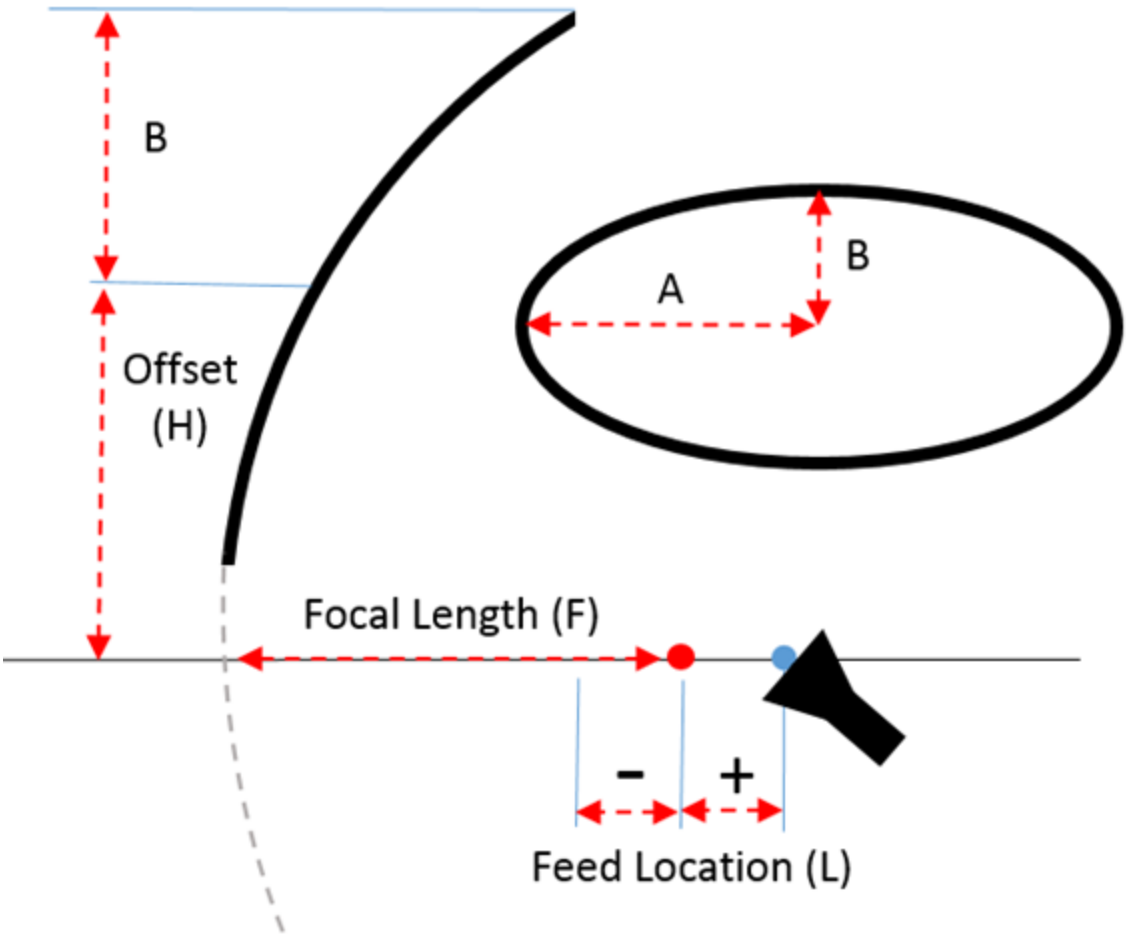


Cassegrain



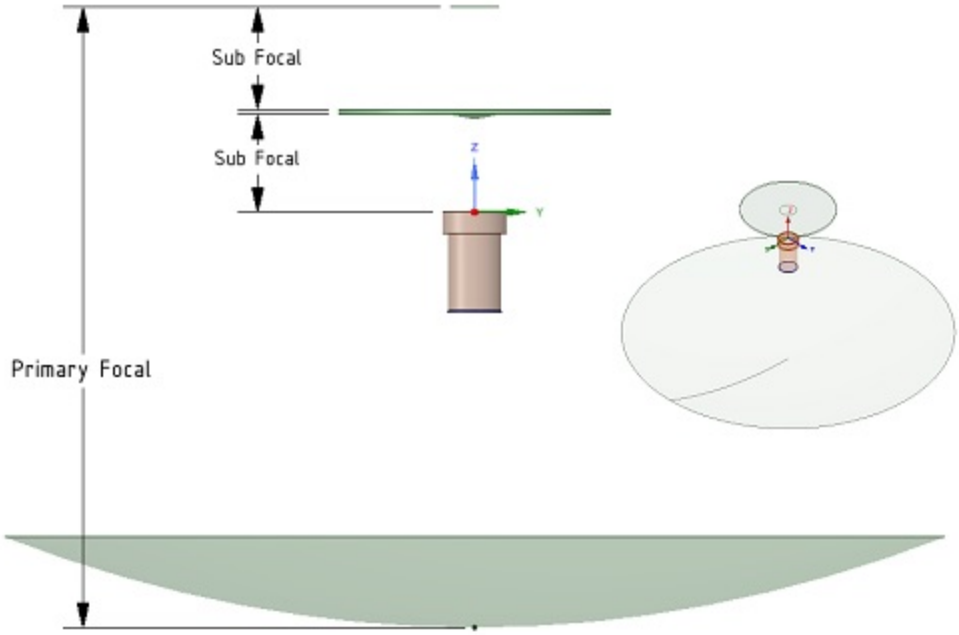
Gregorian

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Parabolic

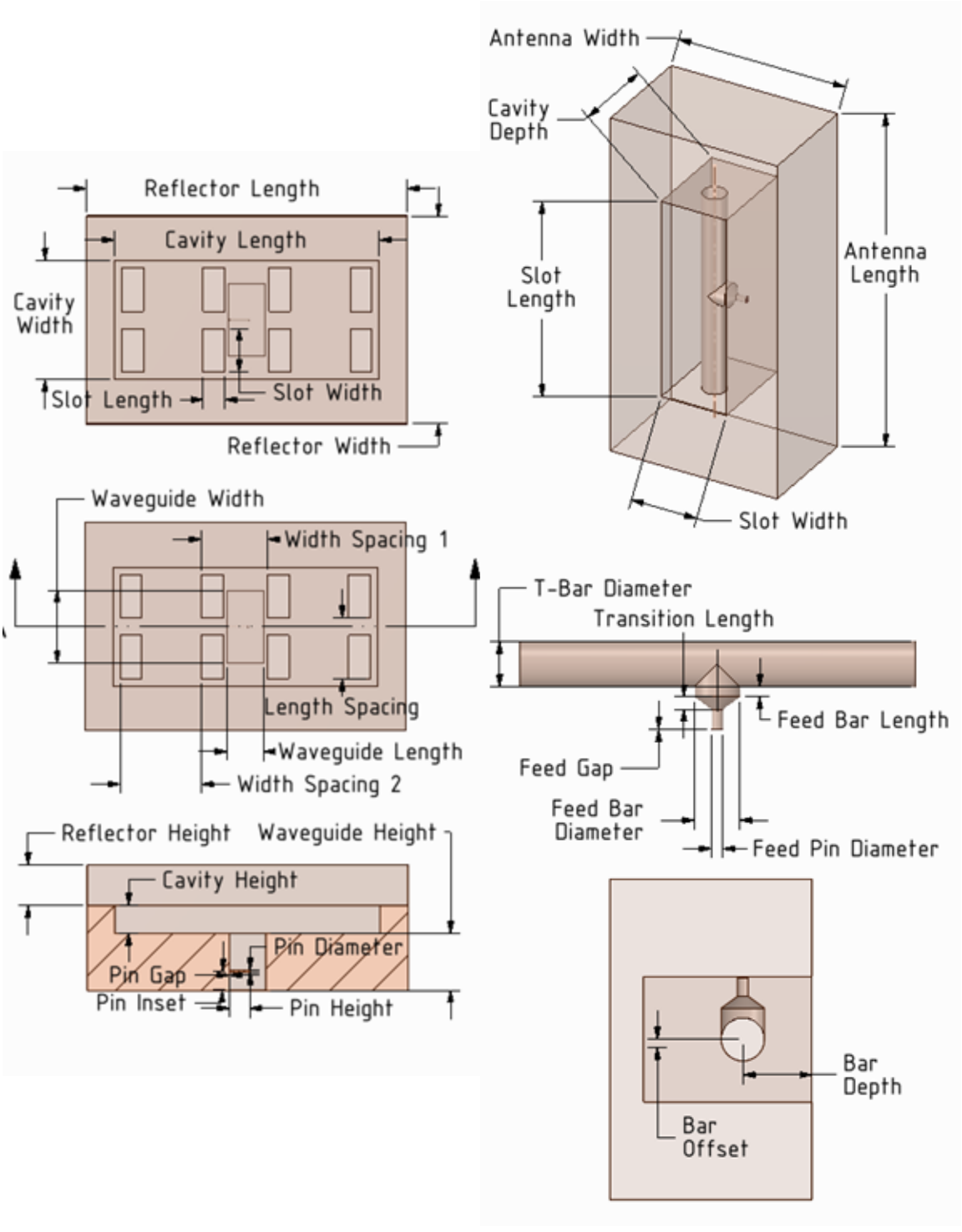
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Splash Plate

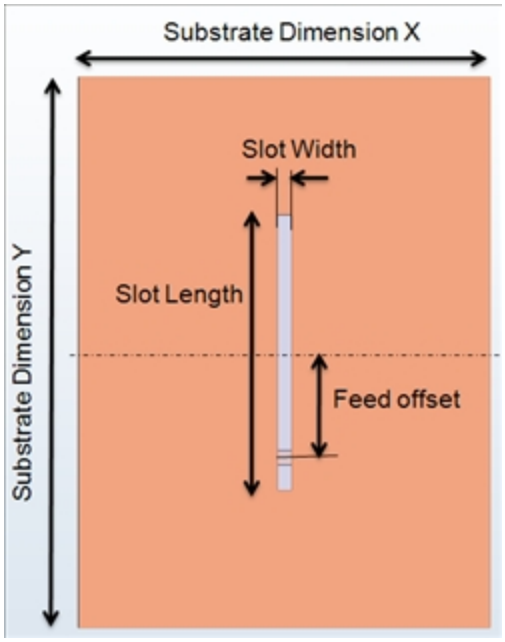
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Slot Antenna**, with four types:

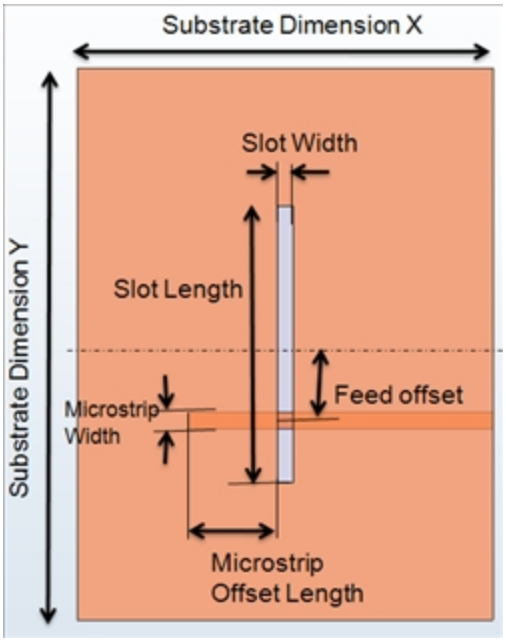


Cavity-backed Slot Antenna Array **Cavity-backed T-bar-fed**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

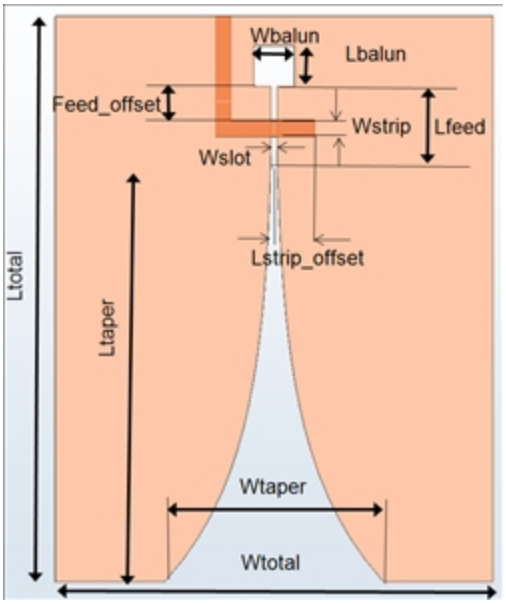


Gap Feed

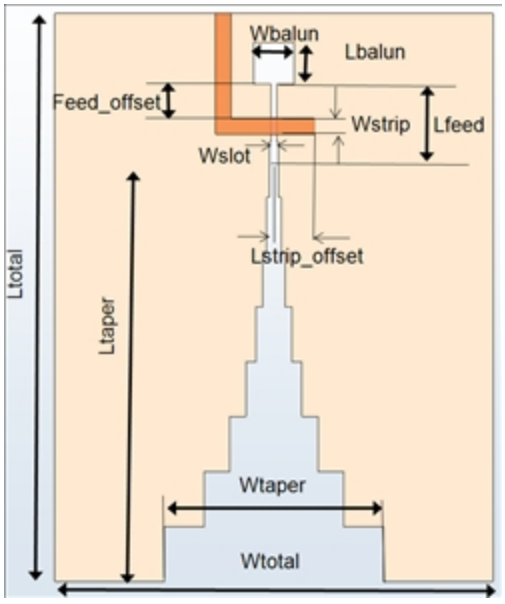


Microstrip Feed

- **Vivaldi Antenna**, with two types.



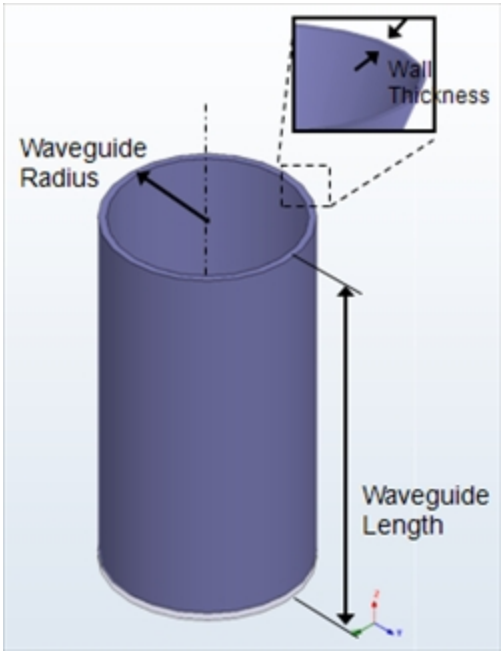
Vivaldi



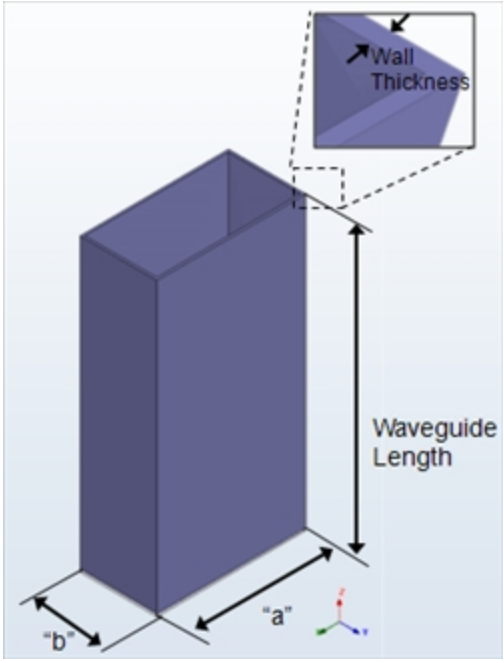
Stepped

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Waveguide**, with three types:

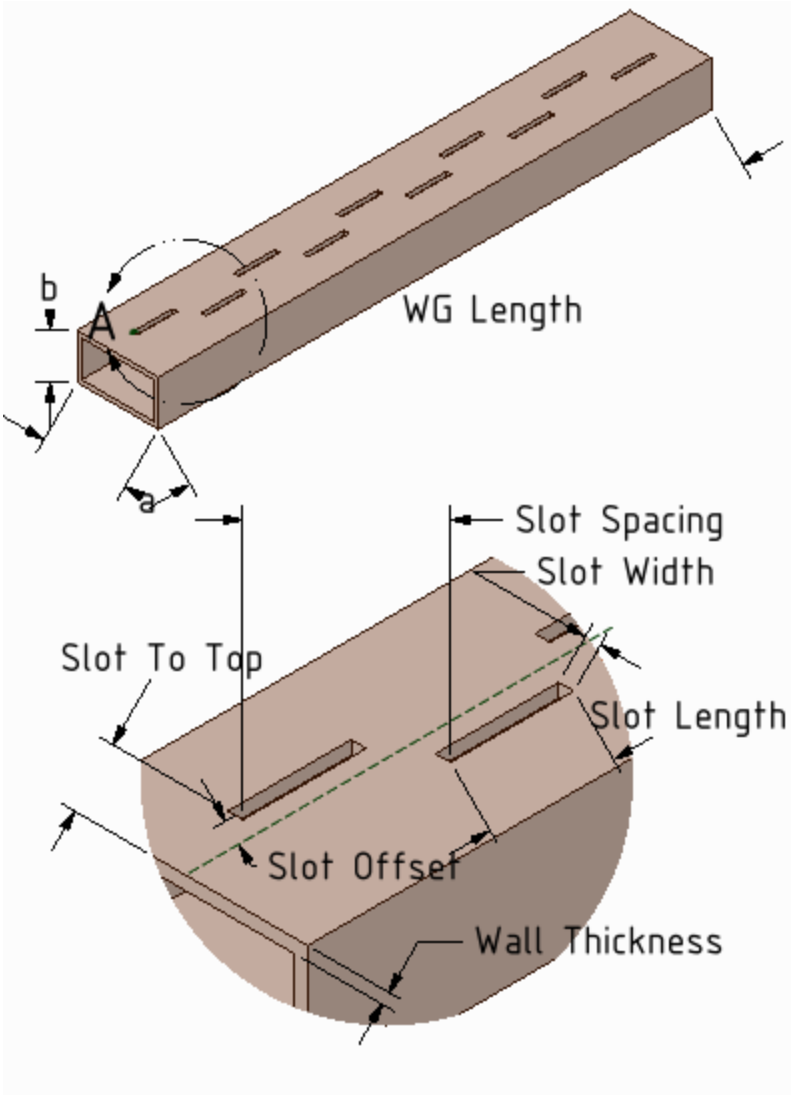


Circular



Rectangular

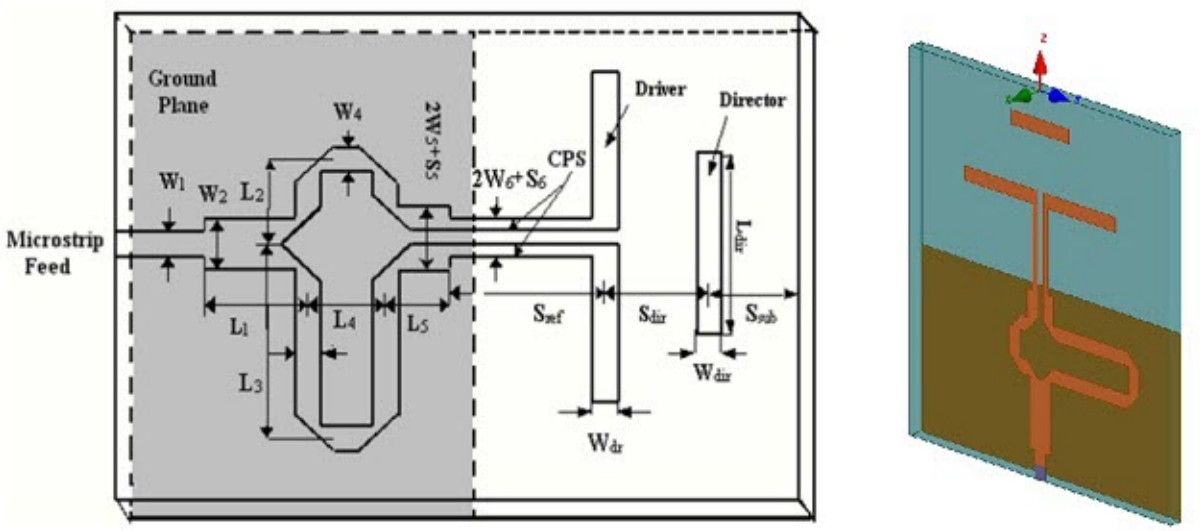
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Rectangular Waveguide Slot Array

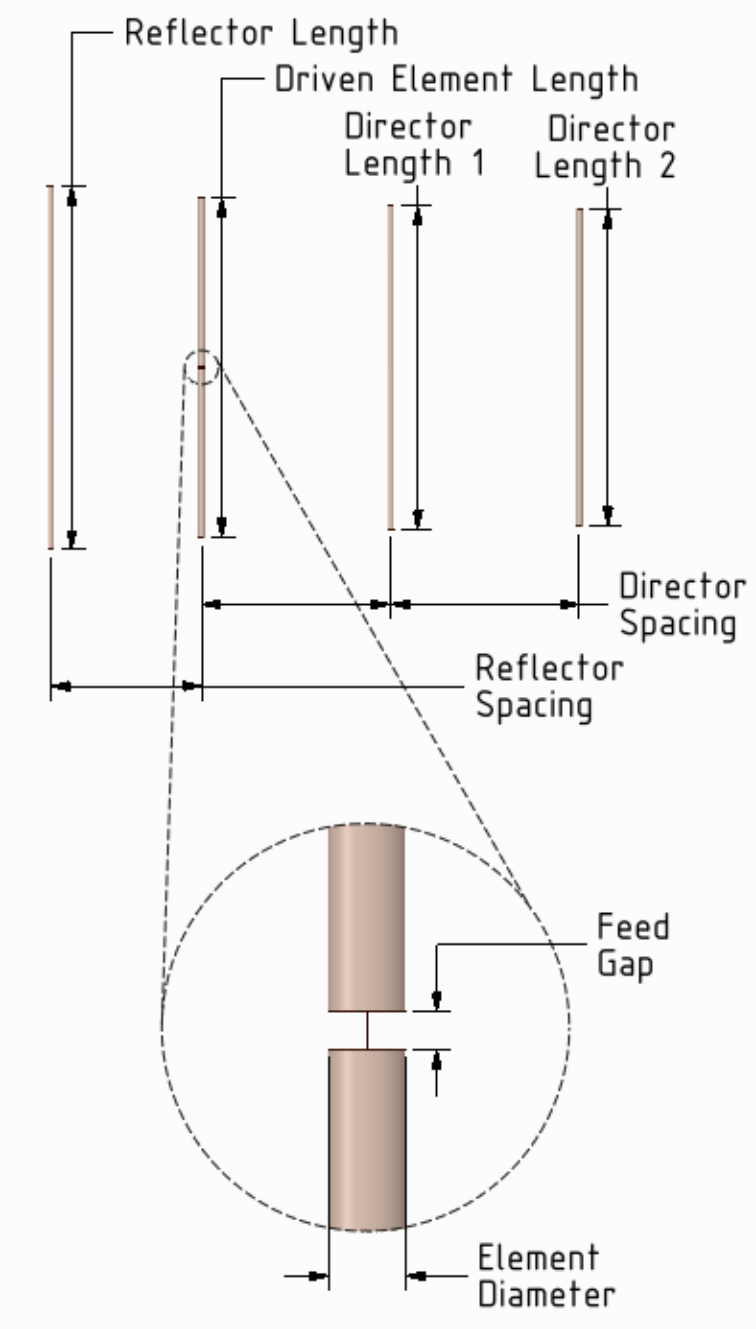
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Yagi-Uda**, with two types:



Quasi-Yagi

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



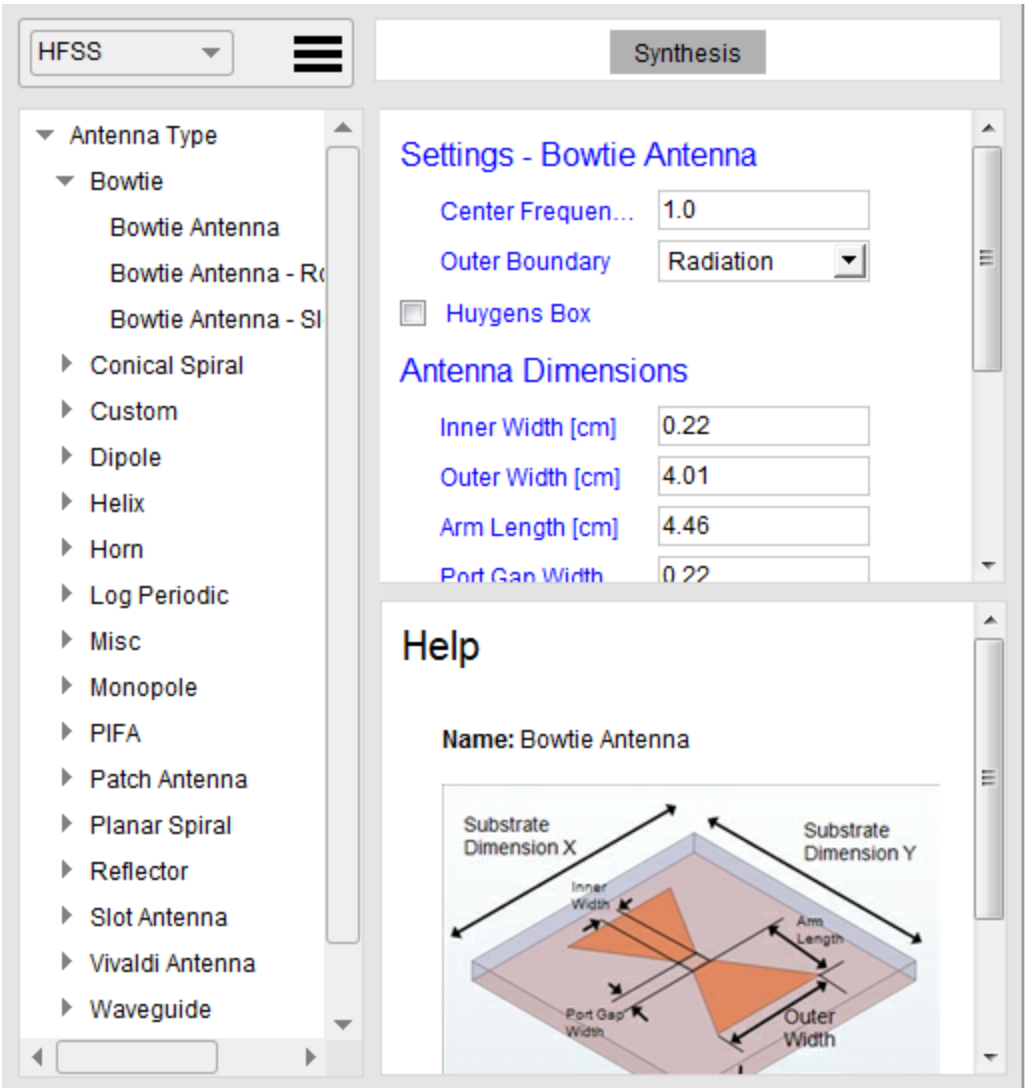
Wire Yagi-Uda

- **PersonalLib**, with any that you have added,

3. When you have selected the antenna, double-click to display its settings, parameters and the help showing the antenna and where the parameters apply.

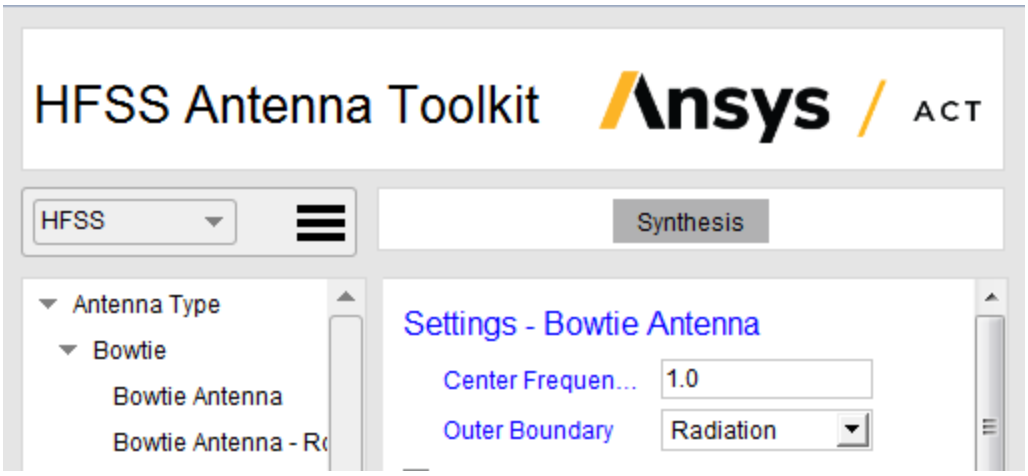
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Settings for each antenna include frequency and various parameters. For HFSS Design types (not HFSS-IE), you can specify the Outer Boundary and check Huygens box. Antenna dimension parameters vary depending on antenna selected. Other dimension parameters for Feed, Slot, Ground plane, Substrate dimensions and such vary somewhat depending on the antenna Type selections. Depending on your window size, you may need to use scroll bars, or resize the window to see all the selections. The Help figure will help you visualize how each parameter applies.



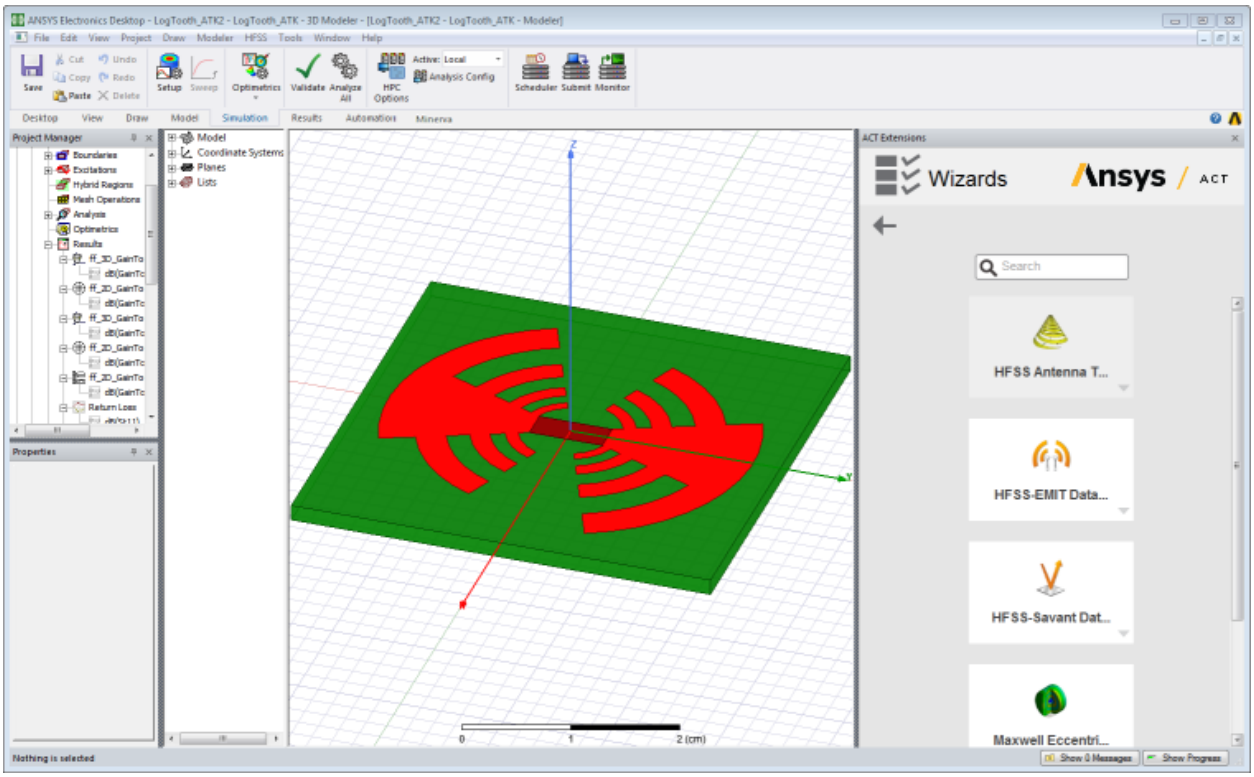
4. If you change the Center Frequency parameter and select **Synthesis**, the antenna dimensions are automatically updated in order to operate at that frequency.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. When you have made your selections click **Finish**.

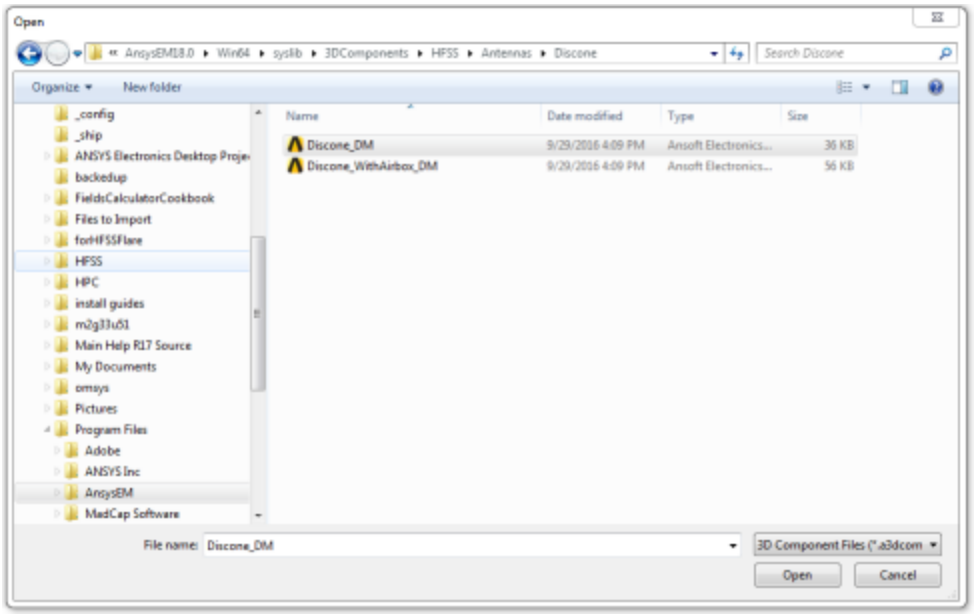
The Antenna Toolkit then creates a project with the antenna design. It displays a message that it is "Executing current step" and some animation indicates progress. As the synthesis completes, the Project tree shows the added Boundaries, Excitations, Hybrid Regions (if specified). It also generates a Solution setup and sweeps specific to the antenna type, as well as a range of predefined 2D and 3D plots, a Port Field Display, and 2D and 3D Radiation Spheres. The Design Properties lists all of the local variables. And the History tree shows the objects created . You can then run analyses and view results.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

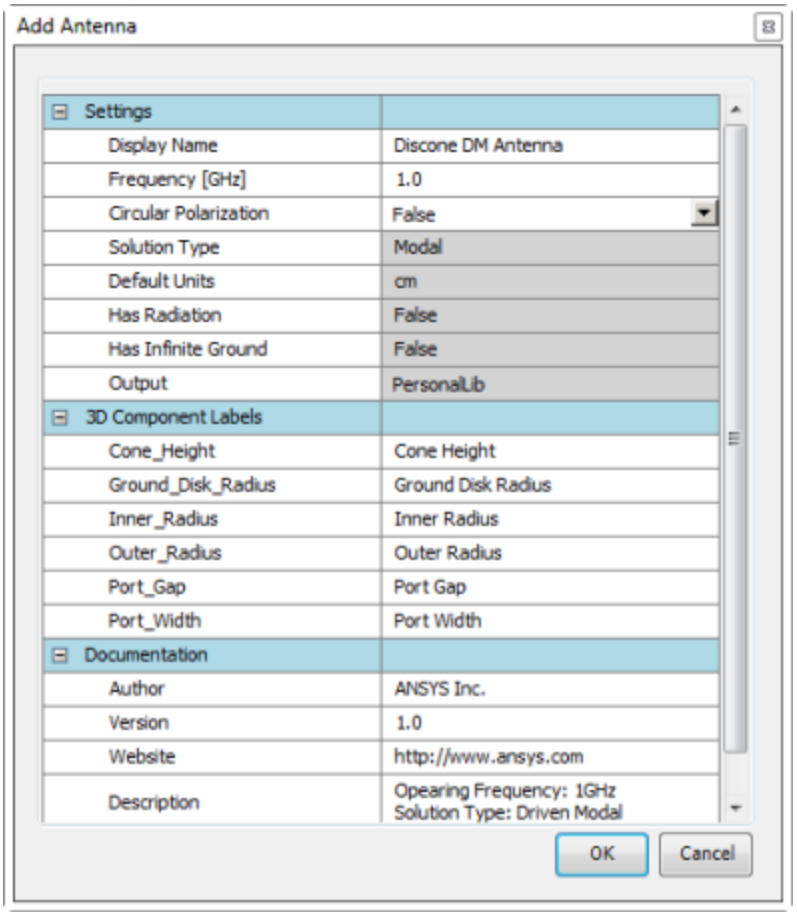
Add Antenna from 3D Component Libraries Button

The Add Antenna button opens a dialog that lets you browse and select from antennas in the 3D Component library (that is, with a *.a3dcom suffix), including example antennas provided and those design you may have created or imported and convert to 3D components.

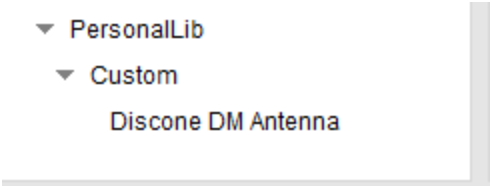


When you select a 3D component file and click Open in the browser, the component is temporarily displayed in the Modeler window, and the Add Antenna dialog displays. Here you can see settings, properties, and documentation, and can click OK to continue.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After you click OK, the antenna is listed under PersonalLib>Custom at the bottom of the Antenna Type list:



One advantage for adding antennas to the Antenna Design Toolkit PersonalLib is that when you then select the antenna from the toolkit library, you get the automated the geometry creation, solution setup, and post-processing reports for over 40 antenna elements.

HFSS-EMA3D Datalink Toolkit

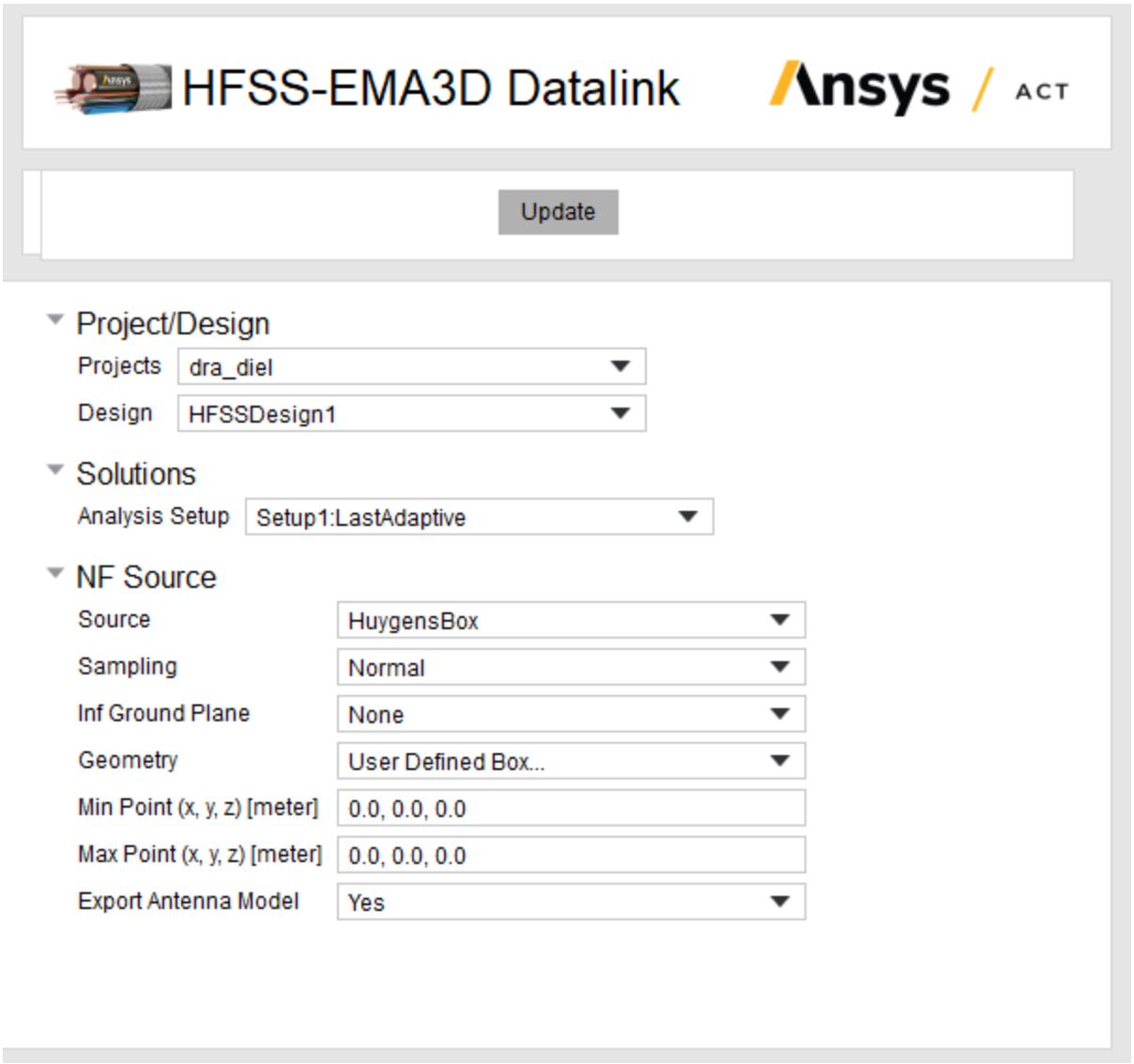
Ansys EMA3D is a simulation tool for installed antenna patterns, including near fields and antenna coupling on electrically large platforms. EMA3D can import antenna data from HFSS to

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

drive a Savant simulation.

The HFSS-EMA3D ACT Toolkit exports antenna geometry, near-field and far-field representation from HFSS. A directory is created so that it can be imported to EMA3D.

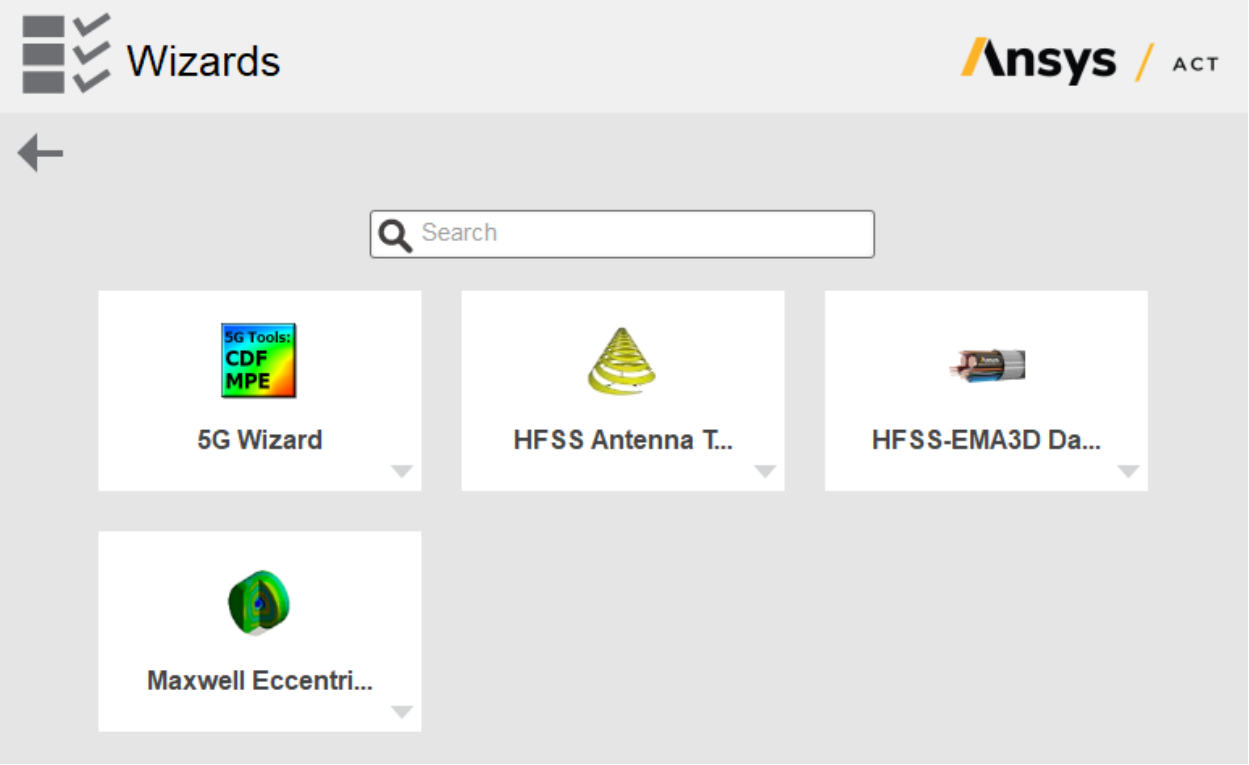
> **Note:**
>
> The **ACT Extensions** window and the design wizards it contains (*5G Wizard, HFSS Antenna Design Toolkit, HFSS-EMA3D Link,* and *Maxwell Eccentricities*) are only available for the Windows version of the Ansys Electronics Desktop software. These items are not available when the software is installed on a Linux platform.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To load the HFSS-EMA3D toolkit:

1. Use the **View>ACT Extensions** command to open the Wizard window for toolkit extensions implemented via Ansys ACT.
2. Click on the Wizards box to load the available toolkits.
3. Load the extension. This can be done with a right-click the newly added square and later selecting the Load Extension option.



4. Return to the main menu by clicking on the back arrow.

To use the HFSS-EMA3D toolkit:

1. Create or open an antenna project with one or more HFSS designs.
2. Analyze the project in HFSS and Save the results.
3. To Export the sample project, open the wizards extension panel by clicking on **View>ACT Extensions** if not opened already.
4. Select the Wizards square from the extension panel. This will open a window with the loaded extensions. Select the **HFSS-EMA3D Datalink** tookit button.
5. The tookit menu will appear in the Wizards panel:

   Project/Design

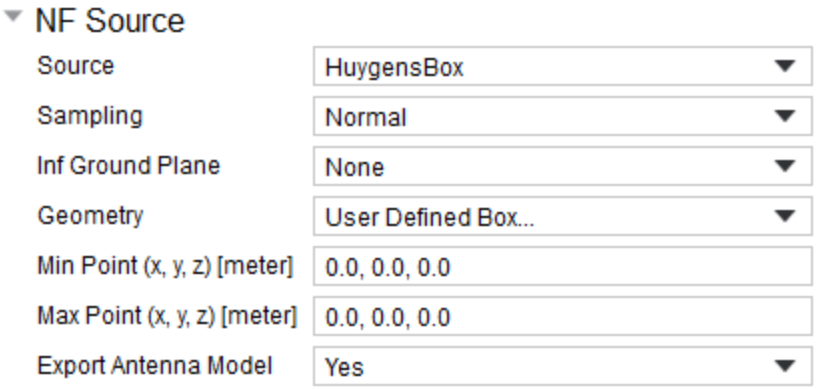
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Project: Select the project. The projects currently open in Desktop are available for selection. The list of projects can be updated by clicking the **Update** button.

Select the design. Choose from the designs within the selected project.

Solutions: Selects the analysis setup to export from the different analysis setups available in the selected Project/Design. Only one analysis can be exported at a time.

NF Source:



This section defines the geometry for the exported near-field data necessary to reconstruct the antenna in EMA3D. The toolkit exports the E and H fields on radiating surfaces or the faces of the Huygens Box based on user selection.

Source: Defines the source surfaces where E and H fields can be exported. There are two available options: **RadiationSurfaces** and **HuygensBox**. When you select **RadiationSurfaces**, HFSS exports E and H fields on Radiation boundaries, PML and FE-BI. When you select **HuygensBox**, the box should be placed close to the antenna, an offset of a λ/10 from the maximum dimensions of the antenna is recommended but larger offsets will work as well. The sample fields should be at least 8 points per wavelength

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ppw) on each face of the box. The HuygensBox option is not applicable when the design has a finite array.

If you choose to export on **Radiation Surfaces**, the only other options for NF Source are Sampling and whether to Export Antenna Model. Geometry and Infinite ground plane selection do not appear. If you have defined an Infinite ground plane, it will be directly taken from the boundary definition.

HFSS provides the actual geometry for one cell for array problem, so export antenna model option will only export the geometry for one cell. HFSS has the mesh exported for radiating surface which could gives an overall outline of the array but not details about every single cell. The antenna CAD model in EMA3D is used for display and placement purpose, to the current on radiation surface achieves similar functionality.

If you select **Huygens Box** the toolkit exports the E and H fields on the faces of the Huygens Box described in this section. The toolkit offers a menu with a calculated Region or **User Defined Box...**. The Huygens Box should be placed close to the antenna, an offset of a λ/10 from the maximum dimensions of the antenna is recommended but larger offsets will work as well. The sample fields should be at least 8 points per wavelength (ppw) on each face of the box.

Sampling: Defines the configuration type for the number of samples/points per wavelength (ppw) and Field Sampling Density for faces from selected Source when applicable. The menu has three options: **Quick**, **Normal** and **Custom**. **Quick** sets the ppw to 8 and the field sample density to 4 respectively; **Normal** sets it to 16 and 8 respectively and **Custom** unveils two new property panels so the user can customize the value of the properties. Only Field Sampling Density is applicable when you select RadiationSurfaces as Source.

NF Sampling: Defines the number of samples/points per wavelength (ppw) for each face of the box. A value of at least 8 is required to produce a faithful result later in EMA3D. This

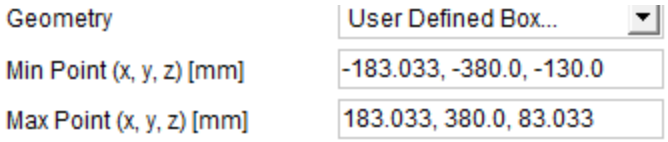
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

property is only available when **Sampling** is set to **Custom** and **Source** is set to **HuygensBox**

Field Sampling Density: Defines the number of samples that will be set by EMA3D in the Antenna's property panel. A value of at least half as much as NF Sampling is required to produce a faithful result later in EMA3D. Visible only when you select **Custom**.

Inf Ground Plane: Exports the ground plane information for the selected face. When loaded into EMA3D, the antenna will present the ground plane information introduced in this toolkit. This is practical to avoid the integration of the EH field in the selected face which eases the computation of the current sources. This property is set automatically if the user picks a specific Geometry. You can change this value at any time.

Geometry: The HFSS-EMA3D Datalink Toolkit defines a region likely to represent the best space to define the Huygens Box. The unavailable Min Point and Max Point values apply to the Region in the Global Coordinate System. If you want to use a customized setting, then you can select the option **User Defined Box ...** to enable the Min and Max Point fields, starting with the dimensions of the default calculated region.



Min Point (x,y,z): The extension of the Huygen Box can be defined through two points with the maximum and minimum extensions of the box respectively. In this selector, the point with the minimum values of the box is entered. The value of the points is entered using 3 floating point numbers separated by a comma ','. The units of the numbers are shown in the panel descriptor. This property is set automatically if you pick a specific Geometry. You can change this value using the option User Defined Box ... in the Geometry selector.

Max Point (x,y,z): Defines the point with the maximum values of the box. The value of the points is entered using 3 floating point numbers separated by a comma ','. The units of the numbers are shown in the panel descriptor. This property is set automatically if the user picks an specific Geometry. You can change this value using the option User Defined Box ... in the Geometry selector.

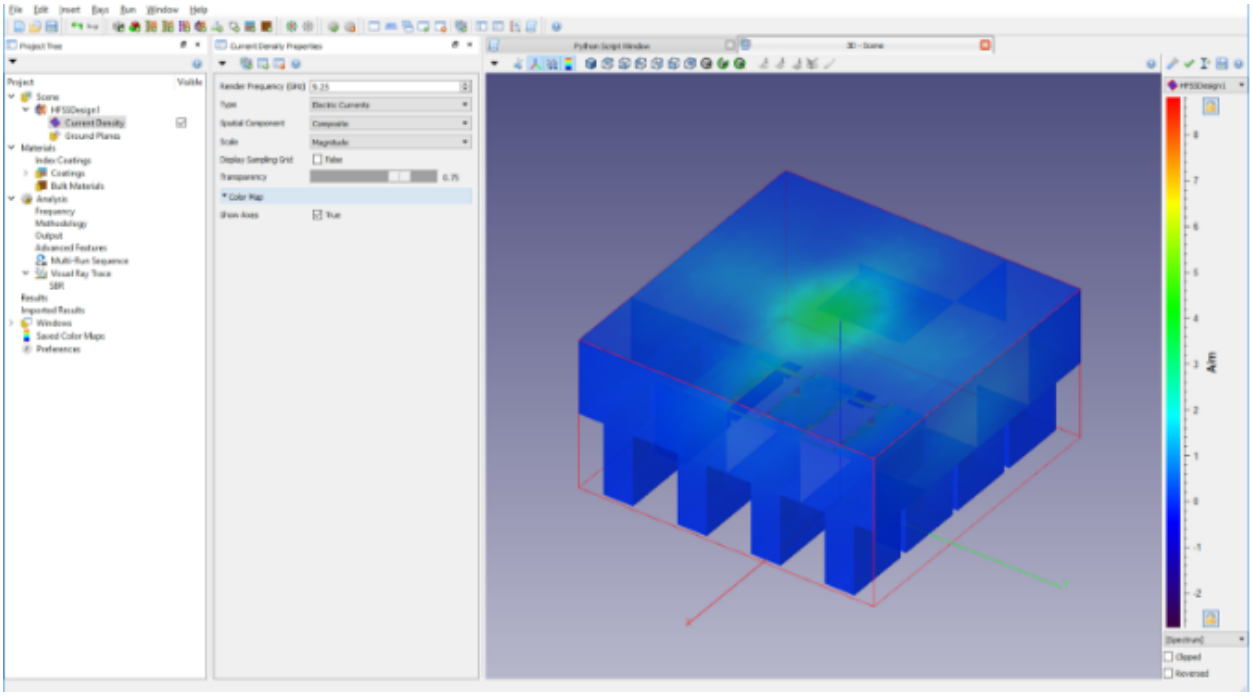
Export Antenna Model: Specify whether to Export the Antenna Model to EMA3D.

Antenna Pattern Source: Infinite Sphere

Far-Field Setup: Selects the setup that will be used to export the far-field data from the toolkit. The options are populated from the specific project and design selected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Set or change the Geometry value. Observe how the Huygens Box extensions change with the new geometry.
7. The tookit analyzes the different geometries present in the project. In most cases it is more convenient to define a box using the HFSS aids rather than writing them on entry.
8. Select **Finish**.
9. The export process begins when you press **Finish**. Once the process finishes, the HFSS-EMA3D datalink toolkit closes itself.
10. The exported project can be found on the same folder where the HFSS project is stored. This path will be used in the following step.



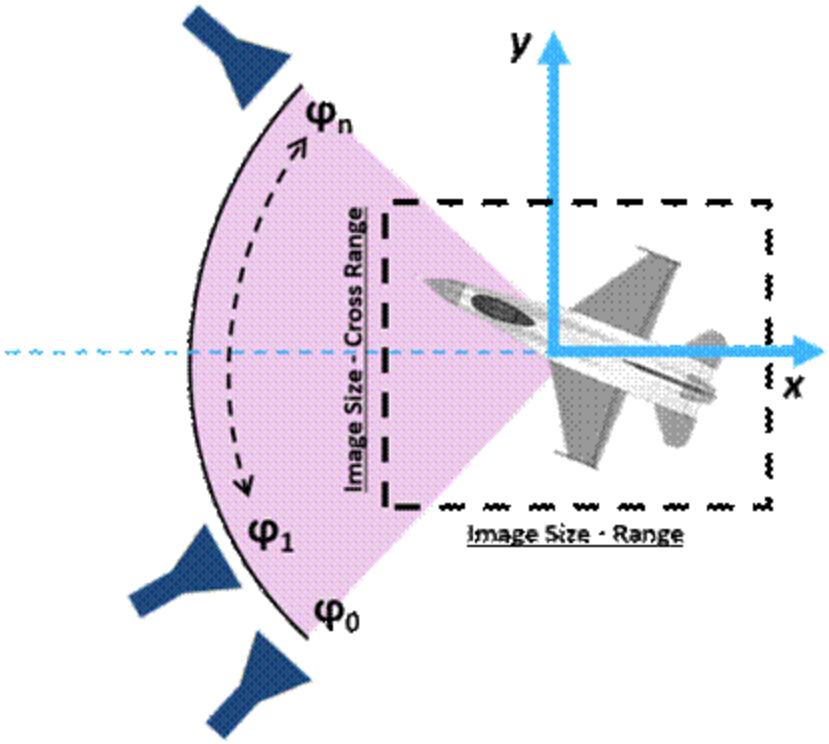
Note: If export data has been generated for multiple designs from the same project, EMA3D will provide a dialog allowing the design of interest to be selected.

HFSS RADARpre ACT Toolkit

The RADARpre ACT generates the appropriate frequency sweep, plane wave incident angles and data reports used for post processing of Range Profiles, 2D Range Profiles (Waterfall) and 2D ISAR images. Specify the type of report and the desired image/range size and resolution, from these inputs the appropriate setup will be created. The results generated from this ACT can be used along with the RADARpost ACT to generate the range profiles and ISAR images.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Down range: Defined as along the X-direction
- Cross range: Defined as along the Y-direction
- Images and Ranges are always assumed to be at Theta = 90deg, or on the XY plane.
- Look angle is defined as a rotation of Phi, starting from the Phi=0deg aligned with the X-axis.



The HFSS RADARpre toolkit is implemented in Ansys ACT.

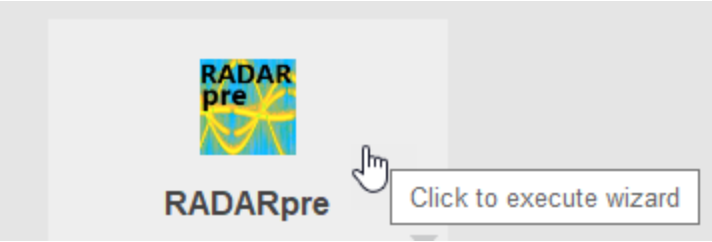
Before launching the wizard, load an appropriate HFSS SBR+ design with one or more antennas.

To launch the Wizard:

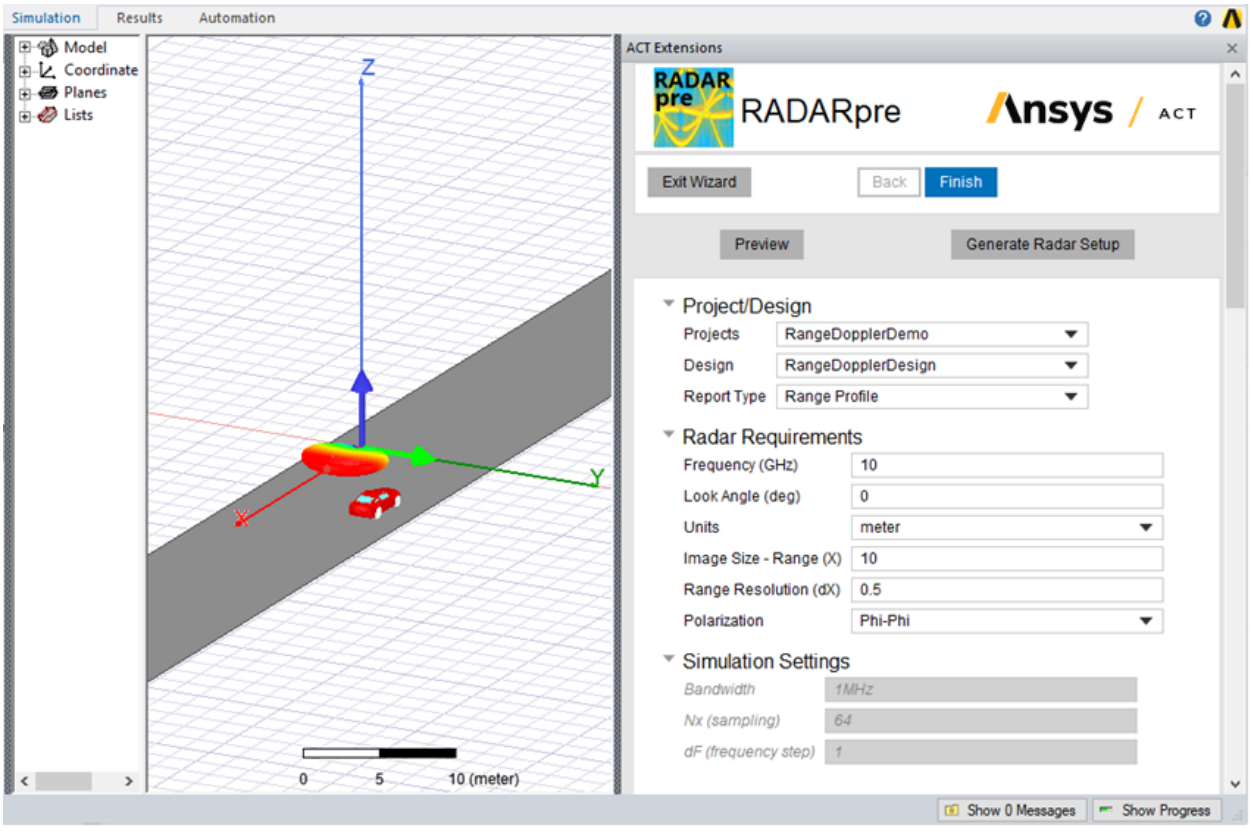
1. Click View>Extensions command to open the Extensions window.
2. Click Wizard to open the ACT Start Page. If you have not loaded the RadarPre extension, use the Manage Extensions feature to do so.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click Launch Wizards to the show the active Radarpre toolkit, and click to execute wizard.



The RADARpre wizard displays.

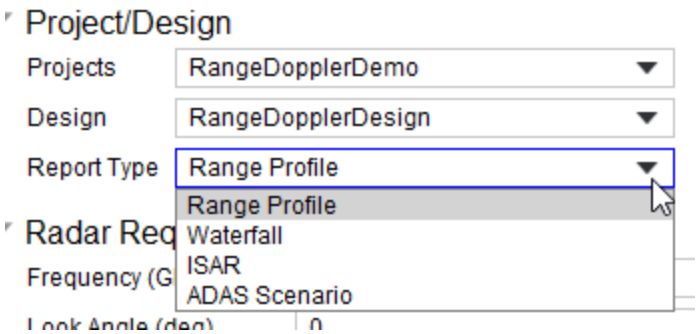


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using the RADARpre Wizard

The RADARpre Wizard contains areas for selecting the Project/Design and Radar Requirements.

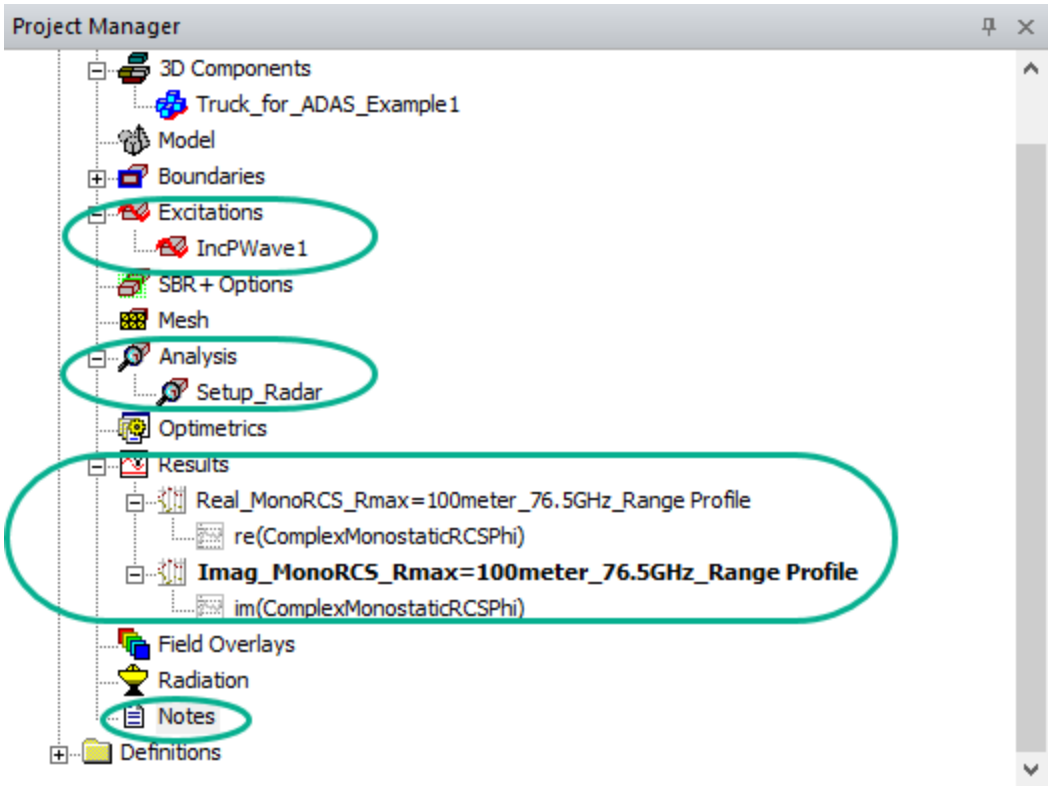
1. If there is more than one suitable pre-loaded design, select the Project/Design.
2. You then select the Report type as Range Profile, Waterfall, ISAR or ADAS Scenario.



3. Specify the Radar Requirements, including:
   Frequency
   Look Angle (deg)
   Units
   Image Size -Range (X0
   Range Resolution (dX)
   Polarization
4. Click **Generate Radar Setup**.

The RADARpre creates the Excitation, an Analysis Setup called Setup_Radar and Reports for Monostatic RSC for Real and Imaginary values.

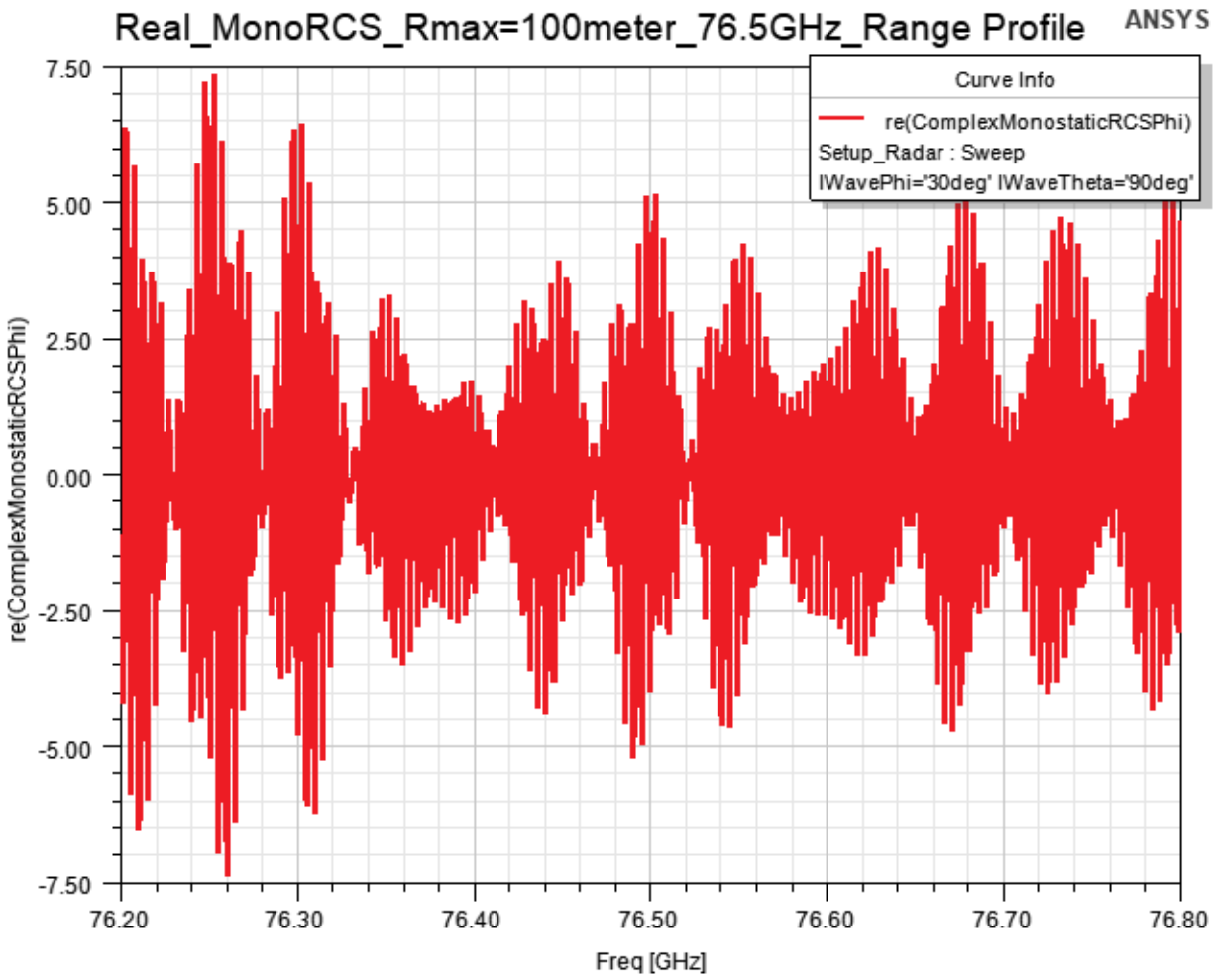
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can then run **Analyze** to generate a solution and populate the reports for the current settings.

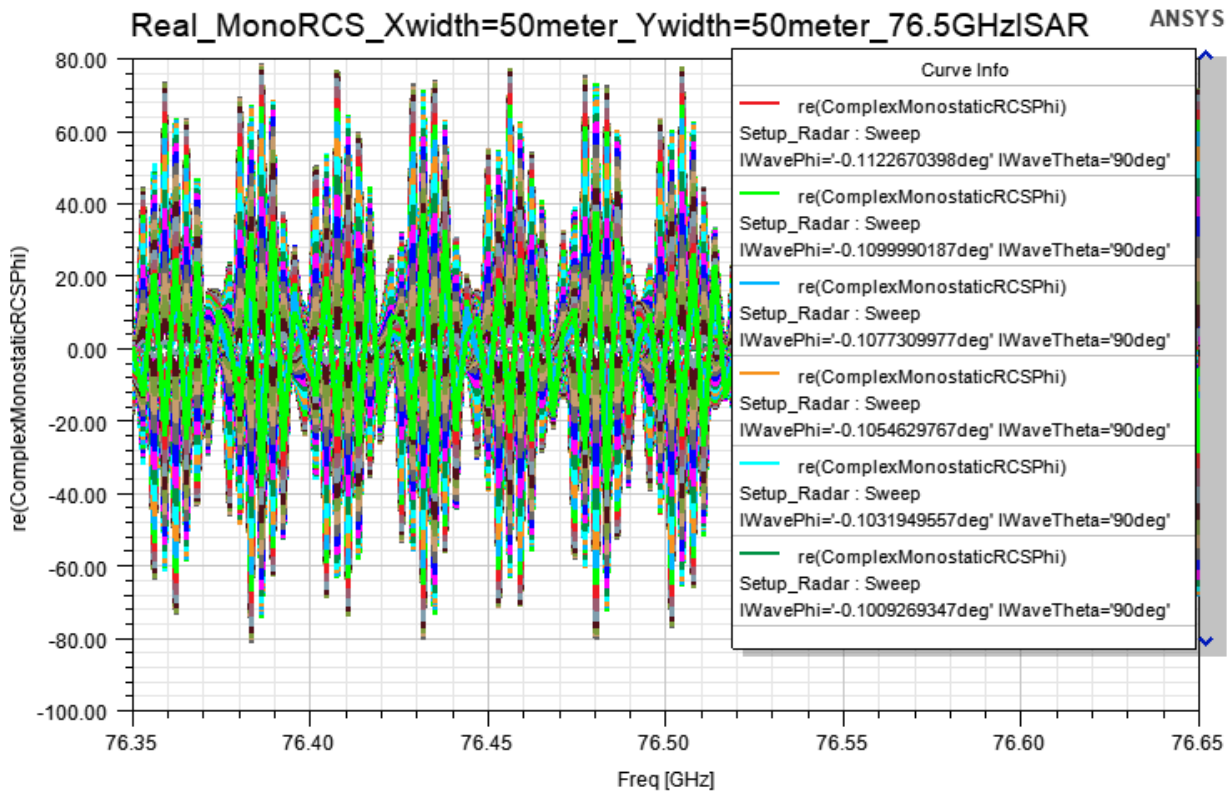
For example, here an example report for the Real_MonoRCS_Rmax=100meter_76.5GHz_ Range Profile.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following is an example of an ISAR report for Real_MonoRCS_ Xwidth=50meter+Ywidth=50memter_76.5GHzISAR.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can use the RADARpre ACT toolkit to generate the correct report formats for the RADARpost toolkit.

HFSS RADARpost ACT Toolkit

The RADARpost ACT extension post processes HFSS simulation results to generate Range Profiles, 2D Range Profiles (Waterfall Plots), ISAR Images and Doppler-Range plots. Select the type of report and corresponding post processing parameters (windowing, dB/mag, Velocity Ambiguity), and the report names used as the source for the data. The format of the data reports can automatically be generated using the RADARpre ACT extension but does not require the use of the RADARpre extension as long as the data is formatted correctly.

Data can be post processed using either S-parameters or RCS results from a simulation, with the exception that waterfall plots are only valid for plane wave excitations (RCS results). The data should be organized as a separate plot for real and imaginary data, with frequency as x-axis (or row if data table). For reports requiring multiple plane wave incident angles (such as waterfall and ISAR), each incident angle should be a different trace (or column of data table). If multiple look angles are present, the 1D Range profiles will require an additional user input to specify the look angle over which to post processing the range profile. Users can also select a windowing function to be applied to the post processed data. An up sampling of 4x is

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

automatically applied to all data during post processing. All data and post processing is assumed to be done in the XY plane.

The HFSS RADARpost toolkit is implemented in Ansys ACT.

Before launching the wizard, load an appropriate HFSS SBR+ design with one or more antennas.

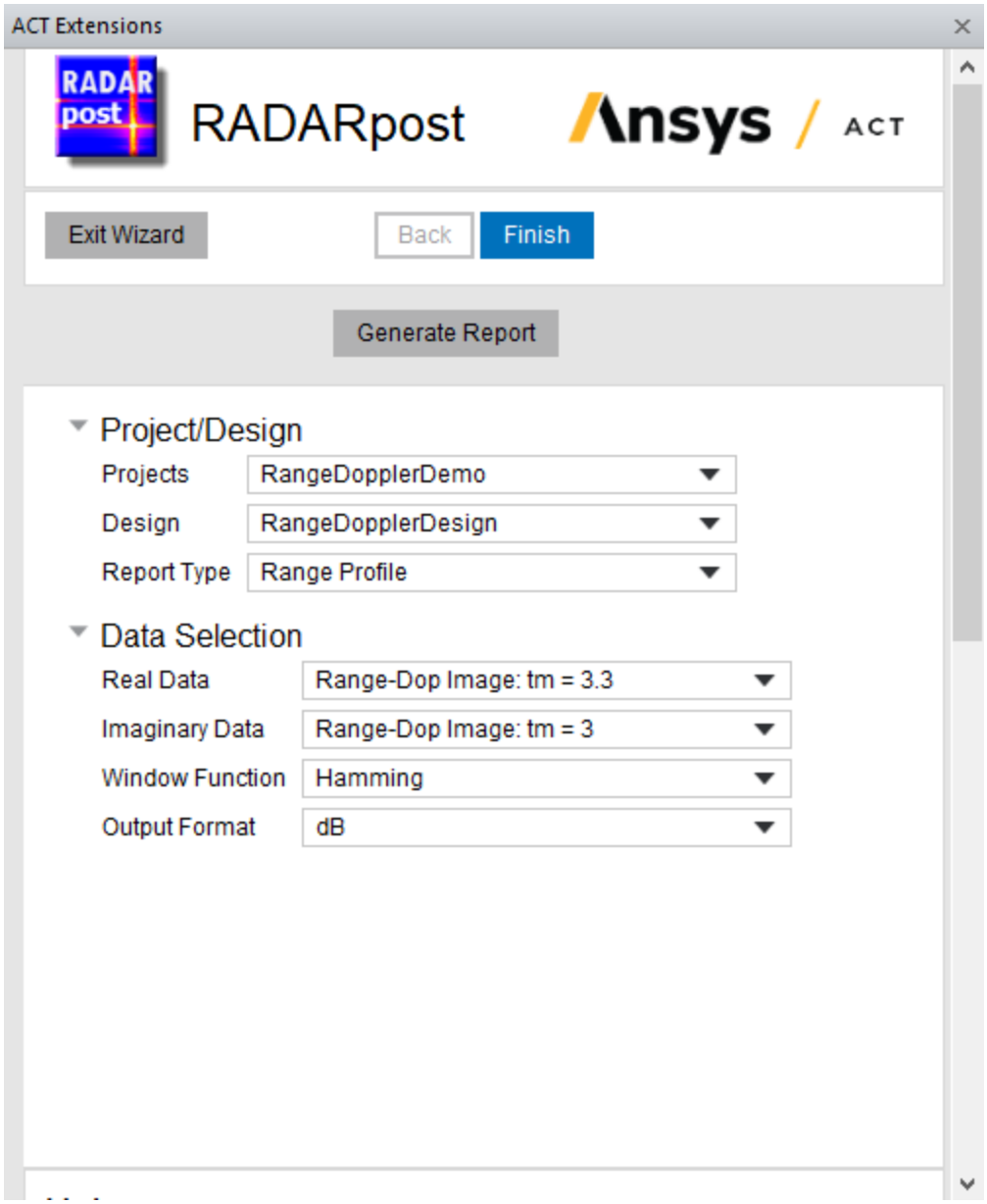
To launch the Wizard:

1. Click View>Extensions command to open the Extensions window.
2. Click **Wizard** to open the ACT Start Page. If you have not loaded the RadarPost extension, use the **Manage Extension**s feature to do so.

3. Click **Launch Wizards** to the show the active Radarpre toolkit, and click to execute wizard.

The RADARpost wizard displays.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



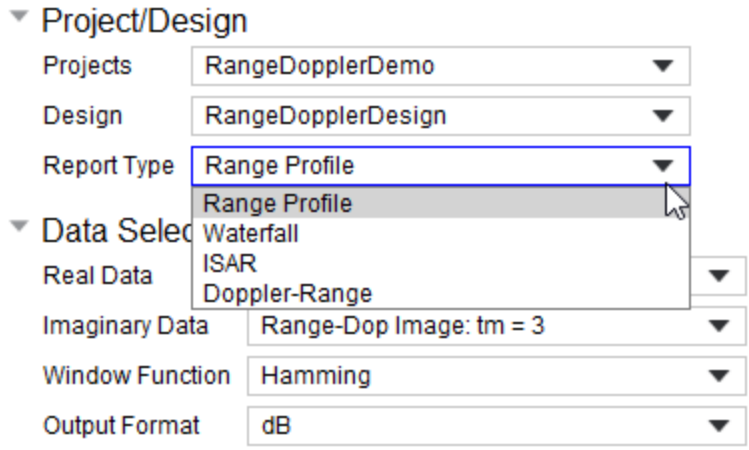
You can scroll down to view the Help overview.

Using the RADARpost Wizard

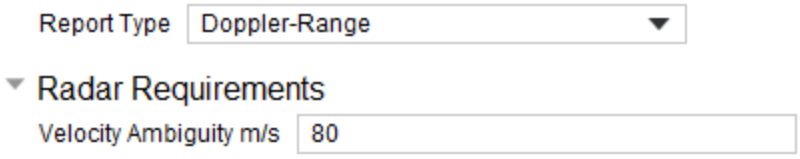
The RADARpost Wizard contains areas for selecting the Project/Design and Radar Requirements.

1. If there is more than one suitable pre-loaded design, select the Project/Design.
2. You then select the Report type as Range Profile, Waterfall, ISAR or ADAS Scenario.

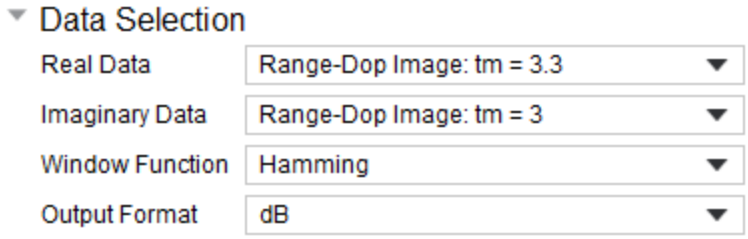
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. If you select Doppler-Range, you also must specify the Radar Requirements.



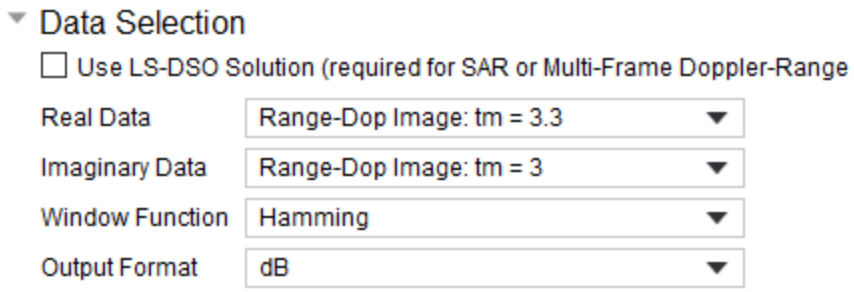
4. Specify the Data Selection Parameters by selecting from the drop down menus:
   Real Data:
   Imaginary Data
   Window Function
   Output format as linear or dB



If you select Doppler-Range, you also can choose whether to Use LS-DSO Solution

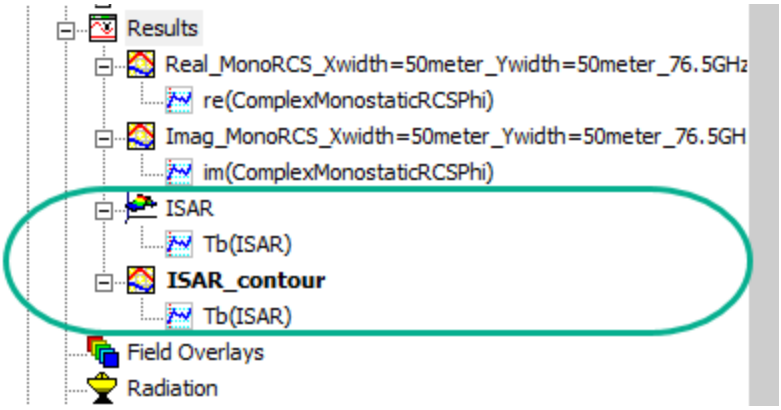
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(Required for SAR or Multi-Frame Doppler Range).



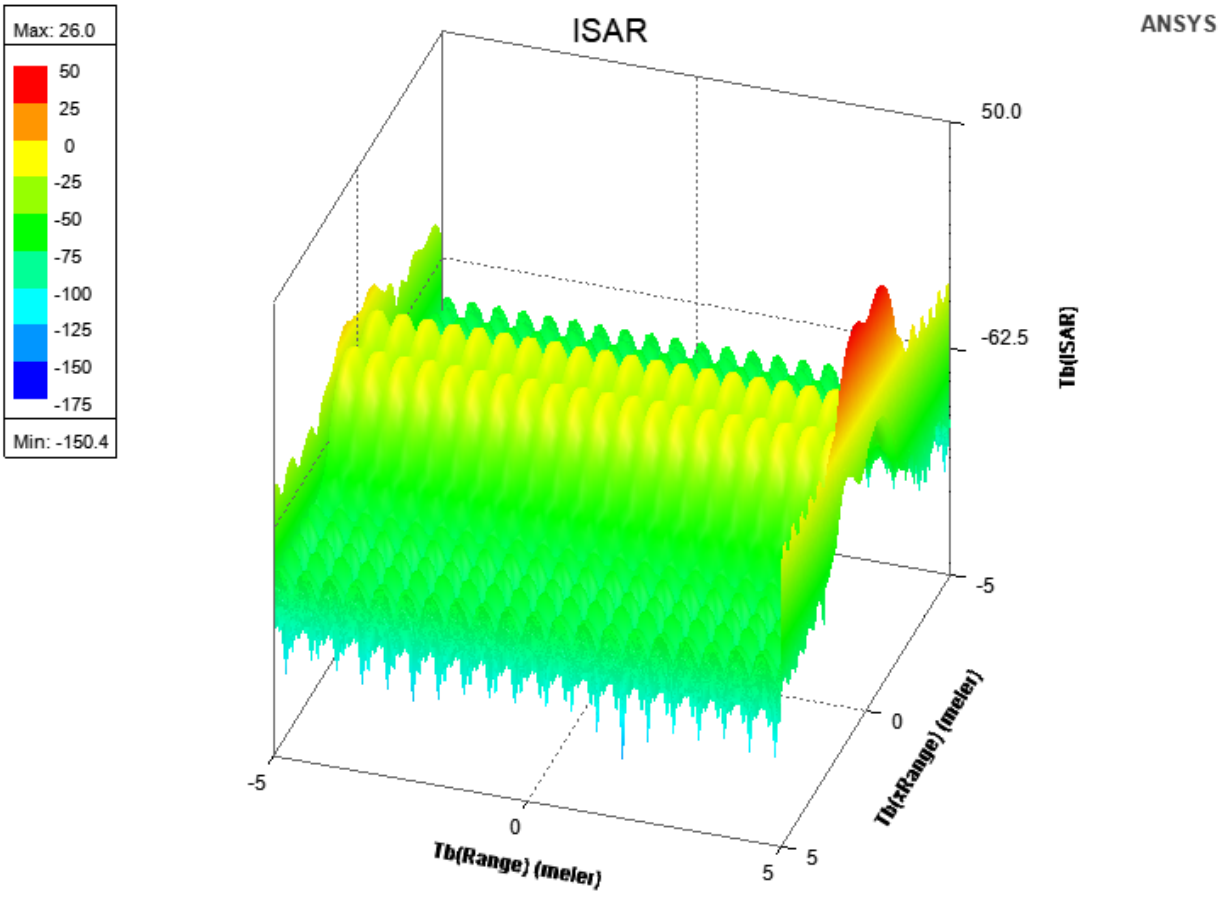
5. Click Generate Report

The RADARpost creates two new reports, based on the data of the existing SBR+ solution,



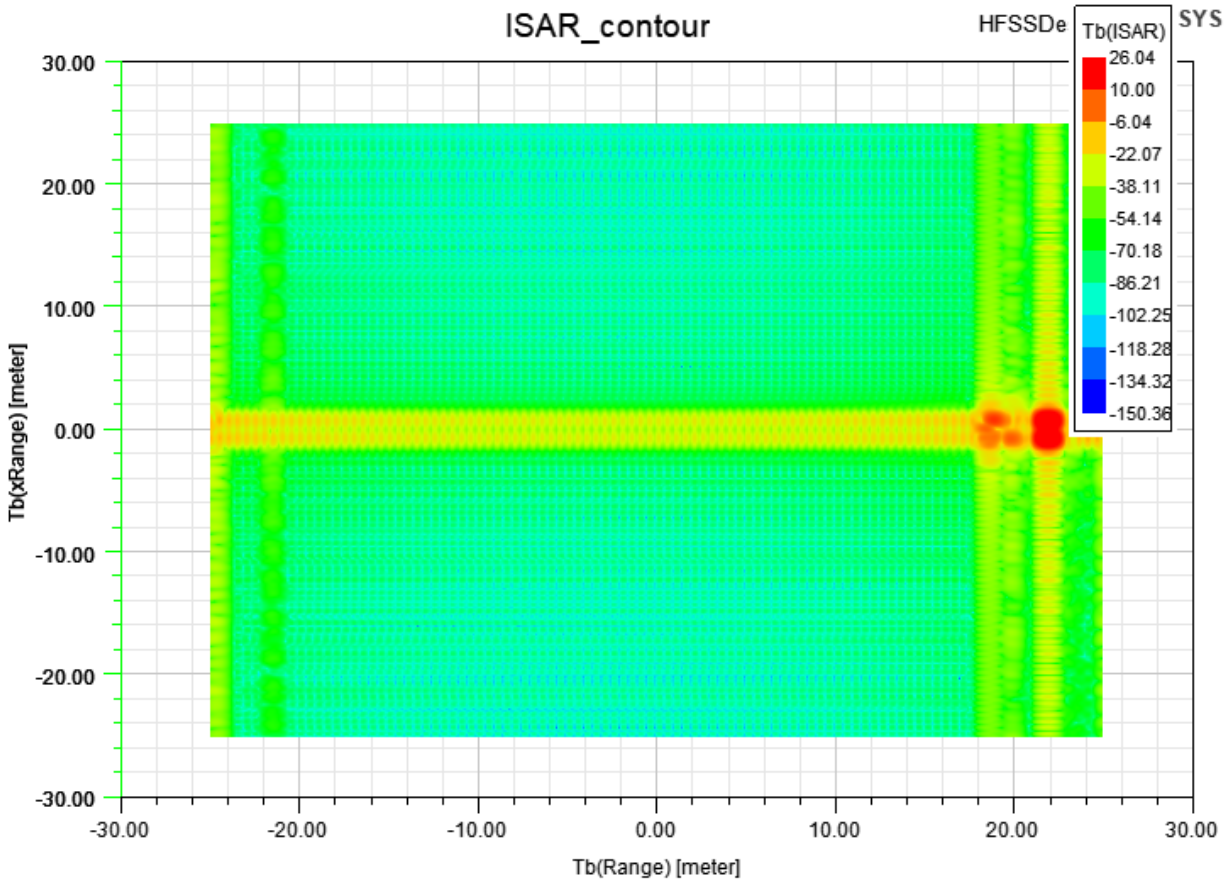
The following is an example ISAR Report.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following is an example ISAR Contour report.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



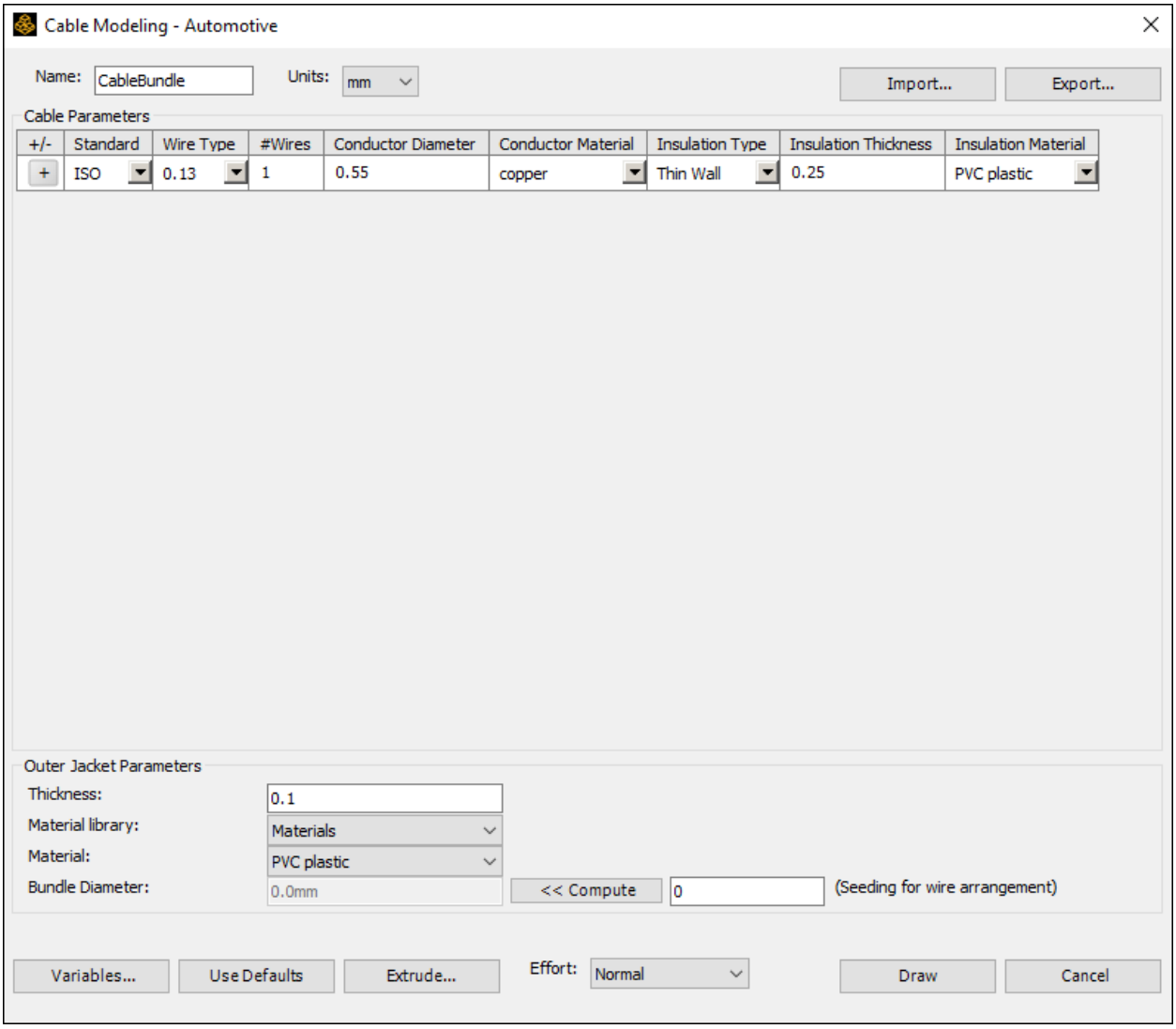
Cable Modeling - Automotive Toolkit

To work with tool kits in the automotive industries:

1. Click **HFSS** > **Toolkit** > **Cable Modeling** > **Automotive Cable Bundle**.

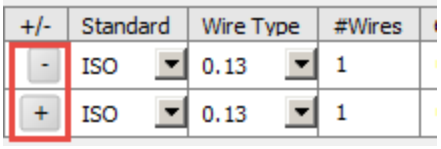
   The **Cable Modeling - Automotive** dialog box appears with the **Cable Parameters** listed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Enter a **Name** for the toolkit, and select **Units**.
3. If you have a *.csv or *.tab file containing cable and outer jacket parameters, you can **Import** it.
4. To manually modify the **Cable Parameters**:
    - Using the +/- column, add or remove wire types to create a bundle. Clicking the **+** button adds a new row. All rows but the initial row will have a **-** button to remove that row.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- In the **Standard** field, choose between the **ISO** or **AWG** standard types.
- In the **Wire Type** field, use the drop-down menu to select the wire type.
- In the **#Wires** field, adjust the number of wires as necessary.
- In the **Conductor Diameter** field, specify the diameter in the unit you selected.
- In the **Conductor Material** field, use the drop-down menu to select a material.
- In the **Insulation Type** field, use the drop-down menu to select either **Thin Wall** or **Ultra Thin Wall**.
- In the **Insulation Thickness** field, specify a value in the unit you selected.
- In the **Insulation Material** field, use the drop-down menu to select a material.

5. To manually modify the **Outer Jacket Parameters**:
   - In the **Thickness** field, specify a value for the inner diameter in the unit you selected.
   - Use the **Material Library** drop-down menu to select the library containing the material you wish you specify, then use the **Material** drop-down menu to select the material.
   - Populate the **Bundle Diameter** field. After specifying all wires to be included in the bundle, click **Compute** to calculate the minimal bundle diameter of the bundle. The value automatically populates.

> **Note:**
>
> The wire radii are adjusted downward very slightly (on the order of 0.1%) at the end of the packing process to eliminate any residual overlaps between the wires and to make the model easier to mesh.

> **Note:**
>
> The Circle Packing algorithm is used to compute the values. It tries to automatically pack the conductors in the bundle tightly with a minimum of empty space between them. It generates and tests a large number of random variations in the conductor positions and attempts to minimize the overall diameter of the bundle.

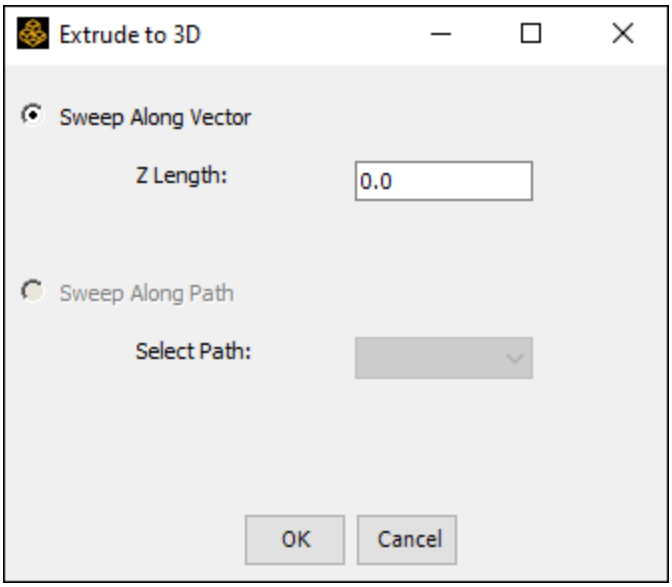
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can adjust the **Bundle Diameter** generated by specifying a **Seeding for wire arrangement** value. The value entered here is the seed value for the pseudo-random number generator used in the conductor packing process. Changing this value results in different arrangements of the conductors. This can be useful for performing statistical analysis of the cables.

6. To save your cable and outer jacket parameters for future use, click **Export** to create a *.csv or *.tab file for later import.
7. If desired, click **Use Defaults** to restore the default values. This removes any rows previously added and clears all custom values.
8. To add new variables, click **Variables**. This opens the **Edit Variables** dialog box, listing all variables that are present at the project and design levels. Click **Add** to create new variables. See: Working with Variables.
9. Click **Draw** to draw the 2D cross-section geometry. A validation check is run. This catches errors such as any variable or parameter missing a value.
10. For 3D designs, you can create a 3D geometry:

    The **Effort** drop-down menu contains four options: Low, Normal, High, and Very High. This controls the trade-off between packing quality (how tightly the conductors are packed together) and the time spent generating the model. Low effort runs relatively fast and may be good enough for small numbers of conductors, but may leave significant space in the model when there are many conductors. High effort may be required for models with large numbers of conductors (100+), but could take several minutes to run.

    Click **Extrude**. The **Extrude to 3D** dialog box appears.



- Click **Sweep Along Vector** to extrude along the z axis. Z length is the input parameter.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click **Sweep Along Path** to extrude along a selected path.

> **Note:**
>
> 3D models of cables are hard to solve if the ratio of extruded length to cross-section diameter is high.

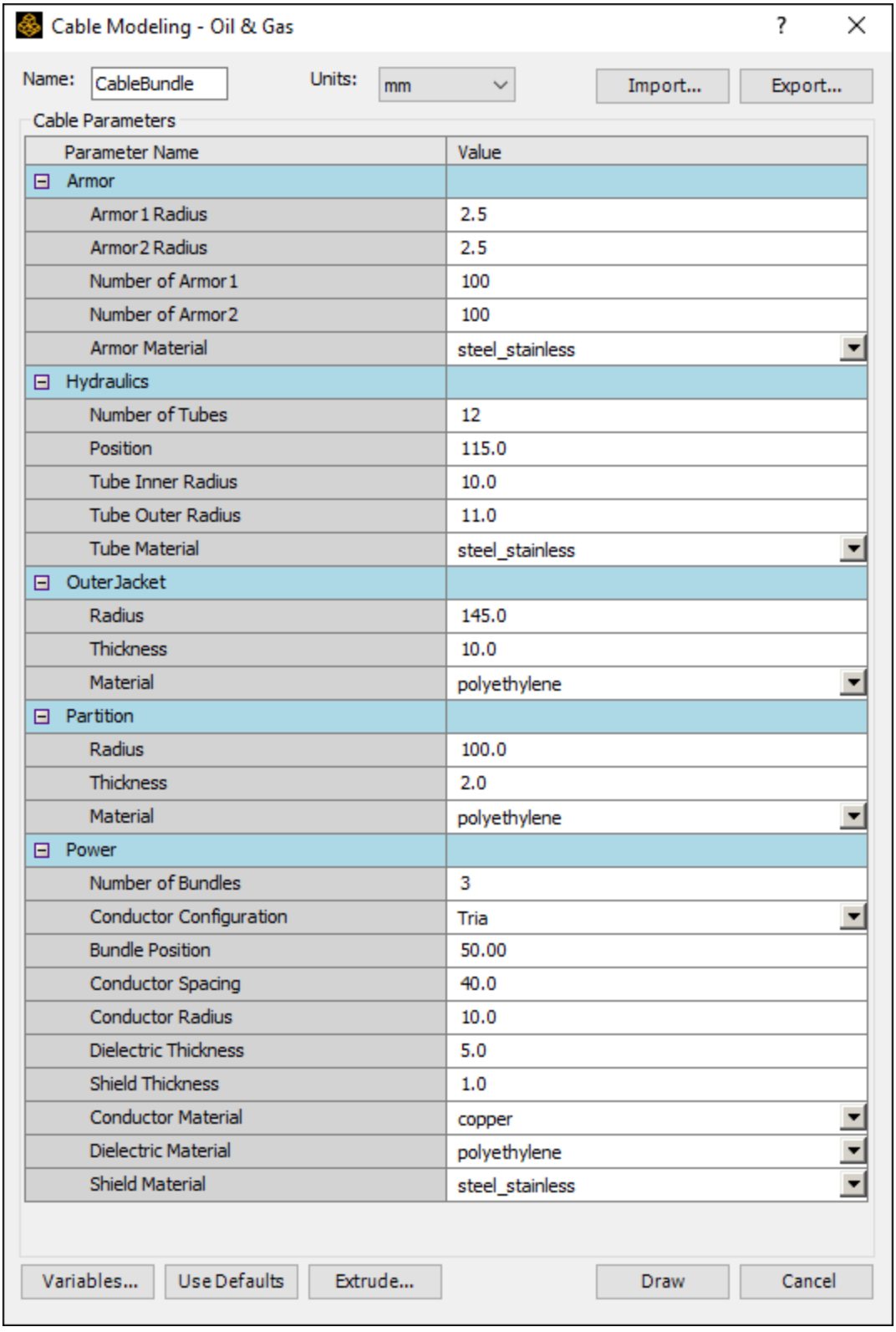
Cable Modeling - Oil and Gas Toolkit

To work with tool kits in the oil and gas industries:

1. Click **HFSS** > **Toolkit** > **Cable Modeling** > **Oil-Gas Cable Bundle**.

   The **Cable Modeling - Oil & Gas** dialog box appears.

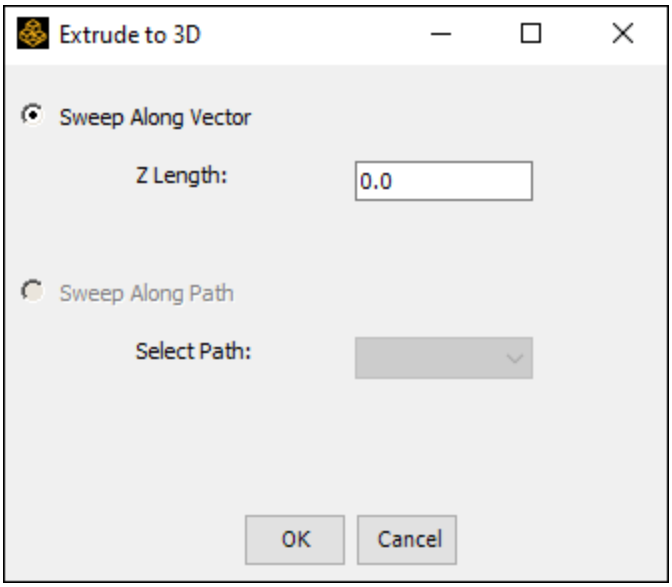
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Enter a **Name** for the toolkit, and select **Units**.
3. If you have a *.csv or *.tab file containing cable parameters, you can **Import** it.
4. To manually modify the cable parameters:
    - Enter and select values for each of the available fields, in the unit you selected.
5. To save your cable parameters for future use, click **Export** to create a *.csv or *.tab file for later import.
6. To add new variables, click **Variables**. This opens the **Edit Variables** dialog box. All variables that are already present in the project and design levels are listed. Click **Add** to add a new row to create new variables.
7. Click **Draw** to draw the 2D cross-section geometry. A validation check is run. This catches errors such as any variable or parameter missing a value.
8. For 3D designs, you can create a 3D geometry.

   Click **Extrude**. The **Extrude to 3D** dialog box appears.



- Click **Sweep Along Vector** to extrude along the z axis. Z length is the input parameter.
- Click **Sweep Along Path** to extrude along a selected path.

> **Note:**
>
> 3D models of cables are hard to solve if the ratio of extruded length to cross-section diameter is high.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Finite Array Beam Angle Calculator Toolkit

The Finite Array Beam Angle calculator provides:

- An interface to calculate the phase shifts along the A and B vectors of the array lattice, given the scan angles for array elements.
- A script as a template for referencing when writing their own scripts to add functionality and automations.
- Post processing variables for driven and design variables for composite data types for all the data fields to allow you to adjust and modify parameters outside the calculator interface and view corresponding results.
- An interface to calculate different types of tapering, such as triangular, cosine, hamming window, and so forth, and apply these tapering and phase angle to each individual source in the form of either equation or calculated value. You can choose to calculate the taper weights in different coordinate systems, like lattice coordinate, absolute XY coordinates and absolute distance, and visualize the result in an intuitive color map. You can apply tapering to the sub-array and create source groups for the sub-array.

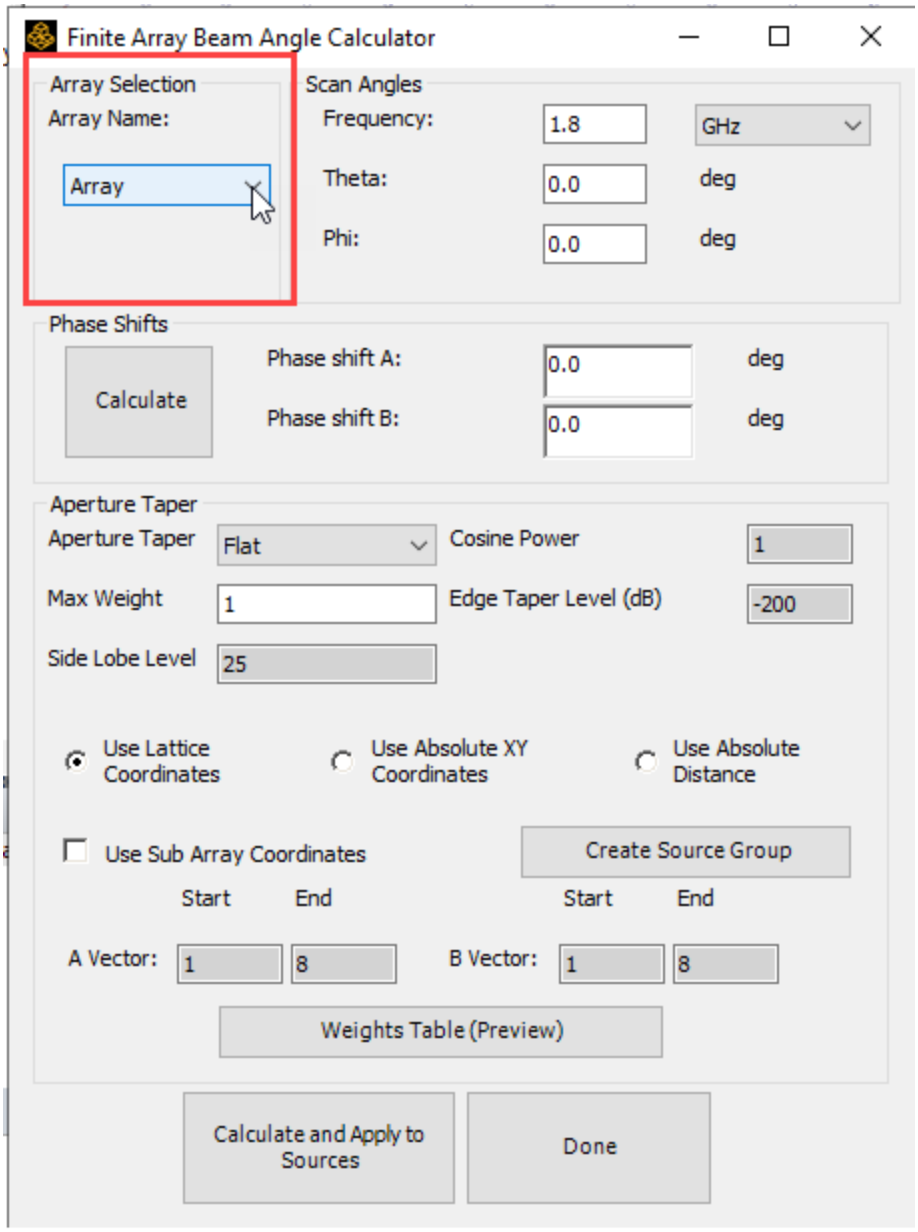
To access the calculator click **HFSS>Toolkit>Update Menu**. This causes the Toolkit menu to display the Finite Array Beam Angle.

1. To use the calculator, you must first load a project that includes a finite array. The design must have at least one port defined in order to apply excitations to Edit Sources. Select a suitable design and click **HFSS>Toolkit>Finite Array Beam Angle.**

   This displays the **Finite Array Beam Angle Calculator** dialog:

   The Array Selection area is for the Beta Feature for HFSS Multiple Component Arrays. If the feature for multiple arrays is enabled and multiple arrays exist in the design, each array can be calculated individually. When the feature flag is off, the Array Selection control box still appears but only one array can be selected.

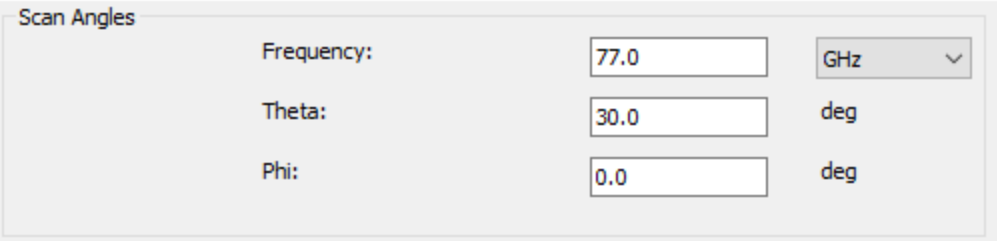
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Frequency** field is automatically populated based on the current frequency value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

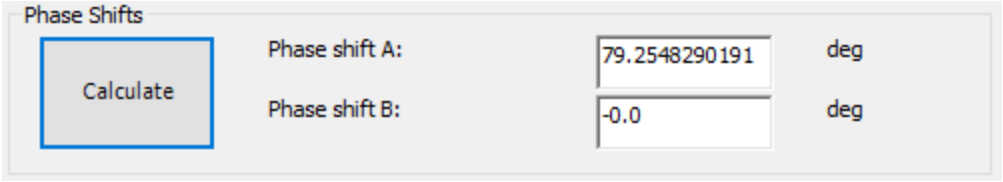
2. You enter values for Scan Angle **Theta** and **Phi** in degrees.



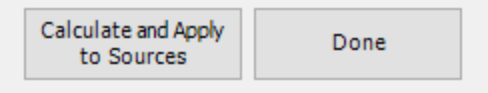
Once you give the scan angle inputs, use the Phase Shifts panel to calculate the phase increment between each adjacent unit cell.

3. Click the **Calculate** button.

The **Phase shift A** and **Phase shift B** fields display the results. If the fields do not initially display, you can size the Toolkit dialog to show all of the necessary digits.

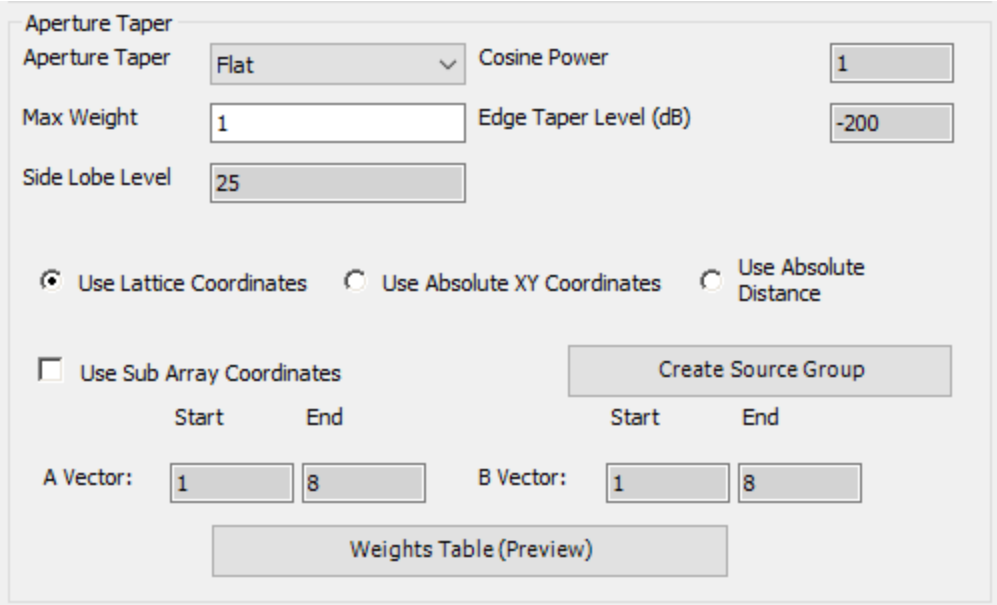


4. Click **Calculate and Apply to Sources**.

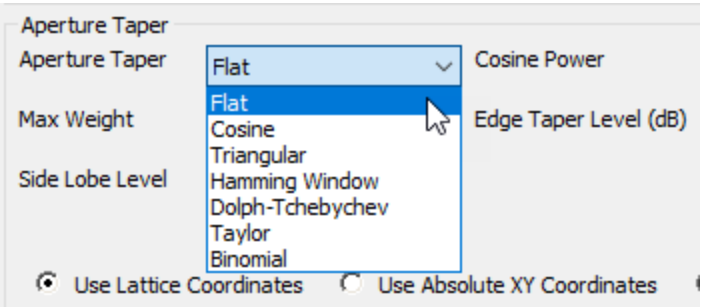


5. The **Aperture Taper** panel contains everything needed to apply taper to each unit cell and create a subarray source group.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Aperture Taper** dropdown menu lets you select a tapering method: Flat, Cosine, Triangular, and Hamming Window, Dolph-Tchebychev, Taylor, and Binomial.

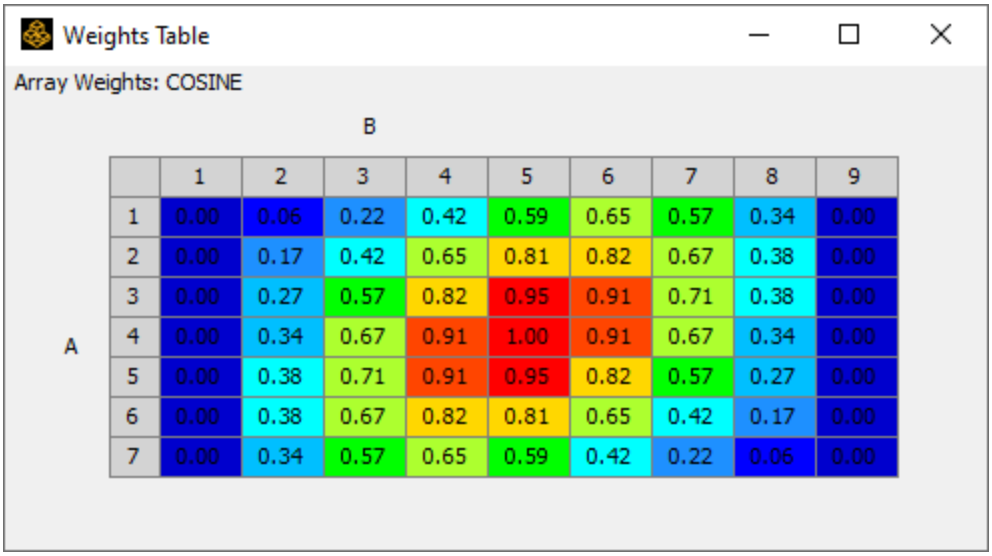


Depending on the method, some variables applicable to the selected taper will become editable on the right side, and you can edit them as they wish.

6. **Max Weight** allows you to input the largest magnitude for the unit cells. The **Max Weight** value occurs at the middle of the array and tapers off at the edges.
7. The three radio buttons below the **Max Weight** text box lets you choose the coordinate system used to calculate weights. The detailed calculation for each tapering method in each coordinate system is described in detail in the next section of this document.
8. **Use Sub Array Coordinates** allows the user to only choose specific parts of the array to apply weights and phases to.
9. The **Create Source Group** button creates a source group with the cell within the Sub Array Coordinates. To use the button, you must check the **Use Sub Array Coordinates** checkbox.

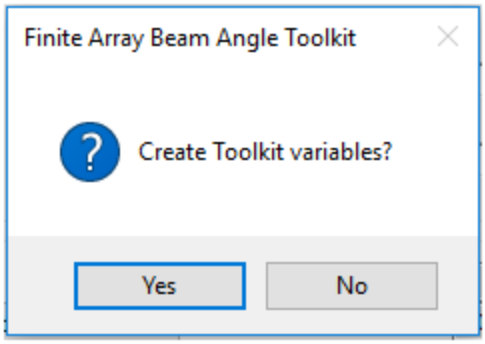
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. The **Weights Table (Preview)** button displays a color map table that gives a preview of the magnitudes that will be applied to each cell. For example:



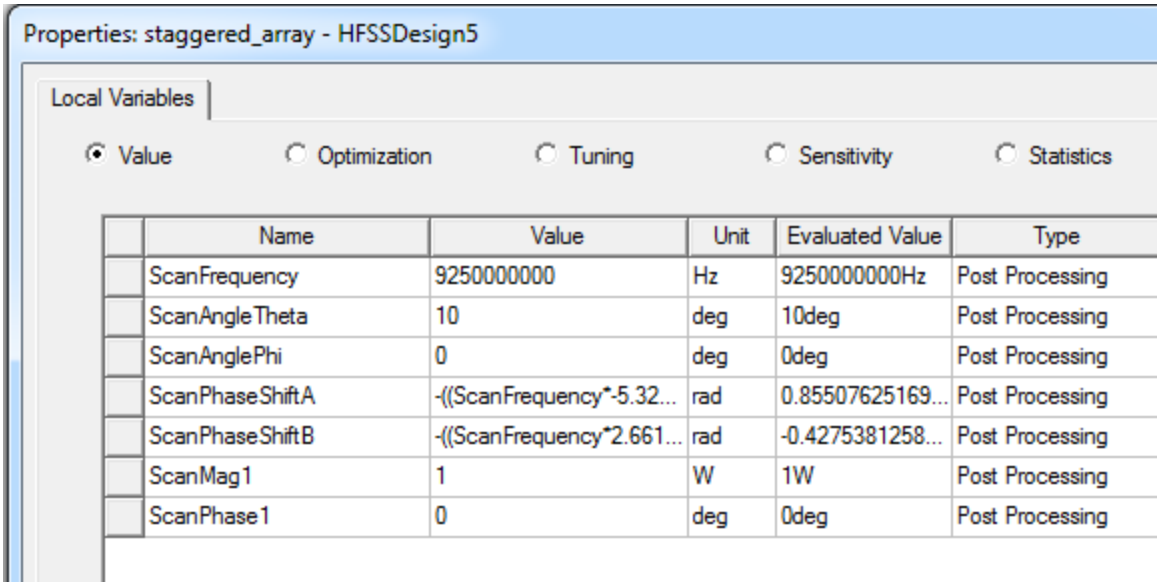
	1	2	3	4	5	6	7	8	9
1	0.00	0.06	0.22	0.42	0.59	0.65	0.57	0.34	0.00
2	0.00	0.17	0.42	0.65	0.81	0.82	0.67	0.38	0.00
3	0.00	0.27	0.57	0.82	0.95	0.91	0.71	0.38	0.00
4	0.00	0.34	0.67	0.91	1.00	0.91	0.67	0.34	0.00
5	0.00	0.38	0.71	0.91	0.95	0.82	0.57	0.27	0.00
6	0.00	0.38	0.67	0.82	0.81	0.65	0.42	0.17	0.00
7	0.00	0.34	0.57	0.65	0.59	0.42	0.22	0.06	0.00

11. Once you have specified all options, click the **Calculate and Apply to Sources** button. A message box asks if you want to save all results as variables [**Yes**] or as numeric values [**No**].



12. Click **HFSS>Design Properties** to view the local variables. Scroll down if necessary to view the newly created variables.

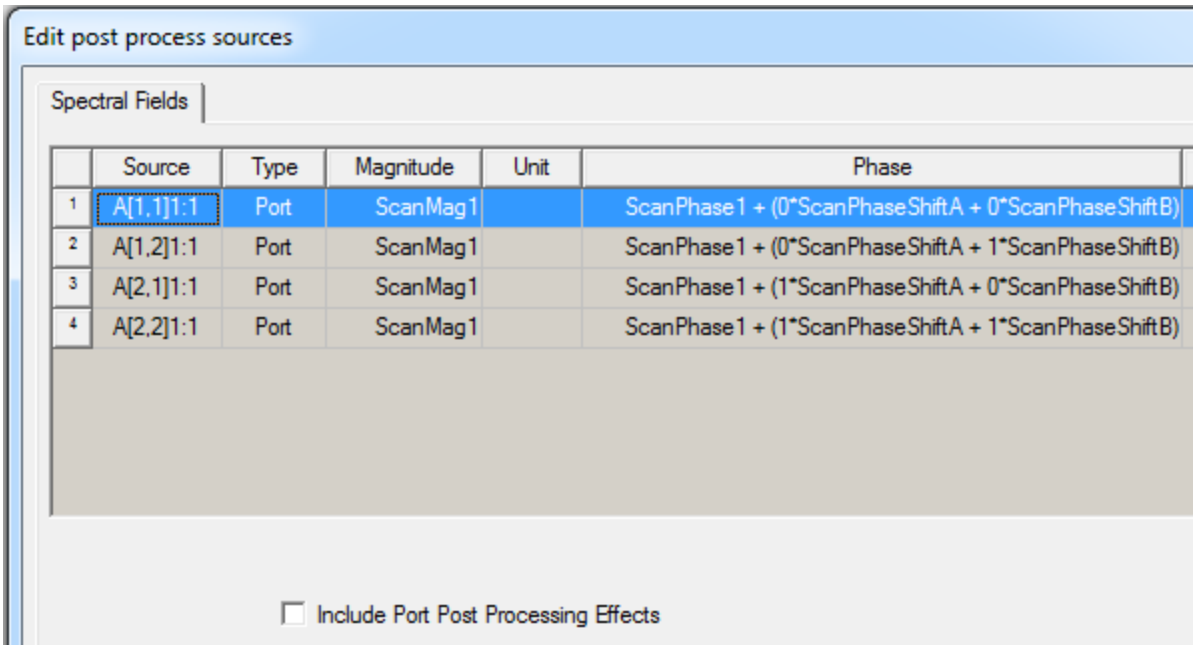
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



	Name	Value	Unit	Evaluated Value	Type
	ScanFrequency	9250000000	Hz	9250000000Hz	Post Processing
	ScanAngleTheta	10	deg	10deg	Post Processing
	ScanAnglePhi	0	deg	0deg	Post Processing
	ScanPhaseShiftA	-((ScanFrequency*-5.32...	rad	0.85507625169...	Post Processing
	ScanPhaseShiftB	-((ScanFrequency*2.661...	rad	-0.4275381258...	Post Processing
	ScanMag1	1	W	1W	Post Processing
	ScanPhase1	0	deg	0deg	Post Processing

These include ScanFrequency, ScanAngleTheta, ScanAnglePhi, ScanPhaseShiftA, and ScanPhaseShiftB.The values for ScanPhaseShiftA and ScanPhaseShiftB will be expressions. There will also be ScanMag*N* and ScanPhase*N*, depending on the number of modes/terminals.

13. Right-click **Excitations>Edit Sources** to view the **Edit Post Process Sources** dialog to view the expressions for each source:



	Source	Type	Magnitude	Unit	Phase
1	A[1,1]1:1	Port	ScanMag1		ScanPhase1 + (0*ScanPhaseShiftA + 0*ScanPhaseShiftB)
2	A[1,2]1:1	Port	ScanMag1		ScanPhase1 + (0*ScanPhaseShiftA + 1*ScanPhaseShiftB)
3	A[2,1]1:1	Port	ScanMag1		ScanPhase1 + (1*ScanPhaseShiftA + 0*ScanPhaseShiftB)
4	A[2,2]1:1	Port	ScanMag1		ScanPhase1 + (1*ScanPhaseShiftA + 1*ScanPhaseShiftB)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14. You can then click **Radiation>Insert Far Field** setups for 3D or XY Plots as needed, and then use **Results>Create Far Fields Report** to define plots.
15. After you run **Analyze** and evaluate the plots you create. You can then change the Frequency, the Theta and Phi values, and the ScanMag and ScanPhase variables to see the effects of your changes.

Phase and Weight Calculations

Phase Increment

The phase for a unit cell is calculated by first finding the phase increment between each adjacent cell in both the a and b lattice directions.

$$k = \frac{2\pi f}{c}$$
$$PhaseShiftA = -1 * ((Ax * k * \sin(\theta) * \cos(\Phi)) + (Ay * k * \sin(\theta) * \sin(\Phi))$$
$$PhaseShiftB = -1 * ((Bx * k * \sin(\theta) * \cos(\Phi)) + (By * k * \sin(\theta) * \sin(\Phi))$$
$$Phase = a * PhaseShiftA + b * PhaseShiftB$$

This uses the *x* and *y* values of the lattice vectors *Ax*, *Ay*, *Bx*, and *By* as well as the scanning angle θ and Φ. The variable *f* is the frequency and *c* is the speed of light. Then, the phase is assigned depending on the *a* and b array position.

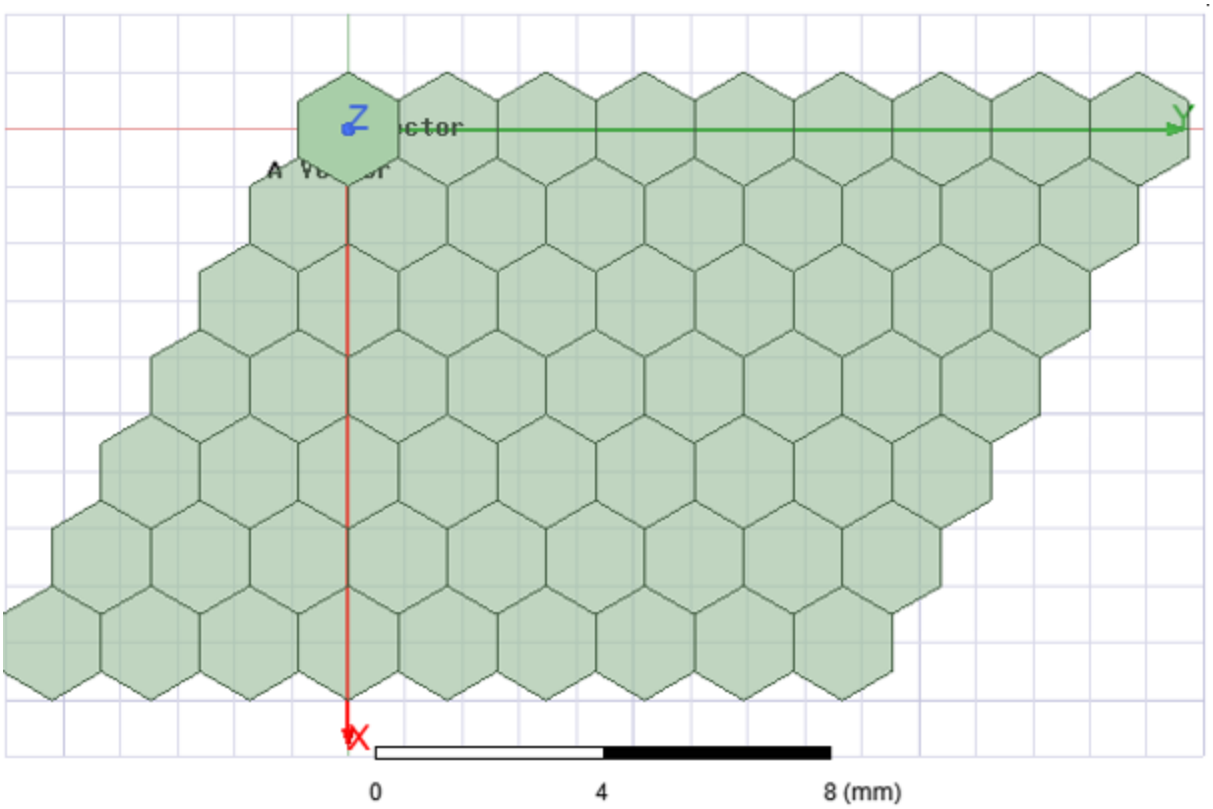
Aperture Taper Weighting Scheme

Calculating Array Center

The array center is calculated by first finding the maximum and minimum lattice coordinates in both *A* and *B* vectors among all cells. Then, the lattice coordinates of the center of the array is calculated by taking the midpoint between the maximum and minimum coordinates. Finally, the global *x*-*y* coordinates of the center of the array is calculated by converting the lattice coordinates into global coordinates.

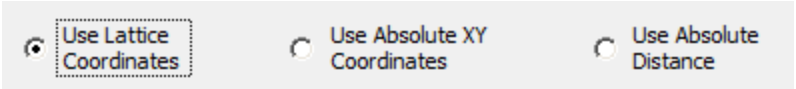
The example weights table is applied to the following array design:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Lattice Coordinate

For lattice coordinate, the final weight is calculated as *Max Weight* * *W*(*a*,*b*), where *W*(*a*,*b*) depends on the chosen aperture taper weighting schemes.

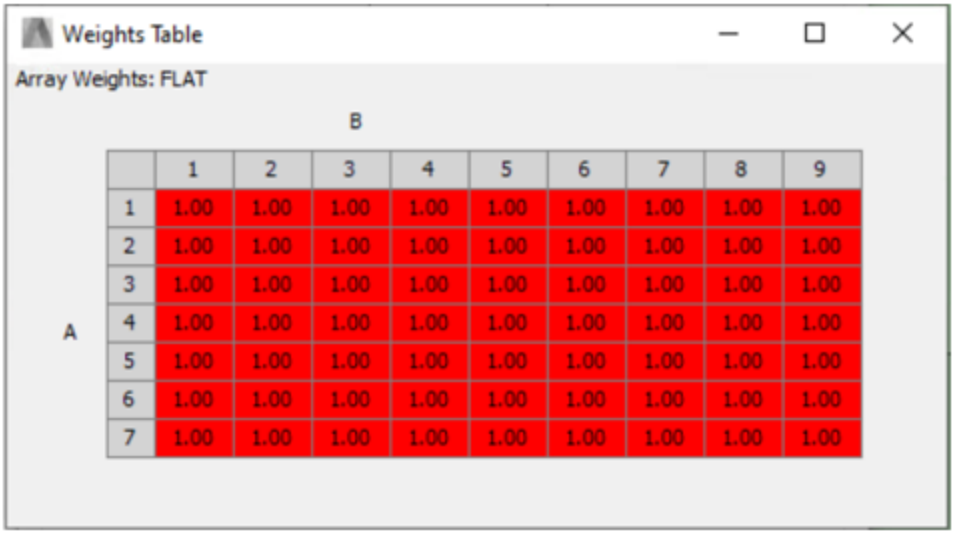


1. Flat:

   W(*a*,*b*)=1

   Weights Table (Preview) for a 7x9 hexagon array:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



	1	2	3	4	5	6	7	8	9
1	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00
2	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00
3	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00
4	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00
5	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00
6	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00
7	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00

2. Cosine:

$$t = 10^{\frac{\text{EdgeTaperLevel}}{20}}$$

$$W(a) = (1-t)(\cos^{n}(\frac{\pi a}{A})) + t$$

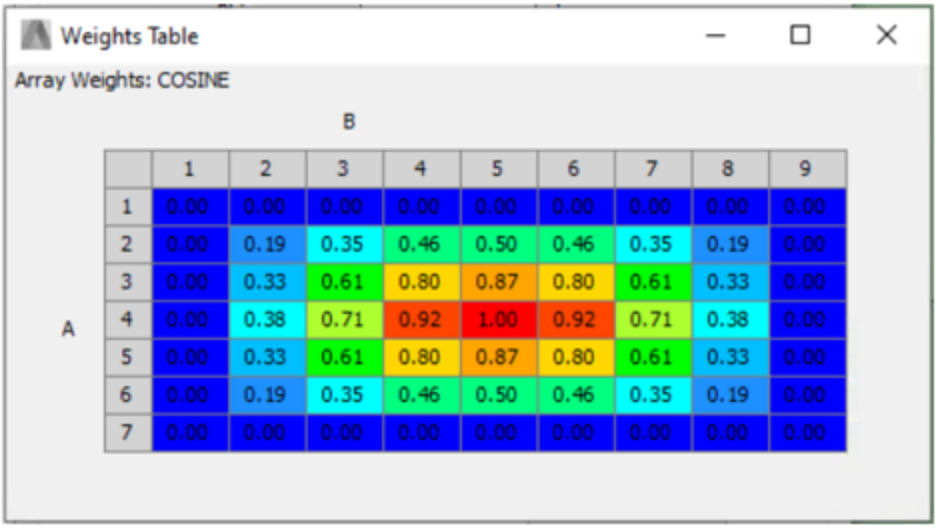
$$W(b) = (1-t)(\cos^{n}(\frac{\pi b}{B})) + t$$

$$W(a,b) = W(a) * W(b)$$

where *EdgeTaperLevel* is given by user, *n* is Cosine Power, variable *a* and *b* is the index difference between the current cell and the center of the array in lattice A and B vector direction, variable *A* and *B* is the maximum index difference among all cells in the lattice A and B vector direction.

Weights Table (Preview) for a 7x9 hexagon array:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Triangular

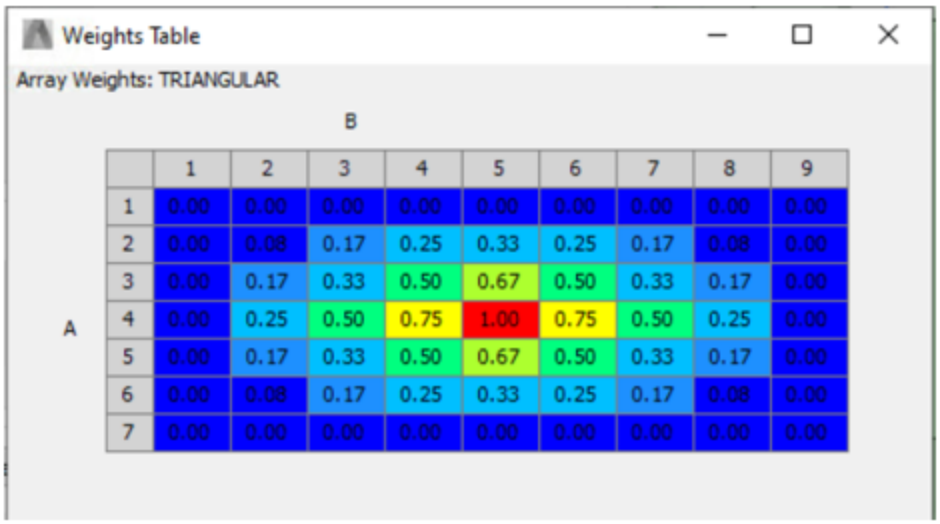
$$W(a) = (1-t)\left(1 - \frac{|a|}{A/2}\right) + t$$

$$W(b) = (1-t)\left(1 - \frac{|b|}{B/2}\right) + t$$

$$W(a,b) = W(a) * W(b)$$

All variables are defined in the same way as those in the previous method

Weights Table (Preview) for a 7x9 hexagon array:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

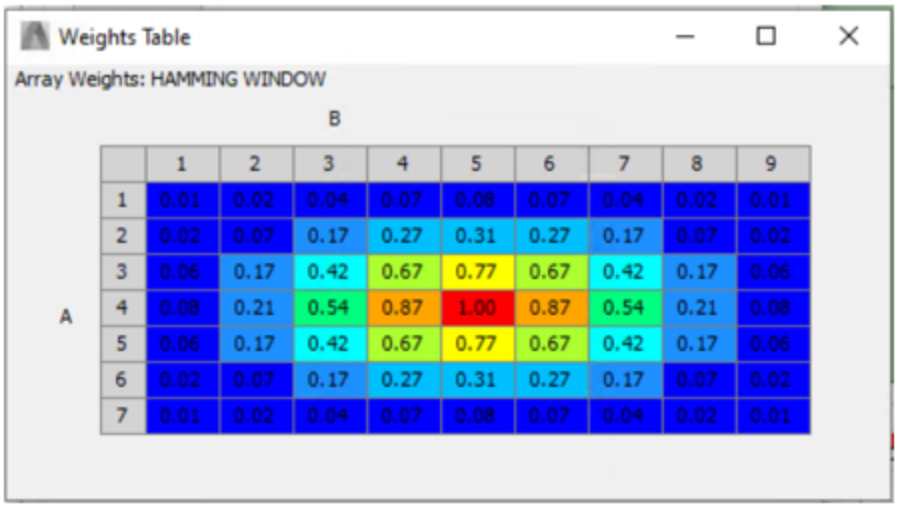
4. Hamming Window

$$W(a) = 0.54 - 0.46\,cos\left(2\pi\left(\frac{a}{A} - 0.5\right)\right)$$

$$W(b) = 0.54 - 0.46\,cos\left(2\pi\left(\frac{b}{B} - 0.5\right)\right)$$

$$W(a,b) = W(a) * W(b)$$

Weights Table (Preview) for a 7x9 hexagon array:



	1	2	3	4	5	6	7	8	9
1	0.01	0.02	0.04	0.07	0.08	0.07	0.04	0.02	0.01
2	0.02	0.07	0.17	0.27	0.31	0.27	0.17	0.07	0.02
3	0.06	0.17	0.42	0.67	0.77	0.67	0.42	0.17	0.06
4	0.08	0.21	0.54	0.87	1.00	0.87	0.54	0.21	0.08
5	0.06	0.17	0.42	0.67	0.77	0.67	0.42	0.17	0.06
6	0.02	0.07	0.17	0.27	0.31	0.27	0.17	0.07	0.02
7	0.01	0.02	0.04	0.07	0.08	0.07	0.04	0.02	0.01

5. Dolph-Tchebychev

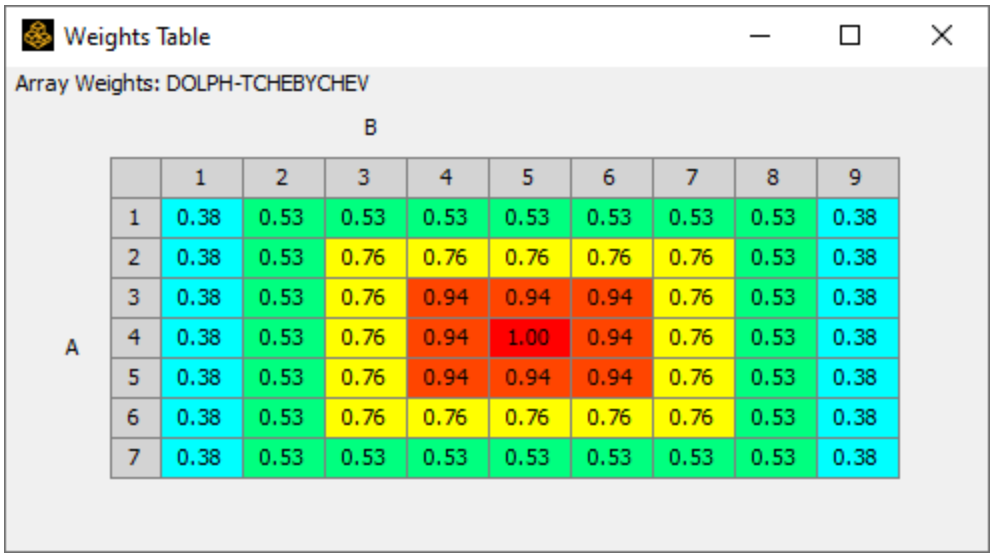
Sidelobe level in absolute units	$R_a = 10^{R/20}$	*R* is the side lobe level
Number of pattern zeros	$N_1 = N - 1$	*N* is the number of array elements
Scaling factor	$x_0 = \cosh\left(\frac{\text{acosh}(R_a)}{N-1}\right)$	
*N*1 zeros of Chebyshev polynomial	$x_i = \cos\left(\frac{(i-1/2)\pi}{N-1}\right)$	$i = 1, 2, \ldots, N-1$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*N*1 array pattern zeros in psi-space $\varphi_i = 2\text{acos}\left(\frac{x_i}{x_0}\right)$

*N*1 zeros of array polynomial $z_i = e^{j\varphi_i}$

Weights Table (Preview) for a 7x9 hexagon array:



	1	2	3	4	5	6	7	8	9
1	0.38	0.53	0.53	0.53	0.53	0.53	0.53	0.53	0.38
2	0.38	0.53	0.76	0.76	0.76	0.76	0.76	0.53	0.38
3	0.38	0.53	0.76	0.94	0.94	0.94	0.76	0.53	0.38
4	0.38	0.53	0.76	0.94	1.00	0.94	0.76	0.53	0.38
5	0.38	0.53	0.76	0.94	0.94	0.94	0.76	0.53	0.38
6	0.38	0.53	0.76	0.76	0.76	0.76	0.76	0.53	0.38
7	0.38	0.53	0.53	0.53	0.53	0.53	0.53	0.53	0.38

6. Taylor

Relative sidelobe level in dB (13<R<120):

$$\alpha = \begin{cases} 0, & R \le 13.26 \\ 0.76609(R-13.26)^{0.4} + 0.09834(R-13.26), & 13.26 < R \le 60 \\ 0.12438((R+6.3), & 60 < R < 120 \end{cases}$$

R is the side lobe level

Row vector of array weights:

$$w(m) = I_0(\alpha\sqrt{1-(\frac{m}{M})^2})$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$m = \pm 1, \pm 2, \ldots, \pm M$ for even number of array elements

$m = 0, \pm 1, \pm 2, \ldots, \pm M$ for odd number of array elements

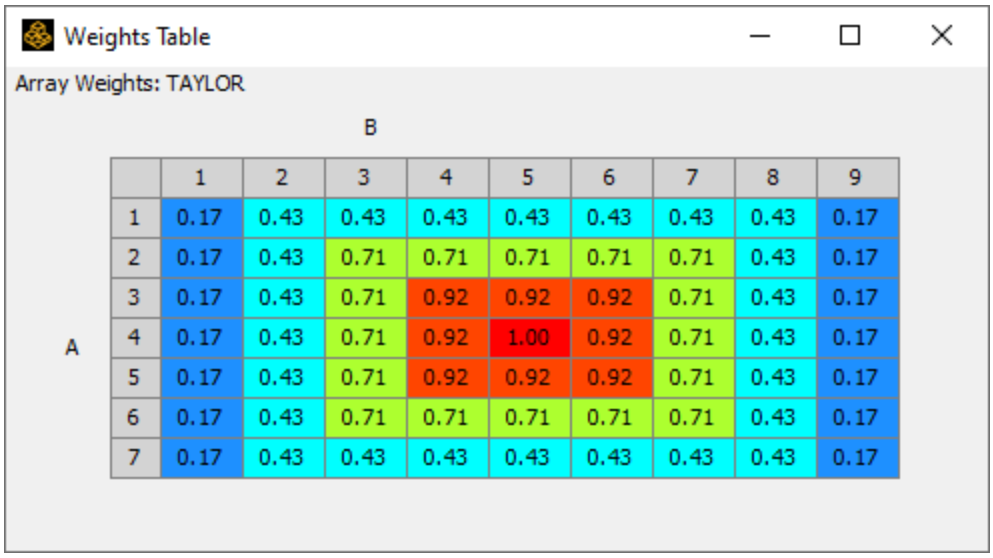
I_0 is the modified Bessel function of first kind and zeroth order.

$M = \frac{N}{2}$ for even elements

$M = \frac{N-1}{2}$ for odd elements

N is the number of array elements

Weights Table (Preview) for a 7x9 hexagon array:



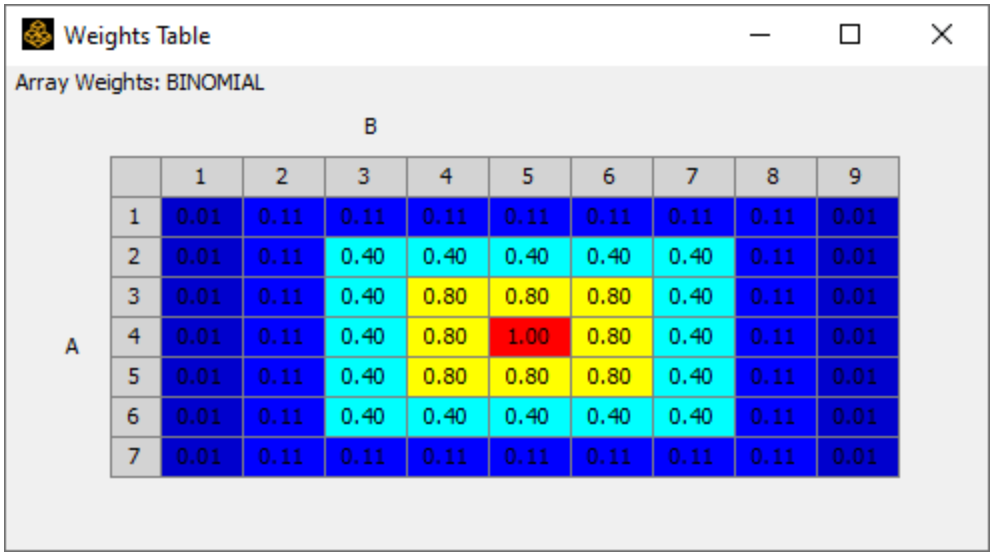
	1	2	3	4	5	6	7	8	9
1	0.17	0.43	0.43	0.43	0.43	0.43	0.43	0.43	0.17
2	0.17	0.43	0.71	0.71	0.71	0.71	0.71	0.43	0.17
3	0.17	0.43	0.71	0.92	0.92	0.92	0.71	0.43	0.17
4	0.17	0.43	0.71	0.92	1.00	0.92	0.71	0.43	0.17
5	0.17	0.43	0.71	0.92	0.92	0.92	0.71	0.43	0.17
6	0.17	0.43	0.71	0.71	0.71	0.71	0.71	0.43	0.17
7	0.17	0.43	0.43	0.43	0.43	0.43	0.43	0.43	0.17

7. Binomial

Row vector of array weights:

$$w(m) = \frac{(N-1)!}{m!\,(N-1-m)!} \qquad m = 0, 1, \ldots, N-1 \qquad m = 0, 1, \ldots, N-1$$

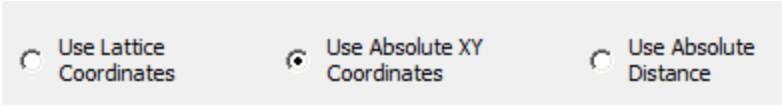
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



	1	2	3	4	5	6	7	8	9
1	0.01	0.11	0.11	0.11	0.11	0.11	0.11	0.11	0.01
2	0.01	0.11	0.40	0.40	0.40	0.40	0.40	0.11	0.01
3	0.01	0.11	0.40	0.80	0.80	0.80	0.40	0.11	0.01
4	0.01	0.11	0.40	0.80	1.00	0.80	0.40	0.11	0.01
5	0.01	0.11	0.40	0.80	0.80	0.80	0.40	0.11	0.01
6	0.01	0.11	0.40	0.40	0.40	0.40	0.40	0.11	0.01
7	0.01	0.11	0.11	0.11	0.11	0.11	0.11	0.11	0.01

Absolute Coordinate - XY

For absolute coordinate, the final weight is calculated as Max Weight * *W*(*x,y*), where *W*(*x,y*) depends on the chosen aperture taper weighting schemes. For this selection Dolph-Tchebychev, Taylor, and Binomial are disabled.



1. Flat:

$$W(x,y) = 1$$

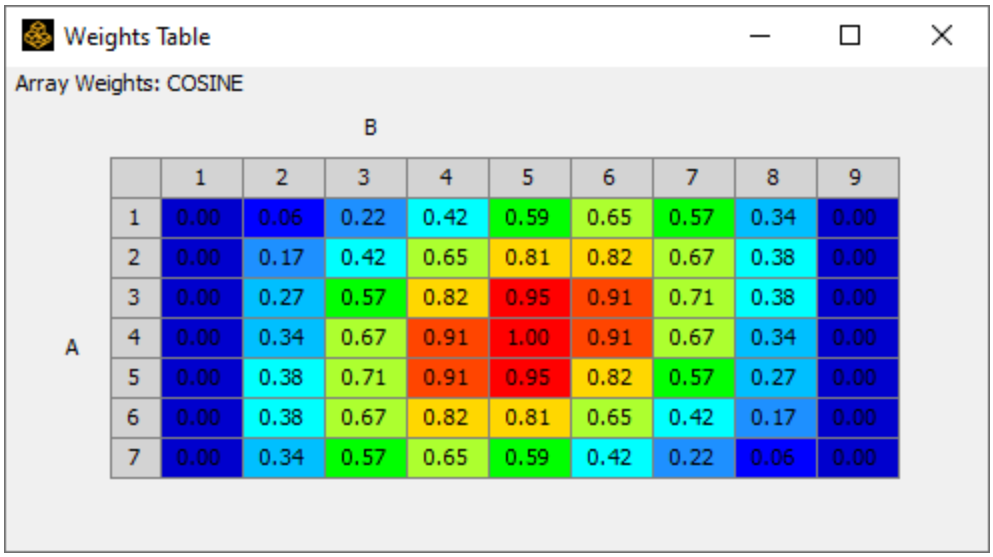
2. Cosine:

$$W(x) = (1-t)(\cos^{n}(\frac{\pi x}{X})) + t$$
$$W(y) = (1-t)(\cos^{n}(\frac{\pi y}{Y})) + t$$
$$W(x,y) = W(x) * W(y)$$

where *x* and *y* are the distance between current cell and the center of the antenna array in global x-y coordinate. X and Y are two times the longest distance between the edge of the array and the center of the array in global x-y coordinate.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Weights Table (Preview) for a 7x9 hexagon array that is aligned with global x-y coordinates:



	1	2	3	4	5	6	7	8	9
1	0.00	0.06	0.22	0.42	0.59	0.65	0.57	0.34	0.00
2	0.00	0.17	0.42	0.65	0.81	0.82	0.67	0.38	0.00
3	0.00	0.27	0.57	0.82	0.95	0.91	0.71	0.38	0.00
4	0.00	0.34	0.67	0.91	1.00	0.91	0.67	0.34	0.00
5	0.00	0.38	0.71	0.91	0.95	0.82	0.57	0.27	0.00
6	0.00	0.38	0.67	0.82	0.81	0.65	0.42	0.17	0.00
7	0.00	0.34	0.57	0.65	0.59	0.42	0.22	0.06	0.00

3. Triangular

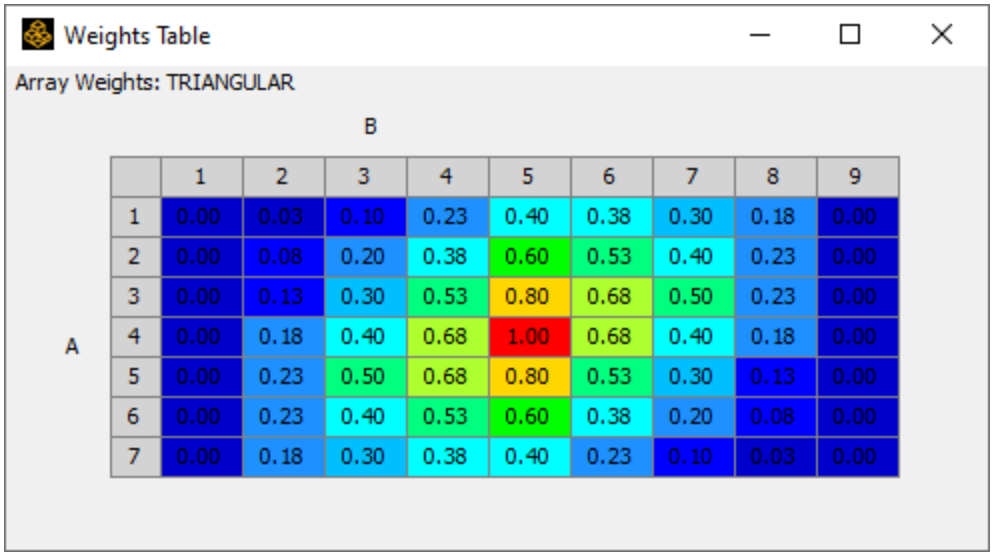
$$W(x) = (1-t)\left(1 - \frac{|x|}{X/2}\right) + t$$

$$W(y) = (1-t)\left(1 - \frac{|y|}{Y/2}\right) + t$$

$$W(x,y) = W(x) * W(y)$$

Weights Table (Preview) for a 7x9 hexagon array:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



	1	2	3	4	5	6	7	8	9
1	0.00	0.03	0.10	0.23	0.40	0.38	0.30	0.18	0.00
2	0.00	0.08	0.20	0.38	0.60	0.53	0.40	0.23	0.00
3	0.00	0.13	0.30	0.53	0.80	0.68	0.50	0.23	0.00
4	0.00	0.18	0.40	0.68	1.00	0.68	0.40	0.18	0.00
5	0.00	0.23	0.50	0.68	0.80	0.53	0.30	0.13	0.00
6	0.00	0.23	0.40	0.53	0.60	0.38	0.20	0.08	0.00
7	0.00	0.18	0.30	0.38	0.40	0.23	0.10	0.03	0.00

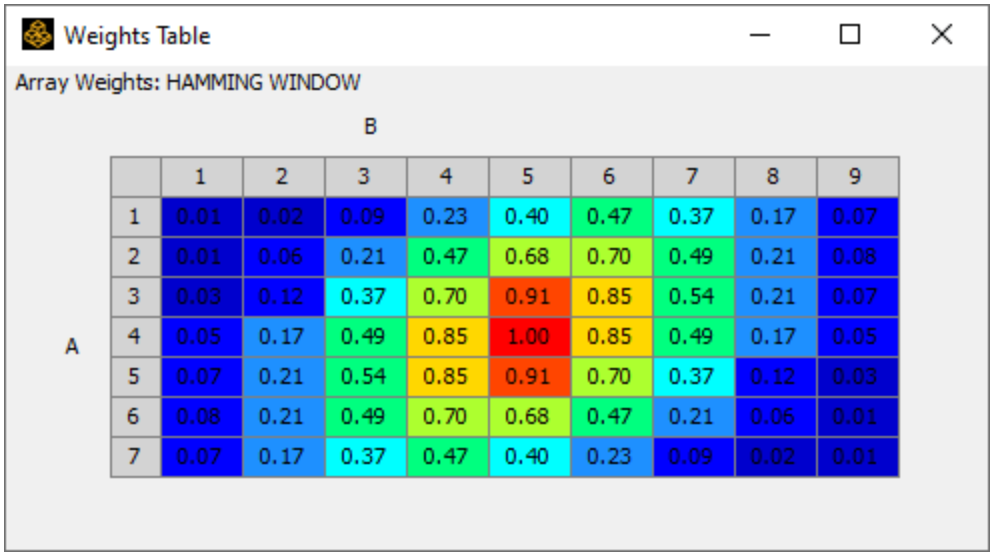
4. Hamming Window

$$W(x) = 0.54 - 0.46\,cos\left(2\pi\left(\frac{x}{X} - 0.5\right)\right)$$

$$W(y) = 0.54 - 0.46\,cos\left(2\pi\left(\frac{y}{Y} - 0.5\right)\right)$$

$$W(x,y) = W(x) * W(y)$$

Weights Table (Preview) for a 7x9 hexagon array:

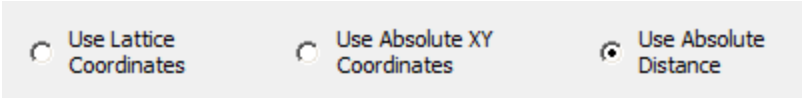


	1	2	3	4	5	6	7	8	9
1	0.01	0.02	0.09	0.23	0.40	0.47	0.37	0.17	0.07
2	0.01	0.06	0.21	0.47	0.68	0.70	0.49	0.21	0.08
3	0.03	0.12	0.37	0.70	0.91	0.85	0.54	0.21	0.07
4	0.05	0.17	0.49	0.85	1.00	0.85	0.49	0.17	0.05
5	0.07	0.21	0.54	0.85	0.91	0.70	0.37	0.12	0.03
6	0.08	0.21	0.49	0.70	0.68	0.47	0.21	0.06	0.01
7	0.07	0.17	0.37	0.47	0.40	0.23	0.09	0.02	0.01

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Absolute Coordinate - Distance

For absolute coordinate, the final weight is calculated as Max Weight * W(r), where W(r) depends on the chosen aperture taper weighting schemes. For this selection Dolph-Tchebychev, Taylor, and Binomial are disabled.



1. Flat:

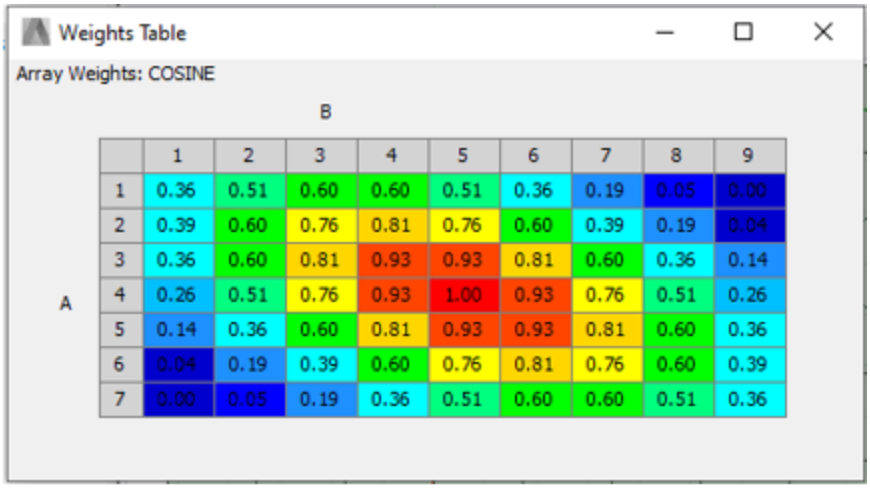
$$W(r) = 1$$

2. Cosine:

$$W(r) = [(1-t)(\cos^n(\frac{\pi r}{L})) + t]^2$$

where *r* is the absolute distance between current cell and the center of the antenna array, and *L* is two times the longest distance between the edge of the array and the center of the array.

Weights Table (Preview) for a 7x9 hexagon array:



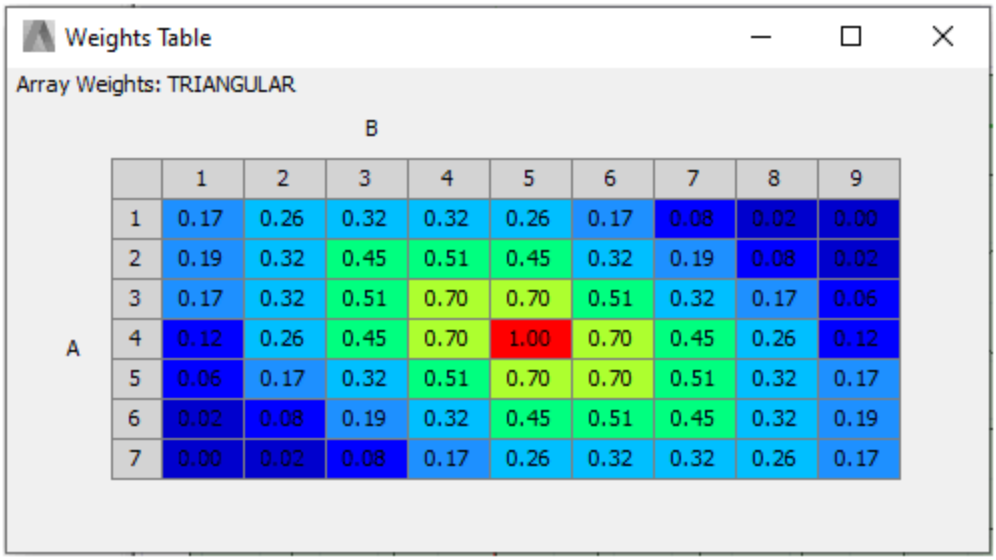
	1	2	3	4	5	6	7	8	9
1	0.36	0.51	0.60	0.60	0.51	0.36	0.19	0.05	0.00
2	0.39	0.60	0.76	0.81	0.76	0.60	0.39	0.19	0.04
3	0.36	0.60	0.81	0.93	0.93	0.81	0.60	0.36	0.14
4	0.26	0.51	0.76	0.93	1.00	0.93	0.76	0.51	0.26
5	0.14	0.36	0.60	0.81	0.93	0.93	0.81	0.60	0.36
6	0.04	0.19	0.39	0.60	0.76	0.81	0.76	0.60	0.39
7	0.00	0.05	0.19	0.36	0.51	0.60	0.60	0.51	0.36

3. Triangular

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$W(r) = [(1-t)\left(1-\frac{|r|}{L/2}\right)+t]^2$$

Weights Table (Preview) for a 7x9 hexagon array:



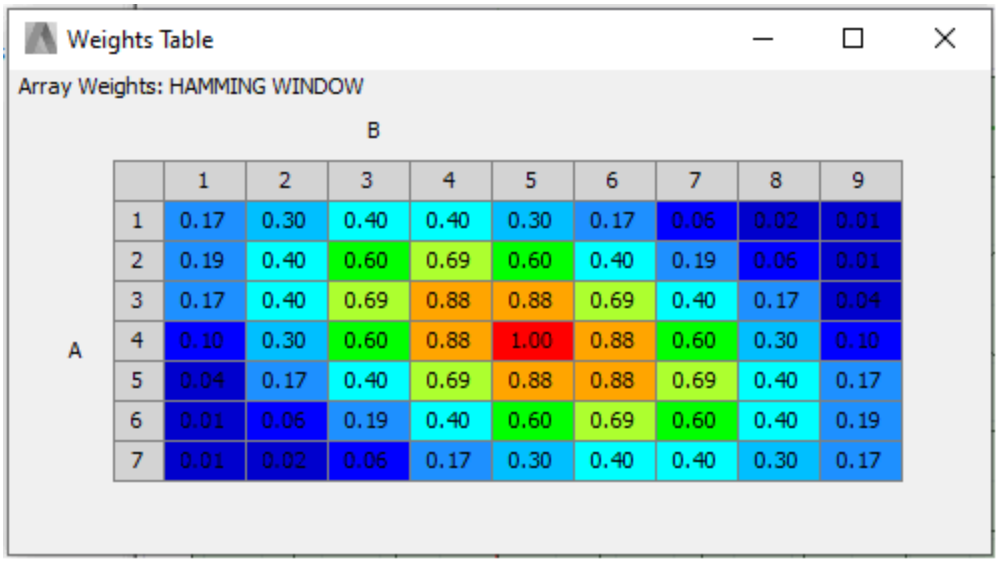
	1	2	3	4	5	6	7	8	9
1	0.17	0.26	0.32	0.32	0.26	0.17	0.08	0.02	0.00
2	0.19	0.32	0.45	0.51	0.45	0.32	0.19	0.08	0.02
3	0.17	0.32	0.51	0.70	0.70	0.51	0.32	0.17	0.06
4	0.12	0.26	0.45	0.70	1.00	0.70	0.45	0.26	0.12
5	0.06	0.17	0.32	0.51	0.70	0.70	0.51	0.32	0.17
6	0.02	0.08	0.19	0.32	0.45	0.51	0.45	0.32	0.19
7	0.00	0.02	0.08	0.17	0.26	0.32	0.32	0.26	0.17

4. Hamming Window

$$W(r) = [0.54 - 0.46\,cos\left(2\pi\left(\frac{r}{L}-0.5\right)\right)]^2$$

Weights Table (Preview) for a 7x9 hexagon array:

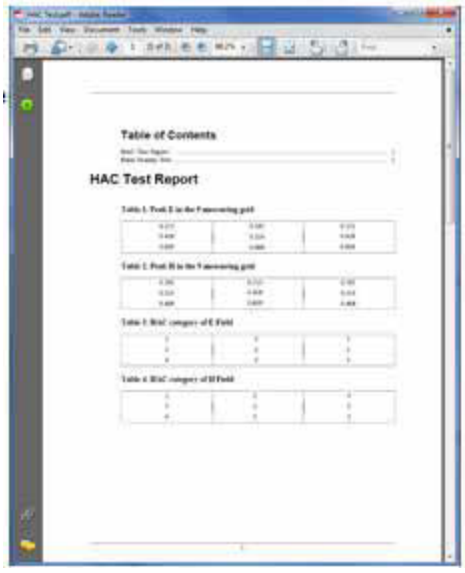
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

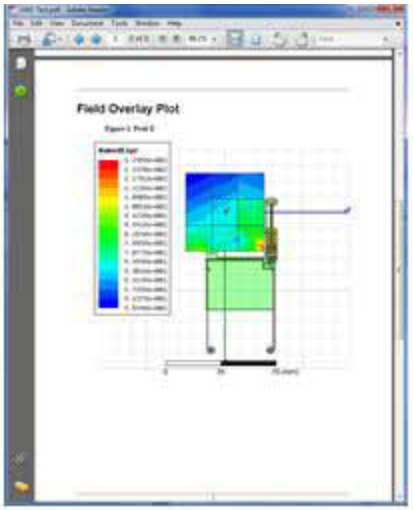


	1	2	3	4	5	6	7	8	9
1	0.17	0.30	0.40	0.40	0.30	0.17	0.06	0.02	0.01
2	0.19	0.40	0.60	0.69	0.60	0.40	0.19	0.06	0.01
3	0.17	0.40	0.69	0.88	0.88	0.69	0.40	0.17	0.04
4	0.10	0.30	0.60	0.88	1.00	0.88	0.60	0.30	0.10
5	0.04	0.17	0.40	0.69	0.88	0.88	0.69	0.40	0.17
6	0.01	0.06	0.19	0.40	0.60	0.69	0.60	0.40	0.19
7	0.01	0.02	0.06	0.17	0.30	0.40	0.40	0.30	0.17

Hearing Aid Compliance Test

Using a toolkit approach, you can test and report the hearing aid compatibility of your designs in HFSS. This involves an easy access to a python script that provides custom GUIs to set up the non-model measurement plane and sweep setup that satisfy the ANSI C63.19 standard. Various reporting mechanisms are created automatically. Reporting mechanisms include field overlay plot and UDD (User Defined Document).





Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The python script is installed with the software. The script assumes that you have already created the phone model and the appropriate relative coordinate system that defines the location and orientation of the ear piece. You must also assign appropriate boundaries/excitations and any other required design data, such as solution setup.

The python script provides default but customizable UI and canned automations. The UDD format can be adjusted/extended as needed. The default UI is a modal dialog.

This allows us to support changing standards, or multiple standards, without necessarily being tied to the product release cycle. Similarly, users can create new UDD to customize the test report. UDD are saved in project result directory. This type of reports can be created prior to simulation, but will only be populated when there is solution.

Once the measurement plane, sweep setup and various reports are created by the Toolkit script, they become part of the design and users will be able to make modifications. However, that might cause the HAC test to fail compliance with the standard.

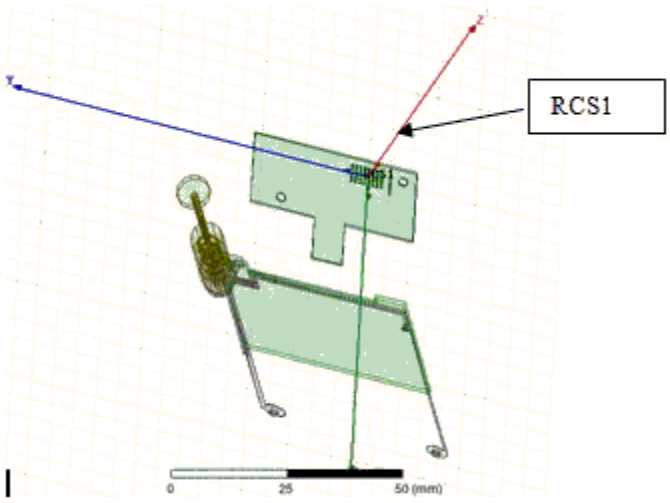
The measurement plane will be parameterized based on the variables used in the RCS definition of the ear piece. The measurement plane is a non-model object and the automated UI operations executed by the python script will not invalidate previously simulated results. This means that users can add the HAC analysis at any stage of their design process.

Using the Hearing Aid Compliance Test

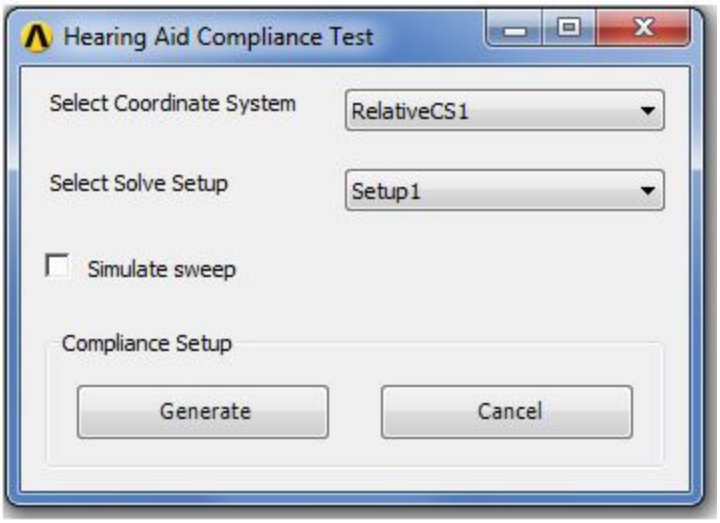
The script assumes that you have already created the phone model and the appropriate relative coordinate system that defines the location and orientation of the ear piece. You must also assign appropriate boundaries/excitations and any other required design data, such as solution setup.

1. Insert a phone model and create the relative coordinate system that defines the location and orientation of the ear piece. The name of the relative coordinate system for the earpiece is predefined.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **HFSS>Toolkits>HearingAidCompliance** to bring up the “Hearing Aid Compliance Test” dialog.



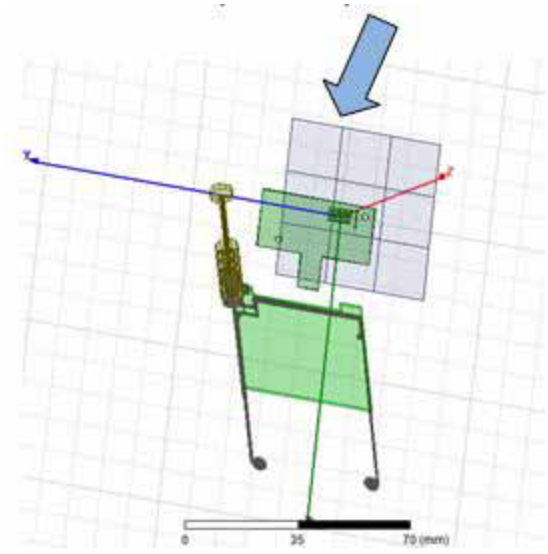
The first combo box is populated with user defined relative coordinate systems and the second is populated with the solve setup in the active design. A sweep will be created within the selected solve setup.

Users can choose to include simulating the created sweep as part of the automation.

3. Click **Generate**.

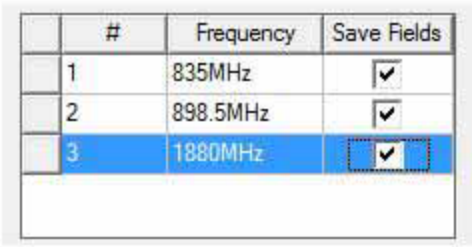
A measurement plane composed of 9 non-model rectangles are inserted into the design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



These 9 rectangles will be named hac_grid1_1, hac _grid1_2, hac _grid1_3, hac _grid2_1, … hac _grid3_3. If existing objects in the design that conflict with any of these 9 names, the python script searches for the next set of names to use, in the format of hac1_grid1_1, hac1 _ grid1_2, hac1 _grid1_3, hac1 _grid2_1, … hac1 _grid3_3, so on and so forth.

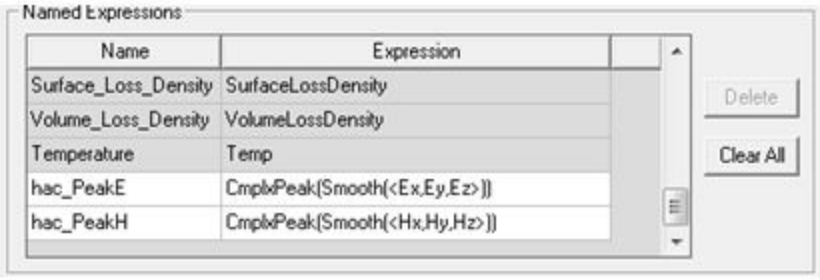
4. A discrete sweep is created for the selected solve setup. The frequencies required in the ANSI standard will be included in the sweep. You can select the testing frequencies. This sweep will be named “HAC_Sweep”. When there is already a sweep of the same name, then the next name to use is “HAC1_Sweep”, “HAC2_Sweep”, so on and so forth.



	#	Frequency	Save Fields
	1	835MHz	☑
	2	898.5MHz	☑
	3	1880MHz	☑

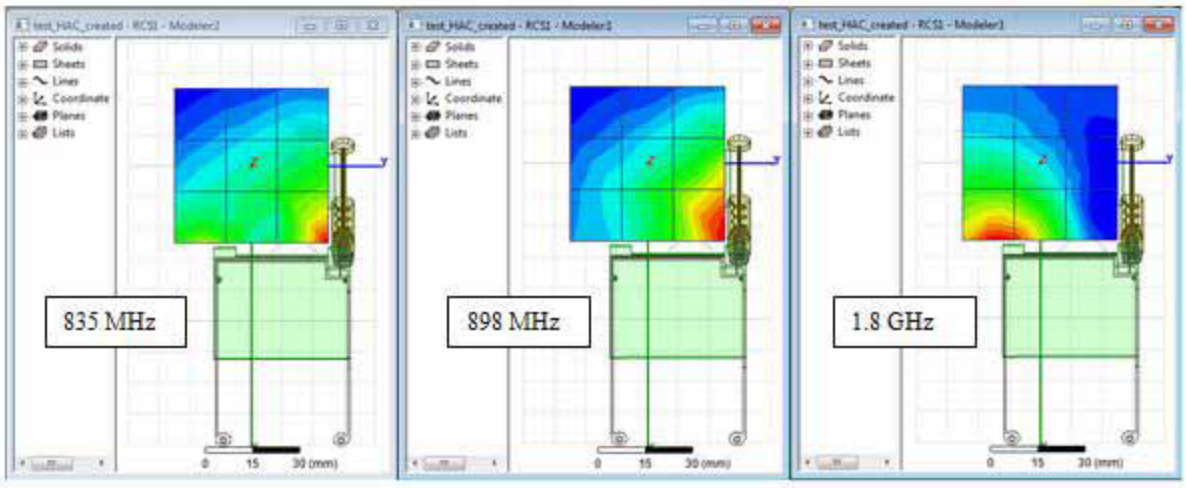
5. You should create the highlighted field calculator expressions.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



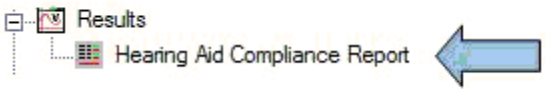
These field calculator expressions can be plotted/evaluated on any user defined polyline.

6. You can plot PeakE and PeakH Field overlay on the surface of the measurement plane, using the sweep solution and at each of the frequencies.



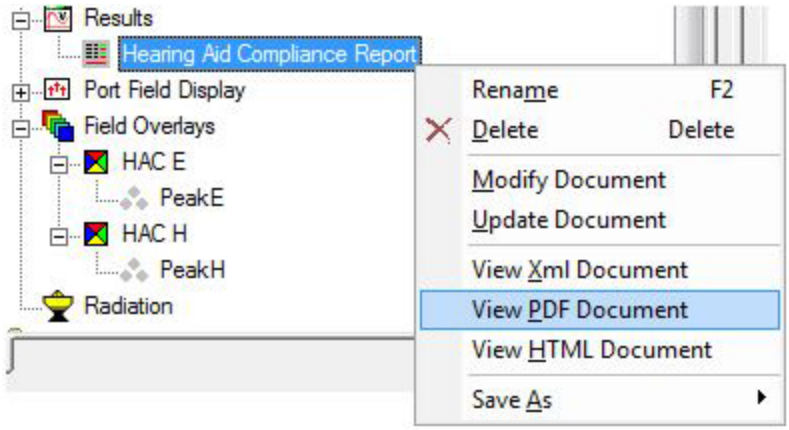
If you did not select the correct coordinate system to generate the measurement grid, and therefore, no solution to populate the plot, there will be an error message.

7. If you select **Simulate Sweep**, a User Defined Document will be created and inserted into the Results folder.



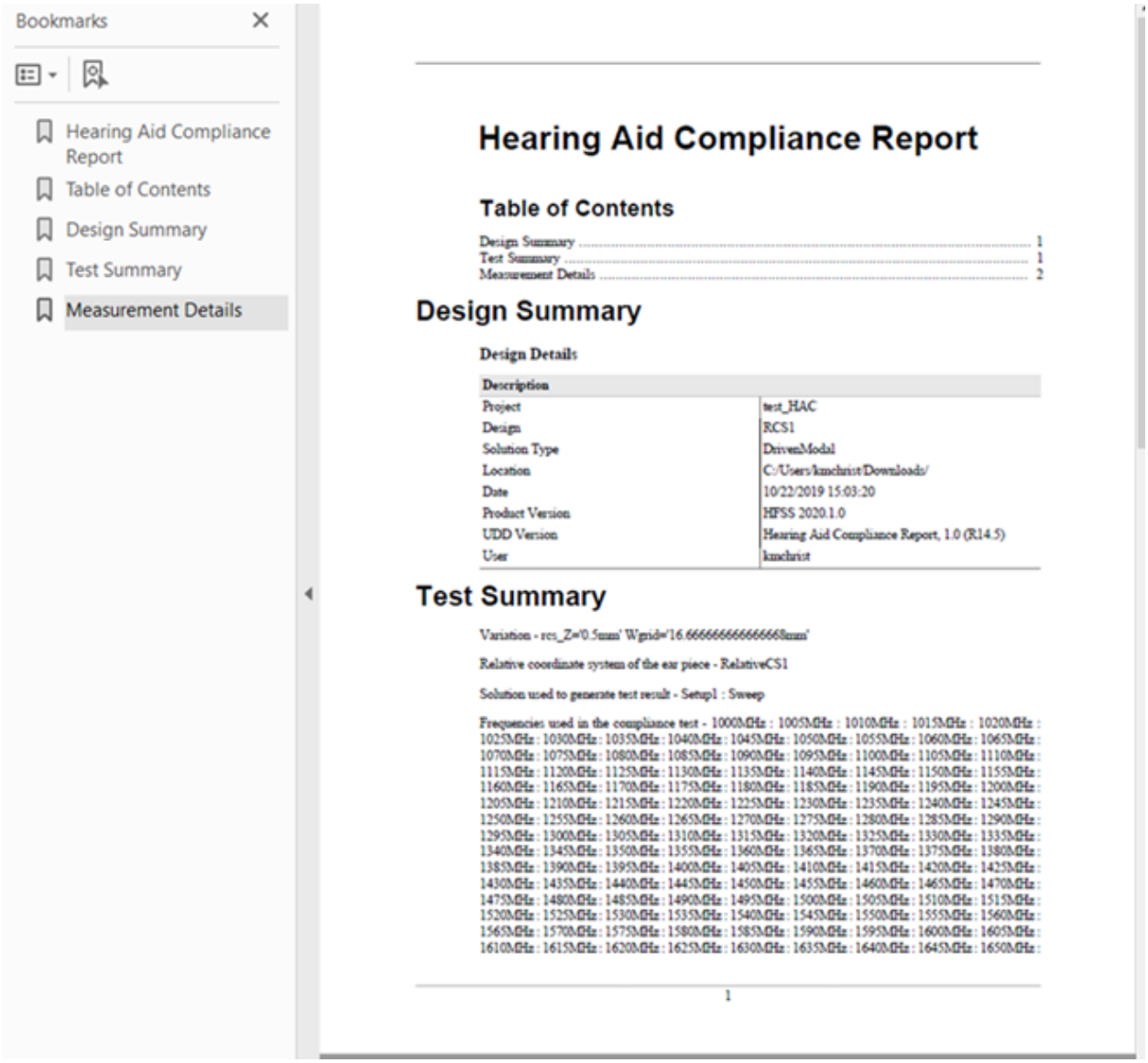
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Right-click and select “View PDF Document” to display the PDF



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Hearing Aid Compliance Report opens:

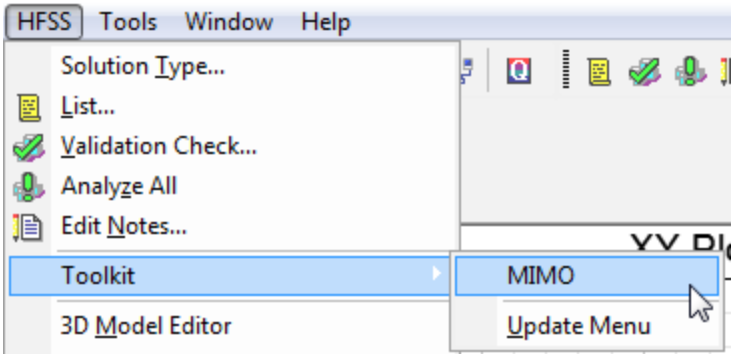


User Defined Solution for MIMO Calculations

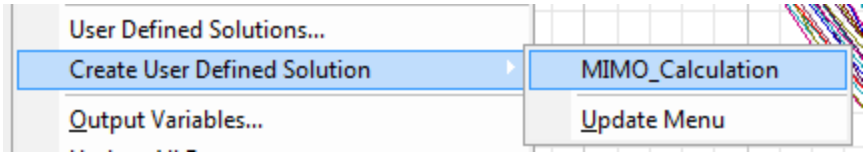
HFSS includes the means to use python scripts for developing Toolkits for User Defined Solutions and User Defined Outputs for specific applications. The MIMO.py script in the syslib>Toolskits directory and the MIMO_Calculation.py script in the syslib>UserDefinedOutputs directory illustrate this technique as applied to an important design application. This toolkit does not work with Linux.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You will need to click **HFSS>Toolkit>Update Menu** to cause the **Toolkit** menu to display the MIMO selection.



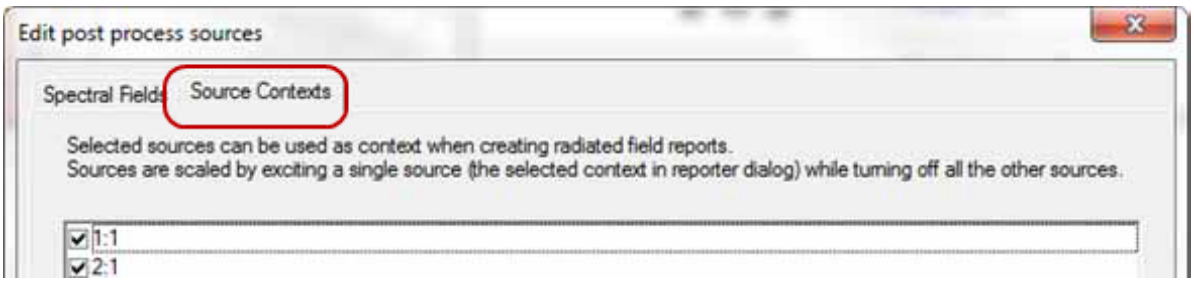
You also click **Results>Create User Defined Solution>Update Menu** to display the MIMO_ Calculation selection.



The use of Multiple Input Multiple Output (MIMO) design in both transmitters and receivers has become an important technique applied toward improved communication. The MIMO calculations also illustrate the process that applies toward generating User Defined Reports. In this case, the Toolkit and User Defined Solutions help you obtain reports of MIMO Calculations of the Envelope Correlation Coefficient and Diversity Gain. You can also use these MIMO quantities in Optimetrics. You can use these scripts with any HFSS Modal or Terminal Network Analysis design with at least two inputs.

Once you have selected a design and have (if necessary) activated the scripts (Update Menu), the MIMO process includes the following steps.

1. In the **Edit Sources** dialog box, use the **Source Contexts** tab to specify the sources to use as context when generating radiated fields.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

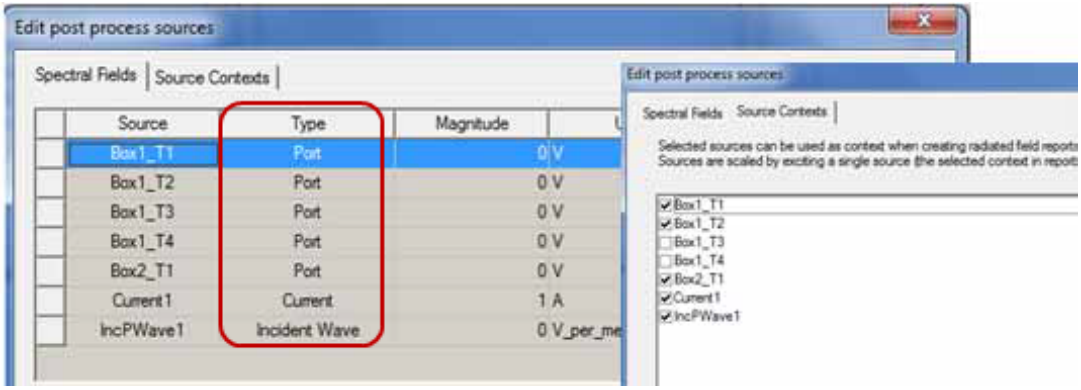
The **Source Contexts** tab is only available in HFSS Driven Modal and Terminal Network Analysis solution types.

It is not available in

- Composite Excitation sub-solution type
- Design with array setup
- Eigen and Transient solution type

To run the MIMO calculations example, you need a driven Modal or Terminal Network Analysis design with at least two sources. The default for all sources is unchecked (disabled).

All types of sources can be enabled as source context.



Each singly selected source is excited with factory default values. The phase will always be 0 deg and the magnitude will be 1 Watt for modal design and 1 Volt (Total voltage) for terminal design. These factory default values cannot be changed by users.

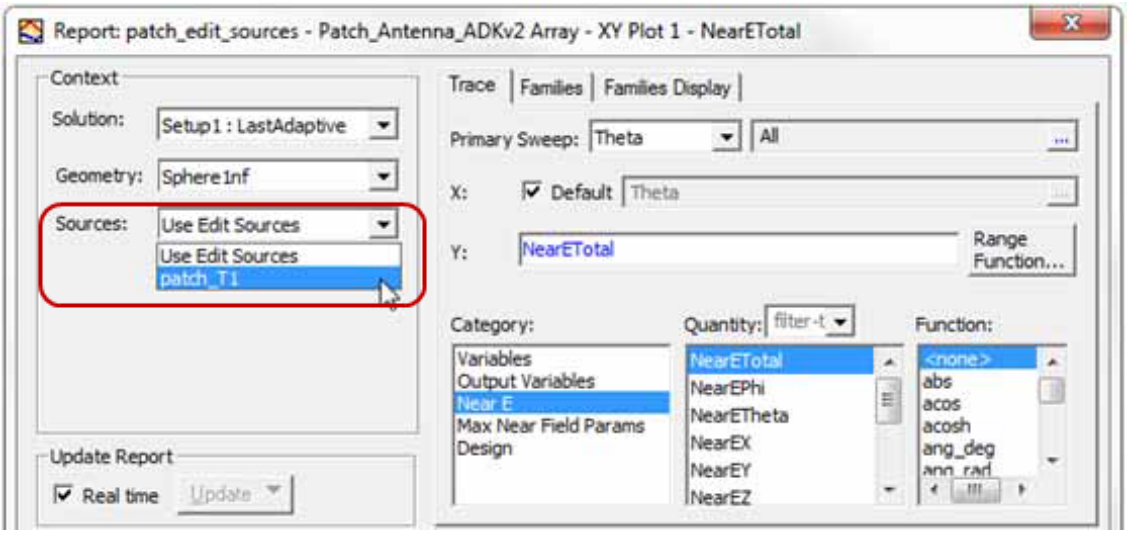
Design **Edit Source** settings such as "Include Port Post Processing Effects" will still be applicable for these singly excited sources. Thus changing any setups on the **Spectral Fields** tab of the **Edit Source** dialog will invalidate all traces that are defined with source context.

Enabling any source as context will not impact traces that are already defined

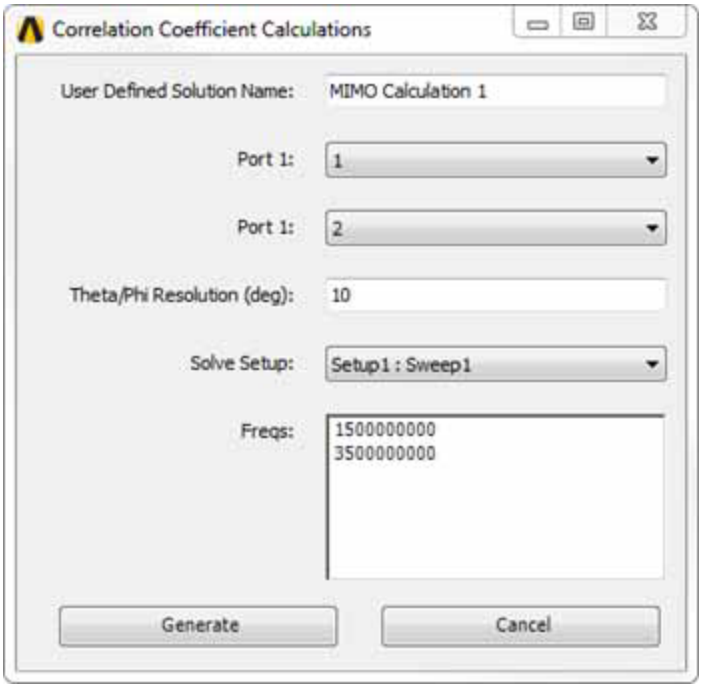
Disabling source as context will invalidate an existing trace that uses that source as context

2. You can scale sources by exciting a single source while using the Reporter. A Sources combo box appears in the Reporter when you have specified at least one source in the **Edit Sources** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



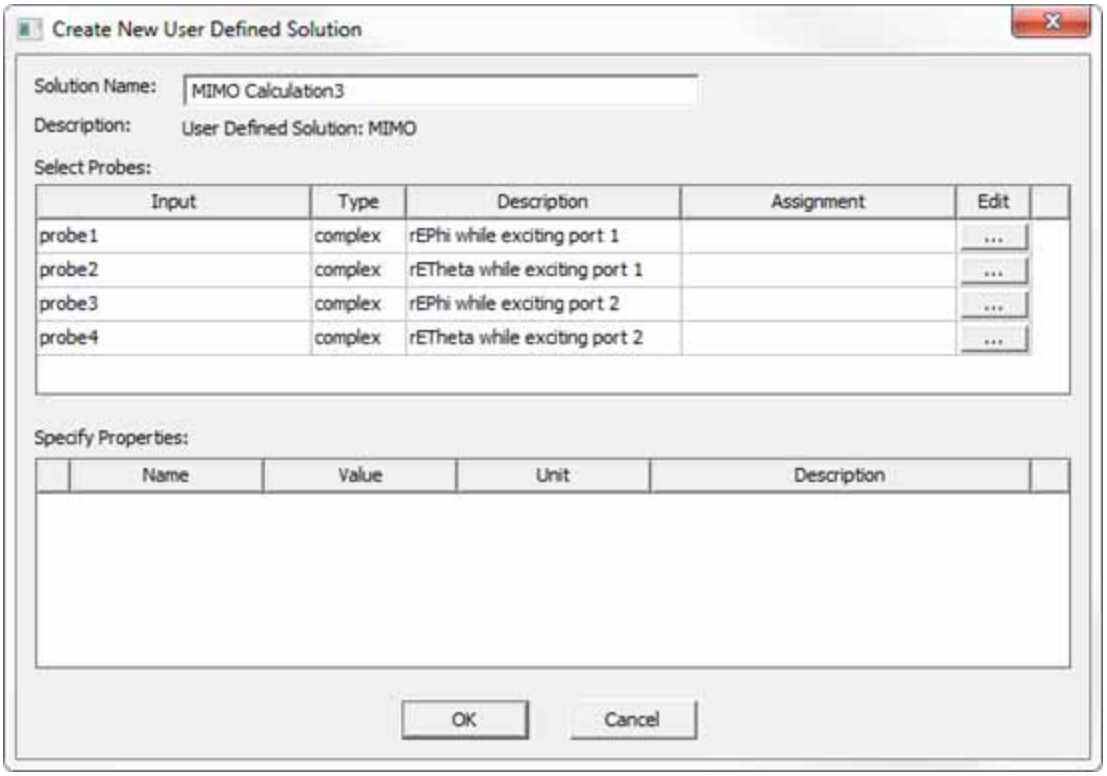
3. Select **HFSS>Toolkit>MIMO** to view the **Correlation Coefficient Calculations** dialog.



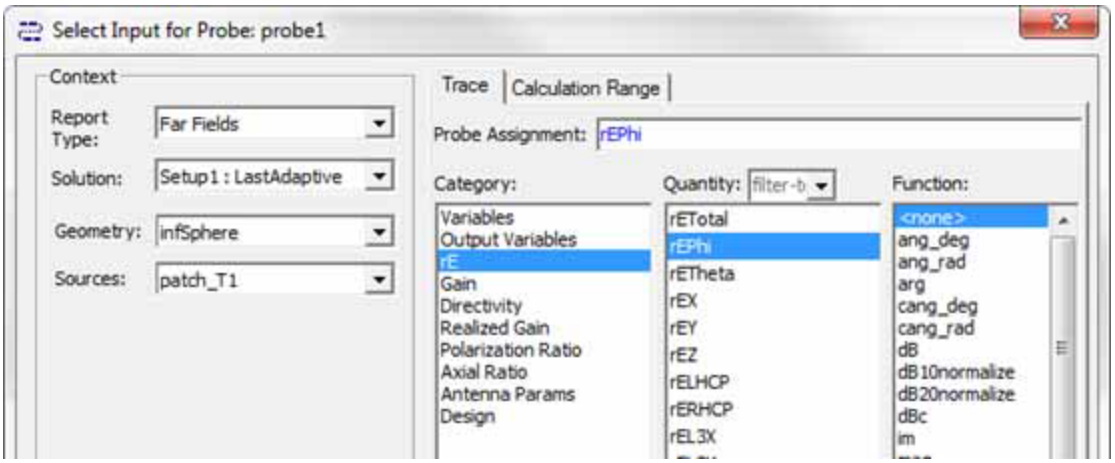
From here you can select Ports, specify the Theta/Phi resolution in degrees, and select the Solve Setup. Click Generate to apply the settings.

4. Select **Results>Create User Defined Solution** to select Input probes in the **Editing User Defined Solution** dialog for the MIMO calculation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



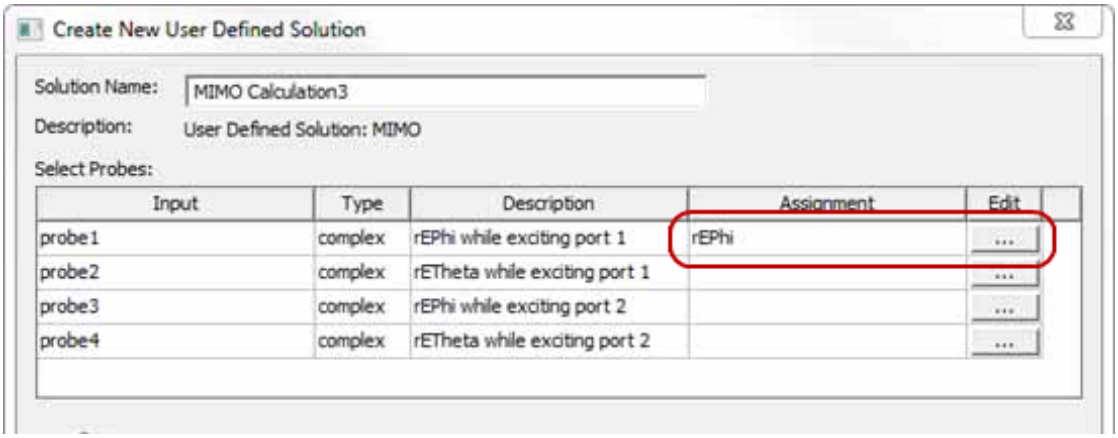
From here, you can also make assignments for the sources. Click the ellipses [...] buttons in the Edit column to open a **Select Input** dialog for the respective probe.



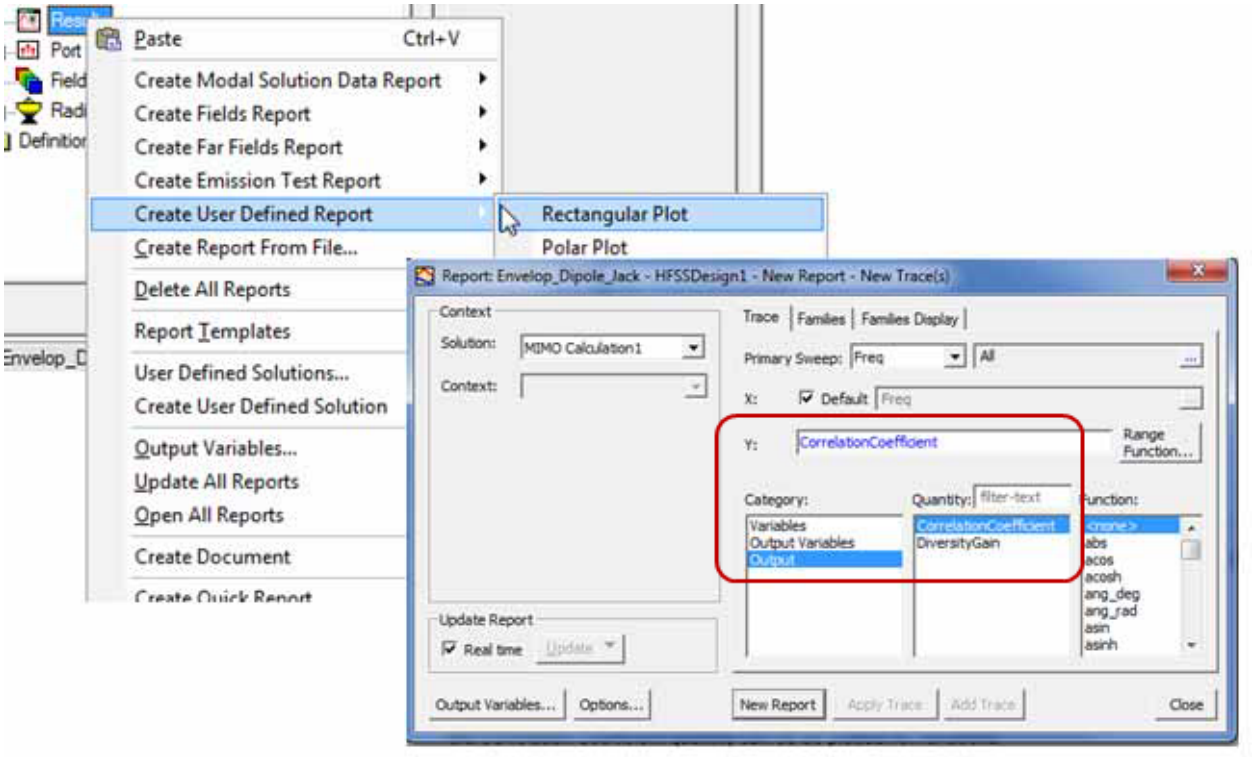
If you edit the Probe assignment here, and OK the dialog box. You select Probe assignment category, quantity and function based on the Description in the User Defined Solution. The Reporter Probe Assignment field restricts the legal values to the Description field. Red text denotes illegal assignments, and blue, legal. The assignment value for that probe appears in

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the **Create User Defined Solution** dialog.

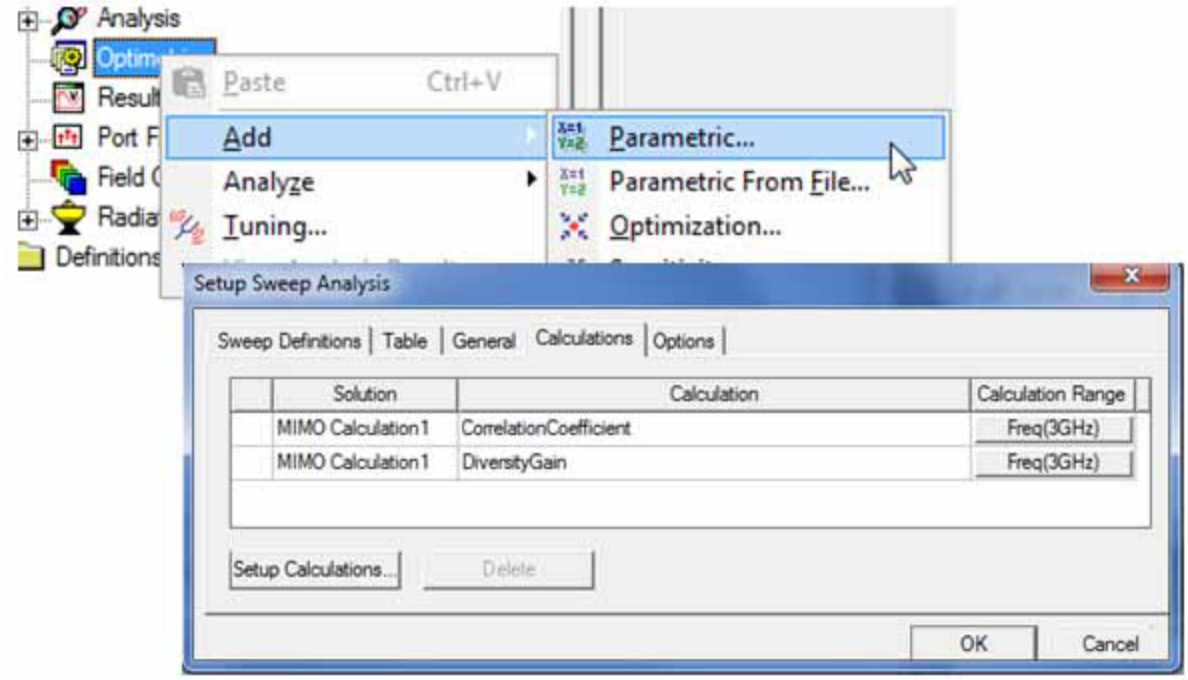


The created solution “MIMO Calculation1” has 2 output quantities, the “CorrelationCoefficient” (Envelope Correlation Coefficient) and “DiversityGain”. These quantities can be used in both reporter and Optimetrics. For the Reporter, you can select Output as the Category to select CorrelationCoefficent and Diversity Gain as Quantities.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

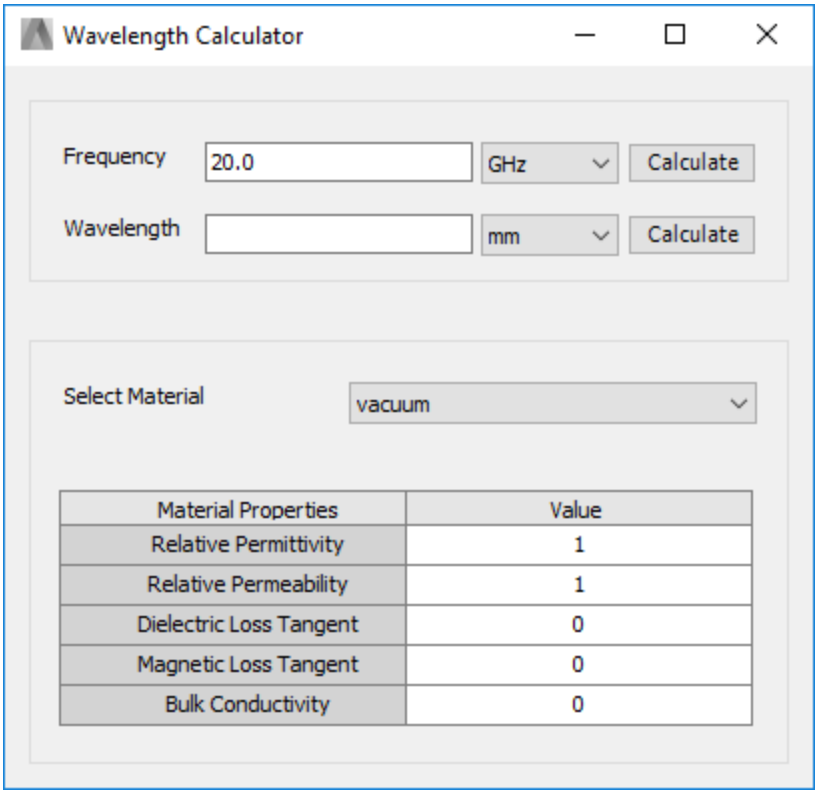
For Optimetrics, the MIMO calculations appear in the Sweep Definitions.



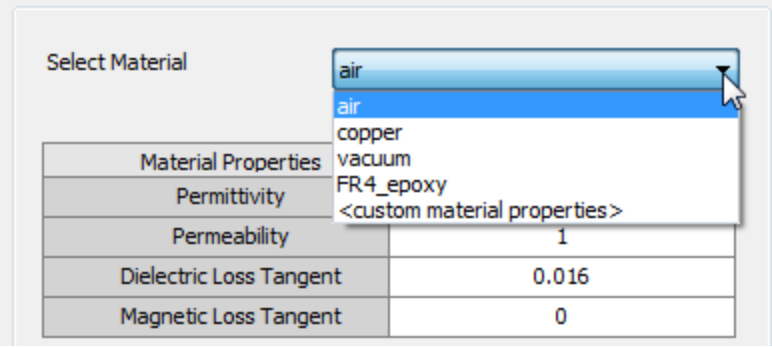
Wavelength Calculator

To use the Wavelength Calculator toolkit, click **HFSS>Toolkit>WavelengthCalculator**. This opens the **Wavelength Calculator** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

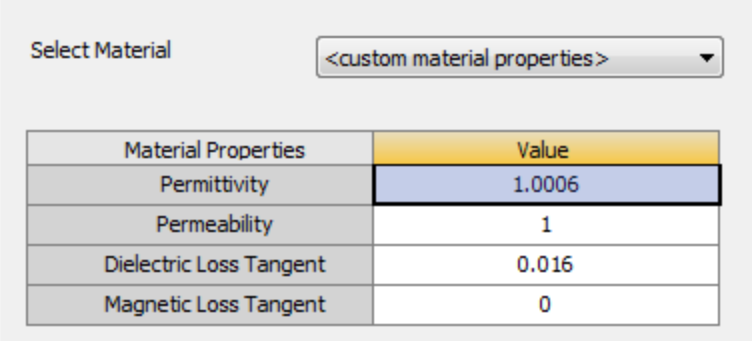


- The Frequency field initially displays the adaptive frequency from the first setup. The field is editable, and you can select the units from a pull down menu.
- The Wavelength field is initially blank, but editable, and you can select the units from the drop-down menu.
- The Select Material menu includes a list of all materials in the current project



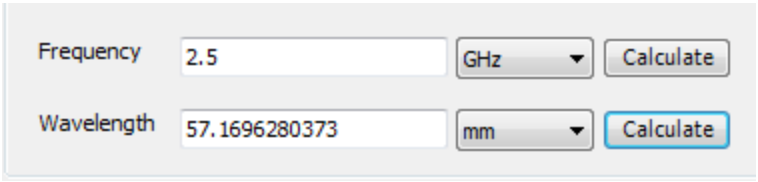
If you select <custom material properties>, the fields for material properties become editable.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- The display shows current material values for Permittivity, Permeability, Dielectric Loss Tangent, and Magnetic Loss Tangent.

To calculate the Wavelength at the given frequency, click the **Calculate** button adjacent to the Wavelength units. This updates the field for the wavelength.



Once you wave calculated or entered a wavelength, and selected or specified for material values, you can use the **Calculate** button adjacent to the Frequency units to calculate a frequency.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12 - HFSS Transient

HFSS Transient is a solution type within HFSS. It employs a time-domain ("transient") solver. Typical applications include, but are not limited to:

- Simulations with pulsed excitations, such as ultra-wide band antennas, lightning strikes, electro-static discharge;
- field visualization employing short-duration excitations;
- time-domain reflectometry.

The interface of HFSS Transient is much like the Terminal Driven solution type in the frequency domain. The geometry creation and the mesh are almost identical. Boundary conditions and definitions of ports or incident waves are almost the same, except for some restrictions in cases where certain frequency-domain options do not apply in the time domain. Therefore, you can often change an existing Terminal Driven design into a Transient design and vice versa. This release offers a beta feature for HFSS Transient Modal Ports to allow Multi-mode Waveguide Simulations in Transient, allowing you to mix wave ports and terminals.

> **Note:**
>
> Before changing an existing design between Terminal Driven and Transient, you must remove the setups and solutions.)

The Analysis Setup for the transient solver is different from that of the frequency domain. Two types of analysis setup are offered:

- **Transient Network Analysis** solution types - excitations are identical on all active ports and are simulated one at a time to facilitate the data collection. Saved fields can be visualized, and voltages and currents can be monitored. Your choice in setting the Transient solution type affects the options for the setup. If you select **Transient Network Analysis** the setup includes a tab for the Excitation for the simulation.
- **Transient Composite Excitation**Composite solution types - Different ports can have different excitations. All Active excitations are launched in one simulation. No S-parameters or TDR results may be available, but saved fields can be visualized, and voltages and currents can be monitored.

Other issues for transient solves include:

- Fields will only be saved on pre-selected surfaces, because saving all 3D fields for all times would take more disk space and I/O time than generally desired.
- If you change the size of the radiation boundary for a Transient simulation, you should expect to see changes in delay for far field results. This is because there are two origins of system for transient: one for space and one for time. For space, the original is defined the same way as frequency domain. For time, the original is located a point infinitely far away.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

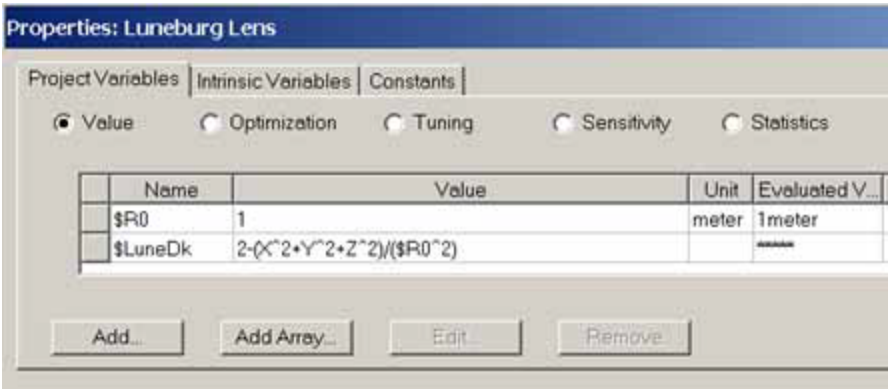
This choice of this time origin is logical because if you place the origin at the original of a space system (in this case the center of air box) then there will be very very long time delay for every transient far field. So for the cases we have here, when air box is larger, the distance from the air box (or current source) is shorter to the time origin. Therefore we are expecting to see different time delays as radiation boundary size changes.

Materials for Transient Solution Types

Materials for transient solutions must meet these requirements.

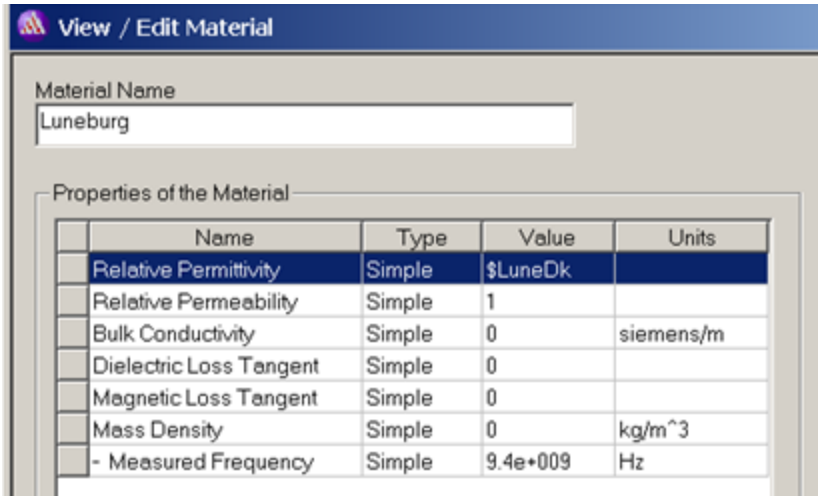
- Materials cannot have arbitrary frequency dependence in a time-domain analysis. For lossy dielectrics, a Debye model is applied to ensure that the loss remain physical across the spectrum. For lossy metals, a Padé approximation is used for the same reason.
- When lossy materials touch a port, the port solver will launch an excitation that fits the lossless case. This is inherent to most time-domain solvers. The 3D part of the simulation will take losses into account correctly.
- Spatially dependent material properties can be specified through user-defined variables in HFSS. Spatially dependent materials are usually applied for modeling dielectric lens, dielectric waveguides, substrates with graded doping profiles, and graded-index fibers.

  For example, to set the dielectric permittivity of a Luneburg lens, you would create a variable, as shown in the following figure:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You would then assign the variable value to the material in View/Edit Material:

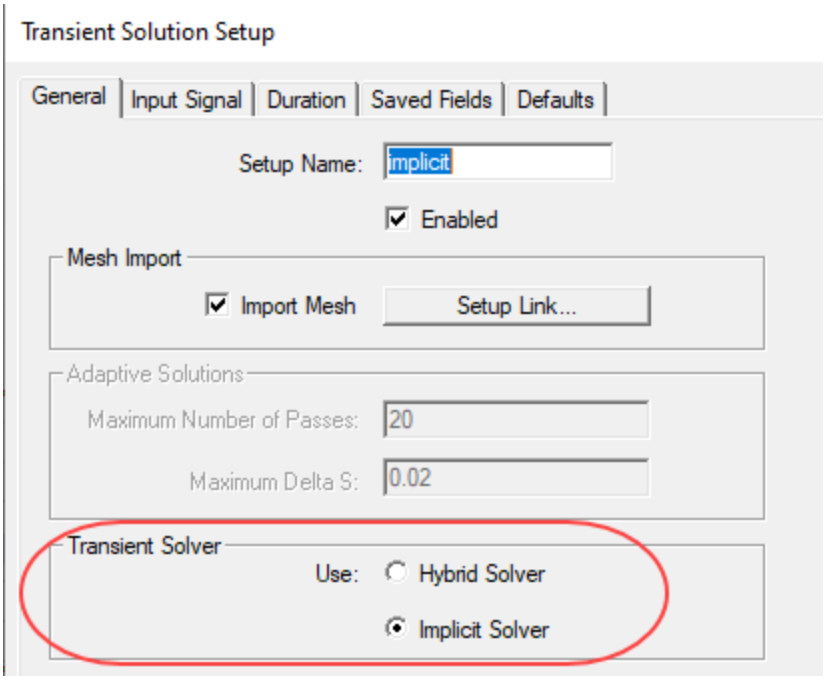


HFSS Transient current design allows ISODIEL and ISODEBYE materials to be spatially dependent. NPOLEDEBYE (poles and residues input from GUI) materials are NOT allowed to be spatially dependent because such models assume homogeneous material properties. The poles and residues are input from the GUI and there is no pole fitting involved. A fixed set of poles and residues cannot be used for inhomogeneous materials.

Diagonal Anisotropic Material Support for the Implicit Transient Solver

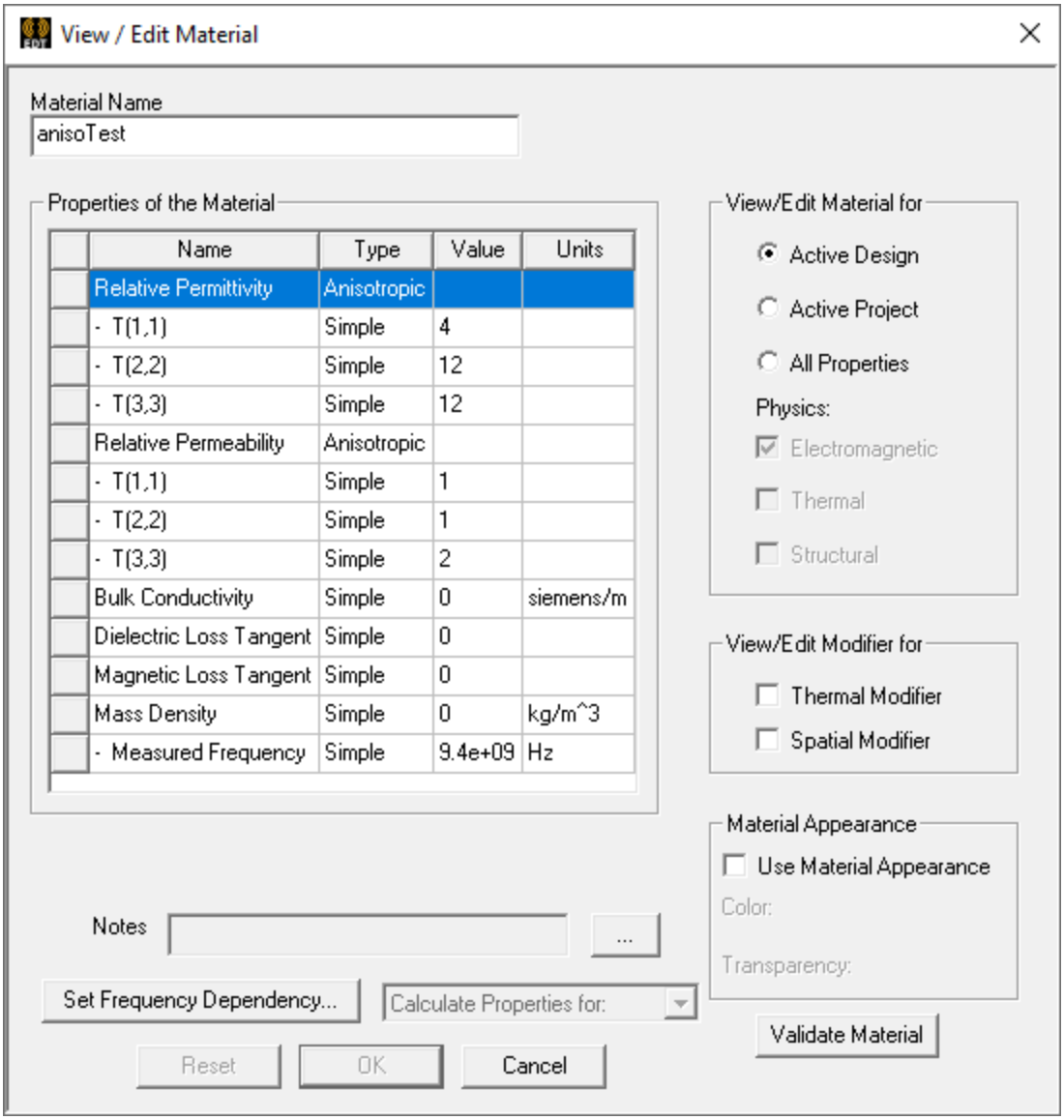
HFSS Transient supports diagonal anisotropic material for the Implicit Solver. If you launch a hybrid solve the Message window displays an error message.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



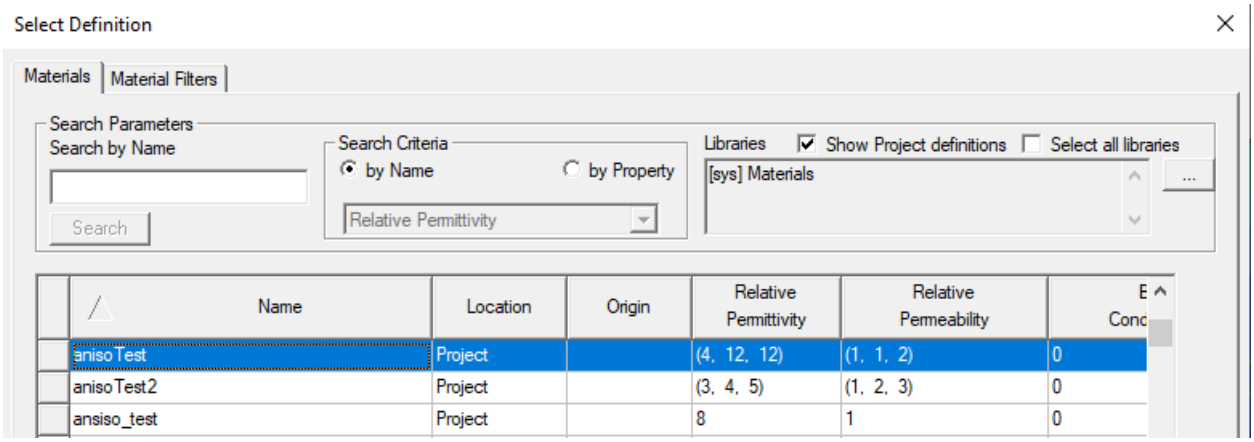
Support is only for diagonal anisotropic materials (not full tensor) and no space dependence or frequency dependence are allowed. Diagonal anisotropic combined with frequency dependent or spatial dependent fails validation for transient. You specify such material using the Materials intereface.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once created, you can select them in the Materials dialog in the library where you create them.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



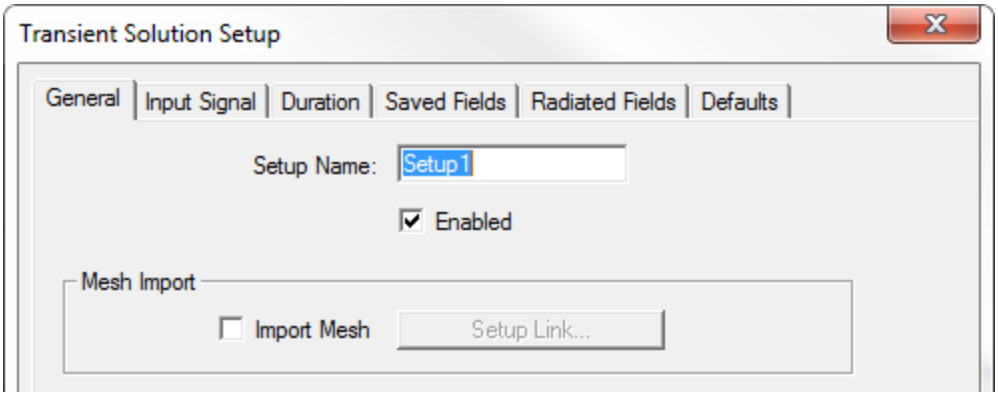
Diagonal anisotropic materials are not allowed to touch port and radiation boundaries. The Message window displays an error if anisotropic materials touch port and radiation boundaries.

Adding a Solution Setup for Transient Solutions

When you specify the Transient solution type, you also specify the focus of this setup as either **Transient Network Analysis** or **Transient Composite Excitation**. The **Transient Network Analysis** setup includes an **Input Signal** tab, as shown in the figure. If the model contains a radiation boundary, the setup includes a Radiated Fields tab.

To add a Solution Setup for a Transient design:

1. Click **HFSS>Analysis Setup>Add Solution Setup** to open the **Transient Setup** dialog with the **General** tab selected.



2. Optionally check Import Mesh for mesh linking.

Otherwise, the mesh for the transient simulation is generated by a regular frequency-domain simulation. For that simulation, the software decides on the appropriate frequency at which to perform the adaptive passes. It uses mixed element orders and the iterative solver.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. For Adaptive solutions, you can specify:
- Maximum Number of Passes
- For **Device characterization** solutions, you specify a Maximum Delta S.

  For **Field Visualization** solutions, you specify a Maximum Delta Energy for convergence per pass.

4. For Transient Solver, select **Hybrid** (the default) or **Implicit**.
- The Hybrid solver is based on the explicit-implicit discontinuous Galerkin time domain method. It is more efficient for electrically large problems. If the GPU is enabled, only the explicit part of the hybrid solver is running and being accelerated by GPUs. For more information, see Technical Notes: Transient Solution Theory.
- The implicit solver is based on the finite element time domain (FETD) method with an implicit time stepping. Change the setting to Implicit for cases which the Hybrid Solver may not perform optimally. This typically refers to structures which are discretized into a small number of tetrahedrons but the required time step for the Hybrid becomes extremely small hindering the performance. Consequently, it takes longer for the simulation to finish. Furthermore, the Implicit Solver is in general better suited for low frequency analysis of electrically small structures with small vias, thin wires, thin slots, narrow gaps, and thin dielectric/metal plates. For more information see Technical Notes: Transient Implicit Solver.

5. For a **Transient Network Analysis** solution, select the Input Signal tab to create a time profile.
6. Select the Duration tab to specify the simulation stop criteria.
7. Use the Saved Fields tab to select the face or object lists for which to calculate fields. By checking the boxes you can enable the Start saving fields time, and Save fields at interval. When you select an object list, you can also specify the **Maximum Number of Samples**.
8. If your design includes a radiation boundary, you can select the Radiated Fields tab to if you intend to view radiated fields.

Transient Network Analysis Input Signal Tab: a Time Profile

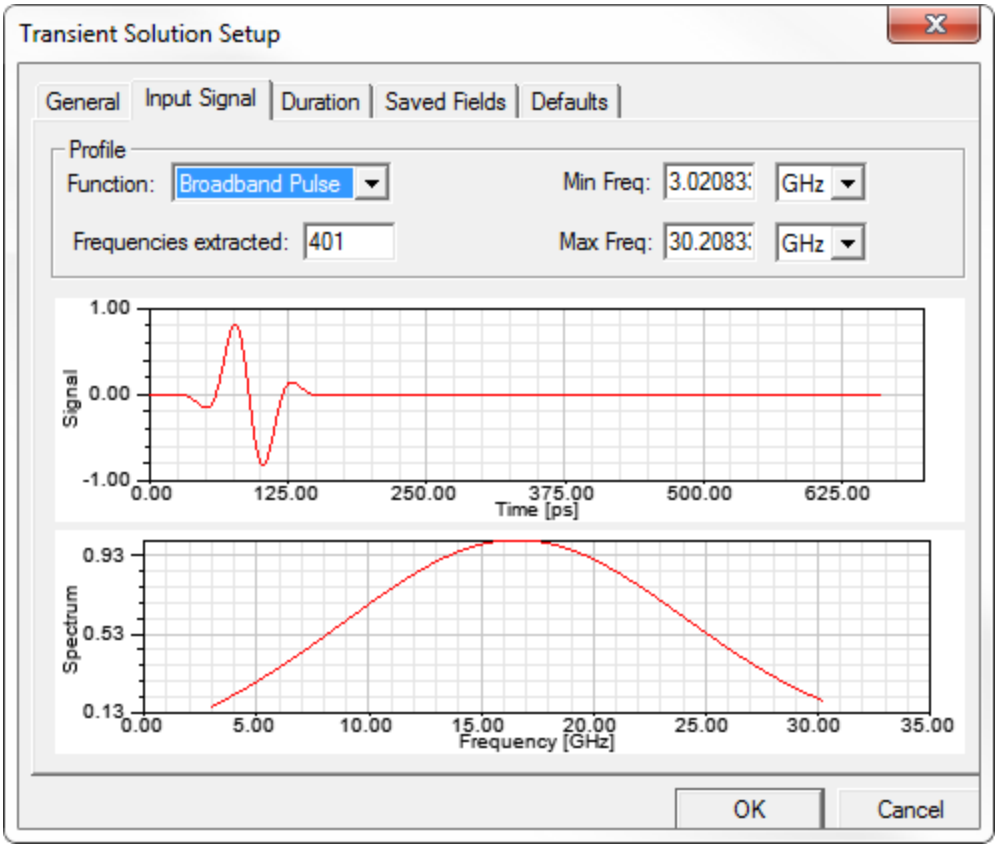
A time profile defines the pulse used to excite a **Transient Network Analysis** solution type. To create a time profile:

1. Select the **Input Signal** tab of a **Transient Network Analysis** setup.

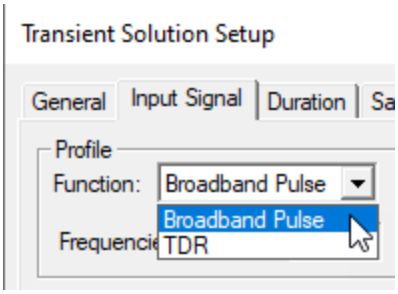
The **Input Signal** tab lets you select the Function set and other parameters. The profile includes two plots. The upper plot shows the excitation of interest given the currently

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

specified parameters. The lower plot shows the energy spectrum of the upper plot



2. You specify the function to use for the Profile by selecting from the drop-down menu.



For network analysis, all excitations are the same.

The wave plots are updated if you select a different function or edit the parameters.

1. Set the parameters for the selected Broadband Pulse or TDR function.

The following table lists the available functions and the parameters that apply to the upper plot. The energy spectrum plot parameters are always Frequency and Max Frequency.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

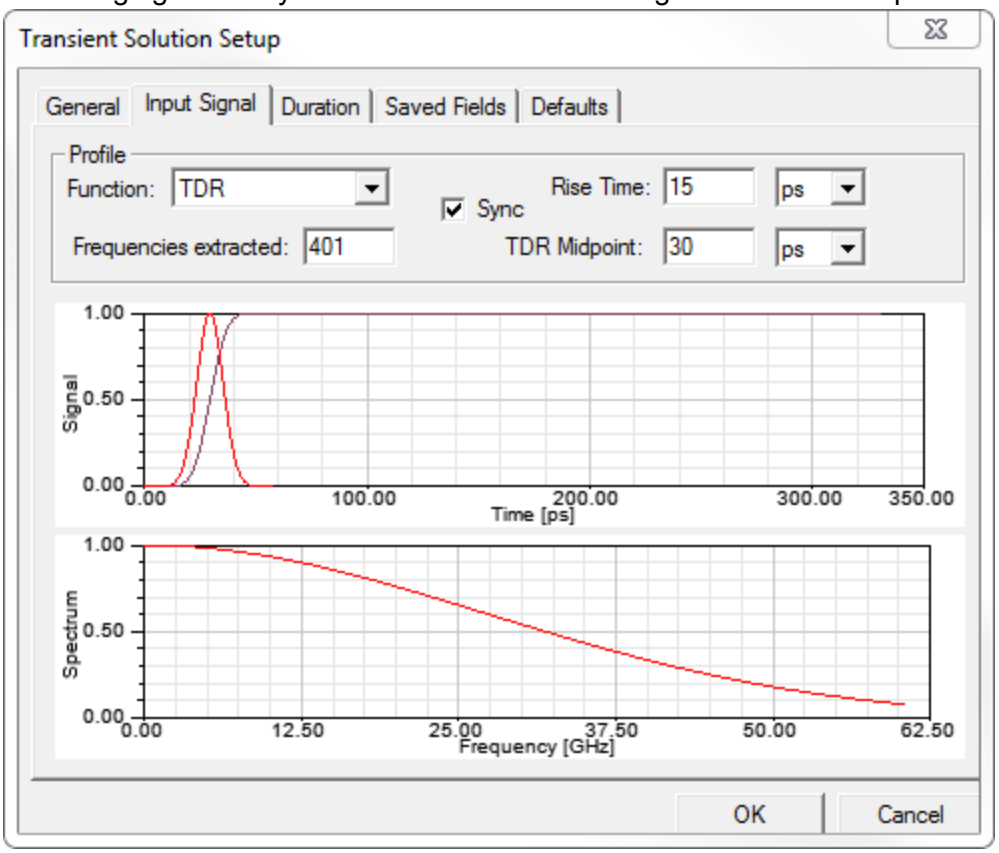
Function	Upper Plot Parameters	Description
Broadband Pulse	Min Freq. Frequencies extracted Max Freq.	A Min Freq. of 0 Hz. is allowed. This changes the shape of the pulse. A time profile that has a strong spectral content between the Min and Max frequencies. This can be used for such purposes as extracting S-parameters in that frequency range.
TDR	Rise Time Frequencies extracted TDR Midpoint Sync	The rise time is defined as from 10% to 90% of the peak signal value. This resembles a Sweep with a Min Freq. of 0 Hz. For convenience, however, this time profile is specified by the rise time rather than the frequency range. This also enables TDR output. TDR Midpoint is the time where the input TDR signal is at 50 % of the peak value. The Sync check box lets you automatically synchronize the signal midpoint and rise time such that minimum allowed midpoint is used for a given rise time or the maximum rise time for a given midpoint. If you leave the box unchecked, you can specify a different delay. If the delay is not valid, the dialog does not close, and you are prompted to provide a value.

You can utilize the TDR Midpoint feature for consistent results when comparing input signals with different rise times. One way to do this is to delay the TDR input signals so that the midpoint of the rise time always occurs at the same time point in the simulation. The major peaks and dips of TDRz for the different rise times will typically be aligned and the main differences of the results is the extra resolution obtained with a shorter rise time. For example, if you want to study the TDRz signal on a device for the rise times 30ps, 40ps, and 50 ps request a TDR Midpoint of 100ps for each signal by unchecking the sync button. Note that 100 ps is the smallest allowed midpoint for the slowest rise time - 50 ps.

The smallest midpoint allowed is 2 times the rise time to better capture the tails of the gaussian pulse used for the simulation. When sync has been checked the software automatically selects the minimum midpoint allowed for a given rise time or the maximum rise time allowed for a given midpoint.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Changing the delay invalidates solutions for the given solution setup.



2. Complete the Solution Setup for Transient Solutions.

Duration Tab for Transient Solution Setup

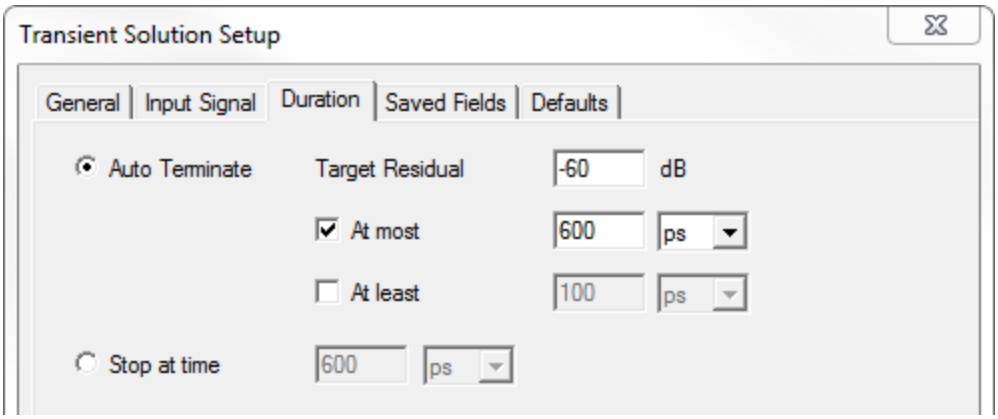
Use the **Duration** tab of the Transient Solution setup to specify the either Auto Terminate and/or At most time and periods.

1. If you select the Auto Terminate radio button, you can also edit the Steady State Criteria value. The Transient Solver can run until the steady state criteria is reached. The value is related to the maximum field remaining in the computational domain at a given time. When the maximum field has fallen to this fraction of its all-time high, the simulation is considered complete and the analysis stops.

   The time duration of a simulation can be controlled by specifying a Target Residual in db20. The residual is defined to be the ratio of the global RMS value of the electric field at time *t* to the maximum RMS value in the time range zero to time *t*. See the technical note here.

2. Optionally, you can also specify and **At most** limit, given in either a Time or periods, or as an At least limit, given in either a Time or periods.

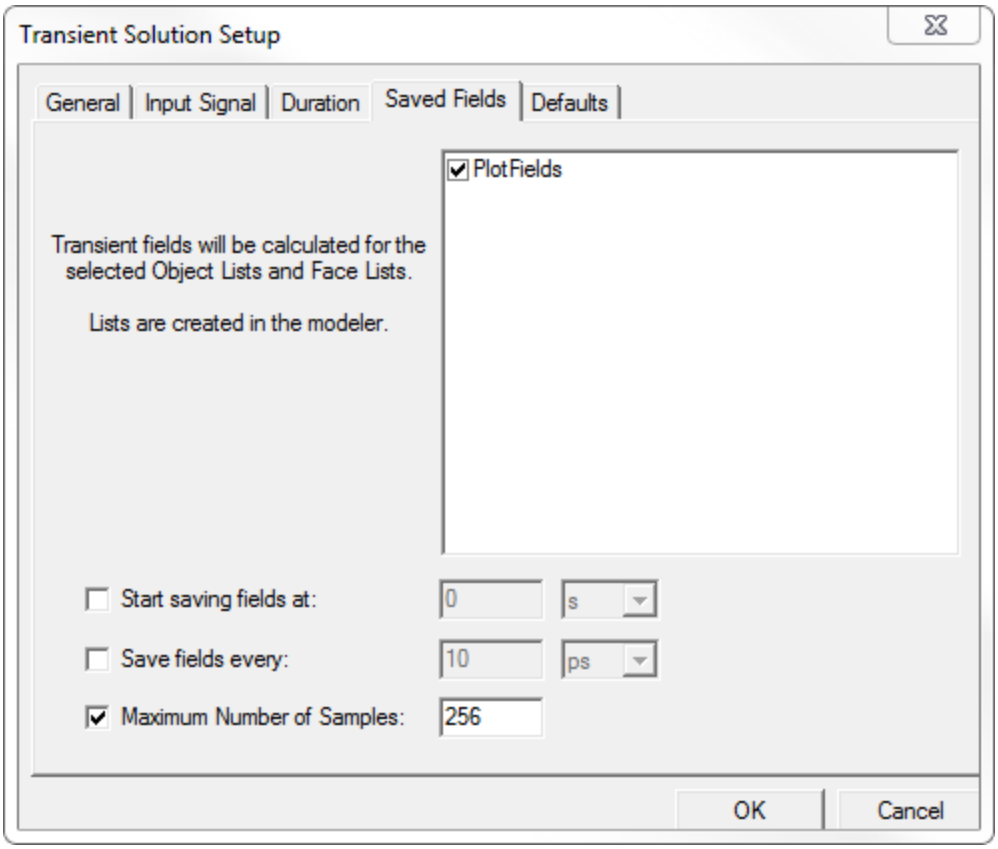
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Saved Fields Tab for Transient Solution Setup

The **Saved Fields** tab for a transient solution setup includes an explanation that **Transient fields will be calculated for the selected**Object lists and Face lists.

Such lists are convenient for specifying the most relevant components.



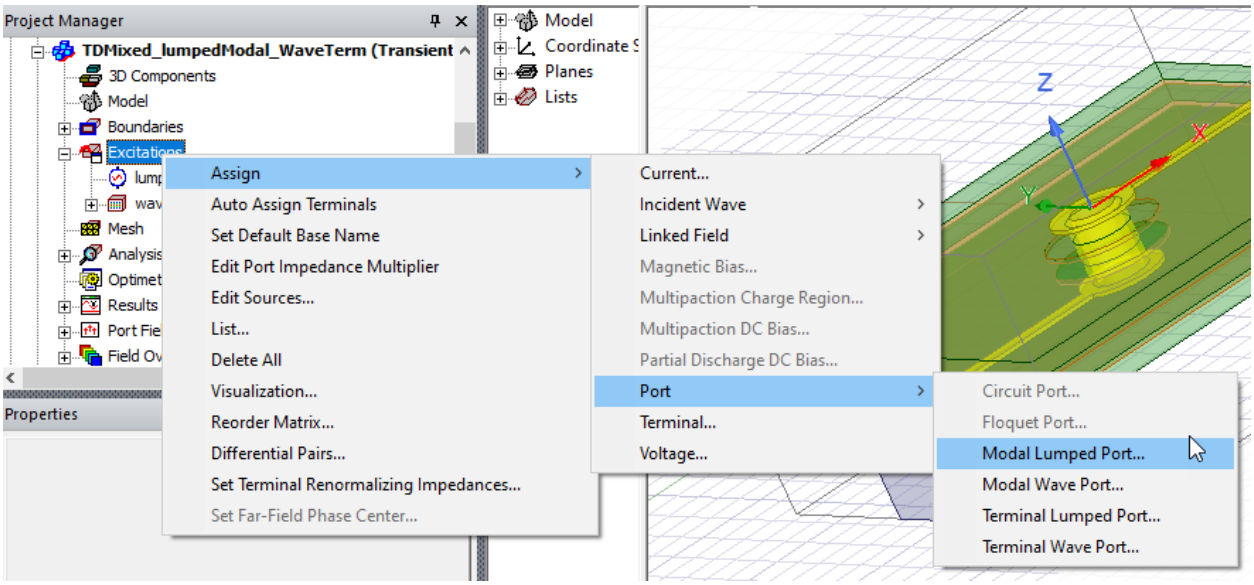
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If lists exist, you can also choose when to Start saving fields, the save interval, and specify the Maximum Number of Samples. Any time request by the reporter in a sweep that is not exactly on the solver setup is either interpolated or extrapolated.

Transient solutions support plotting rE, a real vector versus time, in reporter. Patterns and 3D plots will also be supported.

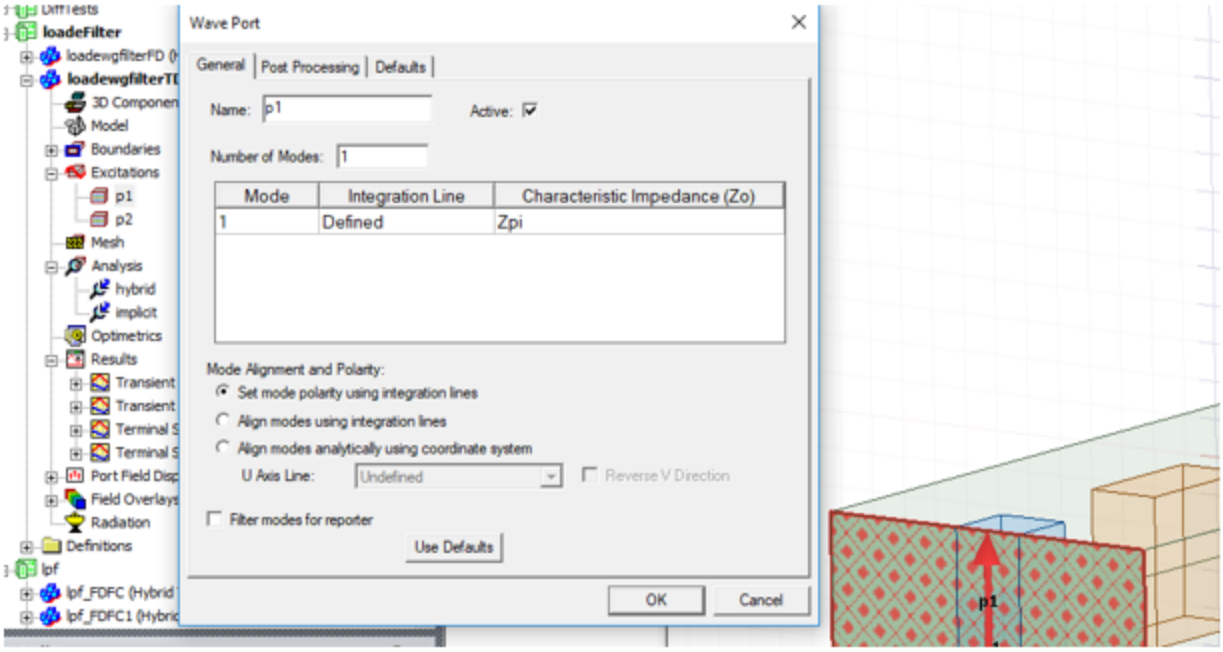
Multimode Waveguide Simulations Transient

This release allows you to mix wave ports and terminals in a transient waveguide simulation. You can add Modal Wave Ports, Modal Lumped Ports, Terminal Wave Ports, and Terminal Lumped Ports to a single simulation. The Excitations menu shows these ports as available.



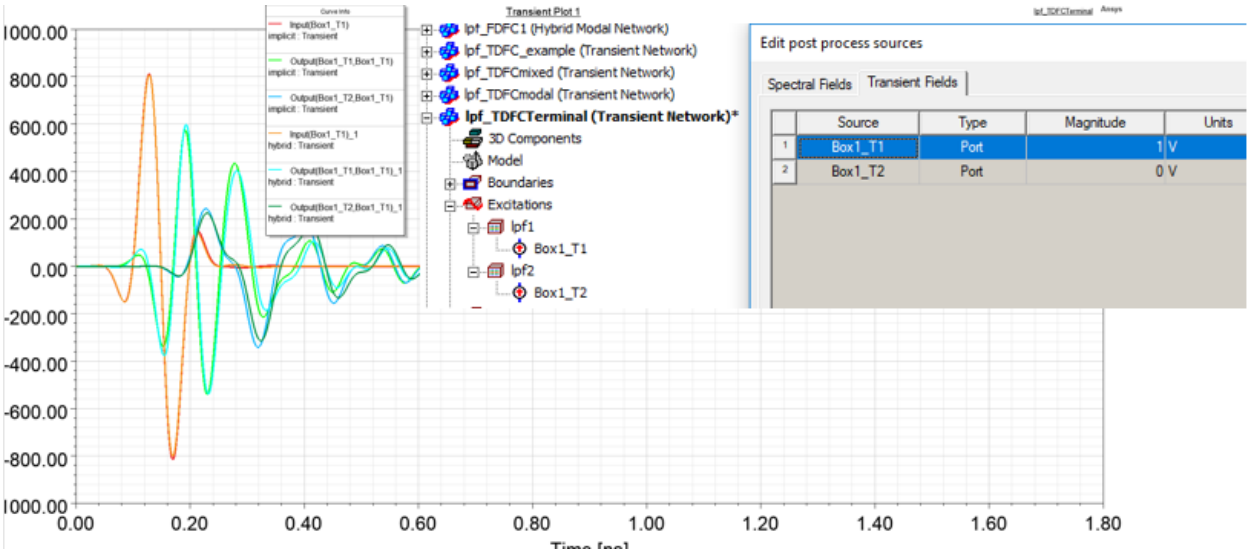
For mixed modal/terminal in transient simulation, the modal Wave Port dialogs are the same as in frequency domain.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



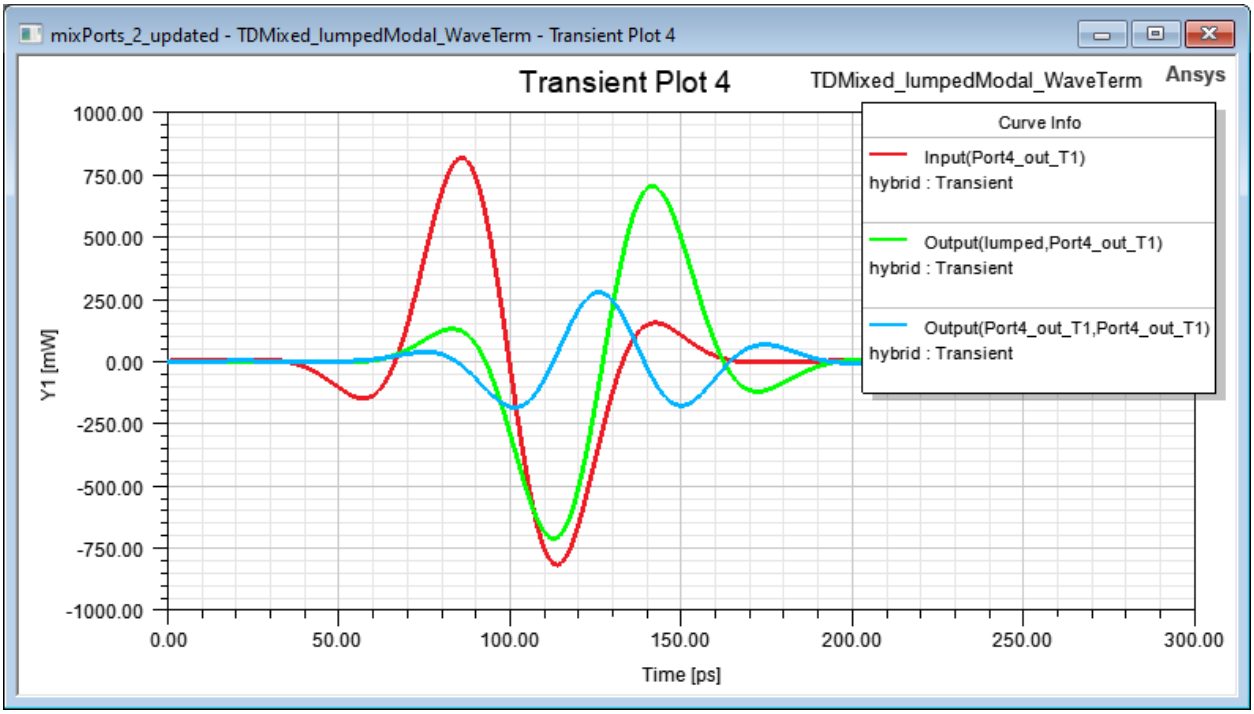
You can mix each kind of wave and terminal ports with the others.

For a transient simulation with only terminal ports, the transient results in reporter are in Volts units, and you edit sources is in Volts units.

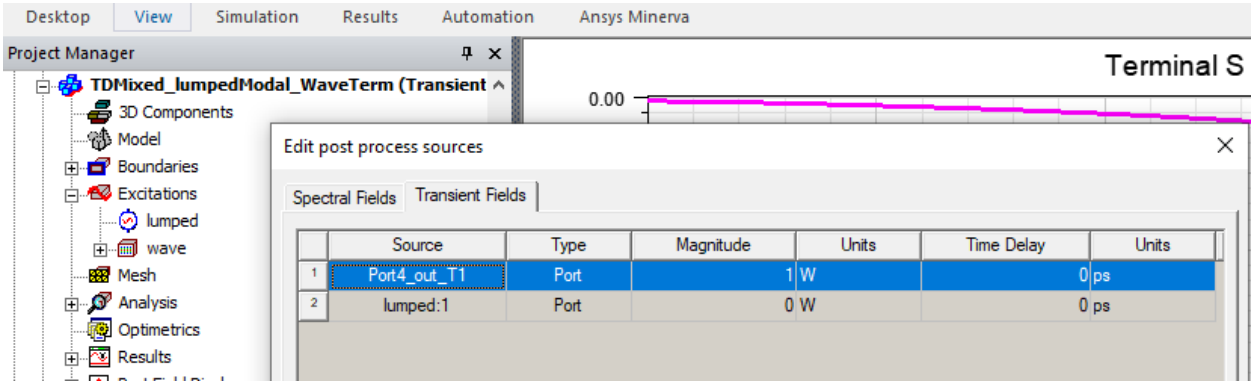


For a transient simulation with all modal or mixed modal/terminal, the transient results in are in Watt units,

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Consequently, for all modal, and mixed modal/terminal transient simulations, you edit sources in Watt units.

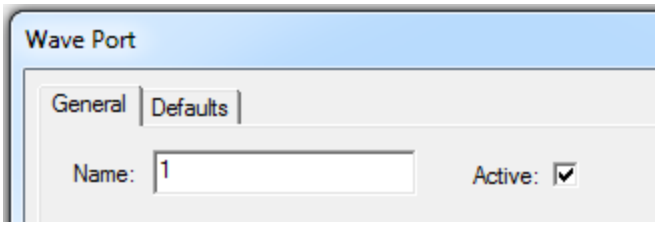


Active or Passive Excitations in HFSS Transient

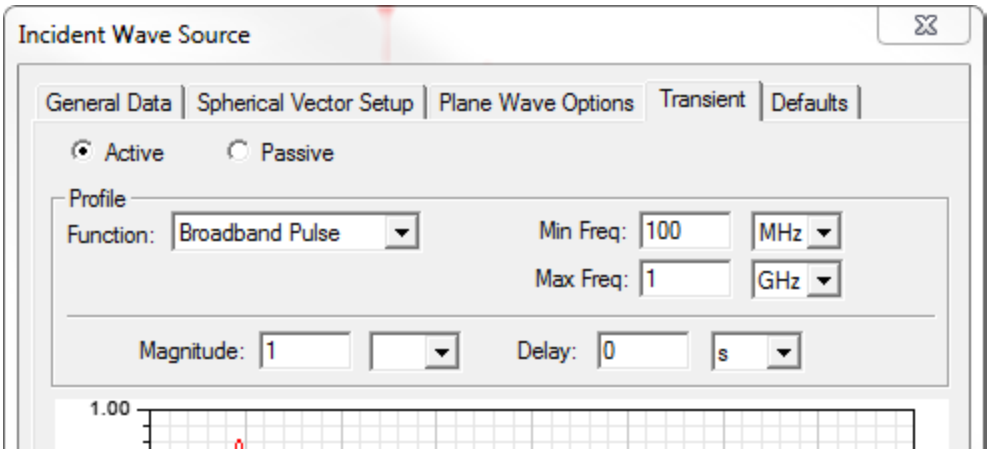
In HFSS Transient, each excitation (port, plane wave, voltage, or current source) shall be "active" (with all ports active they each get an excitation one at a time, and a full S-matrix will be produced). If you want to save simulation time, you can make only one or a few ports of interest "active". The passive ports will act as terminations. You will get only a partial S-matrix.

For wave ports, you specify "Active" using the check box on the **General** tab for the excitation.

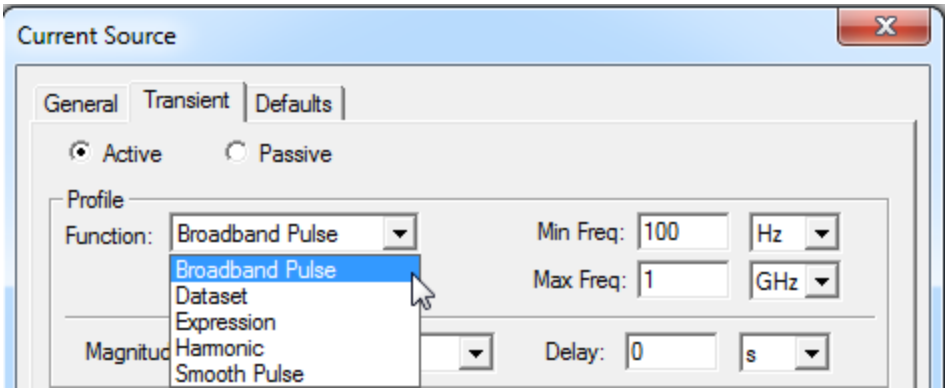
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For an incident wave source, you use the Active or Passive radio buttons on the **Transient** Tab. You can also specify Magnitude and Delay..

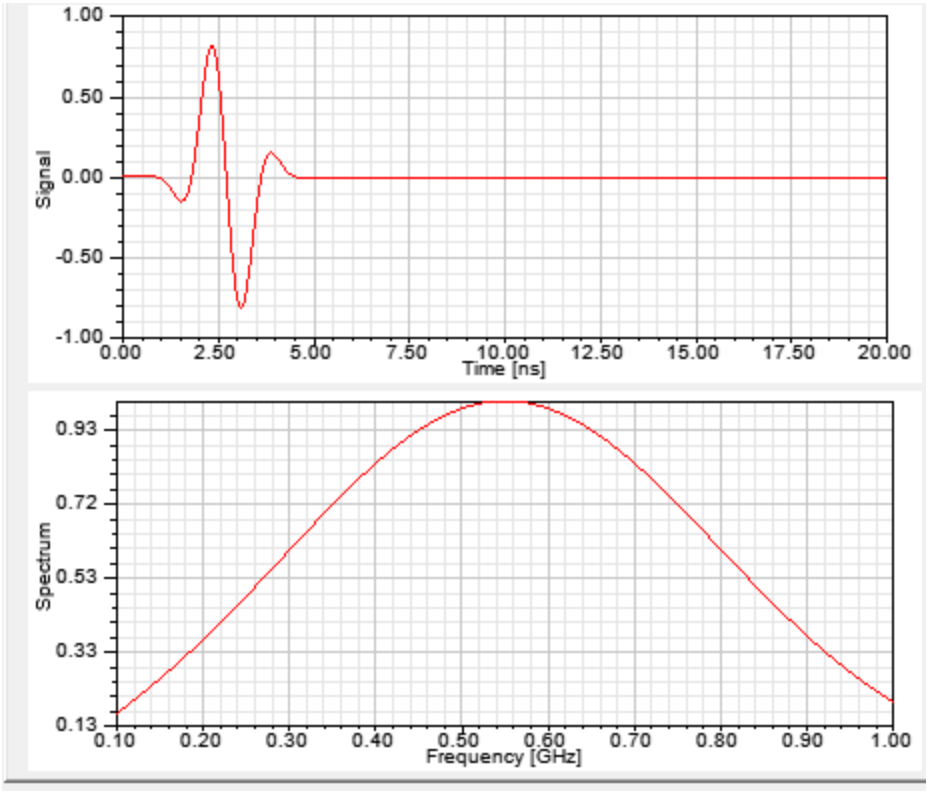


For excitations in HFSS Transient Composite Excitation solutions (and not HFSS Transient Network solutions), if you select Active, you can specify additional parameters for the excitation.



The Transient Composite Excitation Solution type has more time profiles. In this case, the time profiles are defined as part of the excitations, because excitations can have individual profiles in a Composite Excitation solution analysis. The profile includes two plots. The upper plot shows the excitation of interest given the currently specified parameters. The lower plot shows the energy spectrum of the upper plot. The lower plot parameters include the Min and Max frequencies.

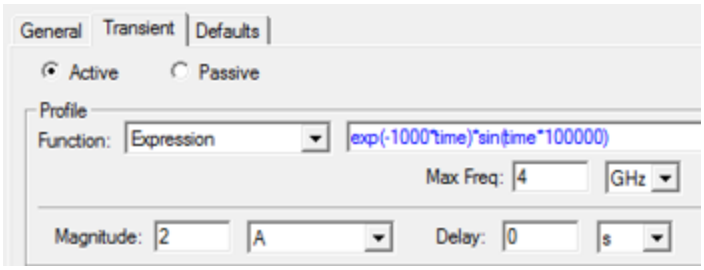
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following table describes the parameters for the upper plots.

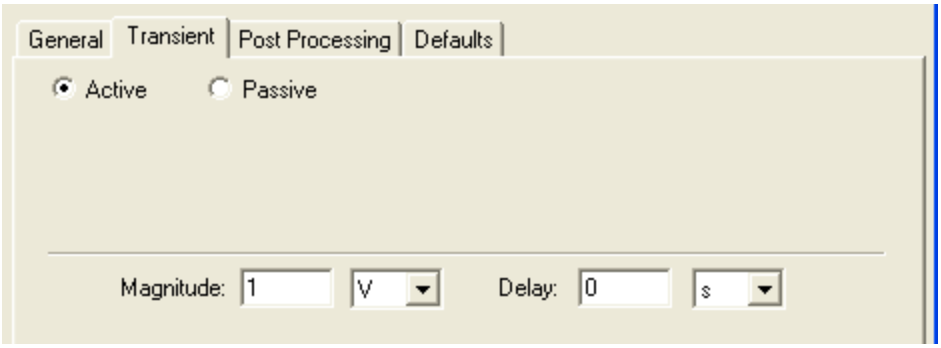
Function	Upper Plot Parameters	Description
Broadband Pulse	Min Freq. Frequencies extracted Max Freq.	A Min Freq. of 0 Hz. is allowed. This changes the shape of the pulse. A time profile that has a strong spectral content between the Min and Max frequencies. This can be used for such purposes as extracting S-parameters in that frequency range.
Data Set	Amplitude Max Freq.	See Adding Datasets for a description of how to create a dataset.
Expression	Expression Max Freq. Magnitude	For lightning strike simulation, it is convenient to define the pulse shape as an equation or algebraic expression. An expression can contain: all standard expressions containing the intrinsic time variable (**time**), HFSS intrinsic function expressions including dataset references, and any user defined variable and dataset (pwl(ds, time)) for the design. An expression cannot contain spatial variables (that is, x,y,z, phi, rho or theta).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Function	Upper Plot Parameters	Description
	Delay.	
Gaussian	T0 Amplitude Width	The T0 parameter specified the start time of the pulse with the given Width and Amplitude.
Harmonic	Ramped Periods Amplitude Frequency	The harmonic displays as a regular sine wave.

Setting Magnitude and Delay for Transient Composite Solution Excitations

In HFSS Transient Composite Excitation solutions, each excitation (port, plane wave, voltage, or current source) can be given a Magnitude and Delay.



For Transient:

- Scaling and delay time can be set on the input signal for each excitation. This appears only in Transient Composite Solution, not Network Analysis. Scaling is a unitless multiplier, and delay time is in appropriate time units.
- Changing scaling or delay time will invalidate solutions.
- Scaling and time delay are passed to the solver and used for the simulation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

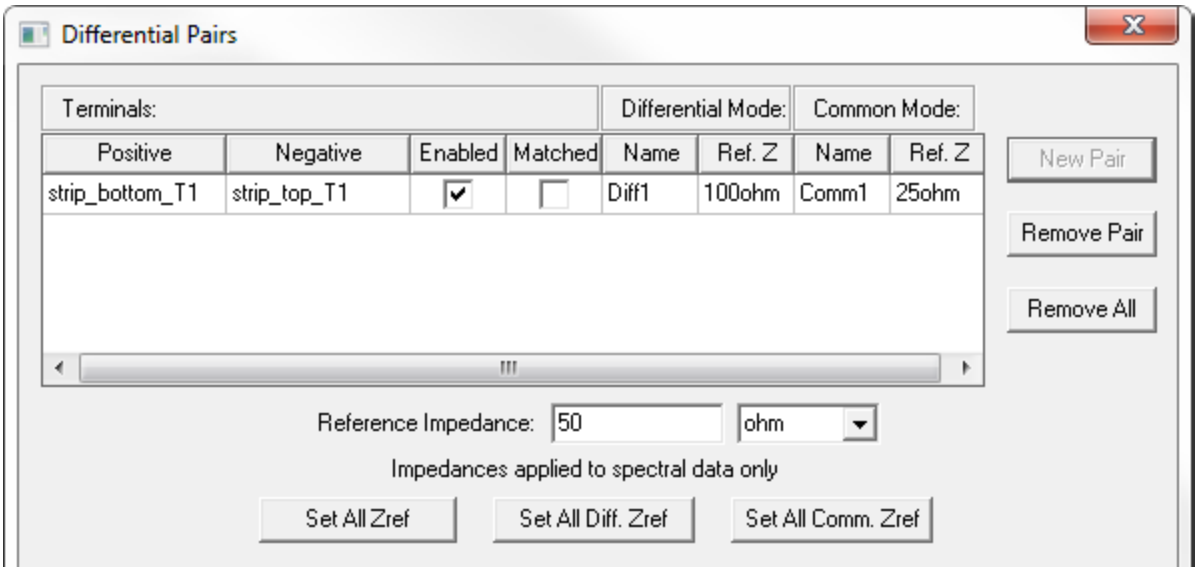
- Legacy projects set the scaling to 1 and time delay to 0. These are also used as the defaults.

Differential Pairs in HFSS Transient Network

For post process calculation of differential pairs in Transient Network problems, you can assign differential pairs in spectral terminal problems for transient designs. For Transient Composite Excitation problems differential pairs are not supported.

In Transient Network, you can solve the single-ended matched terminal problem. HFSS Transient uses the same post processing algorithms that HFSS uses to compute the differential and common modes for spectral values. In addition, a separate algorithm computes the differential pairs values for transient data. You can switch between single-ended and differential views of both spectral and transient data. Renormalization of spectral data is supported, but renormalization does not apply to transient data.

The dialog for transient Differential pairs includes columns for Enabled and Matched. Passive ports are not shown in the diff pairs setup dialog box.



The table row shows the check box for the newly defined pair as **Enabled**. Unchecking the box disables the definition for that pair. This can be useful if you later want to redefine terminal normalization, without having to remove the defined pair altogether.

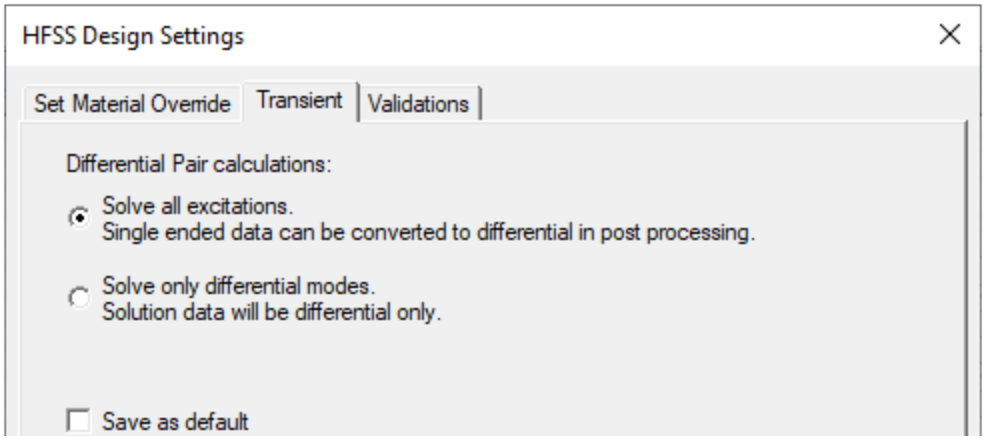
A check box for "Matched" for each differential pair indicates whether to use matched or differential and common impedances. This Matched check box also appears in driven terminal designs and uses the same postprocessing semantics. Matched is the "natural" in general frequency dependent characteristic impedance of the line. For multiple terminals it describes the coupled nature of such lines and is therefore a dense matrix relating voltages and currents on

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the line. A matched terminal project is analogous to an unrenormalized wave port in driven modal where the s-parameter is referenced to the 'natural impedance' of the port cross-section. If you unselect Matched, you renormalize to a set impedance for the diff or common modes.The transient solver uses the Matched value when the Design setting "Apply when solving" is selected.

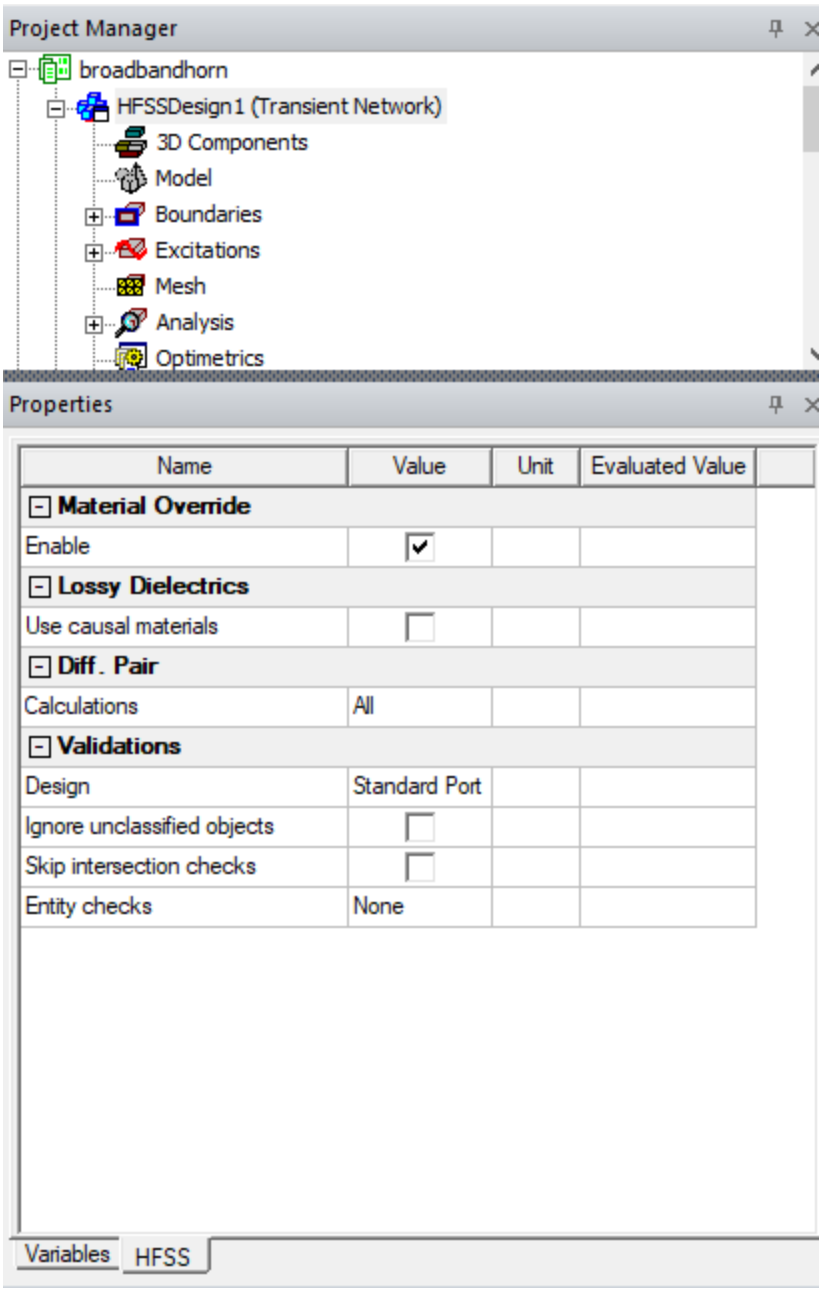
Design Settings for HFSS Transient

The **HFSS>**Design Settings command displays a dialog with tabs for **Set Material Override**, **Transient**, and **Validations**. The Transient tab includes options for Differential Pair calculations.



You can also view and set Design Settings for HFSS Transient by selecting the design in the **Project tree**, and selecting the HFSS tab on the **Properties** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Set Material Override Tab

The **Set Material Override** tab includes text note and a check box to **Allow metals to override dielectrics**. The purpose of this feature is to allow you to avoid doing explicit subtraction in the modeler. One example application is a via that passes through many dielectric layers--with the option turned on, the via does not have to be subtracted from the layers.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Set Material Override** option allows some intersections to be resolved automatically in the mesh. If metal intersects dielectric, the metal overrides the dielectric in the overlap region. (That is, the metal object is subtracted from the dielectric.) If objects with the same material overlap, the small object overrides the larger. (That is, the small object is subtracted from the larger.) All other intersections are treated as errors. Normally, the modeler considers any intersection between 3D objects to be an error.

To use this feature, check **Enable material override**.

In the meshing process, the dielectrics are locally overwritten by the metals in the intersecting region. That is, the part of the dielectric that is inside the metal is removed, and if the dielectric is completely inside, the whole object disappears.

Transient Tab

If your design contains differential pairs, you can select whether to solve all excitations so that single-ended data can be converted to differential in post processing, or to solve differential modes only.

Validations Tab

The **Validation** tab offers choices for Model validation and HFSS validations to control the extent of validations performed, and therefore the time involved.

Model Validation choices are whether to:

- Ignore unclassified objects
- Skip intersection checks

You also control the Entity check level as Strict, Basic, Warning Only, or None.

The HFSS choices are whether to:

- Perform full validations with standard port validations. Standard validation does not check for internal or floating ports.
- Perform full validations with extended port validations. The Extended validation checks to see if a wave port has been applied to an internal face, or if a wave or lumped port does not have solve inside geometry on either side.
- Perform minimal validations. This choice disables port validation options (neither is performed), and skips boundary overlap validation.

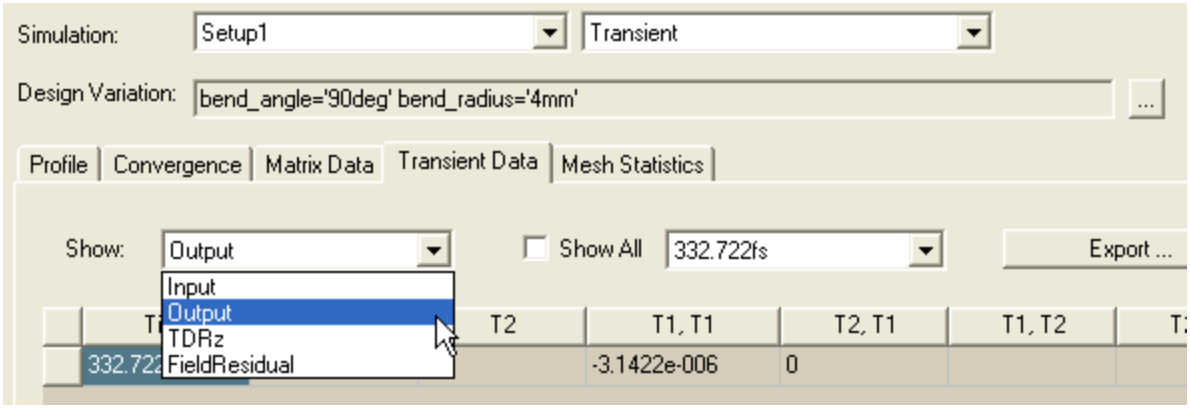
The default is to Perform Full Validations. However you can set your own default choices by using the Save as Default check box.

Transient Solution Data

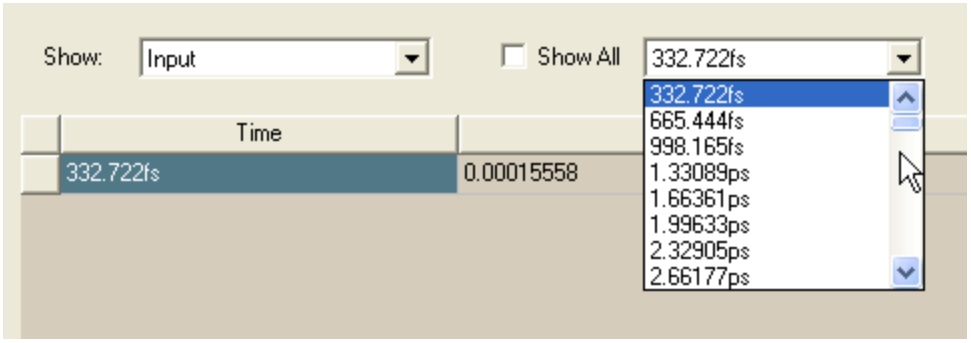
For Transient solutions, the Solution Data dialog includes a **Transient** tab. Your access is by clicking **HFSS>Results>Solution Data**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

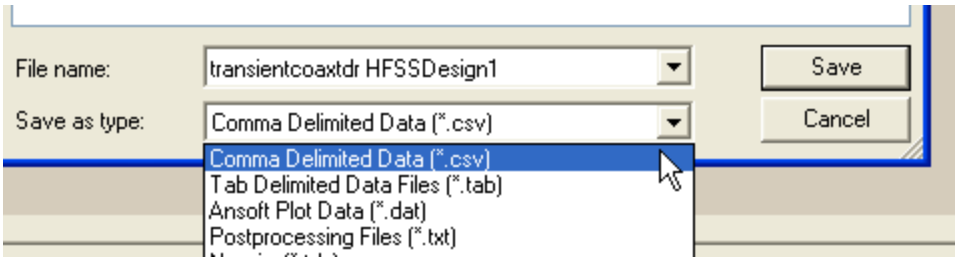
1. To enable the display of transient data on the **Transient** tab, you must first select Transient from the Simulation drop-down menu.



2. Then selecting from the Show drop-down menu, you can select Input, Output, TDRz or Field Residual to display.
3. You can select which time step to display from the drop down, or check Show All.



4. To export the transient data, click Export. This opens File browser winder that lets you specify a File name, location, and data format.



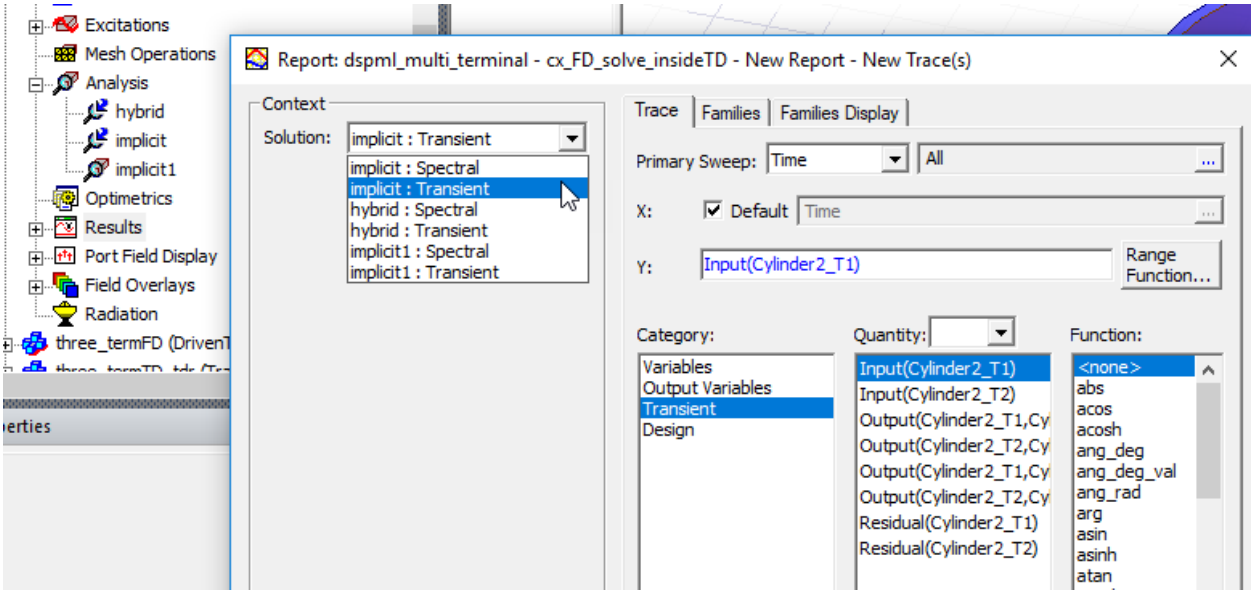
For Transient, the format selections are:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

.csv	Comma Delimited Data.
.tab	Tab Delimited Data files.
.dat	Ansys Plot Data files
.txt	Post Processing Files

Selecting the Report Type in HFSS Transient

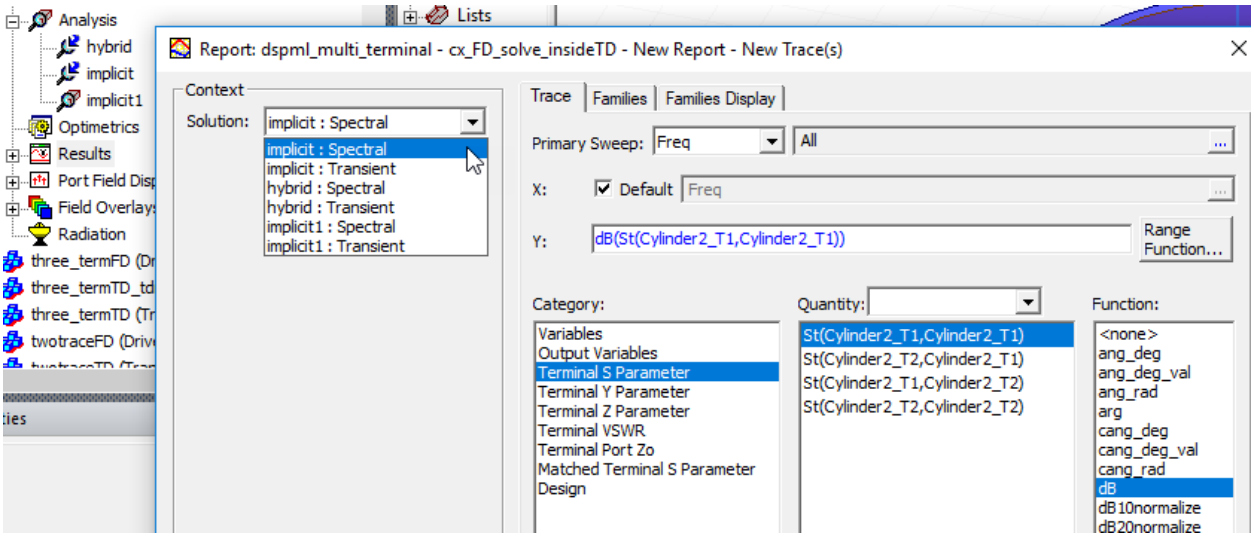
Creating a Report in HFSS Transient differs from standard HFSS in Solution selections. If the Solution Type is Transient, the Solution Context is Transient. If the Solution Type is Transient Network Analysis, you can also select Spectral. If you have created multiple solution setups (in the following example, hybrid, implicit, and implicit with and without imported mesh), the solutions offered will reflect that.



With Transient selected, the Quantities include Input and Output related to the terminals in your design, as well as TDRz and Residual.

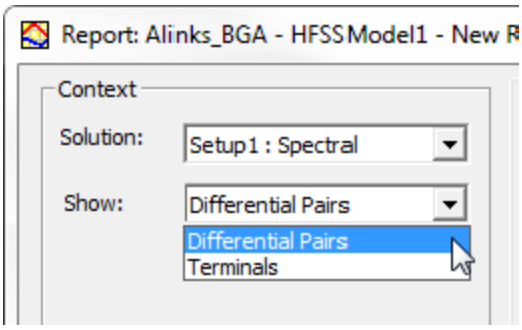
For Transient Network Solution type designs, if you select Spectral, the Category and Quantity lists offer different selections:, including Terminal S Parameters, VSWR, and Port Zo.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Reports for Transient Network with Differential Pairs

For a Transient Network design with differential pairs defined, the Reporter interface allows selection of single-ended or differential signals just as for driven terminal. The Differential Pairs menu is shown for transient and spectral parameters, and allows you to switch between the two representations. The menu does not appear if differential pairs are not defined.



If the Design Setting "Apply when solving" is in force, it will be restricted to 'Differential Pairs' if they are present and 'Single-Ended' otherwise.

For the Transient solution (as opposed to Spectral), selecting "Differential Pairs" will limit the selected quantities to "input, output, and TDR". For the Spectral solution, all quantities will be available.

As with single-ended S parameters, post process differential pair values will not be available until the solve completes. Post process differential transient signals may be available depending on the method produced by the solver group.

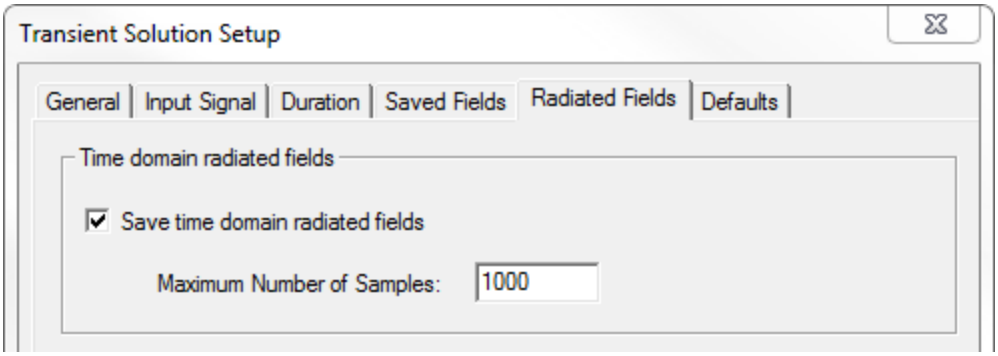
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Transient Display supports the transformation between single-ended and differential transient signals under the same circumstances as the reporter.

Procedure for Viewing Transient Radiated Fields

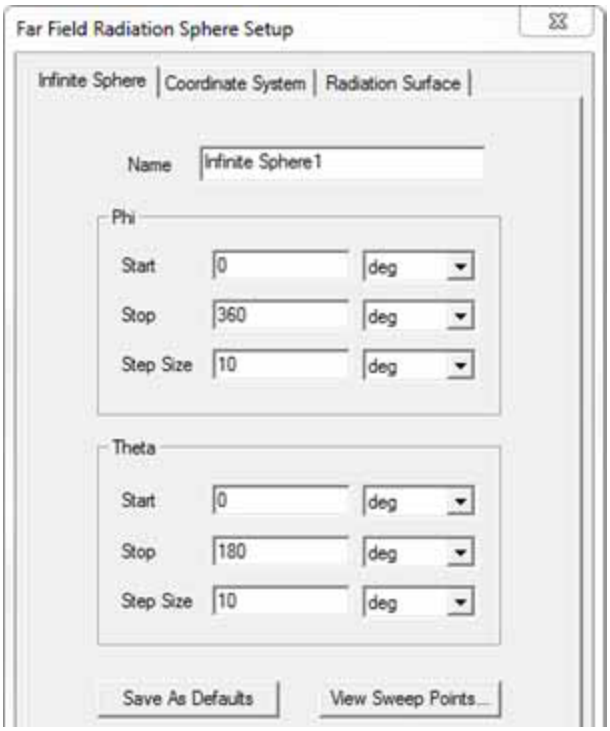
To display transient radiated fields:

1. Add a radiation boundary. Radiated field calculations will only be done for designs with radiation boundaries.
2. If a radiation boundary is present the transient Solve Setup contains the **Radiated Fields** tab with a "Save time domain radiated fields" check box. Select this option to make radiated fields available from a given setup. This applies to Transient with or without Network Analysis.



3. Under **HFSS>Radiation**, you can **Insert Far Field Setup> Infinite Sphere**. This menu is enabled for designs with radiation boundaries, even if no setups are saving radiated fields. The setup dialog resembles the one for frequency domain, but without the Radiation Surface tab. Use this dialog to set up the Theta and Phi sampling and, if needed, the local coordinate system. You can create multiple Infinite Sphere setups in a single design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Once you have created a far field setup AND at least one setup has "Save radiated fields" selected, the **Results** menu will include **Create Far Fields Report**, with all submenus as in Frequency domain.

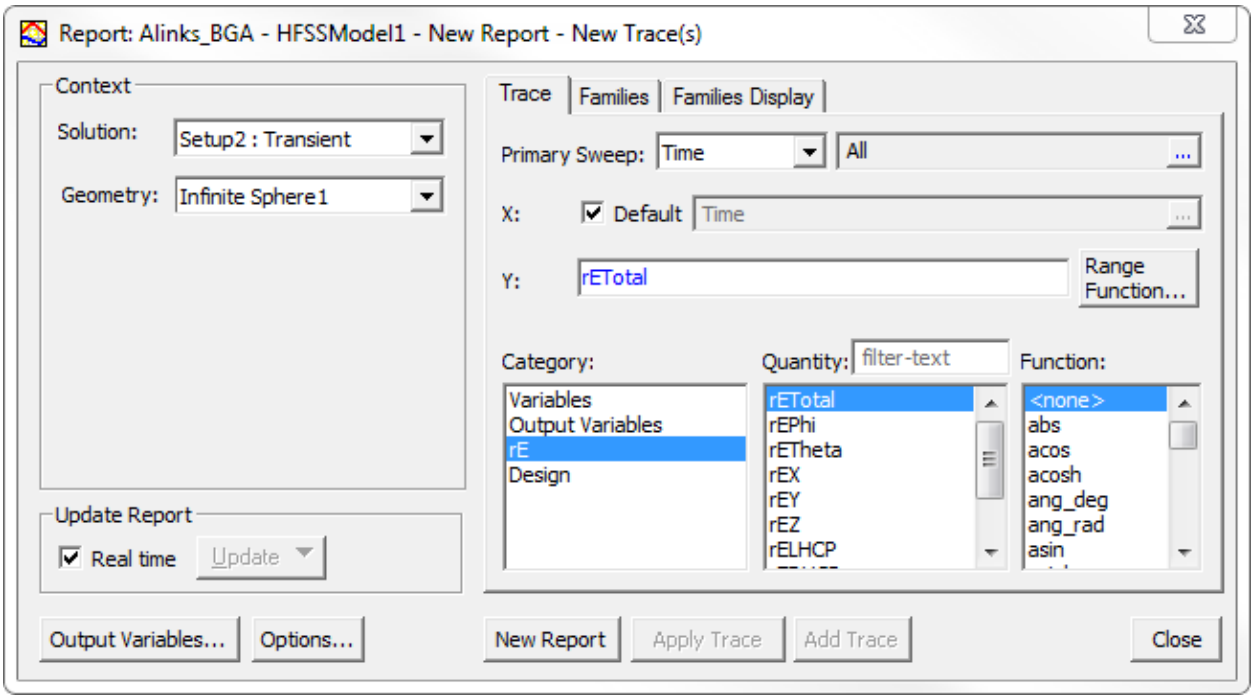
For Rectangular Plot, Rectangular Stacked Plot, and Data Table, the default is "Time" as the primary sweep, with Theta and Phi in Families set to single values corresponding to the first sample point.

For all other plots, the primary and secondary sweeps will be Theta and/or Phi, as in Frequency domain, and the Time is set to a single value - the start time.

5. In the **Report** dialog box, the **Solution** selection includes only setups with "Save radiated fields" checked. The **Geometry** selection will include all far field Infinite Sphere setups. The **Categories** include rE, Variables, Output Variables, and Design. The rE quantities

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are as for frequency domain, but all quantities will be real.



No matter what type of plot is generated, you can access the Time sweep and change the sampling, as with Field reports in Transient.

For 3D patterns, you can overlay the pattern on the geometry, and to animate versus time, as is done in frequency domain.

Once plots have been created, the reporter caches the base radiation field calculation. This means that subsequent plots will be generated more quickly. If you change the radiation setup, or invalidate solutions, the cache is cleared and the next plot takes longer.

For Transient Network Analysis, the radiated fields are based on the setup in Edit Sources. If you change the source excitations that forces recomputation of the radiated fields.

Output variables are supported, as for frequency domain.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13 - HFSS with SBR+

HFSS can use shooting and bouncing ray (SBR) technology to compute the scattered field contribution of defined platform geometry via one-way coupling. This allows you to efficiently calculate installed near-field patterns,far-field patterns and coupling (S-parameters) between antennas in the presence of an electrically large scattering geometry. HFSS integrates shooting bouncing ray (SBR+) analysis in several ways:

- SBR+ Hybrid Regions for HFSS Driven Modal and Driven Terminal solutions.
- SBR+ Solution Types, which permit Parametric Antennas, as well as Parametric Arrays.
- One way link from HFSS Driven Modal or Driven Terminal to SBR+ Designs.
- ACT Toolkit for HFSS to EMA3D Data Link.

HFSS with SBR+ works from within the standard Ansys Electronics Desktop, sharing the GUI and the same 3D modeler and reporting and animation features. HFSS with SBR+ includes:

- Import of Lightweight Geometry for HFSS SBR+ Designs.
- Visual Ray Trace for SBR+ Simulation.
- Select Tx/Rx Antennas for SBR+ Designs.
- Assign Fresnel SBR Perfect Absorber Boundaries.
- Assign a Dielectric or Conducting Material to a Volumetric Region
- SBR Curvature Extraction for Faceted Surfaces.
- SBR+ Range Doppler Solution.
- Creeping Wave for RCS Analysis of Curved Surfaces in SBR+ Simulation.
- Creeping Wave for Antenna Placement on Curved Surfaces in SBR+ Simulation
- Creeping Wave for VRT Plots
- Near Field Analysis in SBR+ Design.
- Exporting ROM Radar to VRXPERIENCE (Beta)

For SBR+ Ray Data Export, you can use Design Settings to specify whether to Export On completion either or both range-doppler data and ray data.

Release R24.1 accounts for geometry blockage per component antenna instance/source, and allows any type of material assignment (perfect E, dielectric, volume material) for these geometry blockage objects. Use of “non-model” geometry is not necessary as part of the preferred workflow – any geometry that should be considered by the SBR+ simulation will be included in the SBR+ region; any geometry that should not be considered will be explicitly excluded from the SBR+ region by defining blockage lists per excitation.

For the SBR+ region, any model geometry that needs to have this special consideration for model blockage (ignored on first bounce, used on multi-bounce) is assigned as such on a per

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

source antenna basis. This is because the model blockage behavior is executed in the SBR+ simulation on a per source basis.

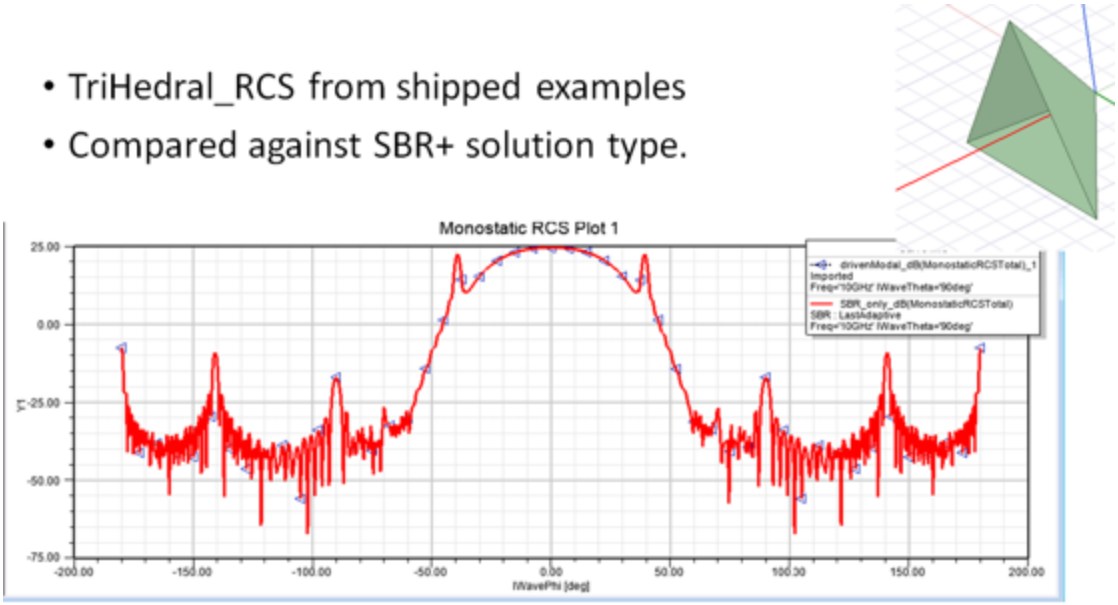
Using SBR+ with HFSS

For using SBR+ Hybrid regions in an HFSS design, see Assigning SBR+ Hybrid Regions. For the SBR+ Solution type, see the Design Flow for SBR+ Solution Type and SBR+ Range Doppler Solution Setup. For linking HFSS with SBR+, see Link from Driven Modal or Driven Terminal Design to SBR+ Design. For the ACT Toolkit, see HFSS to EMA3D Data Link.

Also see, HFSS SBR+ Hybrid Reflector Antenna Simulation example.

Design Flow for SBR+ Solution Type

The SBR+ Solution type simplifies design creation for SBR+ designs for use with Shooting Bouncing Ray technology with optional UTD and PTD correction. With this selection, you do not create explicit SBR+ Regions, and there is a reduced Solution Setup that includes a frequency sweep definition. Near field and far field reports, as well as antenna parameters are available. Matrix reports and Network Data Explorer are available if the design includes a parametric- or file-based antenna. 3D fields, near fields, and port displays are not available. Plot fields, and SAR settings do not apply. An SBR+ only design and its equivalent Modal/Terminal design should have comparable if not identical results.



Prerequisites

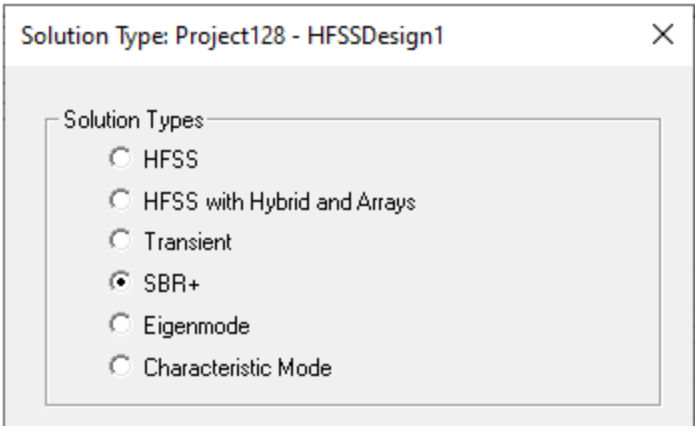
Requirements for SBR+ solutions:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Must have at least one Incident Plane Wave or Antenna Component excitation. An Antenna Component is either a link to a source design, a built-in parametric antenna, or an imported far-field antenna gain pattern (.ffd) file.
- Ports are not allowed.
- Infinite ground planes are not supported.
- SBR+ can trace rays inside dielectric regions of non-uniform thickness, following Snell's laws of reflection and refraction, including for lossy non-conductive dielectrics. For volumetric SBR+, only conformal meshes are supported. There is no lightweight, dynamic or tolerant mesh support. Meshes must represent volume regions (that is, objects) in HFSS. Sheets cannot be assigned materials, but only boundaries. Supported materials are pure dielectrics, with zero conductivity and without excessive loss.
- No spatially dependent boundaries.
- For near fields and far fields, the setup for sample points must be defined before simulation. This can be done through the Field Observation Domain area of the Options tab. You can create new Near Field Lines, Rectangles, boxes, or Spheres, or Far Field Infinite spheres and edit existing domains.
- SBR+ near field analysis will be aborted if one or more encrypted components are included.
- SBR+ designs that include blockages have an enhanced work flow described in "Configure Simulation with SBR+ Antenna Geometry Blockage (SBR+ Solution Type)" on page 13-28

Using SBR+ Solution Type for HFSS

1. When you select the SBR+ Solution type, you do not need to set SBR+ Regions in the design.



2. SBR+ Solutions support the following boundaries:

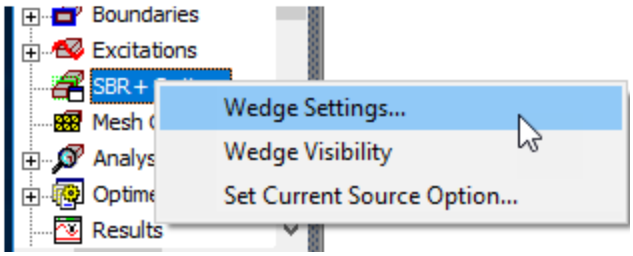
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Perfect E
- Perfect H
- Finite Conductivity
- Layered Impedance
- Impedance

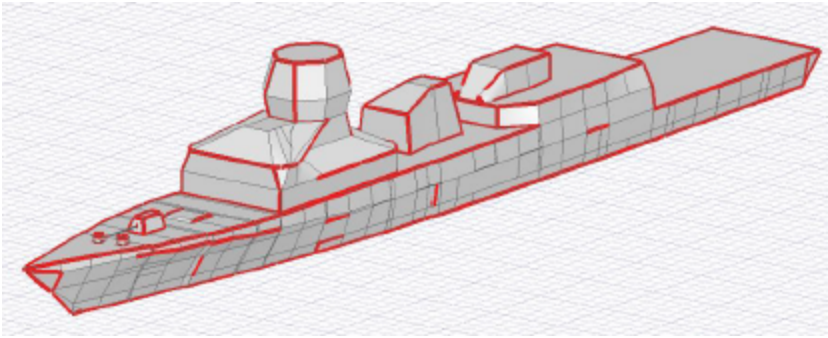
For the supported boundaries, only the material property, impedance, and thickness are used. Surface roughness and shell elements are disabled. If you convert an existing design to SBR+, unsupported excitations and boundaries are deleted, and any unsupported boundary properties are removed. You can **Undo** to restore the original settings.

3. Set the SBR+ Options

The Project Tree for the SBR+ Solution type does not include a Hybrid Regions icon, nor a Port Field display. It does include SBR+ Options where you can specify **Wedge Settings** and enable or disable **Wedge Visibility**. These options are described below. See this link for Set Current Source Option.



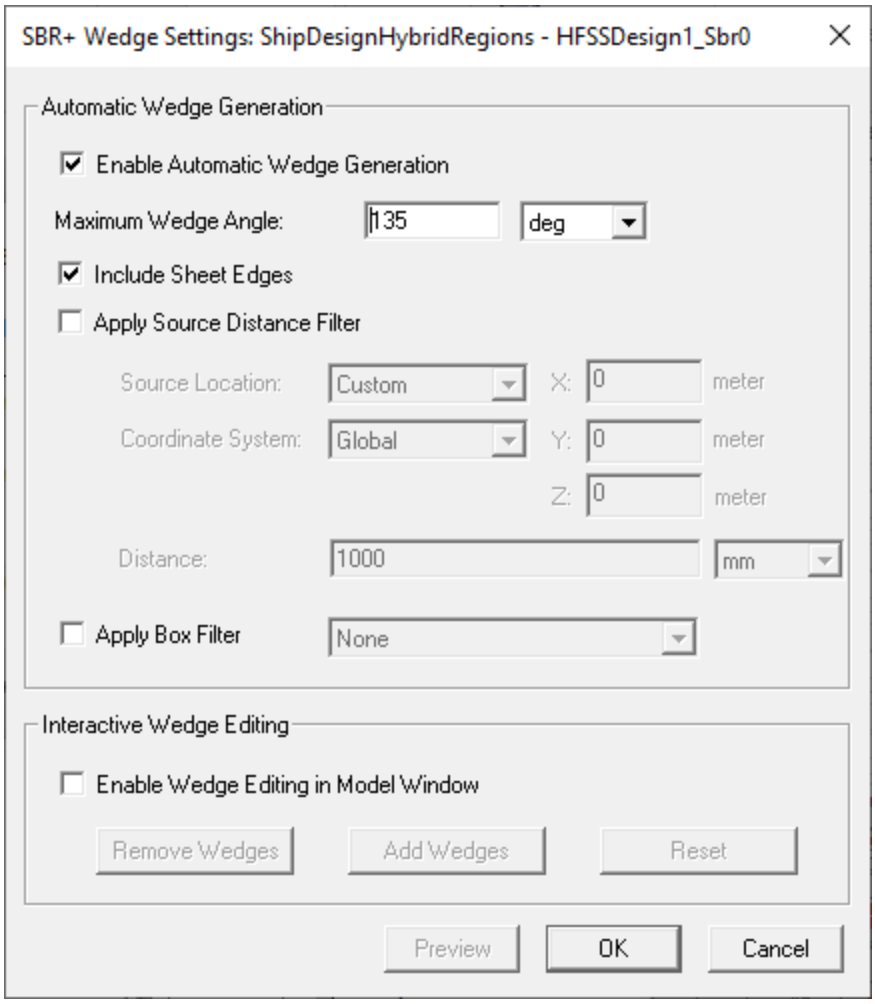
If visible, wedges are shown as thick red lines in 3D model window. Visualization is turned off when there is design edits that invalidate the initial mesh.



The **SBR+ Wedge Settings** dialog contains the Automatic Wedge Generation parameters and the Interactive Wedge Editing feature. Wedges for PTD and UTD are based on the user-specified wedge settings and the initial mesh. The initial mesh is automatically generated on

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

launch of the **SBR+ Wedge Settings** dialog box, if needed. The dialog box will not launch if an initial mesh is not available and cannot be generated.

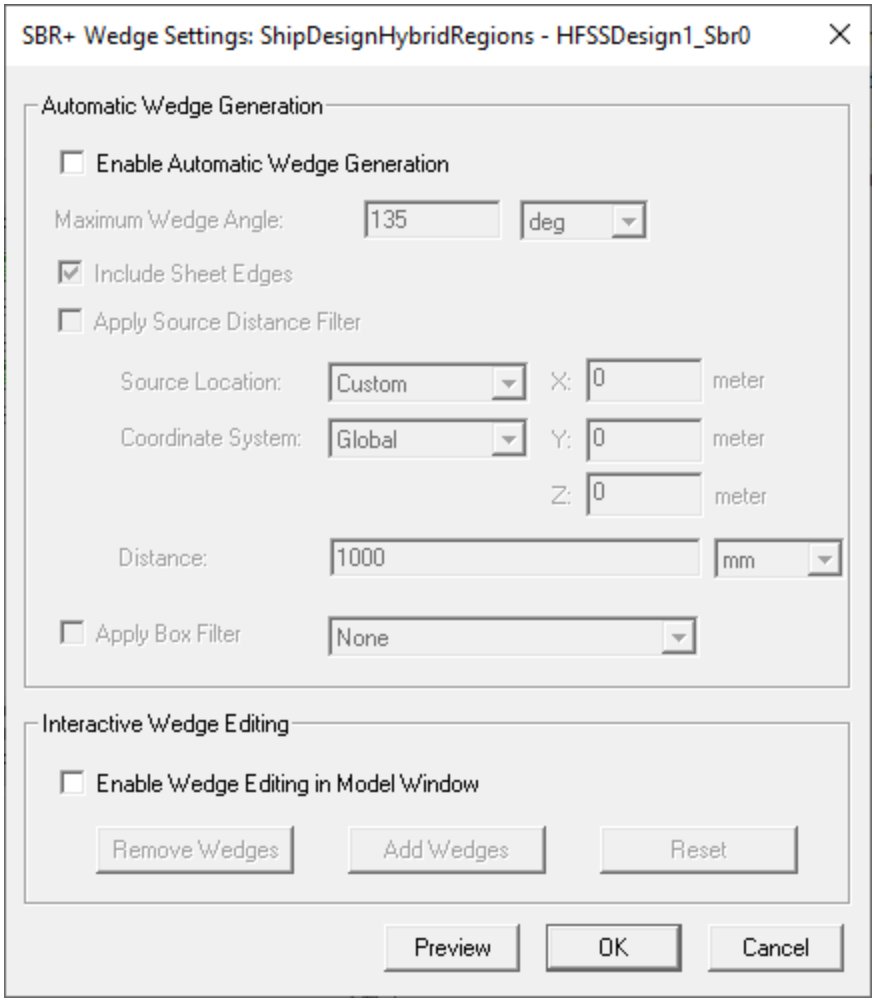


The Wedge Generation parameters are used to determine the candidate set of wedges to be included, with optional filters that can restrict their location on the geometry. The final set of wedges to be used in the SBR+ simulation is the intersection of all the wedge generation criteria, and the effects of any optional Interactive Wedge Editing. complete flexibility in specifying where the wedges are to be used for the simulation. This is important since adding many wedges on very large geometry structures (aircraft, vehicles, etc.) can significantly impact the SBR+ simulation time.

Enable Automatic Wedge Generation.

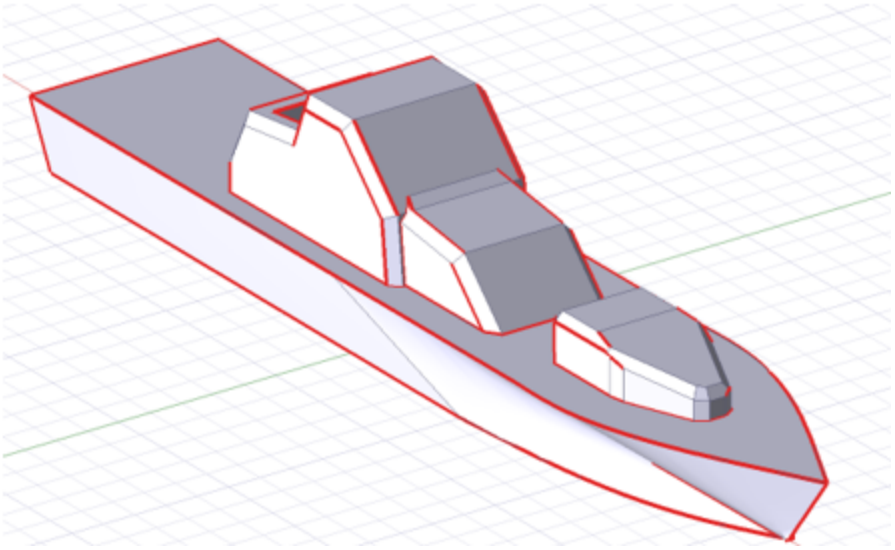
By default, the checkbox is enabled, allowing access to the automatic features and parameters. If you un-check it, the automatic features are disabled. You can independently enable Wedge Editing.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



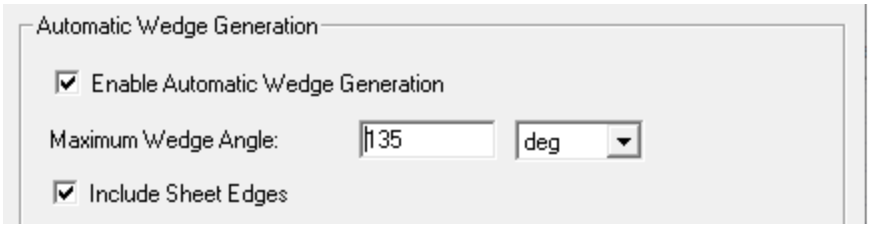
Wedges added through Automatic Wedge Generation are displayed in red.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Maximum Wedge Angle:

An angular criteria can be specified for the maximum wedge angle to help filter the desired set of wedges - default value is 135 deg. This wedge angle ranges from minimum 0 deg to maximum 180 deg. All wedges are included that are less than or equal to the specified maximum angle. An angle equal to 0 deg would be created by a “collapsed” wedge (like a closed hinge), while an angle equal to 180 deg would be created by a completely “open” wedge (like a fully open hinge). You can assign a design variable to Maximum Wedge angle, but you cannot create a design variable within the dialog.



Include Sheet Edges:

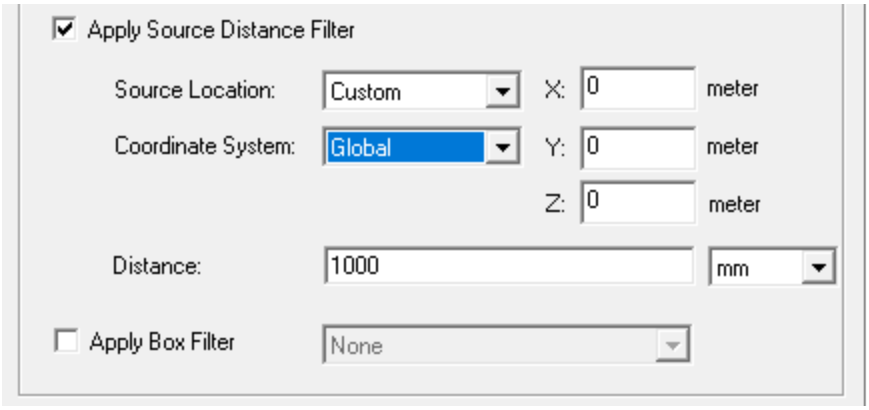
You can choose to include or exclude the edges from sheets (non-connected edges, also known as “knife edges”). The default value is true which includes these edges. Sheet edges are triangle edges with no adjacent/connected triangle along that edge. In the physical object we are modeling these may represent extremely thin surfaces such as metal “fins”, etc.

Optional parameters:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Apply Source Distance filter:

A “source distance filter” is a way to specify the inclusion of wedges within a finite 3D distance from a point (the “source”).



You can choose model Points from drop down as source location, or specify the absolute X, Y, Z in model units. X, Y, Z can be specified in global coordinates or relative to a custom CS. When you select a geometry Point, its coordinate system and X, Y, Z are displayed.

The Distance default is 1 <model unit>. You can assign an existing design variable to the Distance. Default value is false (no Source Distance Filter).

Apply Box Filter:

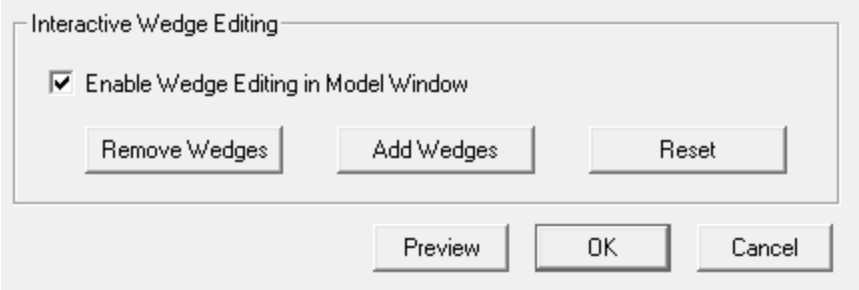
You can select any non-model box that is aligned with the Global Coordinate System axes via a drop-down menu, within which wedges will be included. Default value is false (no Box Filter).

Optional Interactive Wedge Editing

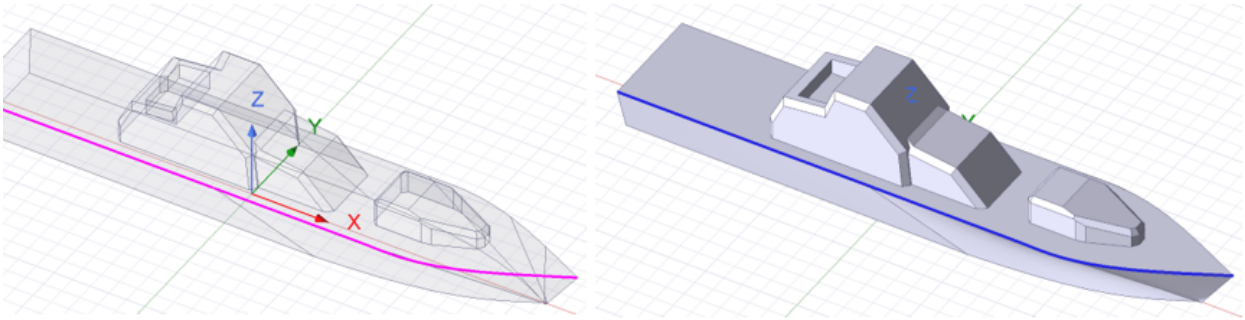
This feature provides complete flexibility in specifying where the wedges are to be used for the simulation. This is important since adding many wedges on very large geometry structures (aircraft, vehicles, etc.) can significantly impact the SBR+ simulation time.

Checking the Enable Wedge Editing the Model Window box enables the buttons for removing wedges from selected geometries (faces, objects, etc.) , adding wedges to selected geometries, and resetting edits.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After you select a geometry and **Add Wedges**, the interactively added wedges appear in blue. Allowed geometry selections include entire geometry objects, faces on objects, and edges on objects.



If you select **Remove Wedges**, all wedges on the selected geometry objects are removed. These can be automatically generated wedges or interactively added wedges.

Click **Reset Wedges** to undo all interactive wedge editing operations. A confirmation dialog appears. Press Yes to reset all wedge editing operations. The reset button is disabled if no wedge edit operations are present.

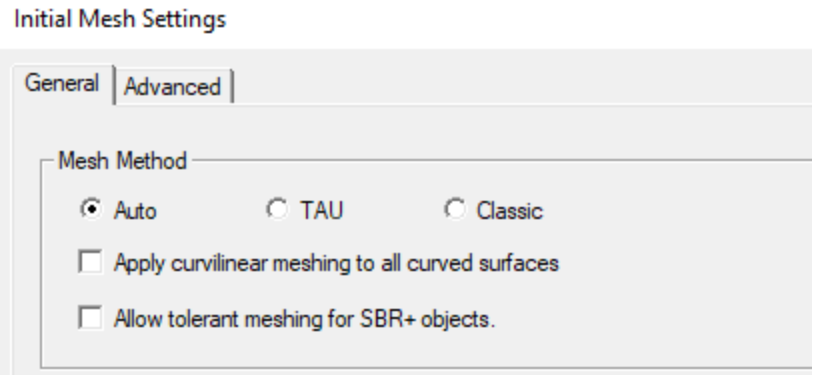
Current Limitations on PTD/UTD Wedge Generation/Editing

Situations where wedges cannot be generated:

Light Weight Imported STL Geometries: STL models imported as light weight geometries are currently not supported for interactive wedge editing (as of R24.1).

In the Initial Mesh Settings for SBR+ Solution type or for HFSS with Hybrid and Arrays that include SBR+ regions, you can turn off tolerant meshing.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

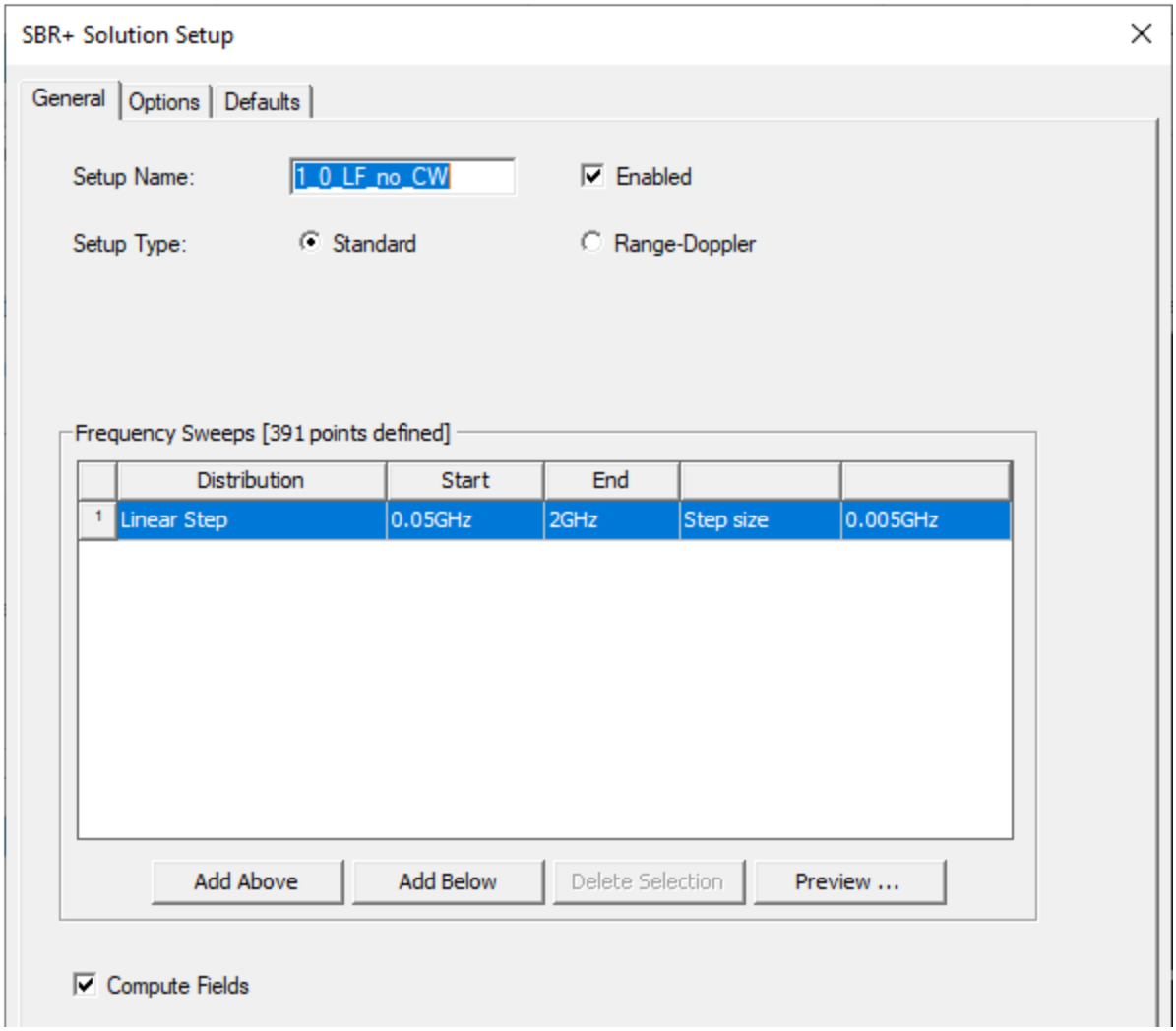


In this case you should consider Current Limitations on PTD/UDT Wedge/Generation Editing.

4. Add an SBR+ Solution Setup by selecting the **Simulation** tab and clicking the Setup icon.

SBR+ solutions offer a reduced solve setup dialog with a frequency sweep definition and SBR+ specific **Options** tab.

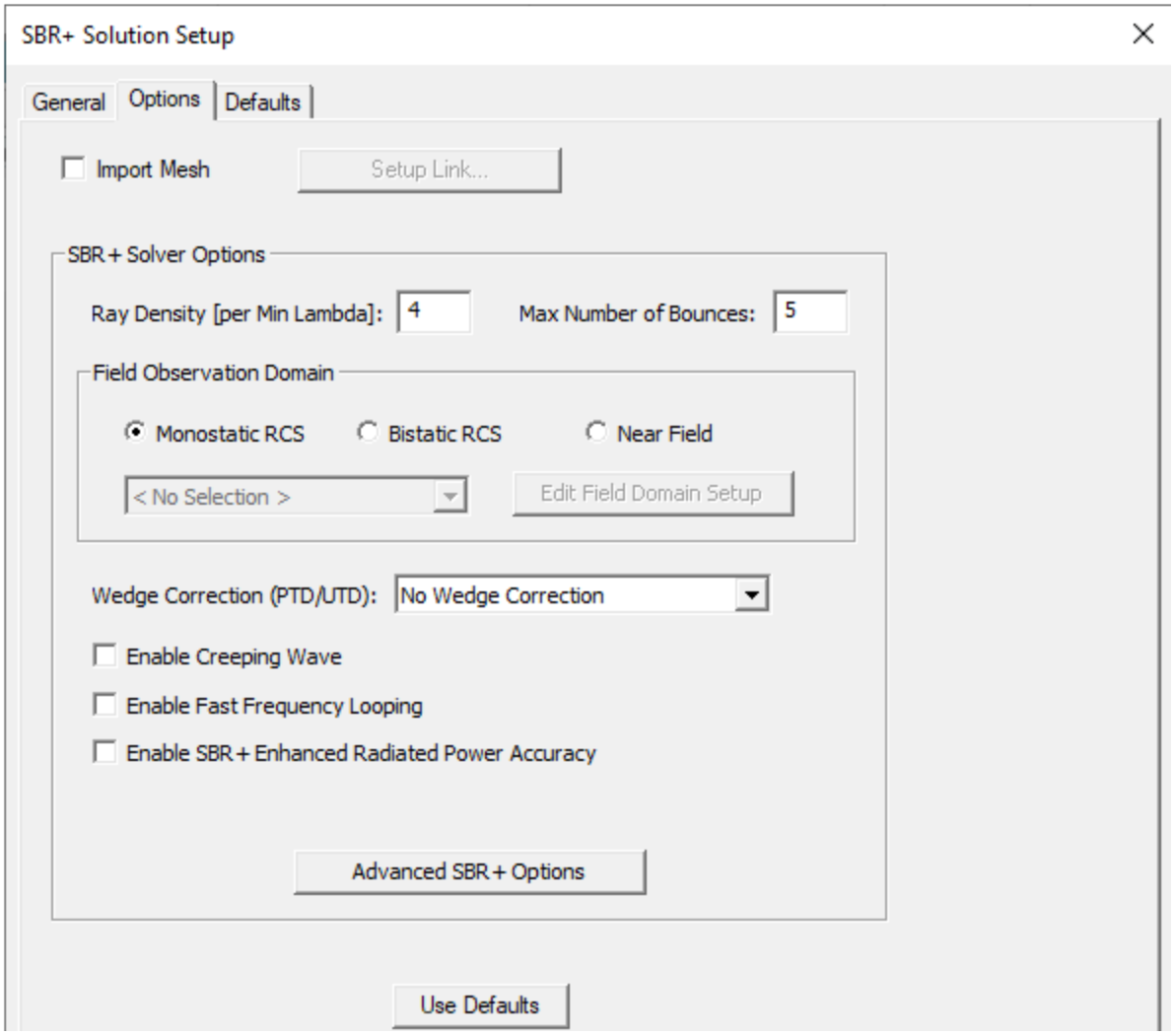
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- There are two options for Setup Type: Standard (default) and Range-Doppler. The settings and specialized capabilities of the Range-Doppler type are described at SBR+ Range-Doppler Solution Setup. The rest of this page pertains only to the Standard setup type.
- Because you can define Frequency Sweeps here, you cannot add Frequency sweeps under the setup.
- Switch on **Compute Fields** to generate installed antenna patterns or perform RCS simulations using SBR+. Switch off **Compute Fields** to restrict simulation results to coupling data (S-parameters) between two or more antenna components configured in the design (i.e., source design links, parametric antennas, imported .ffd far-field gain pattern files).

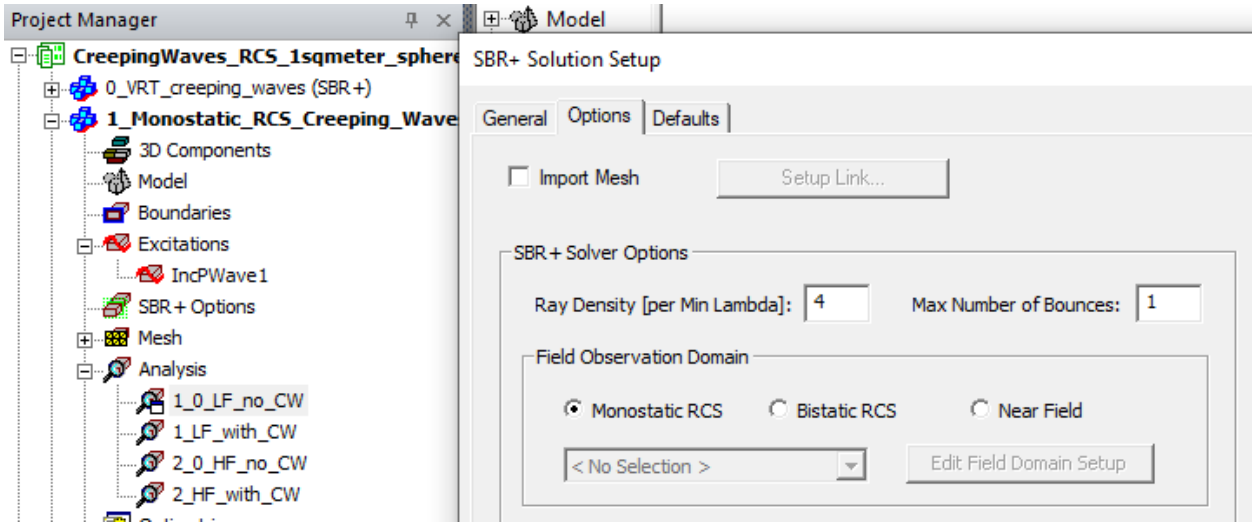
The **Options** tab lets you specify SBR+ Solver Options, including **Ray Density Per Wavelength** and **Maximum Number of Bounces**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



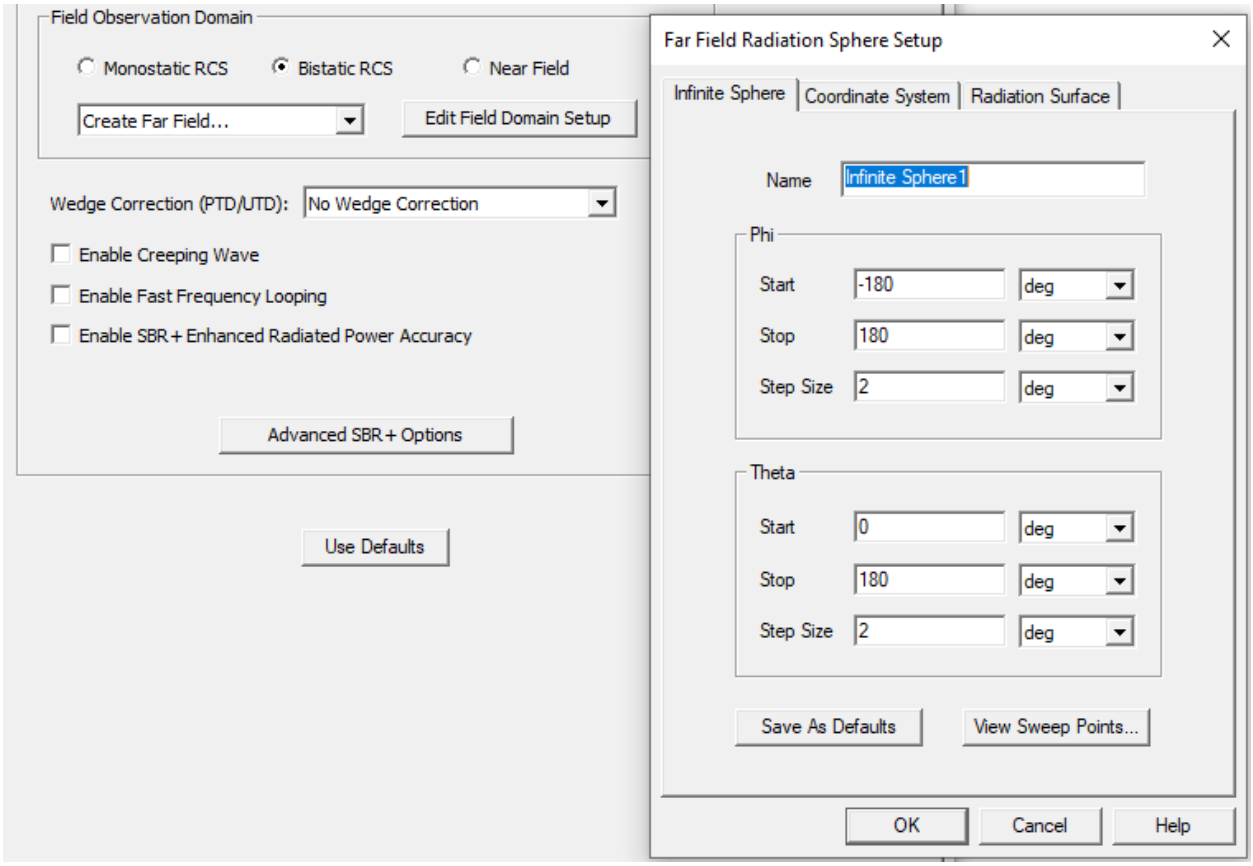
For the RCS Type options for Monostatic and Bistatic to appear, you must include an Incident Plane wave. For Monostatic RCS, you do not need to create a Far Field setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

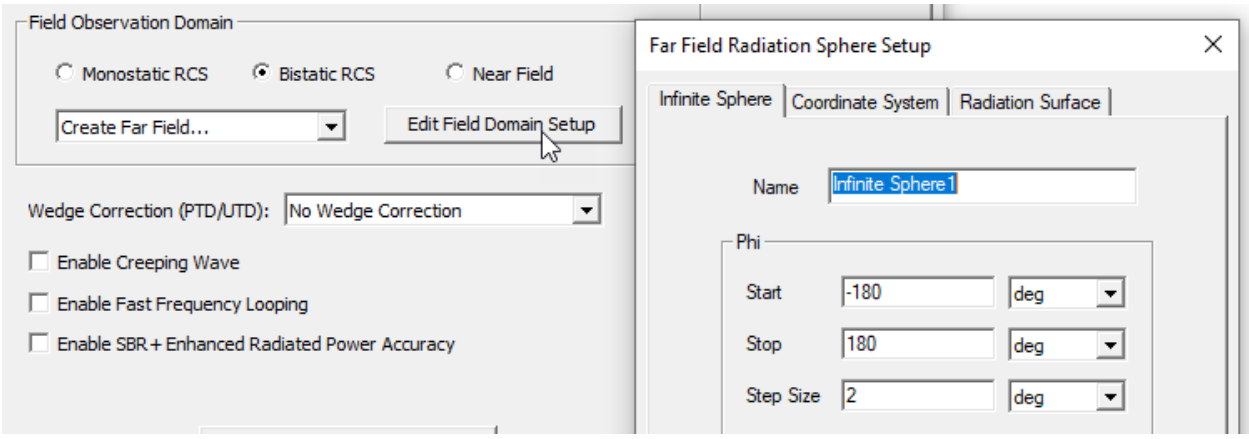


If you have assigned both an SBR+ region and an Incident Plane wave, and select Bistatic as the RCS Type, the Near Field and Far Field Observation domain fields are enabled, and the **Far Field Radiation Sphere Setup** dialog box appears as follows, allowing you to create an infinite sphere for a Far Field or a rectangle for examining near fields. You can edit an existing Far Field Infinite Sphere or Near Field Rectangle setup by selecting from the drop-down menu and clicking **Edit Field Domain Setup**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you have created one or more field observation domains, these are listed on the drop-down menu. You can edit an existing Far Field Infinite Sphere or Near Field Rectangle by clicking **Edit Field Domain Setup**.



For Near Field rectangle setups, you can use the feature described in Overlay Contour Plot of

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

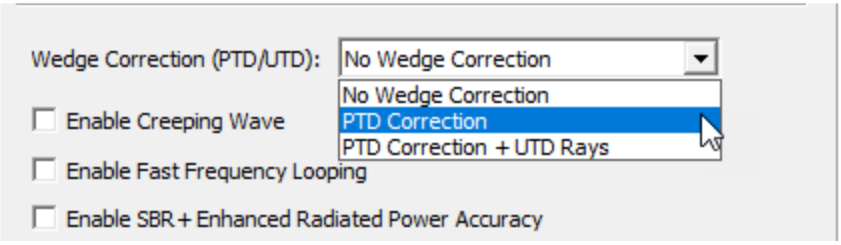
Near Field on Rectangle.

For **Enable Creeping Wave**, see Creeping Waves, Creeping Wave VRT Plots, Creeping Wave for Antenna Placement, and Creeping Wave for RCS of Highly Curved Objects.

The **Fast Frequency Looping** feature is to significantly reduce simulation time for high-density frequency sweeps with tens to thousands of uniform frequency samples. You can select Enable Fast Frequency Looping if you have configured a uniform frequency sweep with at least 2 frequencies. From there, the acceleration is handled automatically, with no other workflow changes in problem setup or post-processing. The acceleration engages for all SBR+ field and signal output types: S-parameters, far-field antenna patterns, near-field E/H-field spatial distributions. See SBR+ Fast Frequency Looping for High-Density Sweeps for details on the benefits and limitations of this acceleration feature.

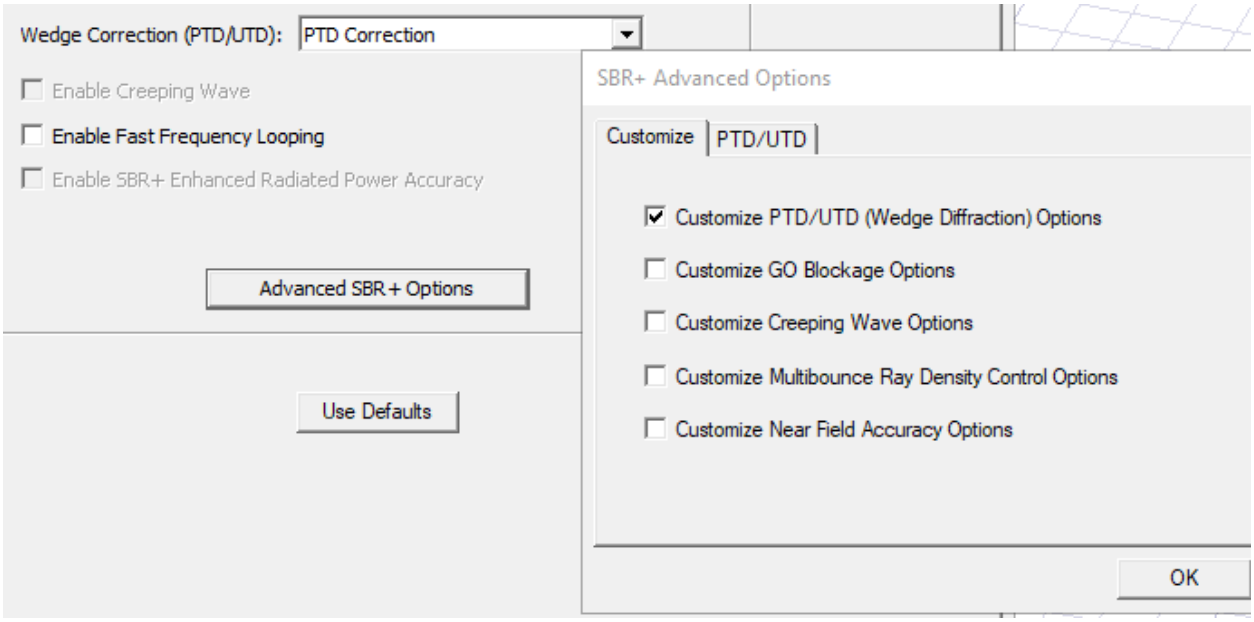
The **SBR+ Enhanced Radiated Power Accuracy** feature can be beneficial when an SBR+ region contains lossy materials (got example, finite conductors, half-space layered impedance boundaries, or absorbers). Enabling this feature produces a more accurate result by having SBR+ compute total fields across the entire far-field sphere. These total field results are then weighted using the configuration specified in the Edit Sources dialog and then integrated over the far-field sphere to determine the total power radiated by the design. The total power is then used to improve the accuracy of the directivity fields and other related antenna parameters such as radiated power.

The **Wedge Correction (PTD/UTD)** simulation settings allow for the inclusion of additional wedge diffraction phenomenology that can improve the accuracy of SBR+ simulations. You can opt out of using the PTD/UTD settings, or select PTD Correction or PTD Correction + UTD Rays. Selecting one of the PTD Correction options enables a field for specifying PTD Edge Density.

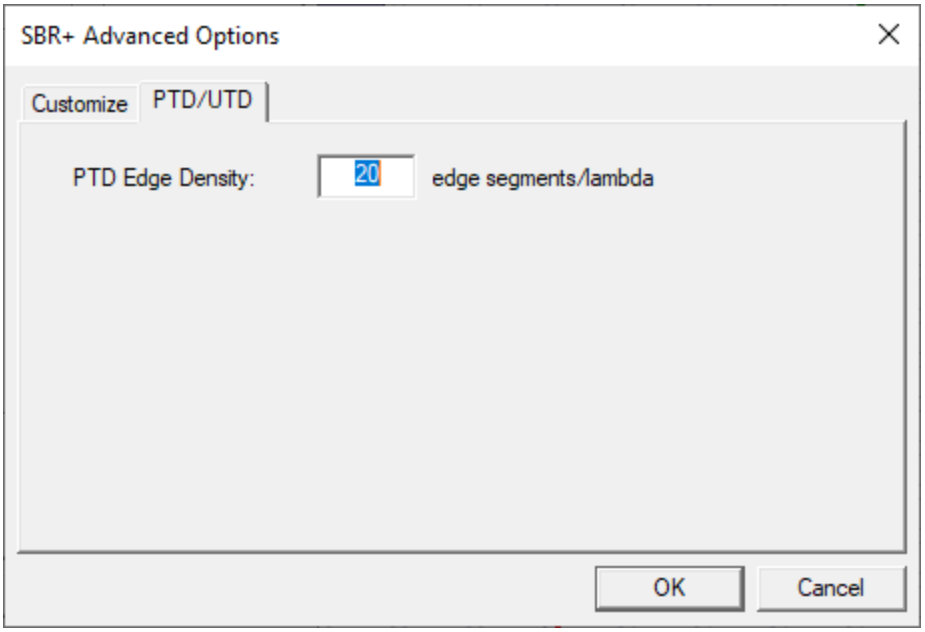


If you select one of the PTD Correction options you can then select the **Advanced SBR+ Options** button to Enable PTD/UTD (Wedge Diffraction) Options.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



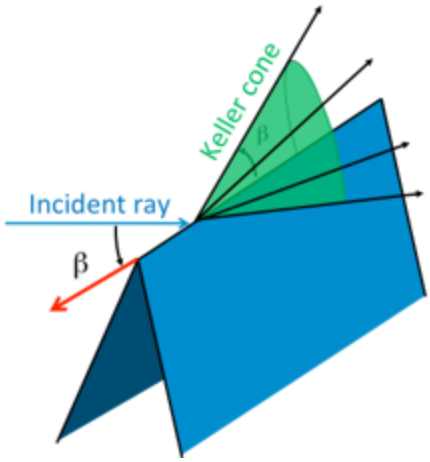
Enabling PTD/UTD (Wedge Diffraction) Options opens **PTD/UTD** tab for specifying PTD Edge Density.



For SBR+ simulations, the Physical Theory of Diffraction (PTD) and Uniform Theory of Diffraction (UTD) features can account for additional phenomenology not well predicted by plain SBR due to truncation of uniform Physical Optics (PO) currents at sharp angular discontinuities (“wedges”) on metallic surfaces and blockage of SBR’s Geometrical Optics

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(GO) rays. PTD is a numeric correction to the scattered fields radiated by PO currents near wedges. UTD launches bundles of edge-diffraction rays from directly illuminated portions of each wedge along the Keller cone. Once launched, the UTD rays behave exactly like regular SBR rays, propagating according to GO and painting PO currents at each bounce that contribute to the scattered field. The UTD rays often illuminate portions of the SBR scattering geometry that are never reached by SBR GO rays launched directly from the field source.



The PTD and UTD wedge features are only deployed for metallic wedges with line-of-sight visibility from the source (Tx) location. If either adjacent surface of the wedge is non-PEC and not within the tolerance for PEC-like, or if the entire edge segment is not visible to the source, the wedge will be ignored in the SBR+ simulation and for visual ray tracing (VRT). For technical details, see Assigning SBR+ Hybrid Regions.

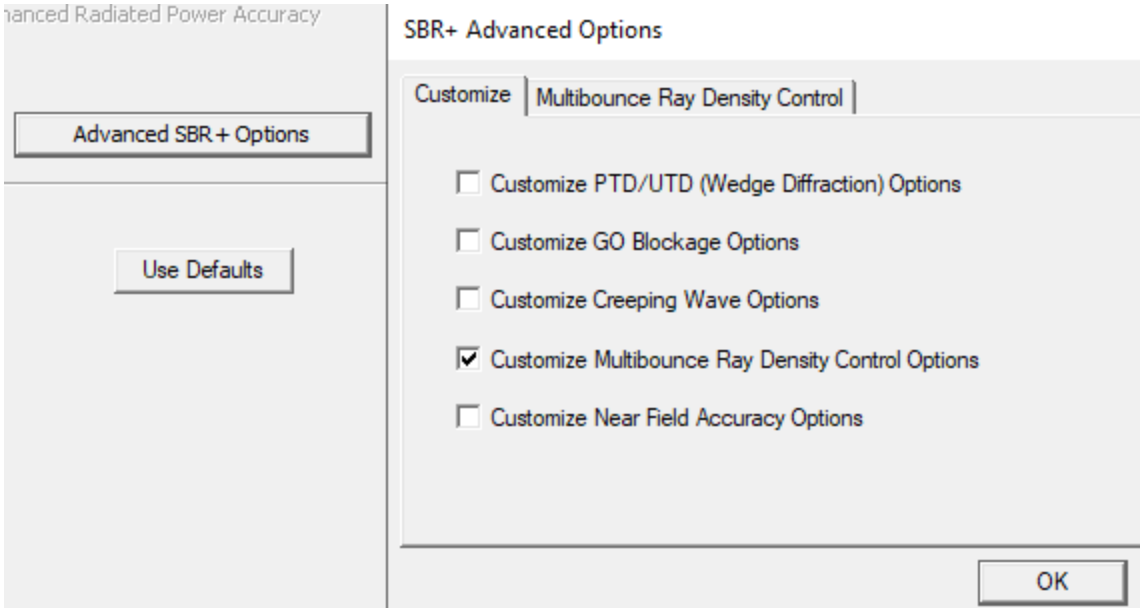
5. Advanced Option for NF Accuracy Settings for SBR+

The NF Accuracy settings (Beta feature, NF Accuracy Controls for SBR+) provide access to methodology settings relating to cases where Tx antennas, Rx antennas, and near-field observation points are in proximity to the scattering geometry (i.e., the platform CAD model). When this feature is switched off, HFSS SBR+ uses default near-field accuracy settings that should be adequate for most situations, including close-proximity Tx/observer conditions that this feature is designed to tune.

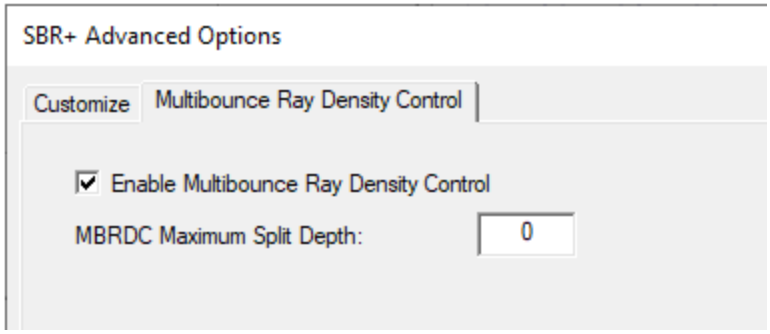
6. Advanced Option for Multibounce Ray Density

By option you can select the **Advanced SBR+ Options** and enable Customize Multibounce Ray Density Control Options.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can then select the **Multibounce Ray Density Control** tab.



Use the checkbox to **Enable Multibounce ray density** and enter the maximum number of subdivisions to be used in the solve. Enabling the feature may increase the solve time.

The **MBRDC Maximum Split Depth** field value increases the number of spawned ray tracks on each ray bounce. The MBRDC algorithm is designed to achieve the ray density per wavelength criterion at any bounce depth. If a footprint at depth *N* in a ray track does not satisfy the ray-density criterion, the MBRDC algorithm attempts a refined ray shoot from an earlier bounce, just for the ray in question. The size of the triggering footprint informs the level of refinement. If the refined shoot is not successful in achieving the desired footprint size, or new footprints are too large, the MBRDC algorithm attempts further, recursive refinements. The **MBRDC Maximum Split Depth** setting limits how aggressively the refinement algorithm is applied by specifying the maximum number of split recursions. For this reason, linear

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

increases in Maximum Split Depth can yield a geometric progression in the total number of rays shot and associated solution time.

When both MBRDC and UTD are enabled, first-bounce UTD rays and bright points on wedges are recomputed during the MBRDC splitting process, but only the original raytrack is rendered up to the 1st bounce. Similarly, length-based ray filters are applied to the original UTD initial ray track up to 1st bounce rather than using the information in the additional MBRDC tracks. This is a known limitation.

7. Advanced Option for Geometrical Optics (GO) Blockage

SBR+ normally applies a physical optics (PO) blockage model where blockage effects emerge from partial cancellation of the incident field by the scattered field. PO blockage not only occurs in connection with the incident field from the Tx antennas, but also within the scattered field itself when reflected fields from the previous bounce are diminished by a blocking surface along the geometrical-optics (GO) reflection path at the current bounce by evaluating the scattering contribution of the incident ray field at that bounce.

In cases of significant blockage by a large obstruction, the PO blockage model requires accurate currents over the extent of the obstruction. Sometimes, this is hard to achieve in the ray tracing approximation. For example, the obstruction may not be well illuminated by direct rays from the Tx antenna or multi-bounce rays after an earlier reflection. An alternative is to use a GO blockage model where scattered field contributions are added to an observation point subject to a line-of-sight (LOS) blockage check performed by the ray tracer. In some cases, this is more accurate than the default PO blockage formulation, while in others it is less accurate. GO blockage also entails a non-trivial computational cost, as the blockage check with the ray tracer must be performed for each observation angle (or point) for each ray hit point.

In addition to GO Blockage for SBR and UTD ray tracks, this feature is also available for PTD and CW.

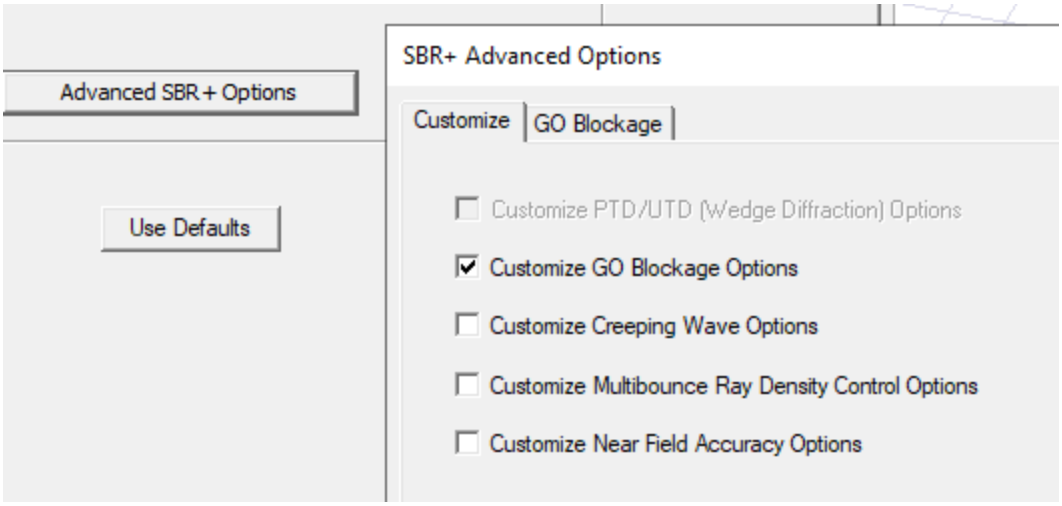
Limitations for GO Blockage:

GO Blockage formulation is suited specifically for SBR+ ray tracks, including UTD ray tracks. While GO Blockage for CW and PTD has been implemented for the first time in this release, results can be surprising and/or inaccurate, as a limitation of the methodology.

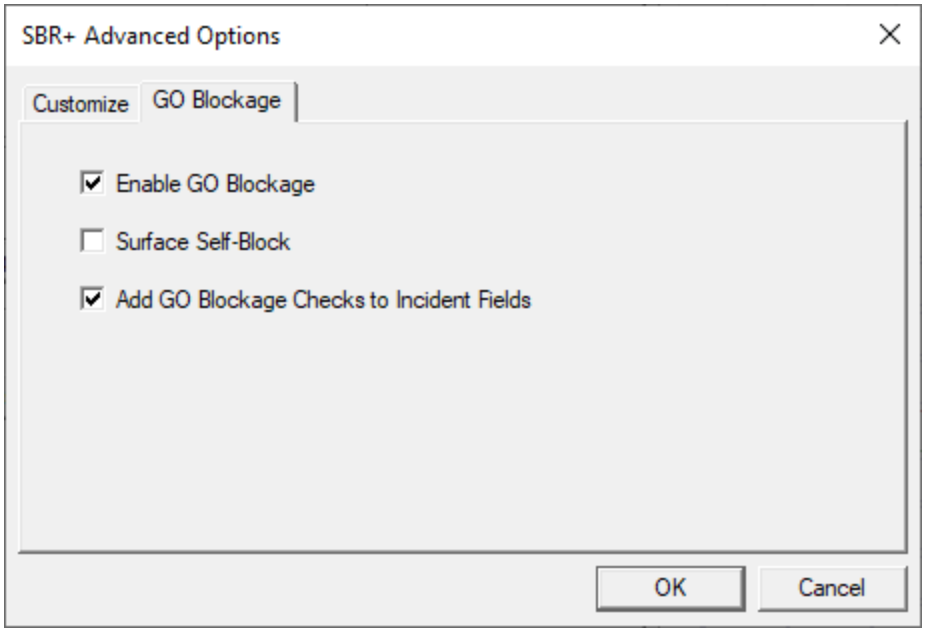
Summary Workflow for GO Blockage:

Click the **Advanced SBR+ Options** button to open the **SBR+ Advanced Options** window. On the **Enable** tab, Check GO Blockage Options to enable the **GO Blockage** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



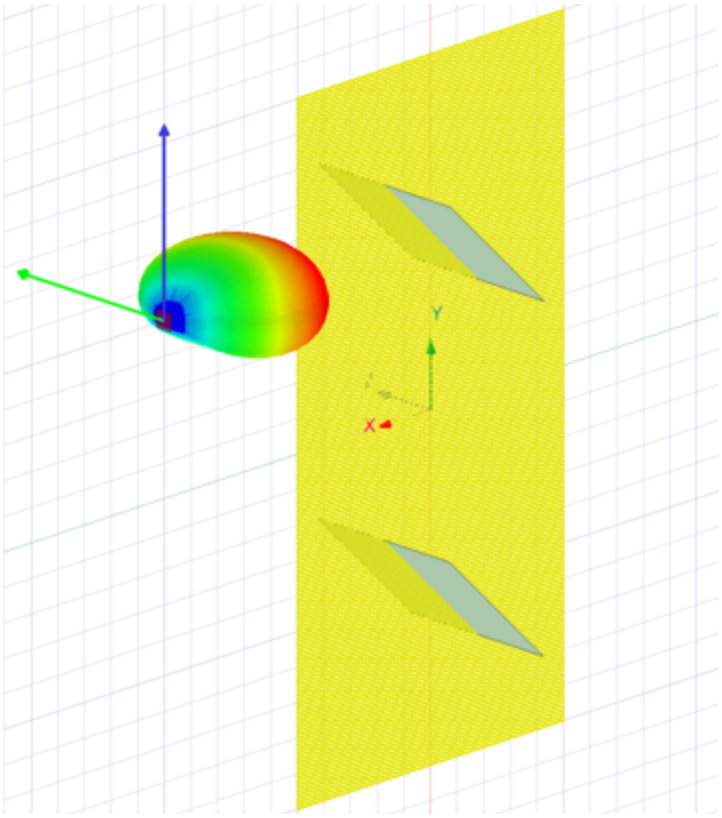
On the **GO Blockage** tab, you have check box options to enable or disable GO Blockage, Surface self-block, and GO Blockage Checks for Incident Fields..



Surface Self-Block: if enabled, the surface where the footprint is radiated from is used for blockage check, that is, anything below the surface with respect to the incoming ray is not in line of sight. If the setting is disabled, the footprint surface is not used for blockage checks (anything below the surface with respect to the incoming ray is in line of sight), but other surfaces are used for blockage.

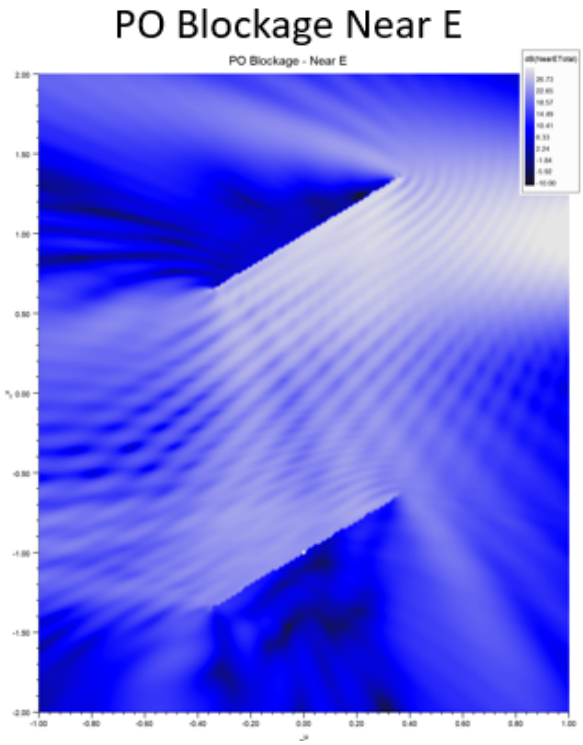
For example, consider a parametric beam antenna illuminating a series of plates.

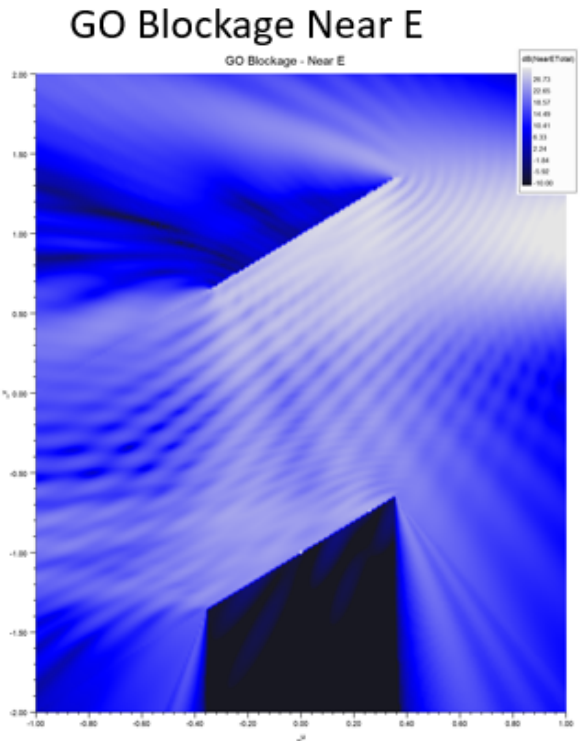
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the plots, the shadows behind the plates are much more pronounced with GO Blockage enabled, but the fields have non-physical discontinuities.

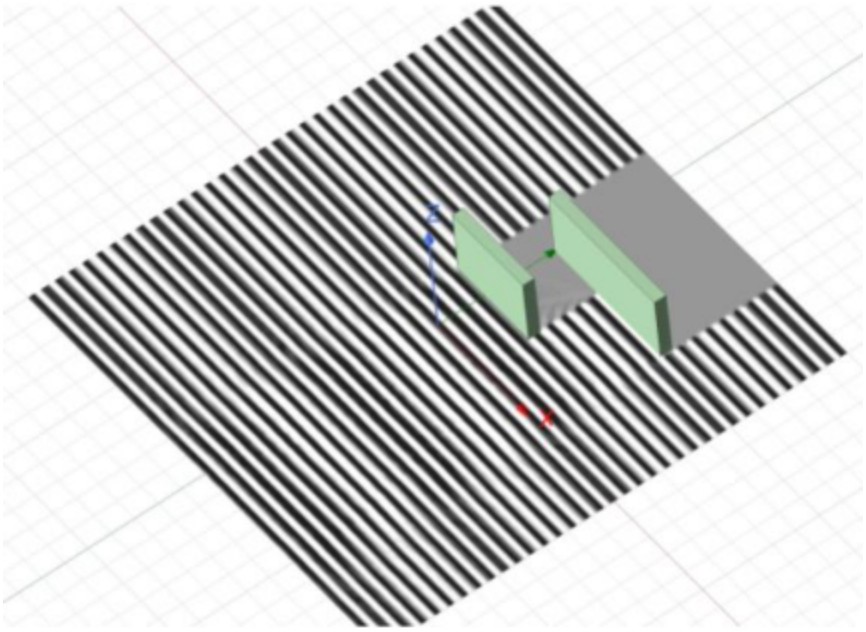
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





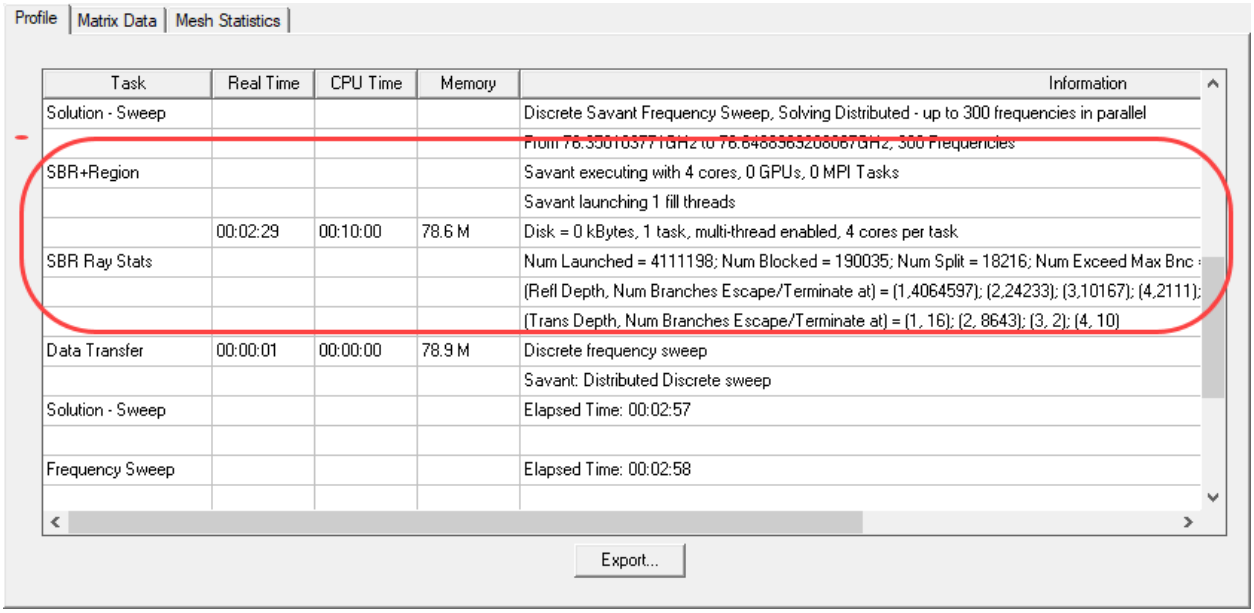
Add GO Blockage Checks to Incident Fields: if enabled,GO blockage effects will also be calculated for the incident field via line-of-sight checks from sources to field points, effectively computing a direct field in the Geometrical Optics sense. For example, the figure below shows the total E-field of a planewave incident on PEC bodies with this option enabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8. Perform Analysis and view Solution Data for SBR+ Solution

After analysis, you can view the **Solution** profile and mesh statistics. Because SBR+ regions cannot contain ports no matrix reports will be provided. Similarly 3D fields, near fields, and port field displays are not available.



SBR+ Ray Stats include:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

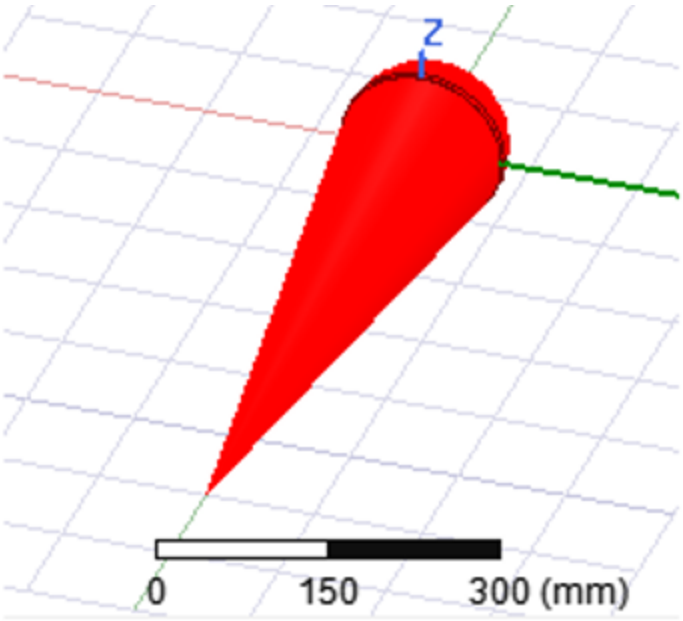
- Number of rays
- Histogram on Reflection
- Histogram on Transmission

If included in the solution, Creeping Wave Stats include

- Number of rays
- Ray lengths
- Source Counts by Initialization Failure Condition
- Ray Counts by Termination Condition
- Histogram On CW ray footprints

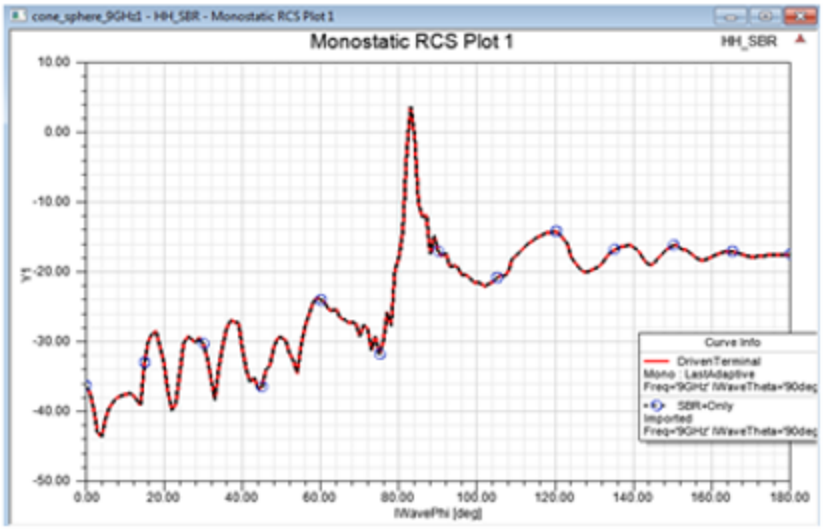
For details see Ray Statistics for SBR+ Analysis.

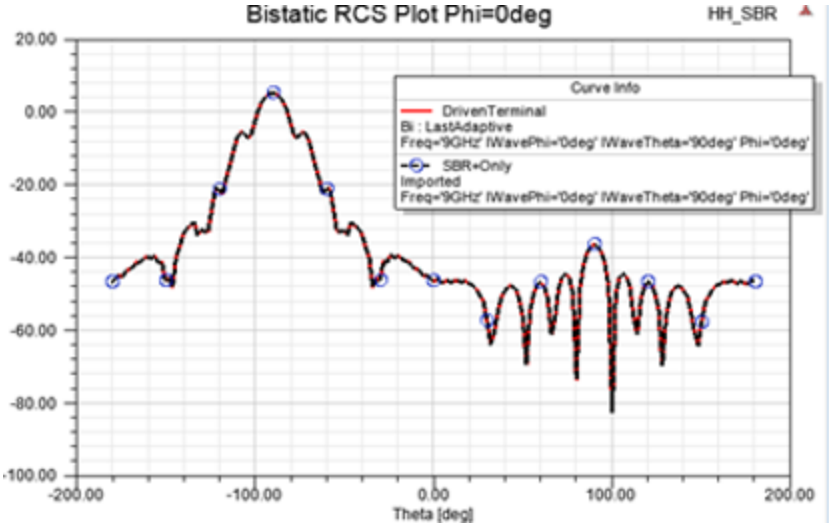
Matrix reports and Network data explorer are available if any parametric antenna or file based antenna is in the design. Far field reports (including antenna parameters) are available. 3D fields, near fields, and port field displays are not available. The following example shows monostatic and bistatic RCS reports for a cone sphere, comparing SBR+ compared to a driven terminal simulation. In a Driven Terminal design, the cone was assigned an SBR+ Region.



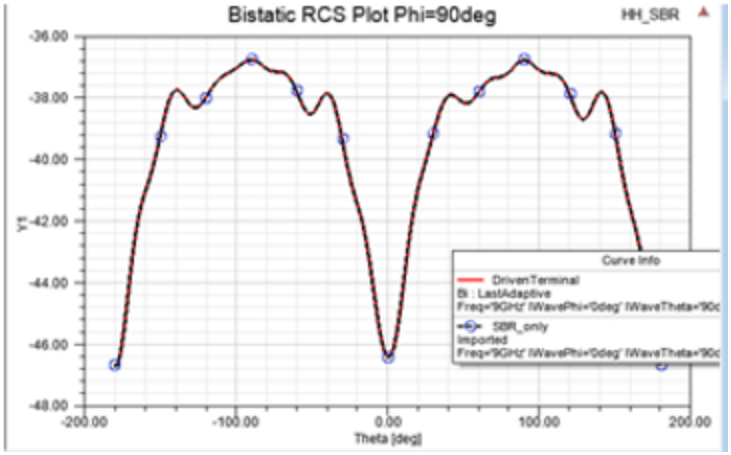
The following plots compare Bistatic RCS and Monostatic RCS in dB. The results match very well.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

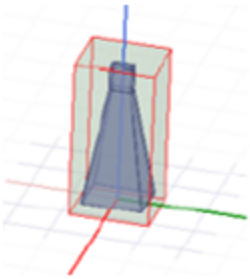




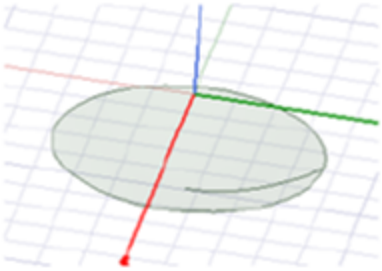
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following design shows Pyramidal Horn Antenna and FEM Parabola: SBR+ vs. HFSS IE Hybrid Region:

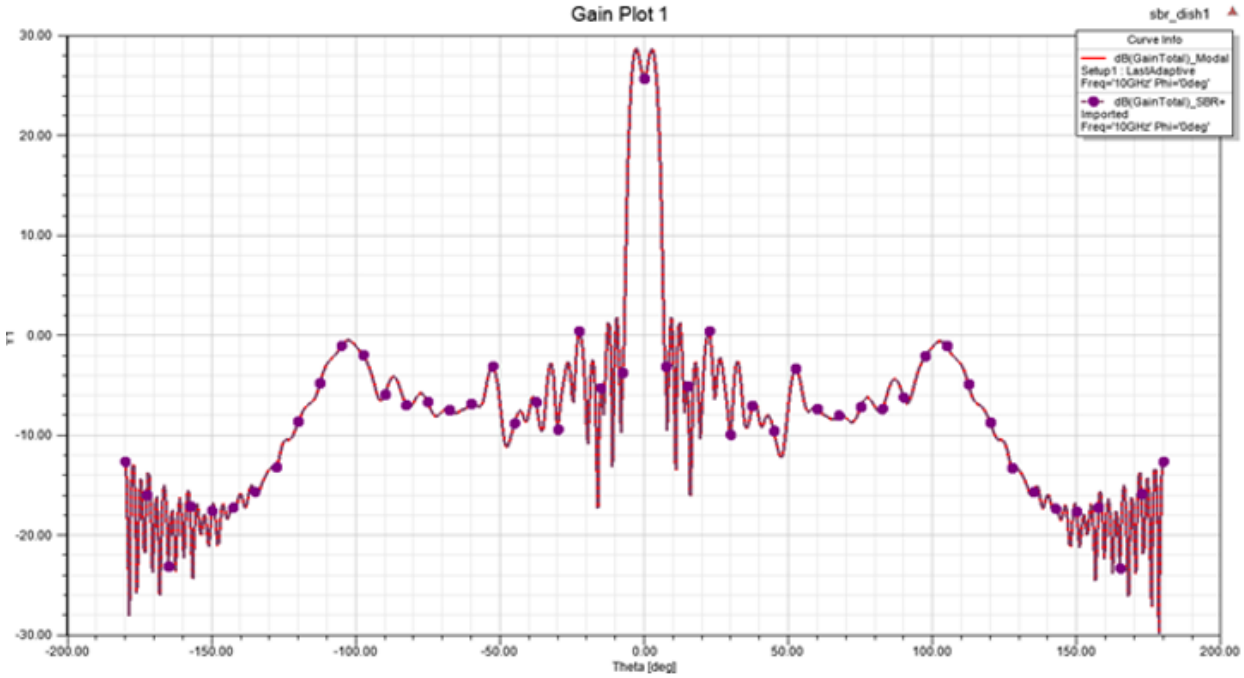


Pyramidal Horn Antenna



Parabola

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3D Components for and from SBR+ Designs

A new SBR+ 3D Component type is available. It can be a component created from an SBR+ design, or from a Driven Model/Terminal design while all solid and sheet geometries included are all assigned SBR+ Regions. These components always have the Mesh Assembly option available.

A 3D component containing only SBR+ region can be inserted into the SBR+ only design, irrespective to the solution type of the source design. A 3D component created from an SBR+ only design can be inserted into any design when SBR+ region is supported in that design's solution type.

If you create a 3D component from an SBR+ Solution type, that component does not contain a **Hybrid Regions** tab. However, if you create a component from an non-SBR+ Solution type, if Hybrid Regions are legal, it will contain a **Hybrid Regions** tab. Editing an SBR+ 3D Component will open the model in the corresponding source Solution Type.

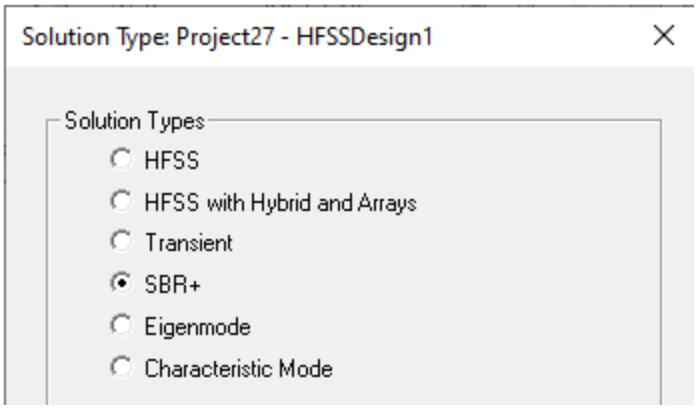
Selecting the SBR+ Solution type also permits you to use Parametric Antennas.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Configure Simulation with SBR+ Antenna Geometry Blockage (SBR+ Solution Type)

The typical configuration begins with a fully integrated model with all geometry. You then select the near antenna region including any nearby geometry interacting with the antenna for analysis by HFSS – either by identifying these objects by hybrid region assignment, or by copying them out to a separate source antenna design to be solved with the HFSS solver, etc. The nearby antenna geometry could even be something different than what is used for the integrated model, such as a flat plate or other structure for considering the nearby antenna interactions. This source antenna design should likely be enclosed in a FE-BI region to allow for easier downstream processing, whether we are in a hybrid or SBR+ solution type. If you use a separate source antenna design, you can re-introduced it to the target design with the full geometry model by creating a 3D component for an antenna that is a linked design to the source design. This means you no longer need to designate any “Model Blockage” when setting up the linked design. You do this as a separate step outside creation of the linked design by selecting geometry from the SBR+ region to be designated as “Model Blockage” associated with a particular source antenna. Note there is no restriction on what objects and/or materials can be designated as “model blockage” geometry. Also note that for either solution type, SBR+ or hybrid HFSS-SBR+, that this selection will be from the available SBR+ region geometry objects, which is different than the previous workflow. Once you select these antenna blockage geometries for each source antenna, HFSS with SBR+ can solve the combined target SBR+ design.The preferred work flow is this.

1. Create/open an HFSS design with SBR+ solution type.

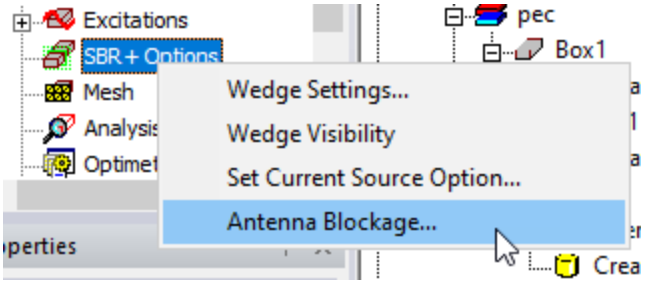


2. Add an Antenna to the design via the menu option for **3D Components > Create Antenna > Link to Source Design...** or **Excitations > Create Antenna Component > Link to Source Design...**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

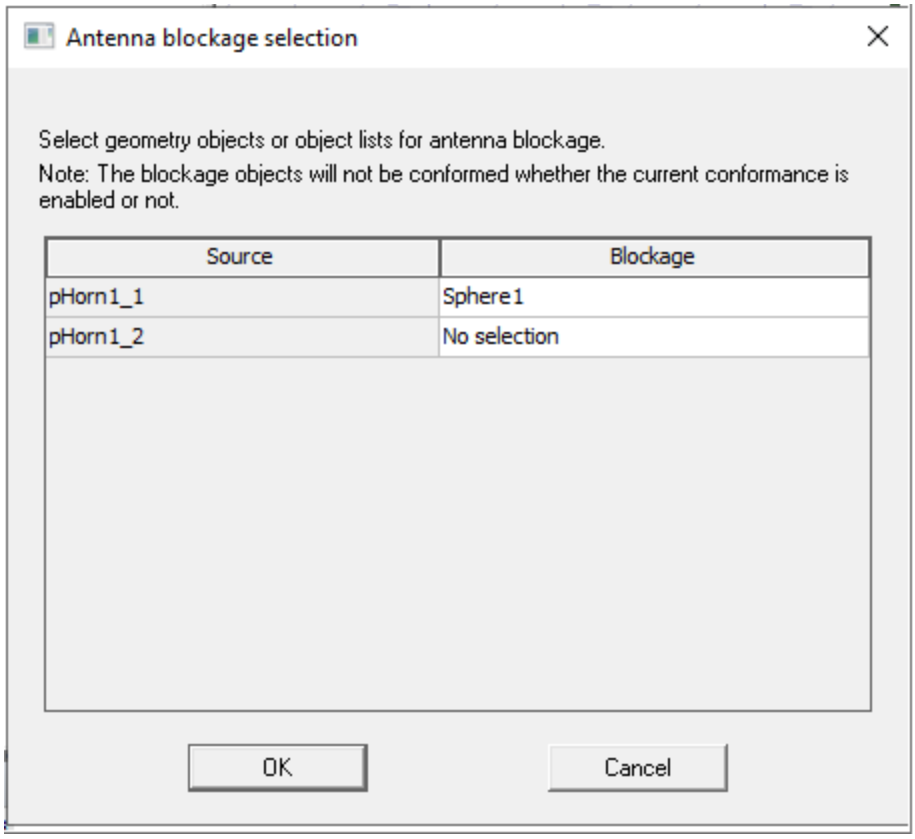
This linked design should be an HFSS design that contains all the relevant geometry to represent nearby interactions with the antenna, such as a radome, enclosure, ground plane, mounting structure, or other nearby (within a few wavelengths) objects.

3. Add the desired SBR+ geometry objects to the design, configuring all materials/boundaries as desired.
4. If you want to designate multiple SBR+ geometry objects as blockage, select them in the 3D Modeler window and create an Object List by selecting **Modeler > List > Create > Object List**. You may want to name the list appropriately. If only a single object is to be designated as a geometry blockage, skip this step.
5. Using the right click menu under **SBR+ Options**, select **Antenna Blockage...**



This opens the **Antenna blockage selection** dialog.

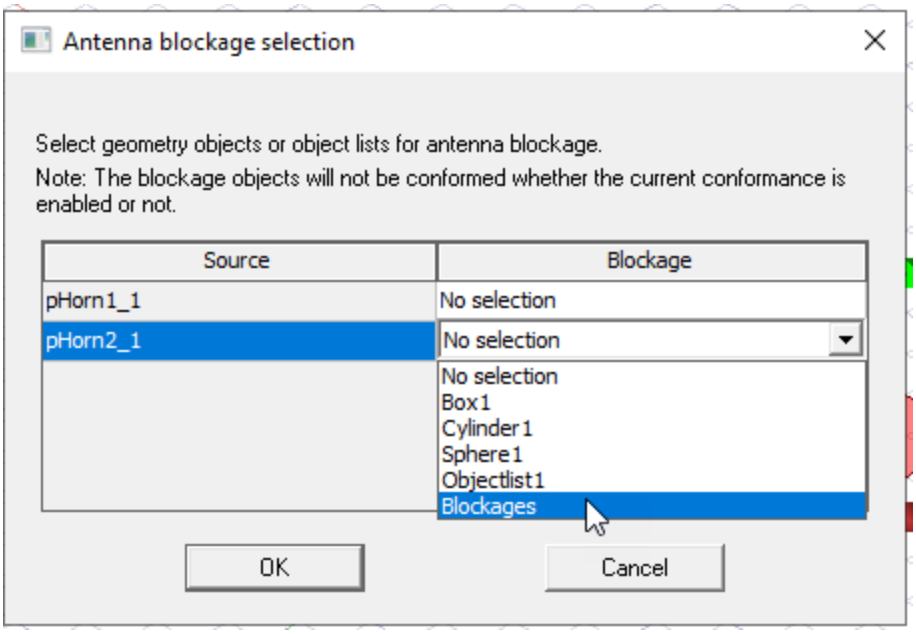
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. For each antenna source, use the dropdown menu for each cell in the column to select the geometry objects to associate with that source as an associated blockage. Each cell in the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

column contains a dropdown list of all geometry objects and all user created object lists.



7. Click OK to save the configuration.
8. Create an Analysis Setup.
9. Run the simulation.

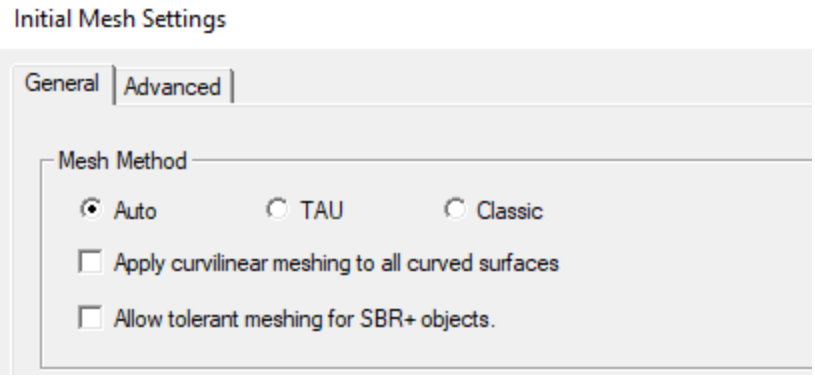
Current Limitations on PTD/UTD Wedge Generation/Editing

Situations where wedges cannot be generated:

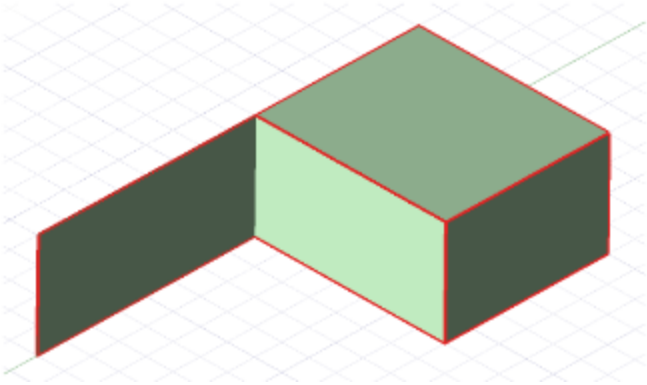
Light Weight Imported STL Geometries: STL models imported as light weight geometries are currently not supported for interactive wedge editing (as of R24.1).

In the Initial Mesh Settings for SBR+ Solution type or for HFSS with Hybrid and Arrays that include SBR+ regions, you can turn off tolerant meshing.

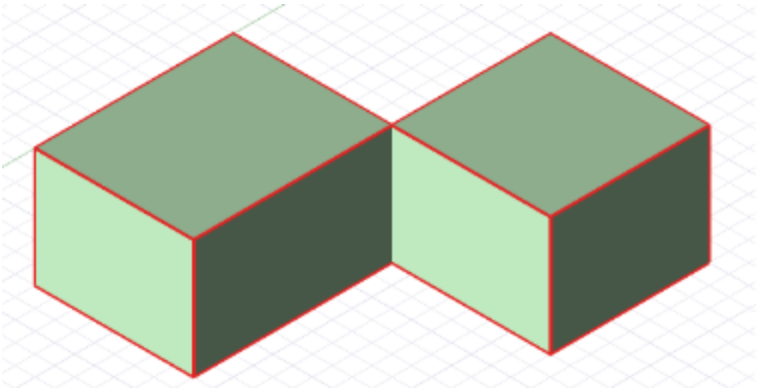
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In these cases, wedges cannot be generated at some locations where different geometry bodies are making contact. More specifically, these are locations on geometries where the mesh edges are in contact with three or more mesh faces.

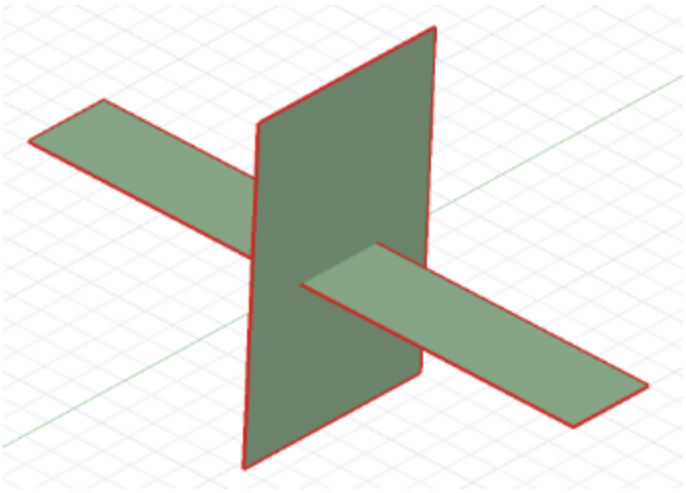


This includes both cases where geometries are merely touching and cases where geometries are outright intersecting.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This includes three dimensional bodies contacting three dimensional bodies, three-dimensional bodies contacting plane objects, and plane objects intersecting plane objects:



Note that on an edge where two plane objects touch (but not intersect), wedges can be generated however the processed wedge angles may be misleading (see next section).

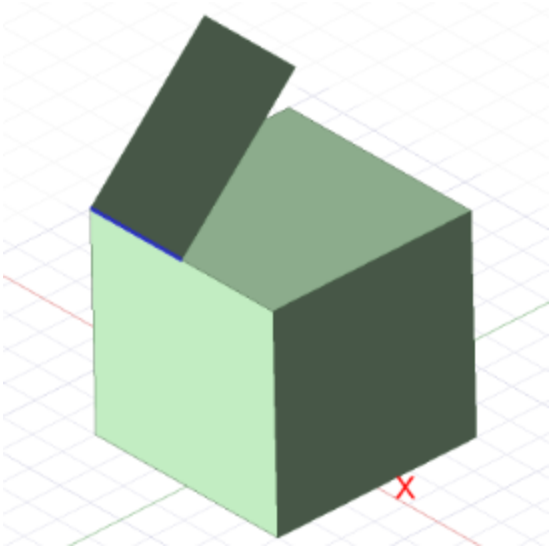
With tolerant meshing, wedges can be generated at points of contact between different bodies. However, these wedges may not have physically correct wedge angles (see next section).

Situations where visualized wedge angles can be misleading

With tolerant meshing, wedges can be generated at the locations of contact between geometries where it would otherwise not be possible with rigorous meshing. However, often in these cases the generated wedges will be processed with interior wedge angles different from the ones visualized in the model window and as such can lead to unphysical simulation results.

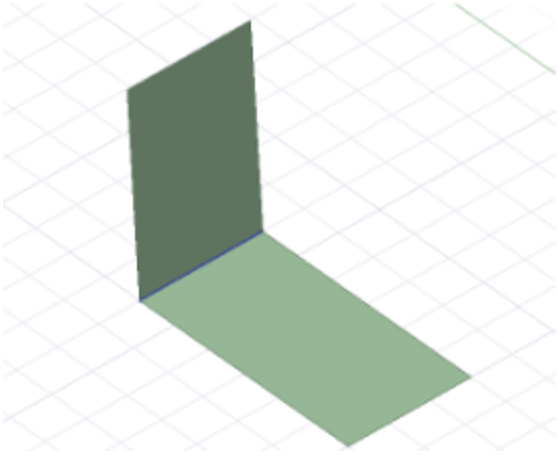
As objects are meshed independently of one another with tolerant meshing, wedge angles on objects will not consider any contact with other objects. This issue is illustrated by the following examples:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



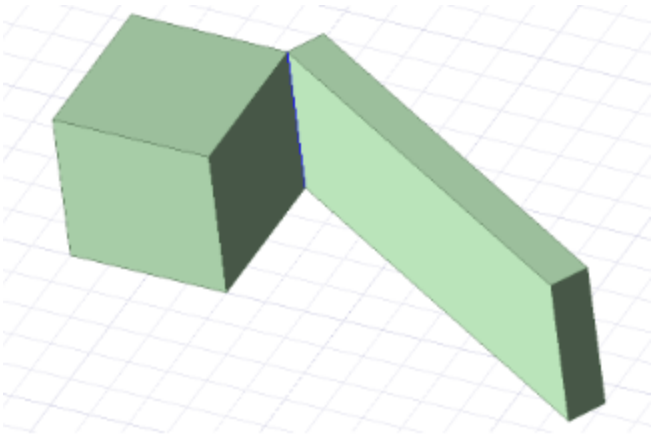
In this example a plane contacts a cube at a 45°angle and a wedge is generated on the edge of contact. From the model window it may appear that this wedge will be simulated with an angle of 45° however depending on whether the edge of the plane or the edge of the cube was selected, a wedge angle of 90° or 360° will instead be used by the solver.

In the next example two planes meet at a 90° angle and a wedge is generated on the edge of contact. Similar to the previous example, the wedge appears to have an angle of 90° in the model window but the solver will actually consider the angle to be 360°. Note that in this case, the user can unify the two planes to get the correct 90°.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the final example a cube contacts a rectangular solid at an angle of 30° from the perspective of the image. As before the wedge generated on the edge of contact will be simulated with an angle of 90° instead of the 30° or 150° which may be intended by the user.

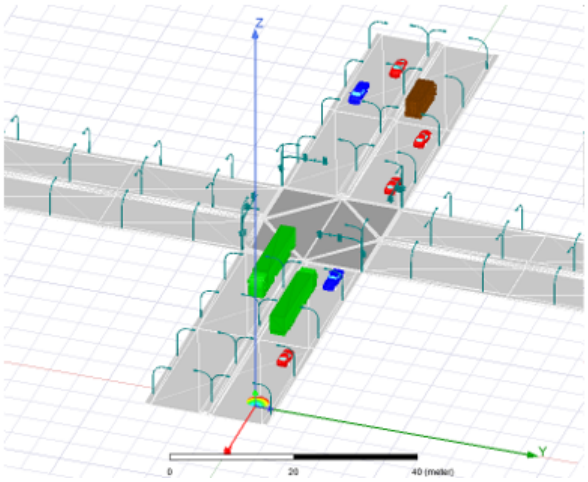


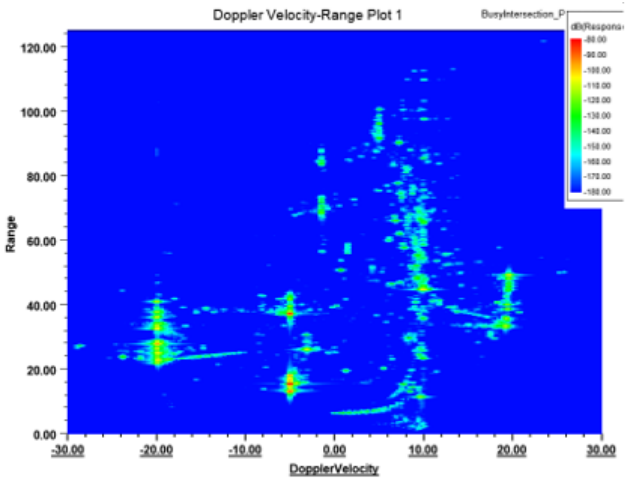
SBR+ Range-Doppler Solution Setup

The Range-Doppler Setup Type option within the SBR+ Solution Type provides a highly optimized simulation capability for range-Doppler radars, such as are commonly found in modern automotive radar sensors and also many aerospace applications. It simulates what a range-Doppler radar will observe looking into a complex environment, such as a traffic scenario or, for an airborne radar, objects on the ground. The results are presented as range-Doppler color-map images. The SBR+ radar simulation is massively accelerated relative to conventional implementations using Ansys's proprietary Accelerated Doppler Processing (ADP) technology. ADP also provides improved image quality by eliminating Doppler artifacts of the ray-tracing solution.

Range-Doppler radars operate by issuing a sequence of many pulses or chirps across a coherent processing interval (CPI). The received signal from each chirp is used to develop a time response, which is then scaled as range (distance) under the assumption of a monostatic radar configuration (that is, one where the radar Tx and Rx antennas are nearly in the same location). The range response is discretized into range bins (pixels).Then, by comparing and Fourier processing the phase response across the multiple chirps comprising the CPI, the radar develops a Doppler velocity response for each range bin. The result is a 2D range-Doppler image. Two radar waveforms are supported: pulse-Doppler and chirp-sequence frequency-modulated continuous wave (CS-FMCW).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





With the Range-Doppler Setup Type, both the radar antennas and scene objects in the surrounding environment are assigned motions (for example, linear velocities, accelerations, rotations) in the **History** tree that depend on a user-defined time variable. Through the main dialog of this setup type, radar processing specifications are configured in terms of center frequency and range-Doppler performance specifications: range and velocity resolutions and periodic (aliasing) ambiguities. Simulation can either be performed at a single time step or, via Optimetrics, a sequence of time steps. ADP massively accelerates the simulation by eliminating the need to separately solve the hundreds of individual chirps across the radar's CPI in microscopic time-stepping of the dynamic scene. This acceleration technique carries the side benefit of eliminating Doppler simulation artifacts caused by abrupt changes in some ray paths between chirps of the CPI.

The Range-Doppler Setup Type has a special Range-Doppler report type for displaying the range-Doppler images in color scale. These can animated according to the Optimetrics time sweep, either in a separate window or together with the 3D scene.

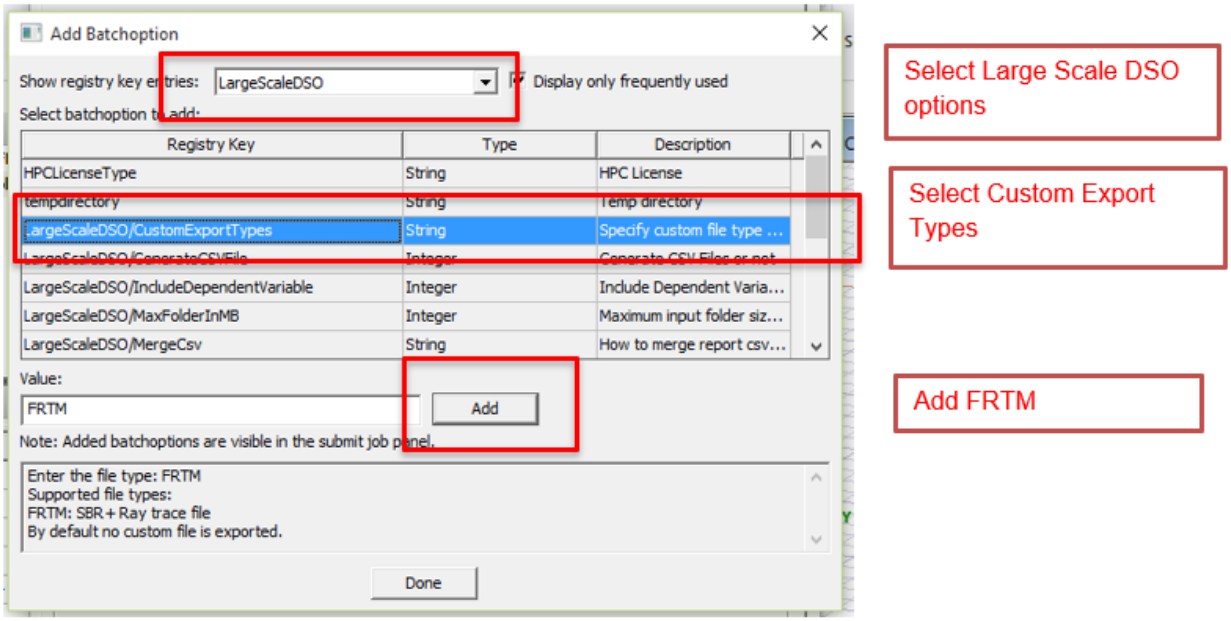
Prerequisites

Requirements and restrictions for Range Doppler Setup Type in SBR+ Solution Type:

- A user-defined time variable that governs the motion of objects through expressions in coordinate system definitions or Modeler tree motion operations.
- Two or more radar antenna ports defined as either built-in parametric antenna with Far-Field Source, imported far-field antennas, or far-field links to a source design. Near-field links are not supported with the Range-Doppler Setup Type.
- The range-Doppler setup is not compatible with Incident Plane Wave excitation and RCS simulations.
- Moving objects that interact with the radar must be imported as 3D Components.
- Stationary objects may defined within the range-Doppler design in the usual fashion.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Material treatments applied to all scattering objects in the usual fashion of the SBR+ Solution Type (including via imported 3D Components).
- PTD and UTD options of SBR+ are available with the Range-Doppler Setup Type.
- The Creeping Wave option of SBR+ are not available with the Range-Doppler Setup Type
- When examining range-doppler problems, the SBR+ solver produces its raw data in a proprietary file format with the extension ".frtm". You have the option to export these files for local solves through **Export on Completion** option in the Design Settings and for DSO parametric solves by specifying a batch option through the Job Management interface for Large Scale DSO. The relevant batch option for a DSO parametric solve for a range doppler problem is FRTM, as follows:



Using SBR+ Solution Type for HFSS SBR+ Range Doppler Designs

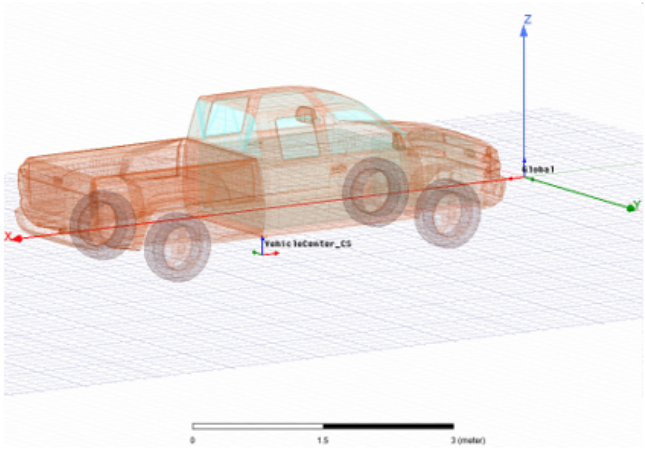
1. Moving Objects: Create or obtain 3D Components for any moving objects you intend to include in the scene as scattering geometry. These should be created within an HFSS Design set to the SBR+ Solution Type.

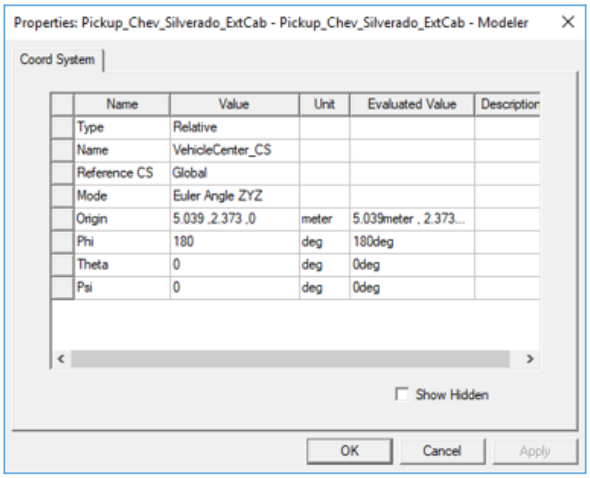
   Stationary Objects: These can be 3D Components, but they can also be defined directly in the HFSS range-Doppler design if preferred.

   The 3D Component should include material definitions and boundary assignments for all surfaces and bodies that are consistent with an SBR+ simulation.

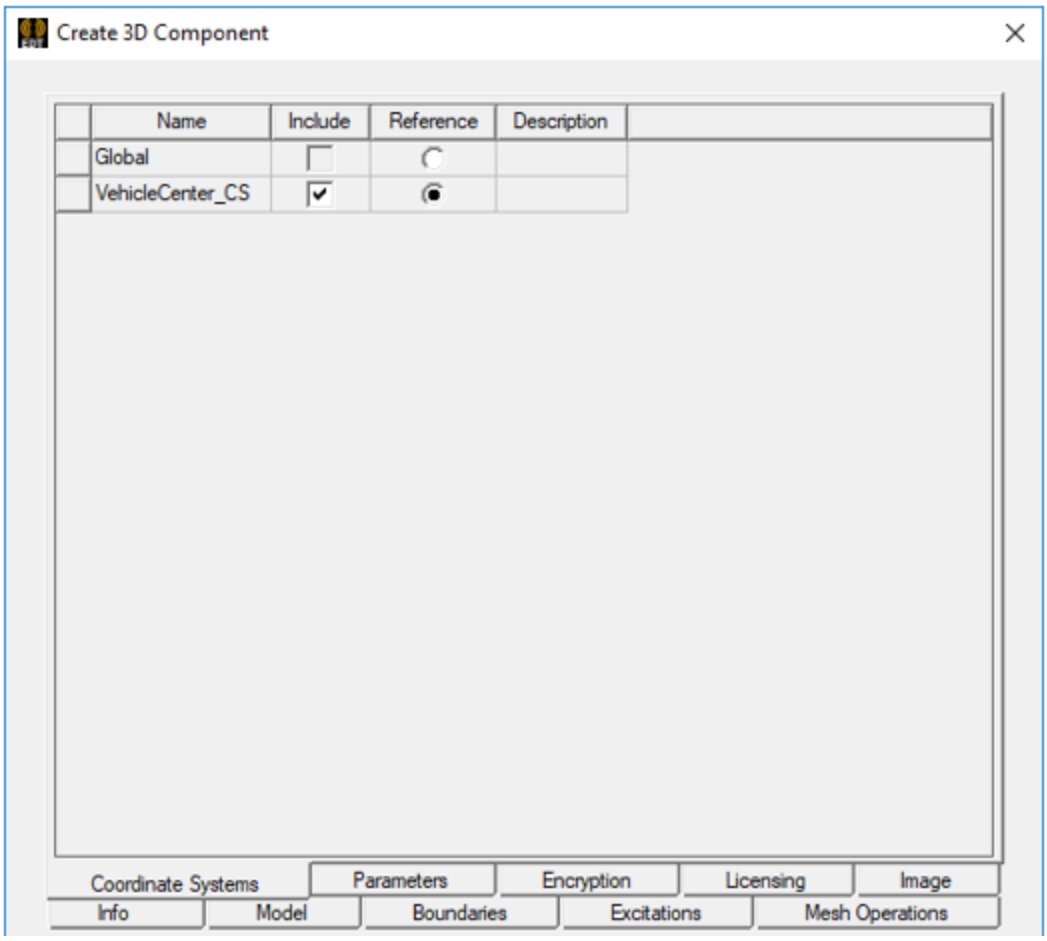
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Choose a reference coordinate system that is convenient for the placement of the object within the larger scene. For example, with vehicles to be placed on the ground, it is convenient to define a local coordinate system that is laterally centered within the vehicle, with the z=0 plane at the bottom of the vehicle wheels, and oriented for forward travel in the +x direction. Be sure to include this local coordinate system definition and also select it as the reference coordinate system when the 3D Component is created.





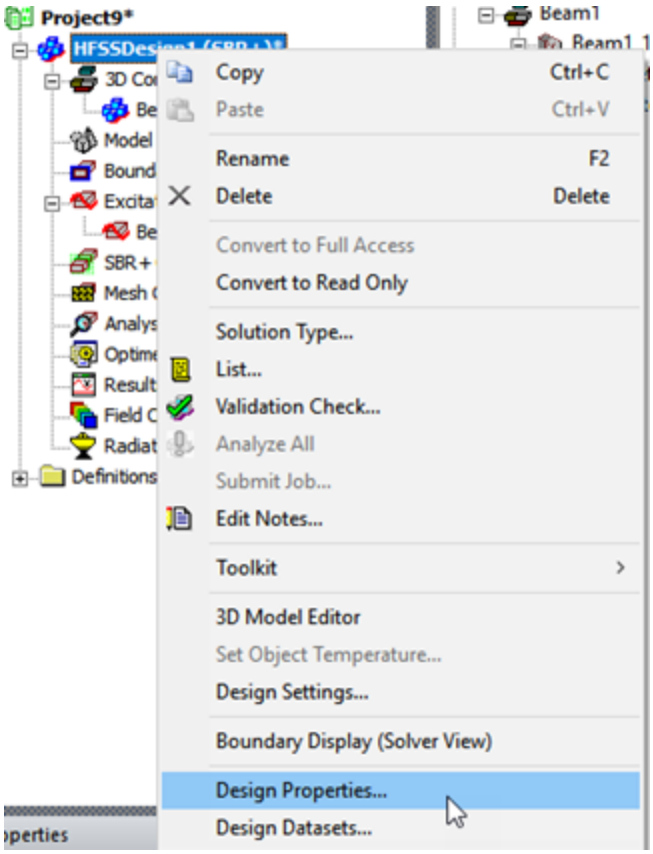
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



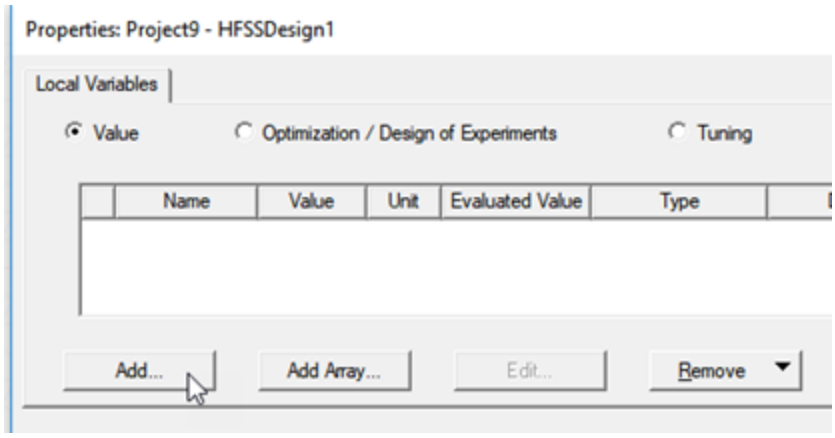
2. Create a new HFSS Design for the radar scenario and set its solution type to SBR+.
3. Create a time variable that will drive the motion of all objects in the scene, including the radar antennas.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Project** tree, right-click on the design node and select **Design Properties…**



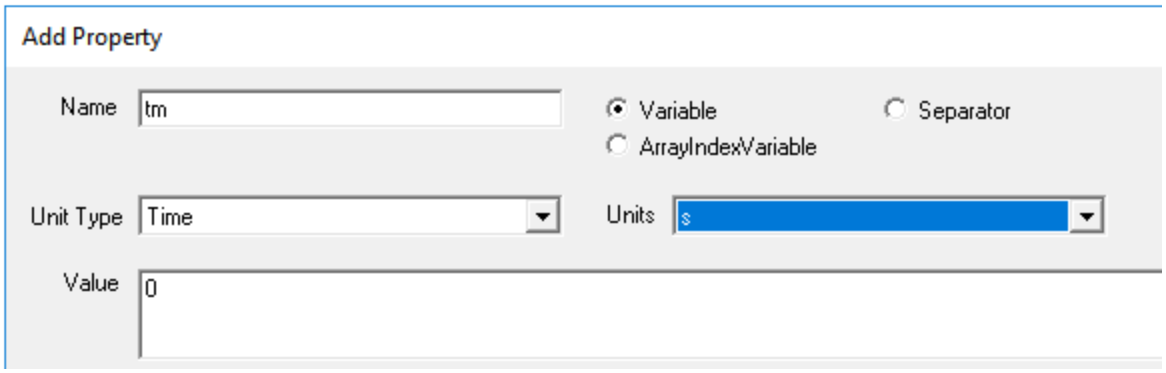
In the **Properties** dialog that appears, click **Add…**



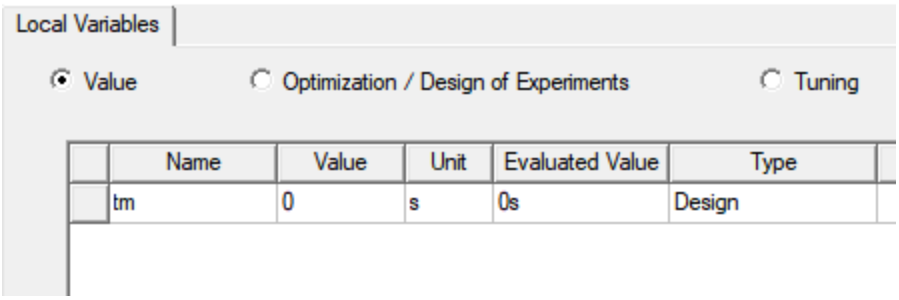
In the **Add Property** dialog that appears, set the Name to “tm” (or whatever you prefer to call the time variable) , select the property type as Variable, select the Unit Type as Time, and select a time unit (e.g., “s” for seconds).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Leave its Value as 0. You can later change it to advance the scenario or base its behavior on another variable through an expression in the variable definition.



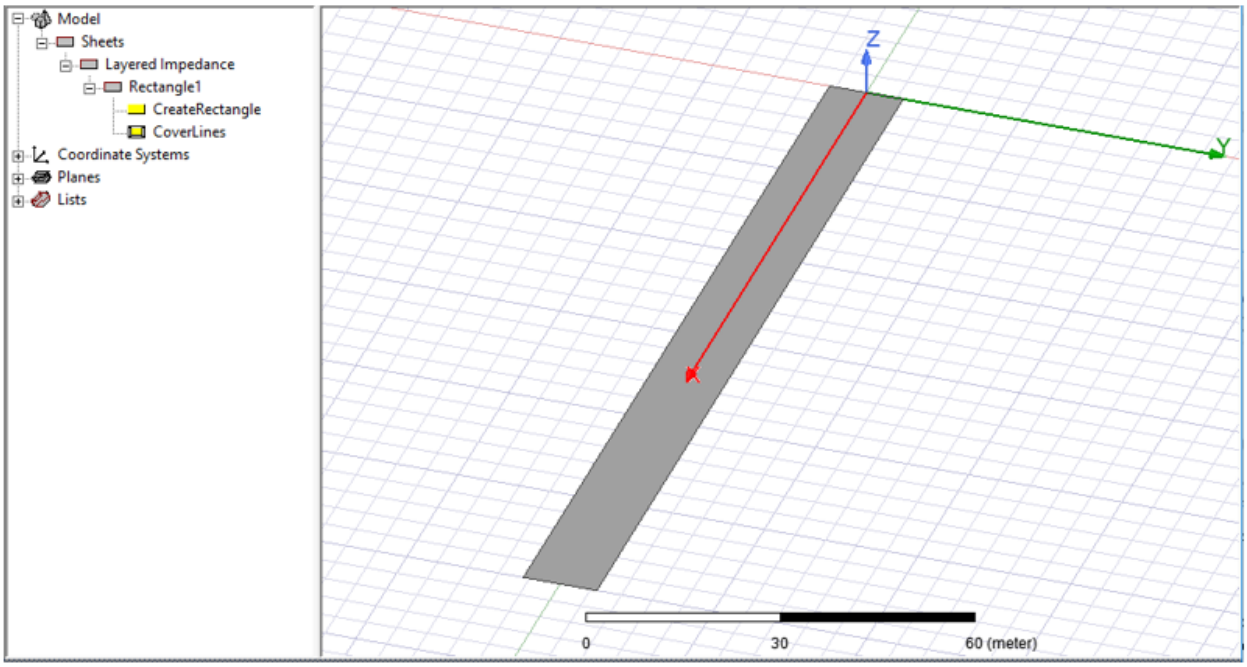
OK the dialog box. The new time variable is listed in the Design Properties.



4. Optional: Create or import geometry for stationary objects in the scene and assign boundaries to their surfaces.

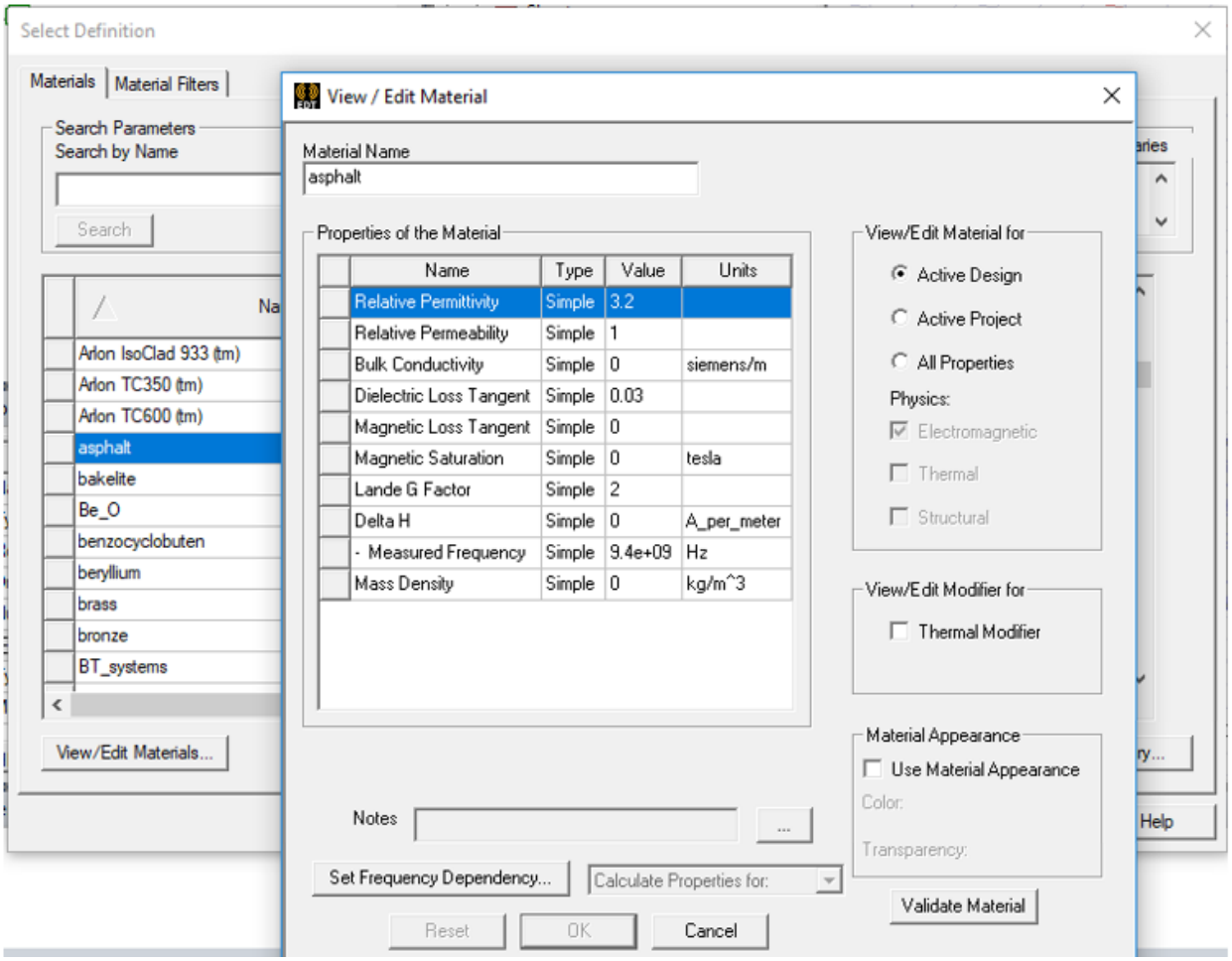
   In this example, we've created a 150 m x 12 m rectangle to represent a road surface.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



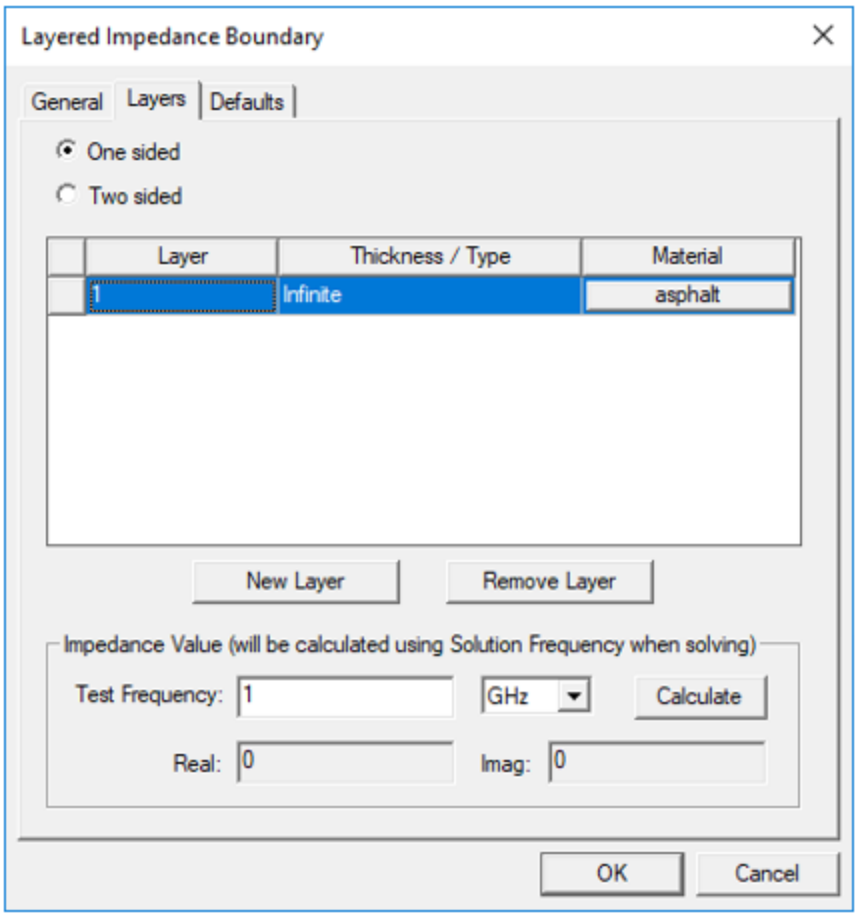
The material, “asphalt” has been added to the materials library, defined with a dielectric constant $\varepsilon_r = 3.2$ and loss tangent $\tan\delta = 0.03$.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The assigned boundary is Layered Impedance: one-sided, one layer of “asphalt” with Infinite thickness. Because it is one-sided, it behaves as an asphalt half-space; SBR rays incident upon it will generate attenuated reflection rays but no transmission rays, with some energy lost to the half-space region.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Import 3D Components for moving scattering geometry and configure their time-dependent motion.

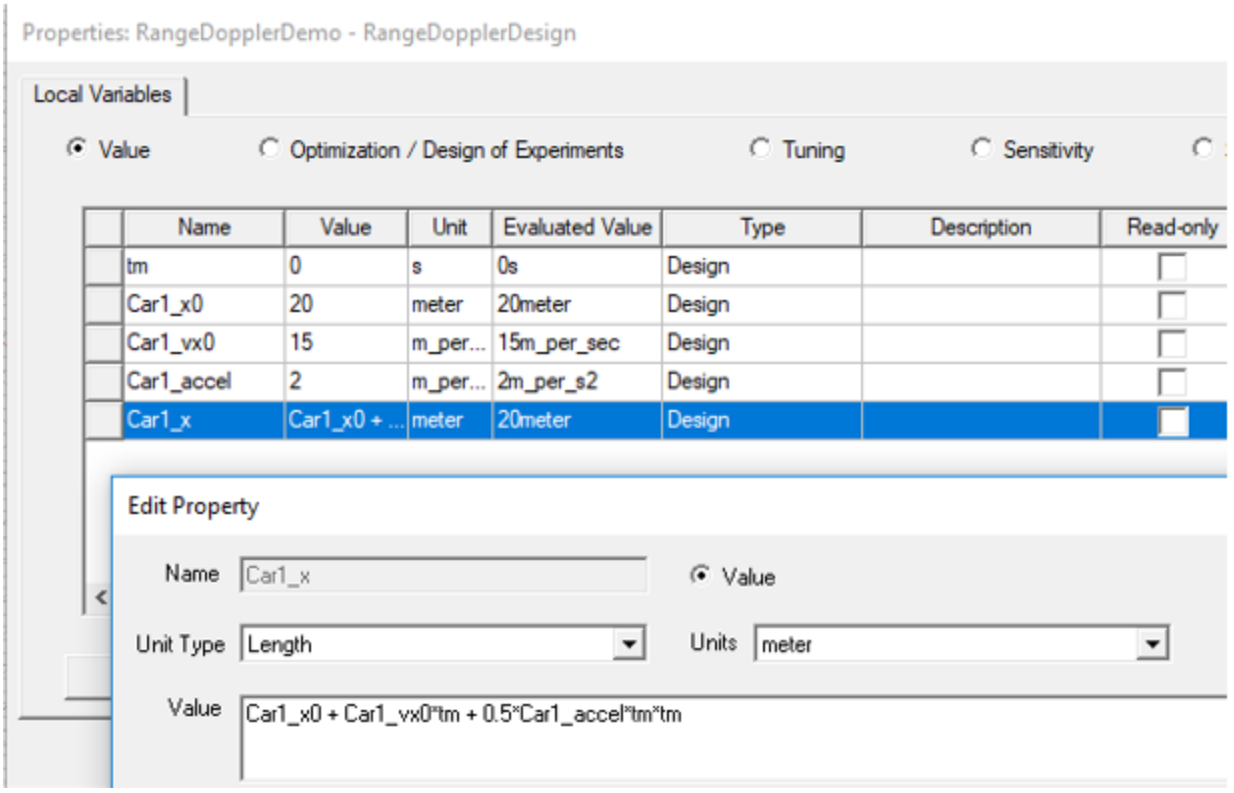
   Optional: Create variables that define motion behavior. This will simplify the subsequent definition of coordinate systems. Here, for a vehicle, we specify:

- initial position: (Car1_x0, Unit Type = Length, Unit = meter),
- initial velocity: (Car1_vx0, Unit Type = Speed, Unit = m_per_sec),
- constant acceleration: (Car1_accel, Unit Type = Acceleration, Unit = m_per_s2).

These variables are then used in an expression for a fourth Car variable that depends on the time variable (tm) for the x-position (Car1_x, Unit Type = Length, Unit = meter) of a coordinate system that will carry the 3D Component for a car in the scene:

$$x(t) = x_0 + v_x\, t + a_x\, t^2/2$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

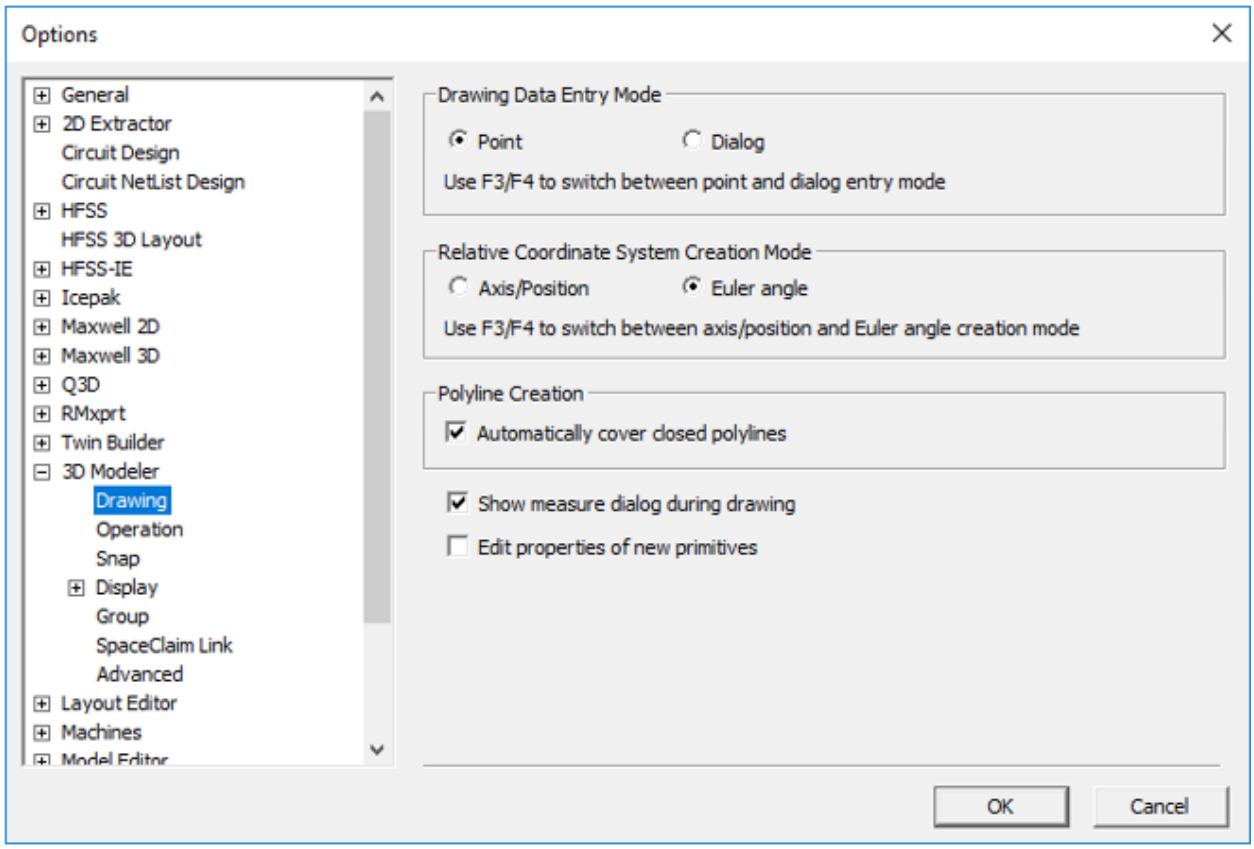


Verify correct entry of the expression for Car1_x by adjusting the input Value for the tm variable in the properties table and observing the Evaluated Value of Car1_x.

You then create a target coordinate system for the 3D Component: Car1_CS.

To follow this example, set the Relative Coordinate System Creation Mode to “Euler angle” in user preferences: click **Tools >Options>General Options…**, and then expand the 3D Modeler options group and select the Drawing options sub-group.

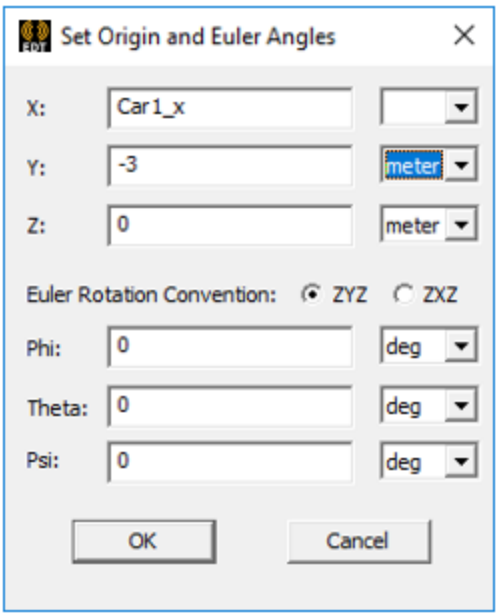
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



From the **Toolbar** menu, click **Modeler>Coordinate System>Create >Relative CS>Both**.

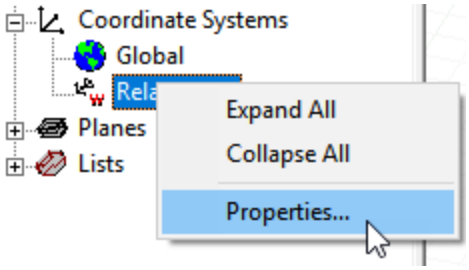
In the **Set Origin and Euler Angles** dialog box, set the X value to the Car1_x variable that you defined earlier and set the Y value to -3 m.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



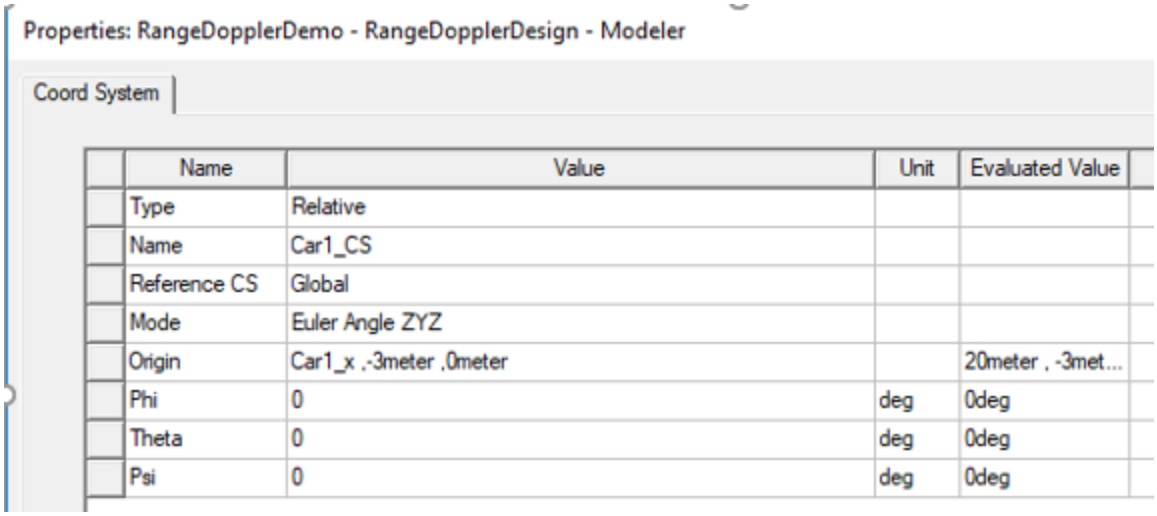
The created coordinate system is given a default name like RelativeCS1.

To access its properties and rename it, expand Coordinate Systems node in the **History** tree, right click on the newly created coordinate systems, and select **Properties…**

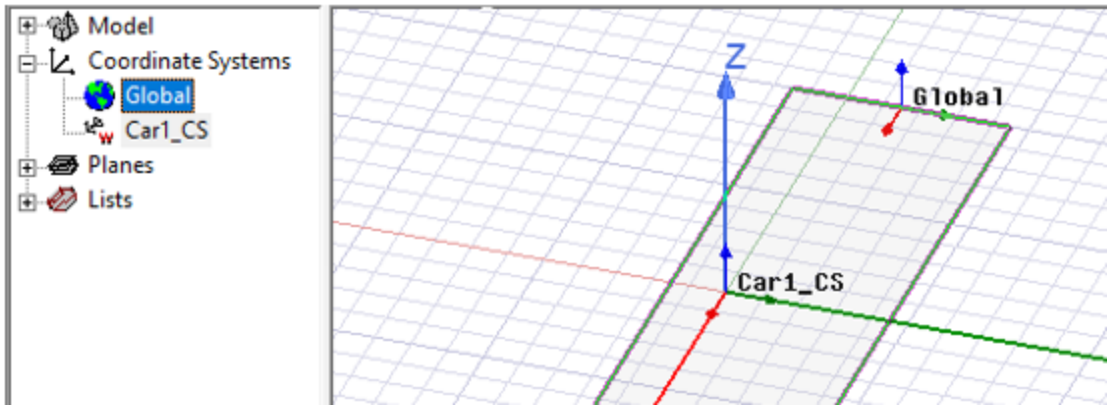


Rename it to Car1_CS in the Name field.

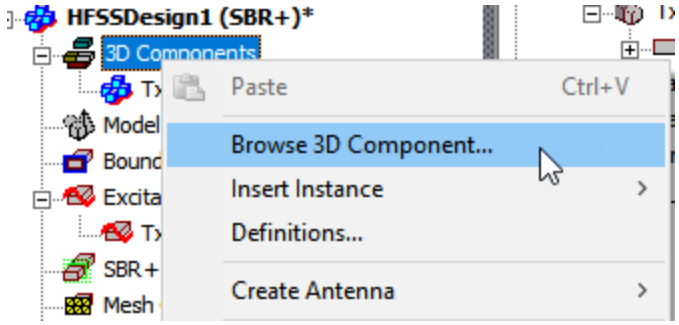
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



OK the dialog to show RelativeCS1 renamed as Car1_CS.



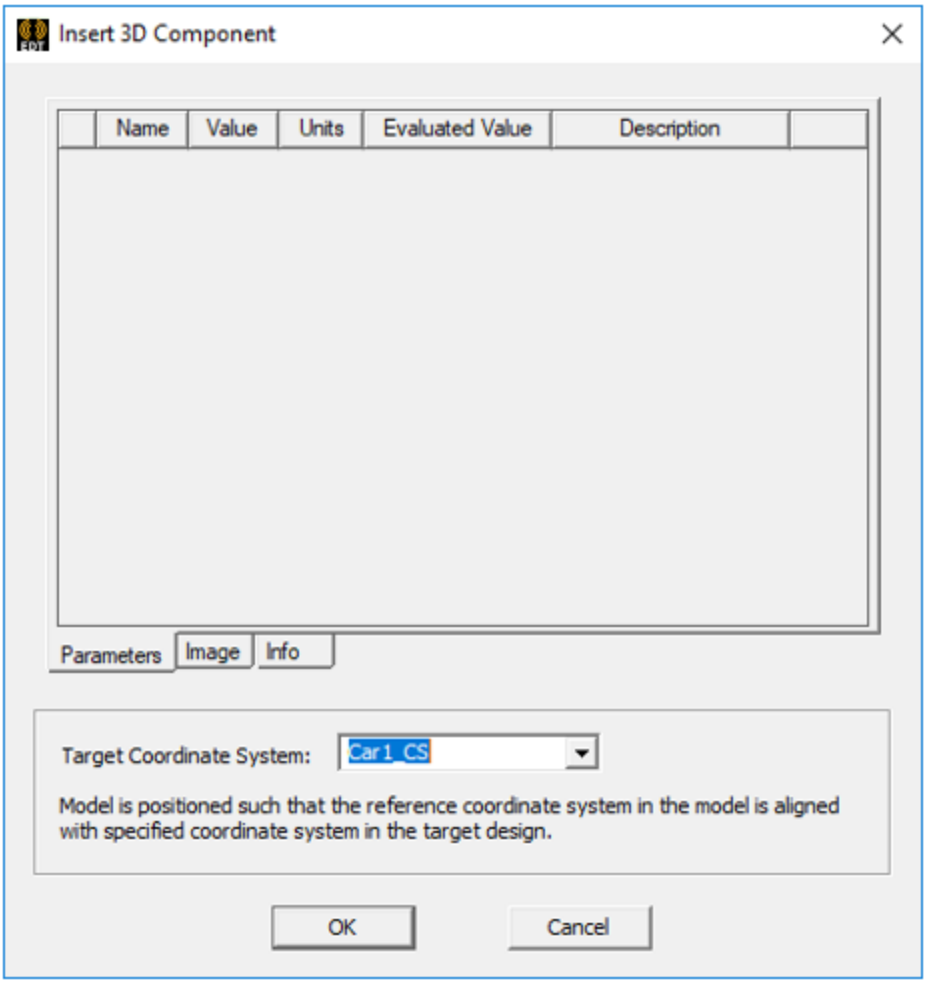
To import a 3D component, right-click the **3D Components** icon in the **Project** tree of the range-Doppler design in the project tree and select **Browse 3D Components…**



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

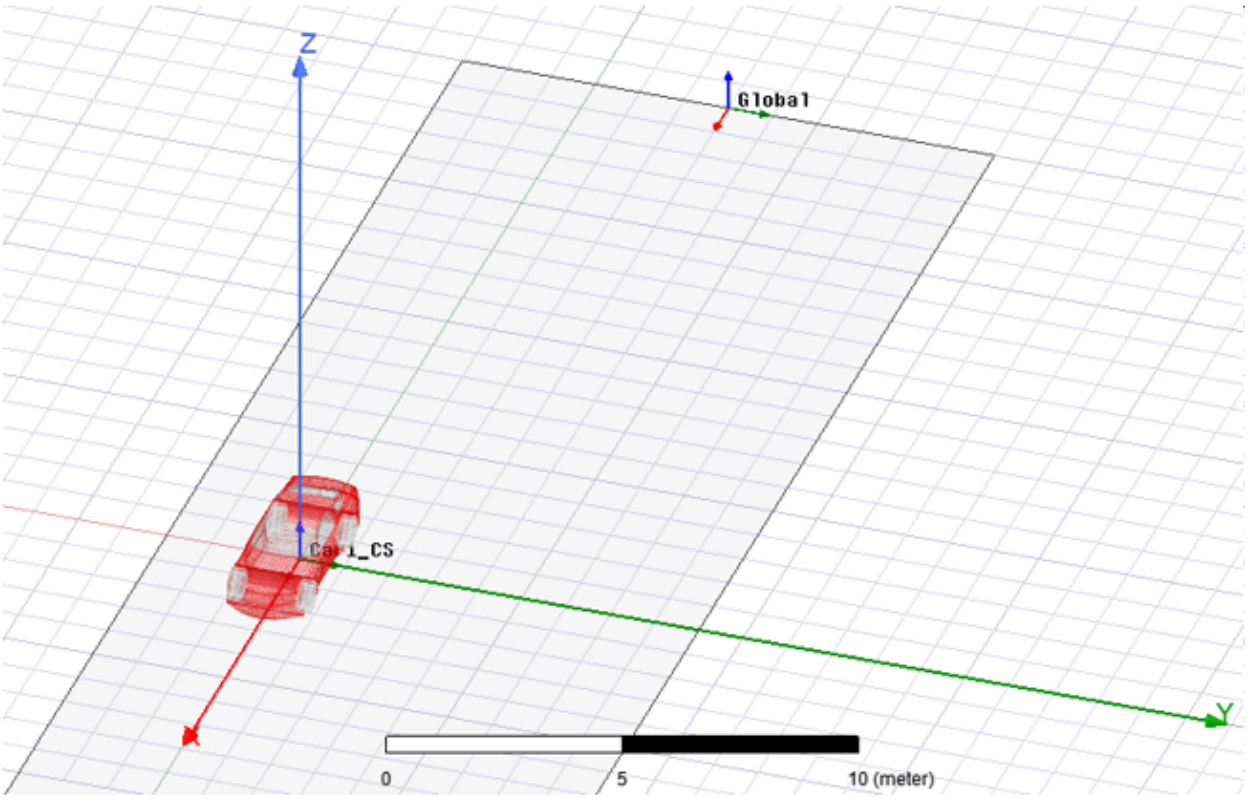
Use the resulting file browser to select the 3D Component file (*.a3dcomp) of an object to include in the scene and click **Open**.

In the **Insert 3D Component** dialog that appears, select Car1_CS as the target coordinate system. This will cause the reference coordinate system of the 3D Component (see Step 1) to be fixed to and move with Car1_CS.



Click OK to insert your model at the Car1_CS. If your adjust the time variable, the imported component moves with Car1_CS.

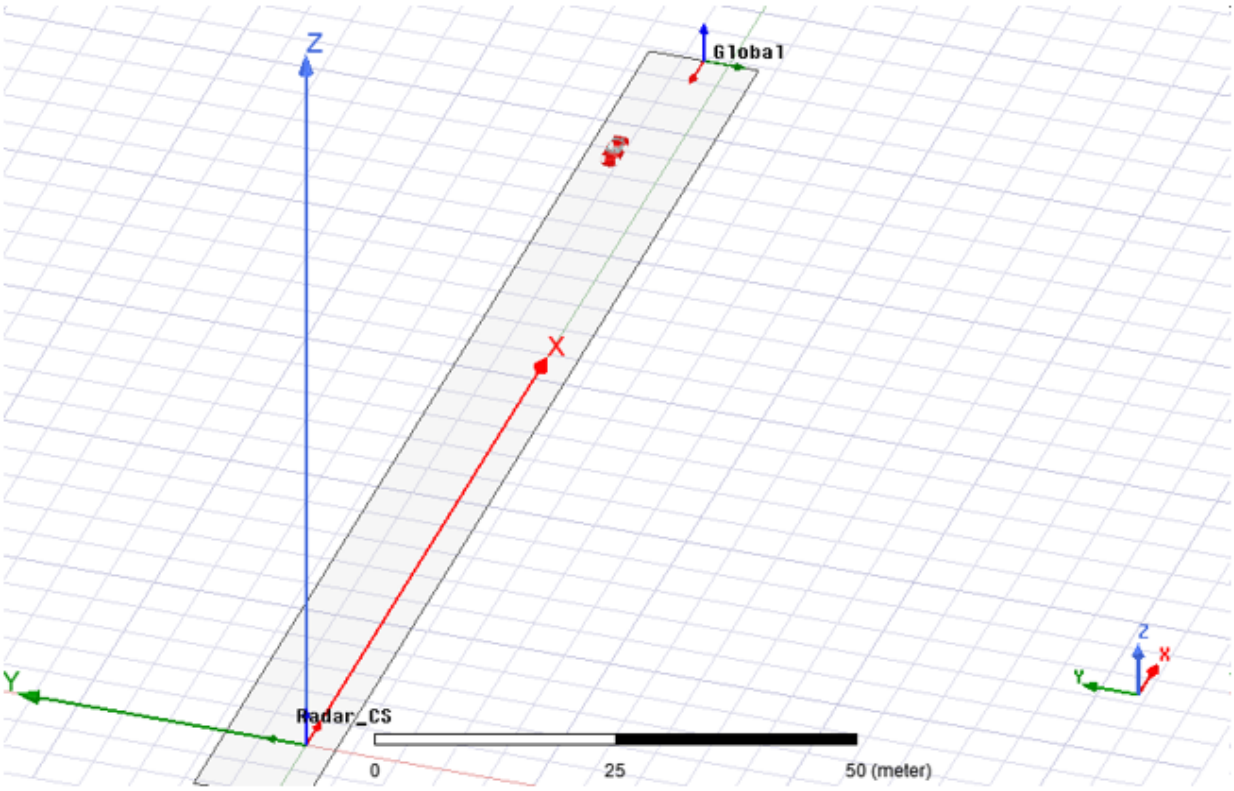
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. Create and configure radar antennas.

   Create a moving coordinate system for the radar antennas.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Below, we have defined Radar_CS relative to Global coordinates.

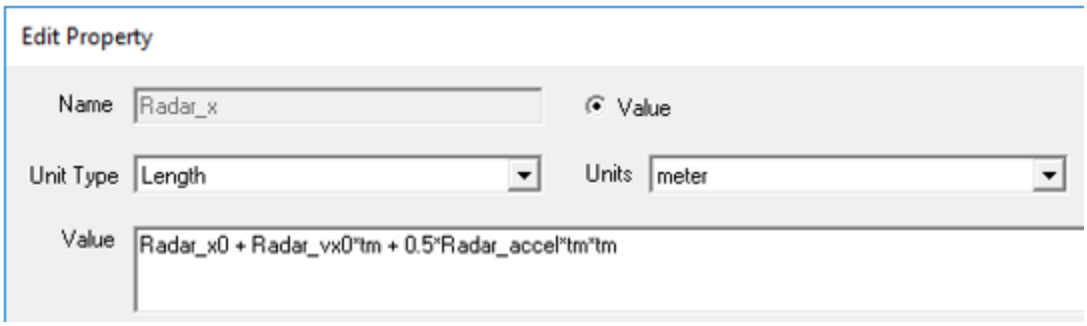
Properties: RangeDopplerDemo - RangeDopplerDesign - Modeler

Coord System

Name	Value	Unit	Evaluated Value
Type	Relative		
Name	Radar_CS		
Reference CS	Global		
Mode	Euler Angle ZYZ		
Origin	Radar_x ,3meter ,0.5meter		140meter , 3me...
Phi	180	deg	180deg
Theta	0	deg	0deg
Psi	0	deg	0deg

Its X component is set to a variable, Radar_x, which itself is defined in terms of other variables. Radar_x0 = 140 m, and Radar_vx0 = -15 m/s, Radar_accel = 0, and the time variable tm.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



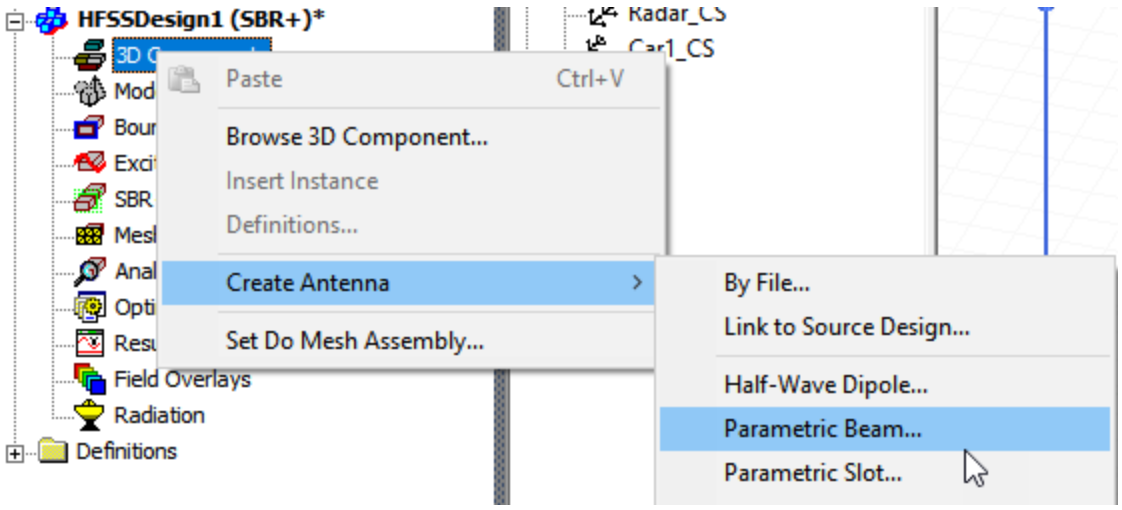
Its Y and Z components are set to 3 m and 0.5 m, respectively, the latter to elevate the radar antennas above the road.

By setting Phi = 180^{o}, we've oriented its X-axis to point in the -X direction of global coordinates.

Make the radar coordinate system (Radar_CS, in this example) the working coordinate system by selecting it in **History** tree. The working coordinate system becomes the target coordinate system of the radar antenna created in the next step.

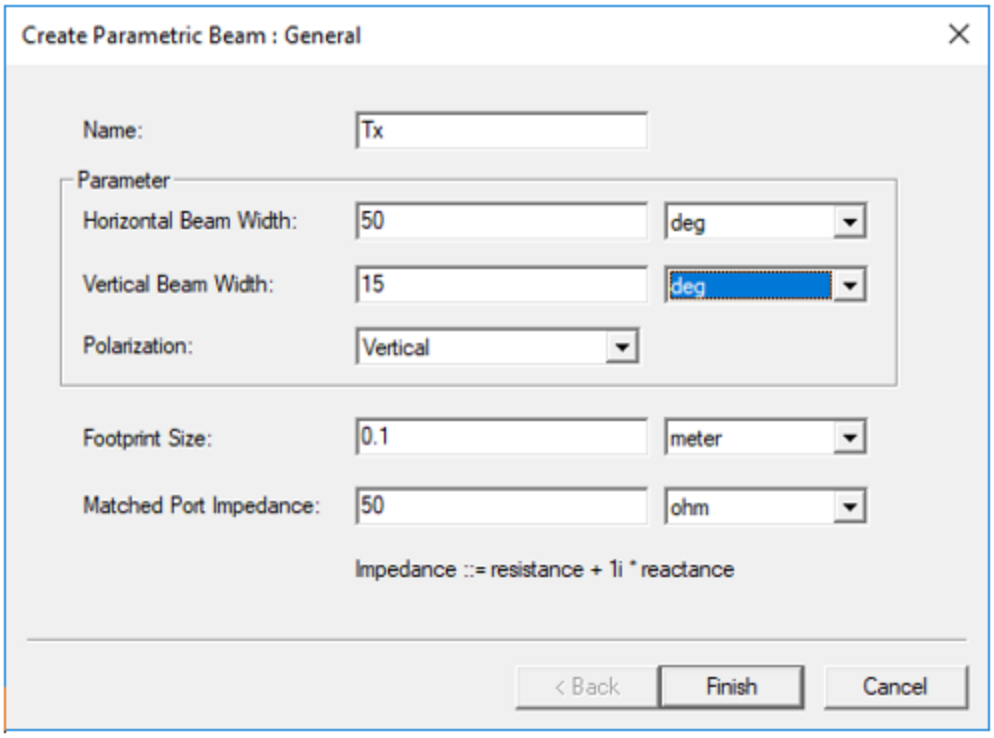
Create the radar transmitting (Tx). A good choice for radar antennas is the built-in parametric beam, which example we will follow here. Alternatively, you can setup a link to a source design or import far-field gain pattern (.ffd) file. In the case of a one-way link from a source design, the Range-Doppler Setup Type only supports far-field links.

To create a parametric beam antenna, right-click on the 3D Components icon in the Project tree for the range-Doppler design and select **Create Antenna>Parametric Beam…**



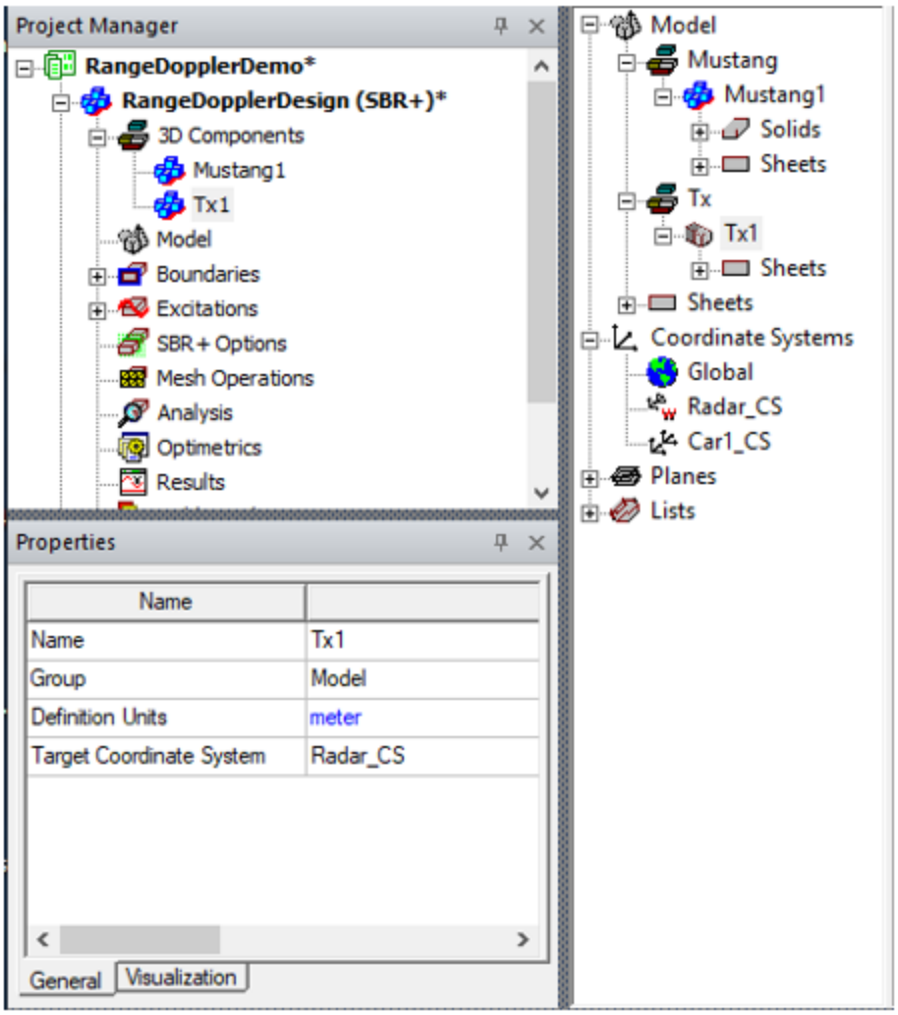
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Create Parametric Beam** dialog that appears, configure the beam widths and polarization. In this example, we have renamed the antenna to Tx. Then click **Finish** and the SBR+ source is created.



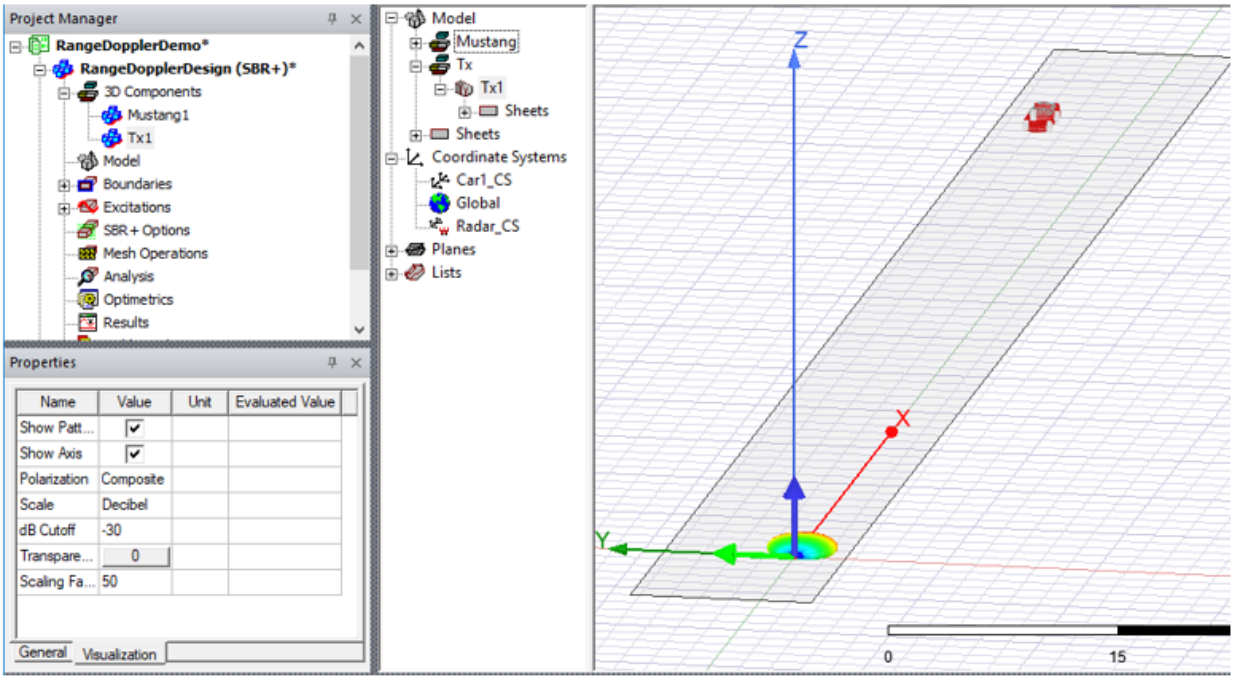
The following dialog shows the **Project** tree and **History** tree for the RangeDoppler demo with the 3D Components for car and antenna, and the coordinate systems.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To increase the display size of the beam pattern to better match the scale of the scene, select the antenna's object instance in the **Project** tree or **History** tree (Tx1 in this example), select the **Visualization** tab in the **Properties** window, and then adjust the **Scaling Factor** property as needed. Here, we have set **Scaling Factor** to 50.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

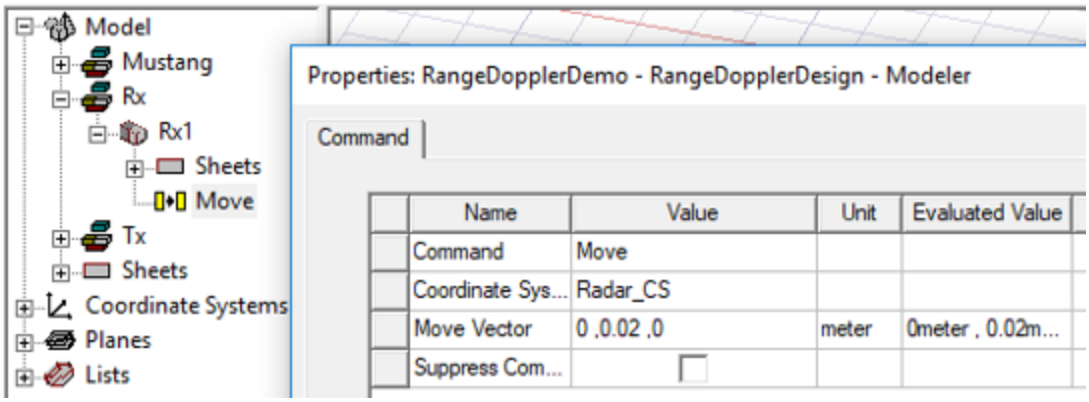


Create at least one radar receiving (Rx) antenna. In this example, we configure the Rx antenna as a vertically polarized parametric beam, but this time with a horizontal beam-width of 70^{o} instead of 50^{o} and naming it “Rx”. So that it moves with Tx1 antenna, we set its target coordinate system to also be Radar_CS. A move operation is then applied to the antenna instance (Rx1) to shift its position horizontally 2 cm relative to Tx1 (and the origin of Radar_CS). For a radar operating at 76.5 GHz (next step), this is a typical displacement between Tx and Rx antennas.

To effect the move operation to separate the Tx and Rx antennas:

- Select the Rx1 instance under the Rx definition node in the **History** tree.
- Select the **Draw** ribbon at the top of the main window.
- From the Operations group in the **Draw** ribbon, select Move.
- Press function key F4 to use dialog entry mode.
- Enter the Move Vector as 0, 0.02, 0 m. This will shift Rx1 by 2 cm along the Y axis of Radar_CS.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



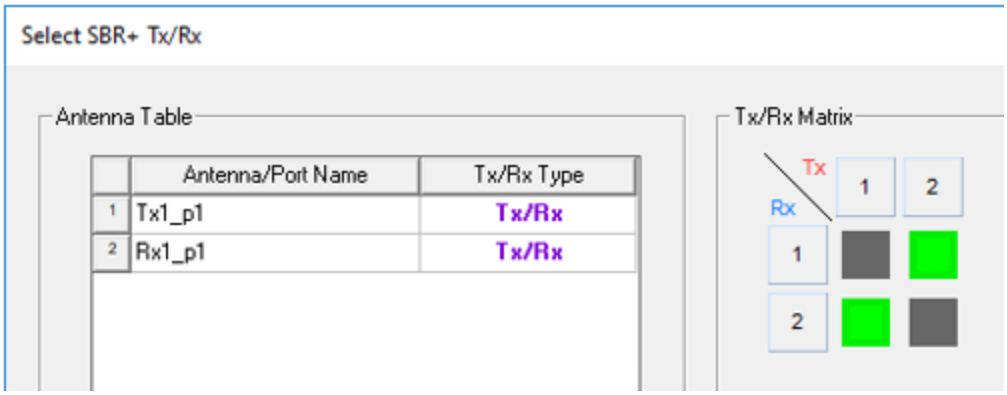
7. Configure Tx and Rx roles of antennas and port coupling pairs.

   After all radar antennas and ports have been defined, configure which ports attach to transmitting (Tx) antennas and which to receiving (Rx) antennas. By default, the SBR+ solution type will compute the S-parameter coupling in both directions for each port pair. Unless all of these results are actually wanted, this causes wasted computational effort. Instead, use the **Select SBR+ Tx/Rx** dialog to select the particular coupling paths to be simulated. In the above example, where we only have two antennas, Tx1 and Rx1, and assuming we only want to transmit on Tx1 and receive on Rx1, meaning SBR rays will be launched only from Tx1 and their scattered field only observed on Rx1, the setup steps are as follows:

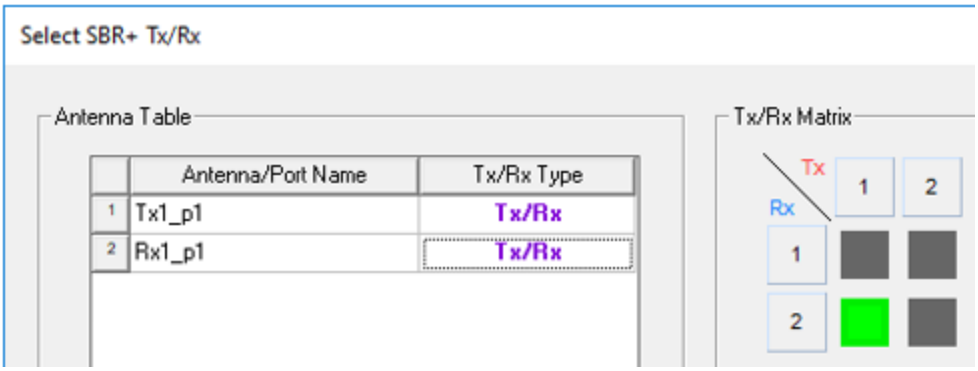
   In the range-Doppler design, expand the Excitations node and notice it has two ports: Tx1_p1 and Rx1_p1.

   Right-click on the Excitations node and choose the menu item, Select Tx/Rx… .The Select SBR+ Tx/Rx dialog appears. The Antenna Table on the left side of the dialog identifies the port names and assigns them an integer port Index: Tx1_p1 = Port 1, Rx1_p1 = Port 2. It also lists the potential role (Tx/Rx Type) of each port: Tx/Rx, Tx, or Rx. By default, each port has a Tx/Rx role, which means it can *potentially* act as both a transmitter and receiver. The Tx/Rx Matrix on the right side of the same dialog identifies which port coupling combinations are actually active. Tx-capable ports (that is, those with a role of Tx or Tx/Rx) are listed by index left-to-right across the top of the matrix. Rx-capable ports are listed by index top-to-bottom along the left side of the matrix. Active coupling paths are indicated by green squares in the matrix. By the default, all port pair combinations S_{ij} that are allowed by the assigned port roles are active, with the exception of self-coupling (that is, S_{ii}). In this example, we see that both S_{12} and S_{21} will be computed.

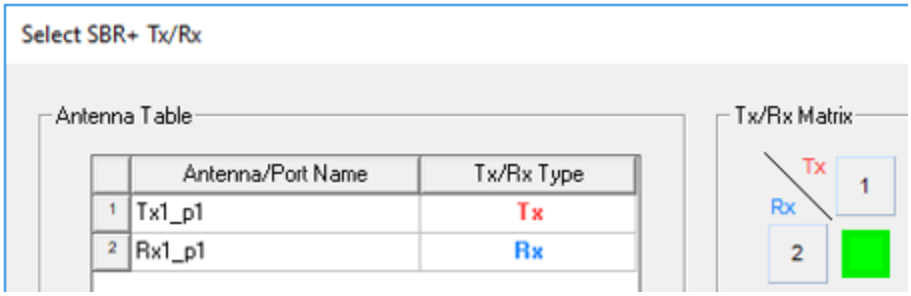
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To switch off S_{12}, which means transmitting on port Rx1_p1 and receiving on port Tx1_p1, click on the corresponding (upper-right) square to turn it grey. This leaves only S_{21} active.



Alternatively, when there are many antennas and ports, and the matrix would grow large, it can be convenient to limit the roles of some ports. In this case, we can set Tx1_p1 to the role of Tx and Rx1_p1 to the role of Rx. This reduces the matrix to a 1 x 1, with a single green cell. Reducing port roles when they are known to be exclusively Tx or Rx for a particular system is especially helpful for MIMO radars with a large number of ports.

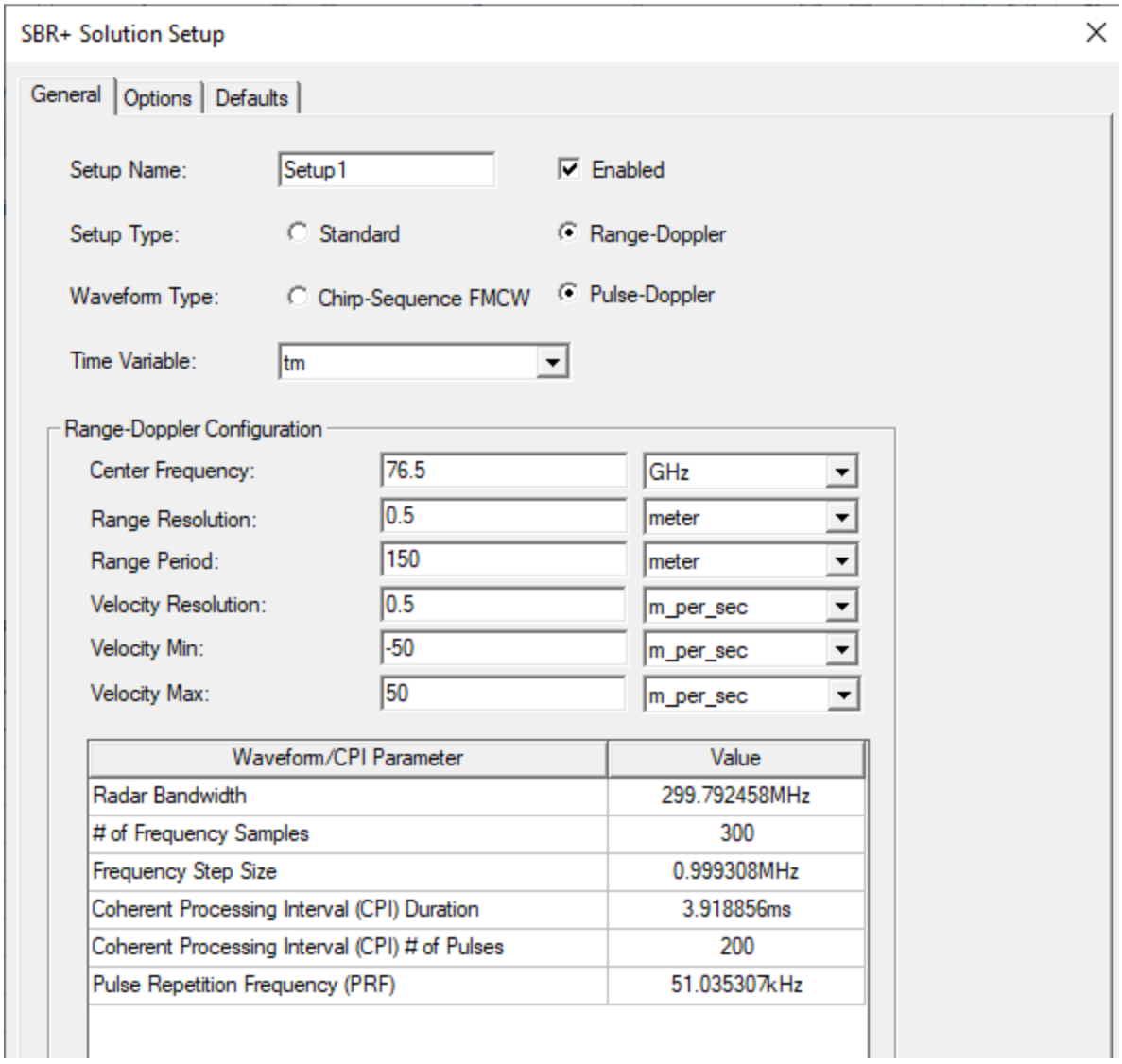


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Configure the Range-Doppler Setup Type.

Right-click on the **Analysis** icon in the Project tree and select **Add Solution Setup…**

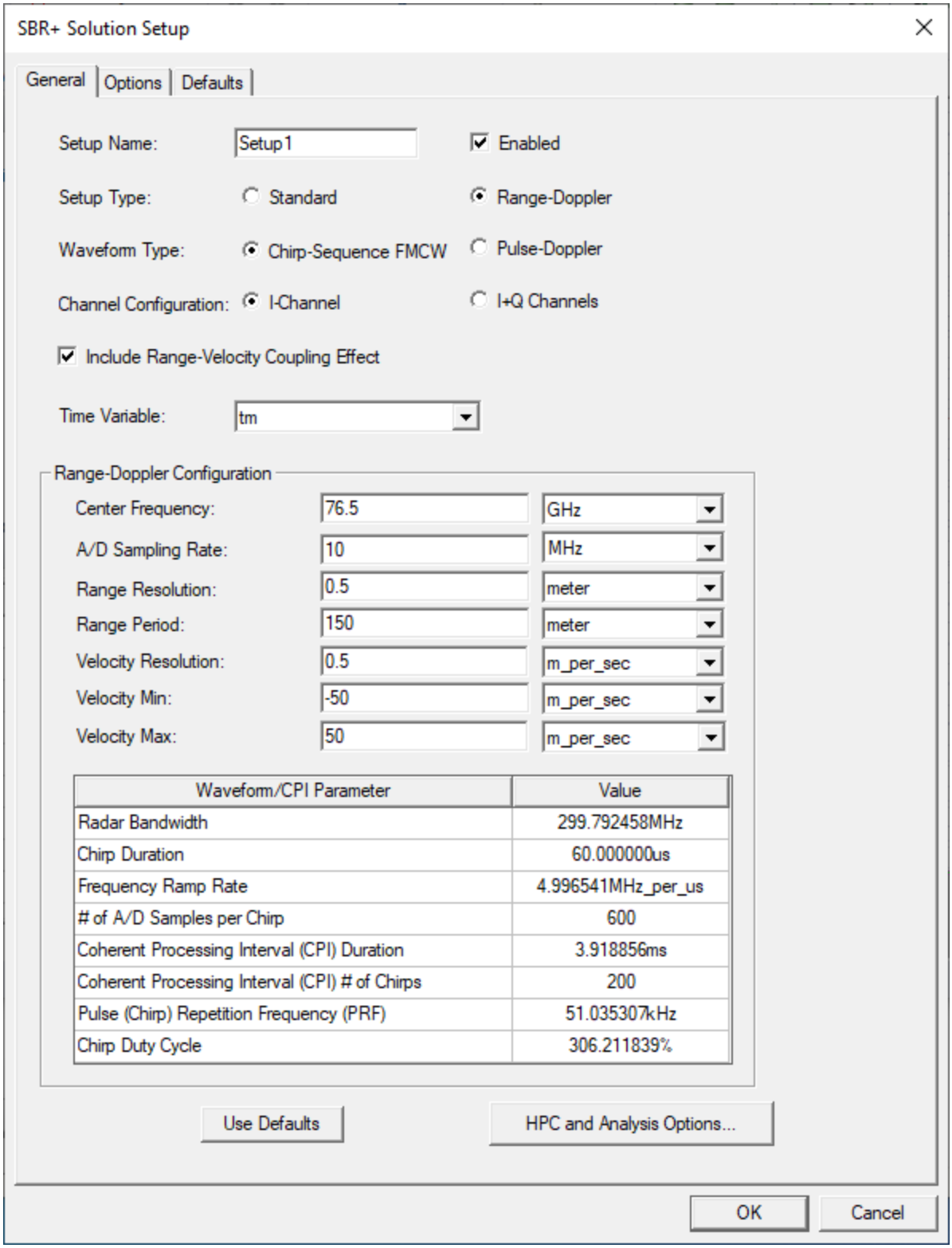
In the **SBR+ Solution Setup** dialog that appears, set Setup Type to Range-Doppler. This lets you select a Waveform Type of either Chirp-Sequence FMCW (default) or Pulse-Doppler. The simulation results (range-Doppler images) for Pulse-Doppler do not include the range-velocity coupling effect inherent in chirp-sequence FMCW (CS-FMCW). This effect is a shift in the perceived range of scattering features that is proportional to their radial (Doppler) velocity. The following figures show the Pulse-Doppler selection and configuration.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting Chirp-Sequence FMCW offers additional options and settings: Channel Configuration (I-Channel or I+Q Channels), A/D Sampling Rate, and whether the waveform model should Include Range-Velocity Coupling Effect. See Chirp Sequence FMCW for SBR+ Range Doppler Solution Setup for further detail on configuring this waveform option and its range-velocity coupling effect.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select the **Time Variable**. In this example, it is tm. Setting the Time Variable cannot be deferred.

Configure the Center Frequency and Range-Doppler performance specifications of the radar. This configuration will be applied to all port combinations selected in Step 7. When setting the Range-Doppler performance properties, there are several things to consider:

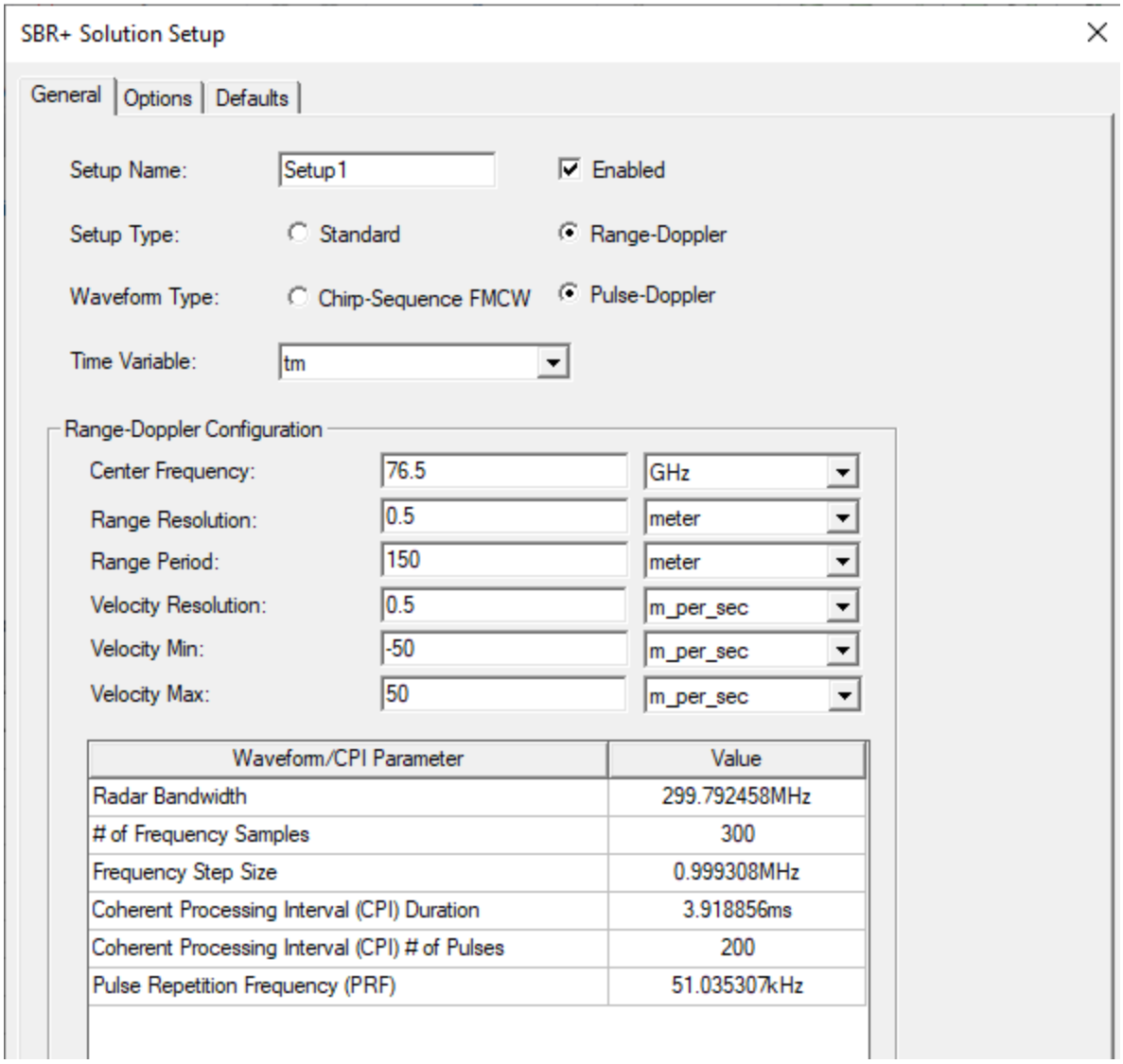
- Through the combination Center Frequency, Range Resolution, and Range Period, you are indirectly configuring a uniform frequency sweep. The Center Frequency (f_c) needs to be high enough to accommodate the Radar Bandwidth (*BW*) implied by the Range Resolution (Rr), which is obtained by the formula $BW = c / (2 R_r)$, where c is the speed of light. The setup requires $f_c > BW/2$. For example, with a Range Resolution = 1 m, the frequency bandwidth is **BW** ~= 0.15 GHz. This means the Center Frequency needs to be 75 MHz or higher. Note that at the default frequency of 76.5 GHz, a Range Resolution of 1 m is easily accommodated. The number of frequency samples (N_f) in the implied sweep is then obtained as Range Period (P_r) divided by Range Resolution: $N_f = P_r/R_r$.
- The range response is periodic and, thus, ambiguous. If the Range Period is set to 100 m and a feature is observed at 40 m, this could mean the scattering feature is physically located 40 m from the radar, but in theory it could be 140m, 240 m, etc. from the radar. In radar data sheets, the range period is sometimes given as maximum range or range ambiguity.
- The Doppler velocity response is also periodic (and ambiguous). The Doppler velocity period P_v is determined from the dialog inputs as P_v = Velocity Max – Velocity Min. Suppose Velocity Min = -50 m/s and Velocity Max = 70 m/s. This means the reported Doppler velocity period is Pv = 120 m/s, centered at 10 m/s. Suppose a scattering feature is observed to have a Doppler velocity of 50 m/s. In theory, this could mean its actual Doppler velocity relative to the radar is -70 m/s, +50 m/s, +170 m/s, or any other value of 50 m/s + n 120 m/s, where n is an integer. In radar data sheets, the Doppler velocity period is often expressed through a minimum and maximum velocity, but it can also appear directly as the velocity ambiguity or period. *With range-Doppler radars, the ambiguous (periodic) quality of range and velocity is an unavoidable design limitation.*
- There is a tradeoff between range and velocity ambiguity. In general, the higher the range ambiguity (period), the lower the maximum theoretically achievable Doppler velocity ambiguity (period), and vice versa. The input dialog for the Range-Doppler setup will not permit a violation of this theoretical limitation, as it would then result in a meaningless range-Doppler performance specification that is not physically realizable in any radar. The constraint is given by the following inequality:

$$P_r P_v < \frac{c^2}{4 f_c}$$

In this example, we select the following range-Doppler settings:

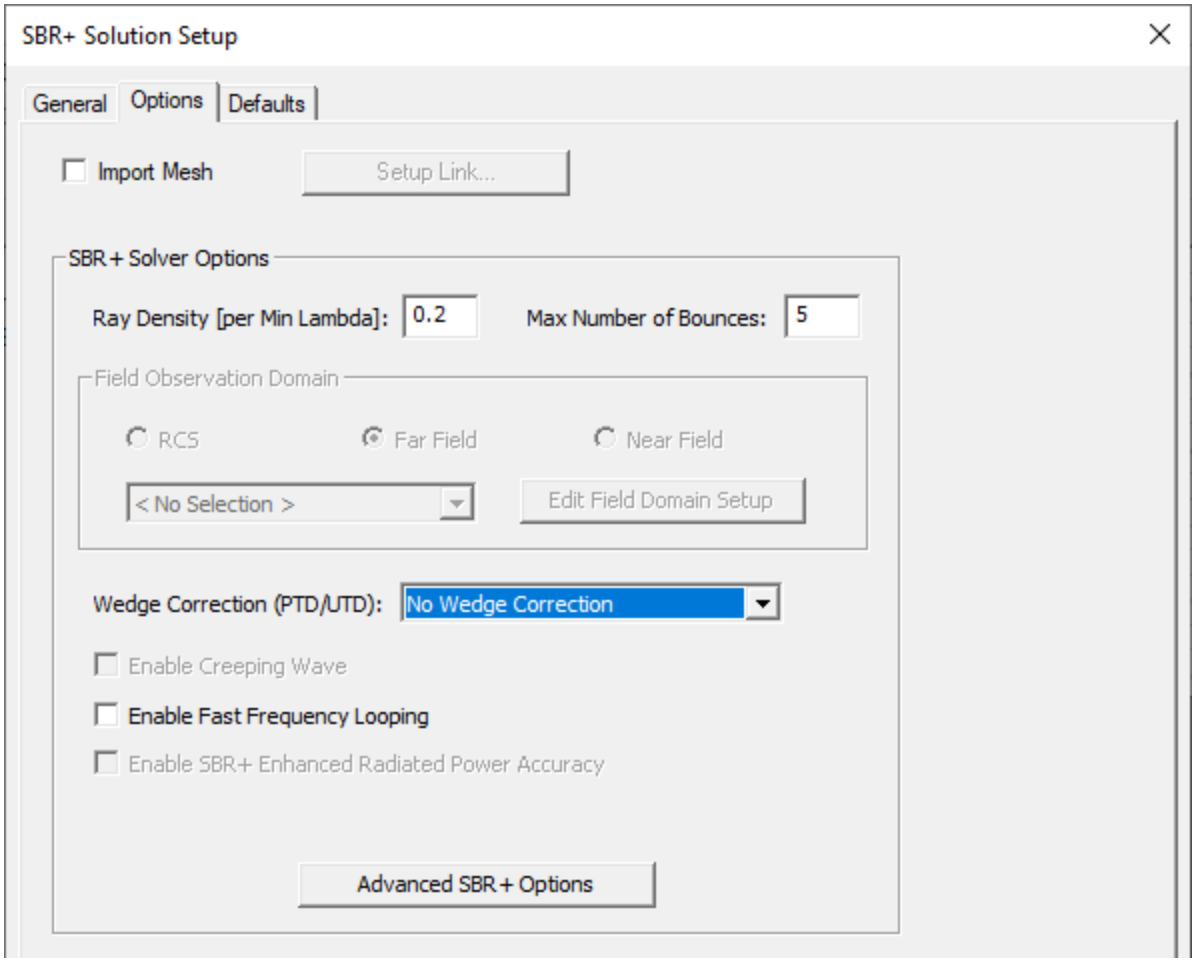
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Center Frequency = 76.5 GHz
- Range Resolution = 0.5 m
- Range Period = 150 m
- Velocity Resolution = 0.5 m/s
- Velocity Min, Max = -50, 50 m/s



Select the **Options** tab to set the SBR+ Solver Options: Ray Density and Maximum Number of Bounces.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

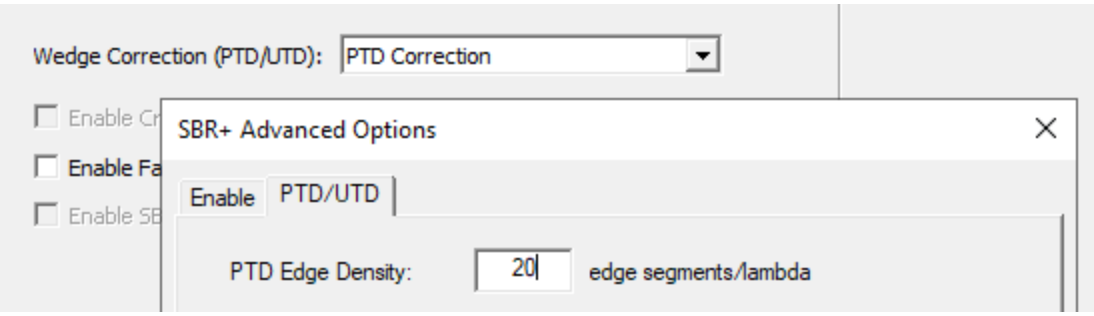


Note that the default Ray Density for the Range-Doppler Setup Type is 0.2 rays/wavelength whereas its default for the Standard Setup Type is 4.0 rays/wavelength. The lower default Ray Density for the range-Doppler setup reflects the fact that it is optimized for millimeter-wave radars operating in scenes spanning hundreds of meters (typically tens of thousands of wavelengths). For example, at the default center frequency of 76.5 GHz, the wavelength is 3.9 mm. A ray density of 0.2 rays/wavelength therefore corresponds to a first bounce ray-spacing of about 2 cm. Unlike matrix-based solutions, SBR has no inherent requirement for sub-wavelength sampling of the scattering geometry. Rather, it is sufficient if the rays adequately sample the geometric features of the scatterer irrespective of wavelength. For this example, we use the default SRB+ Solver Options: Ray Density = 0.2 rays/wavelength and Maximum Number of Bounces = 5.

Wedge Correction can be selected from the dropdown for PTD and UTD. When two different bodies (with different body IDs) with possibly different velocities join during a simulation to form a valid wedge, the two sides of the wedge are treated as

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

knife wedges. Wedges are extracted as expected for triangles belonging to the same body. For example, imagine two separate plates moving towards each other that at time *t* join to form a dihedral. The extracted wedges will be two distinct knife edges rather than one dihedral wedge. If you select **PTD Correction** or **PTD Correction + UTD Rays**, the **Advanced SBR+ Options** includes an option to **Enable** PTD/UTD (Wedge Diffraction) options. You can then specify PTD Edge Density as edge segments/lambda.



The **Enable Fast Frequency Looping** (FFL) option is available for range-Doppler setups. FFL will significantly accelerate a range-Doppler simulation in most cases, usually with little loss of fidelity. FFL accelerates high-density frequency sweeps with uniform frequency intervals. All range-Doppler setups imply uniform frequency sampling and typically also require a large number of frequency samples to achieve a **Range Period** spanning many **Range Resolution** intervals. FFL begins to show significant speed-up when the number of frequency samples exceeds 10, improving to 5x – 10x or greater when the number of samples exceeds 100. The number of frequency samples of your setup can be determined by consulting **# of Frequency Samples** (for **Pulse-Doppler**) or **# of A/D Samples per Chirp** (for **Chirp-Sequence FMCW**) in the table at the bottom of the **General** tab of the setup dialog box. Typical range-Doppler setups also have a relative bandwidth (Radar Bandwidth divided by Center Frequency) well under 10%, making them ideal for the accuracy of FFL acceleration. See SBR+ Fast Frequency Looping for High-Density Sweeps for details on the benefits and limitations of this acceleration feature.

9. Advanced Option for Geometrical Optics (GO) Blockage

SBR+ normally applies a physical optics (PO) blockage model where blockage effects emerge from partial cancellation of the incident field by the scattered field. PO blockage not only occurs in connection with the incident field from the Tx antennas, but also within the scattered field itself when reflected fields from the previous bounce are diminished by a blocking surface along the geometrical-optics (GO) reflection path at the current bounce by evaluating the scattering contribution of the incident ray field at that bounce.

In cases of significant blockage by a large obstruction, the PO blockage model requires accurate currents over the extent of the obstruction. Sometimes, this is hard to achieve in the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ray tracing approximation. For example, the obstruction may not be well illuminated by direct rays from the Tx antenna or multi-bounce rays after an earlier reflection. An alternative is to use a GO blockage model where scattered field contributions are added to an observation point subject to a line-of-sight (LOS) blockage check performed by the ray tracer. In some cases, this is more accurate than the default PO blockage formulation, while in others it is less accurate. GO blockage also entails a non-trivial computational cost, as the blockage check with the ray tracer must be performed for each observation angle (or point) for each ray hit point.

In addition to GO Blockage for SBR and UTD ray tracks, this feature is also available for PTD and CW.

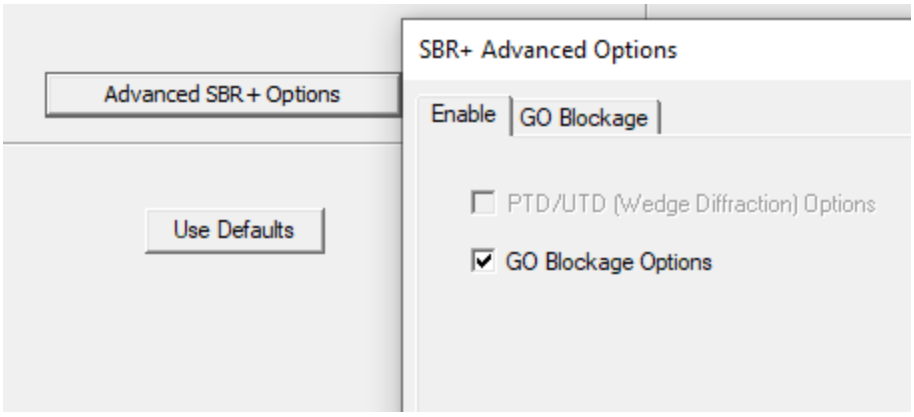
Limitations:

GO Blockage formulation is suited specifically for SBR+ ray tracks, including UTD ray tracks. While GO Blockage for CW and PTD has been implemented for the first time in this release, results can be surprising and/or inaccurate, as a limitation of the methodology.

Summary Workflow:

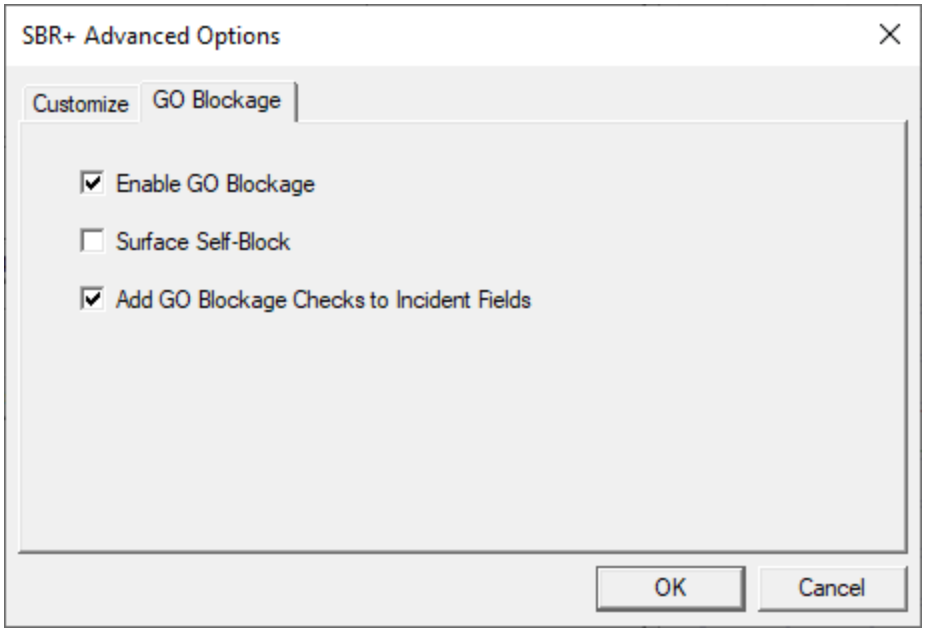
GO Blockage works with every SBR+ modality, in Hybrid and SBR+ solution type, with any type of source (plane wave vs antenna), and even in ADP mode.

Click the **Advanced SBR+ Options** button to open the **SBR+ Advanced Options** window. On the **Enable** tab, Check GO Blockage Options to enable the **GO Blockage** tab.



On the GO Blockage tab, you have check box options to enable or disable GO Blockage and enable or disable Surface Self-block.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

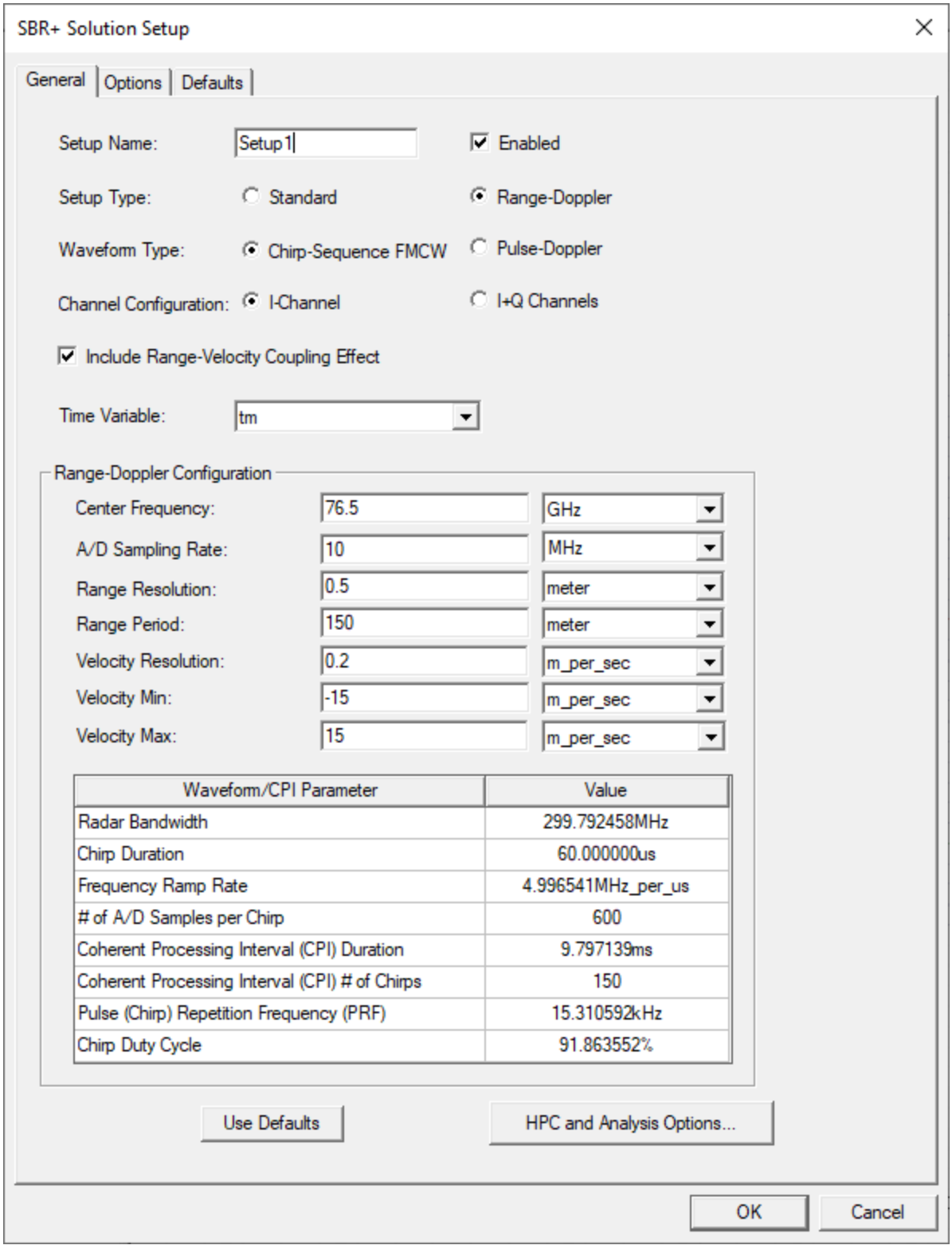


Surface Self-Block: if enabled, the surface where the footprint is radiated from is used for blockage check, that is, anything below the surface wrt respect to the incoming ray is not in line of sight. If the setting is disabled, the footprint surface is not used for blockage checks (anything below the surface wrt the incoming ray is in line of sight), but other surfaces are used for blockage.

10. Generate Range-Doppler report.

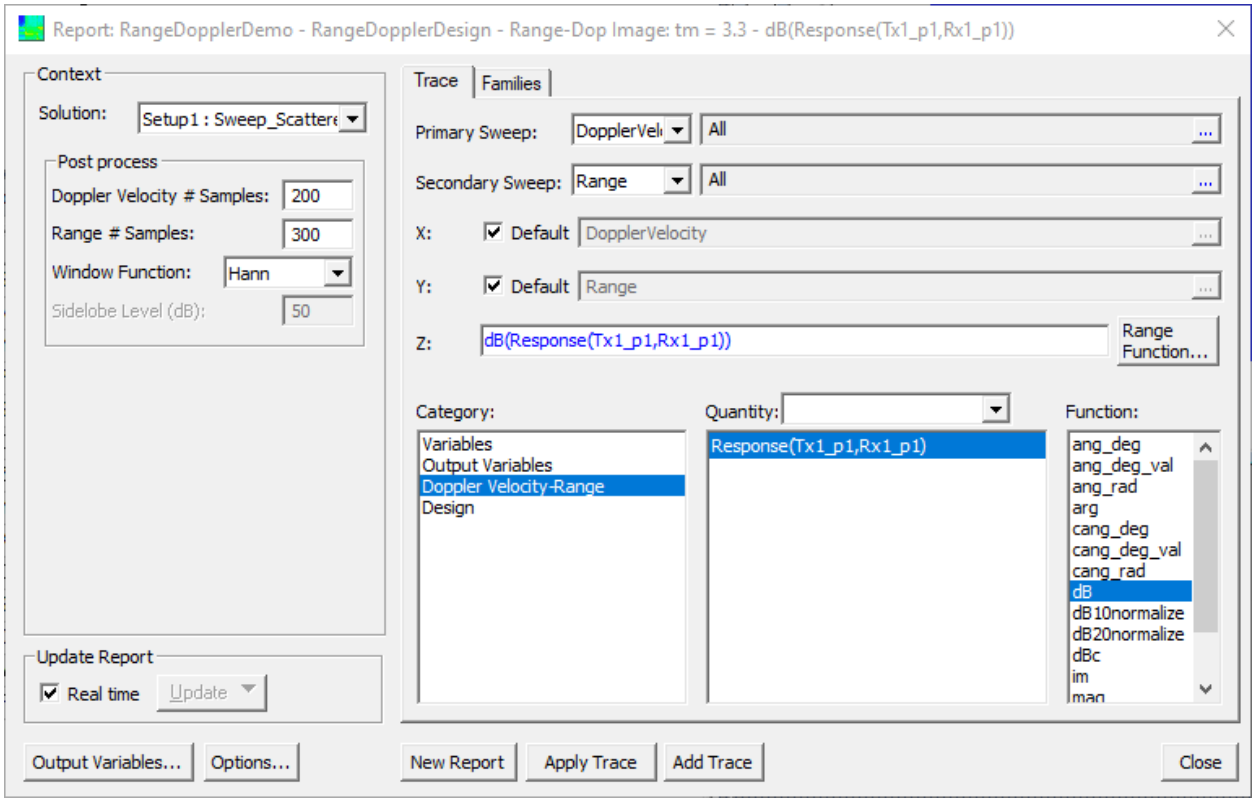
    After validating the solution setup and executing the analysis, you can generate a range-Doppler image for each configured port-coupling pair by right-clicking on the **Results** icon in the **Project** tree and selecting **Create Range-Doppler Report>Rectangular Color Map Plot** or select the Results tab of the ribbon and click **Range-Doppler Report>2D Color Map**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **Report** dialog that appears, first configure the post-processing settings.

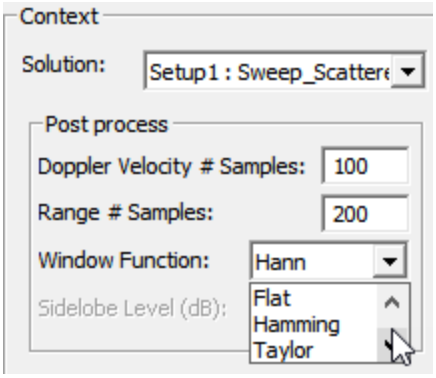
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The width and height of the image in pixels is set by **Doppler Velocity # Samples** and **Range # Samples**, respectively. By default these correspond to the settings in the Range-Doppler setup. For example, since the Velocity Min and Max were -50 m/s and +50 m/s, this means the velocity period is Pv = 100 m/s. The Velocity Resolution was set to Rv = 0.5 m/s. Therefore, the default number of Doppler velocity samples is Pv/Rv = 200. Similarly, the Range Period and Range Resolution were set to Pr = 150 m and Rr = 0.5 m, respectively. Therefore, the default number of range samples is Pr/Rr = 300. The default number of samples preserves all information but the results might appear choppy. You can increase or decrease the number of samples (image pixels) relative to the default to either over-sample or under-sample the image. Over-sampling smooths out the image without adding information, while down-sampling causes information to be lost. In this example, we stick with the default number of samples.

The **Window Function** selection controls the tradeoff between feature blurring and signal-processing sidelobe suppression. There are four options: Hann (default), Flat, Hamming, and Taylor.

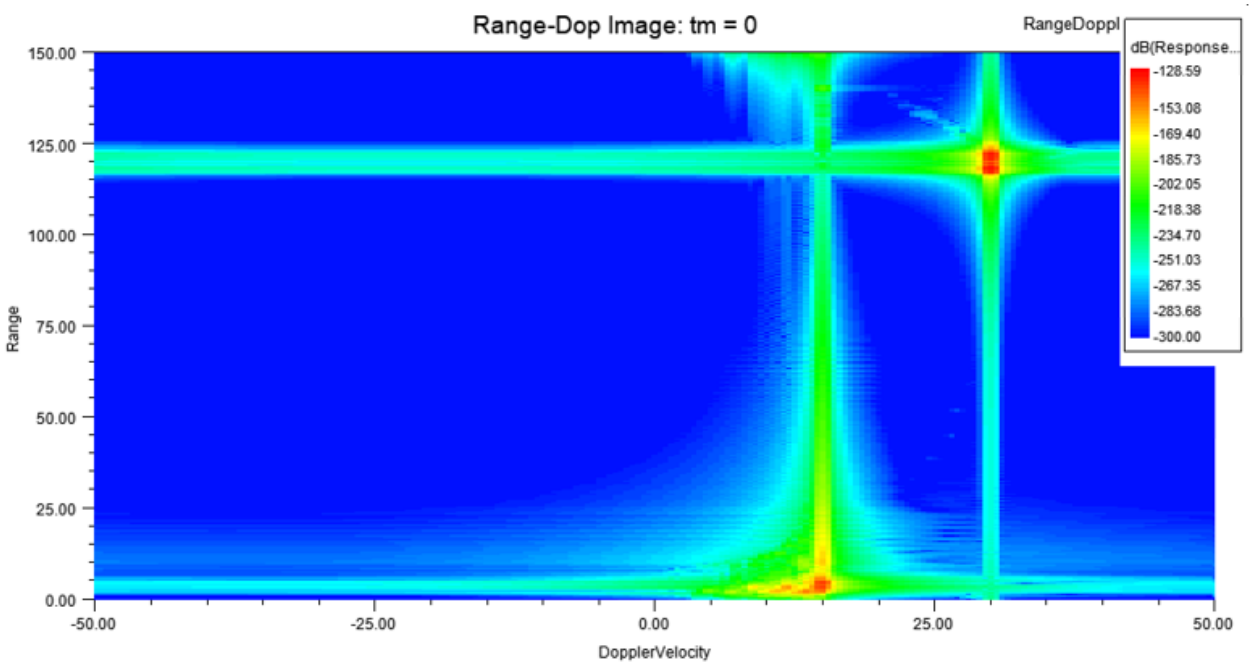
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In radar imaging applications, Hann provides a good compromise between blurring and sidelobe suppression. The Flat window provides the best resolution (no blurring), matching the range and velocity resolution specifications in the Range-Doppler Setup. However, its first sidelobe is only -13.3 dB down from the feature peak, and subsequent sidelobes decay slowly. This means the sidelobes of strong features in the image tend to mask weaker ones. The Hann (aka Hanning) window has a first sidelobe that is -31.5 dB down from the feature peak, and subsequent sidelobes decay quickly from there, but the achieved resolution is twice as large (blurry) as that specified in the Range-Doppler Setup. The Hamming window has very similar resolution to the Hann window, and its first sidelobe is -42.7 dB down from the feature peak, which is better than Hann. However, subsequent sidelobes decay very slowly from there, so strong features in the image have a greater tendency to mask weak features compared to the Hann window. The Taylor window provides the option to customize sidelobe levels, and the specified level will be roughly identical for all sidelobes. As with the other window functions, the higher the sidelobe suppression that is configured for Taylor, the greater the amount of blurring. In this example, we stick with the default Hann window.

- Under the **Trace** tab, leave the Category selection as its default: Doppler Velocity-Range.
- Under the **Quantity** label, select the port-pair combination to display. In this example, there is only one option: Response(Tx1_p1,Rx1_p1).
- The range-Doppler image is complex-valued pixels. Select the **Function** to apply to the complex data in order to convert it into a real number that can be mapped to a linear color map. The default Function is dB. This is recommended for most cases since typically a large dynamic range is required to simultaneously observe multiple features.
- Click the **New Report** button to generate the range-Doppler plot.

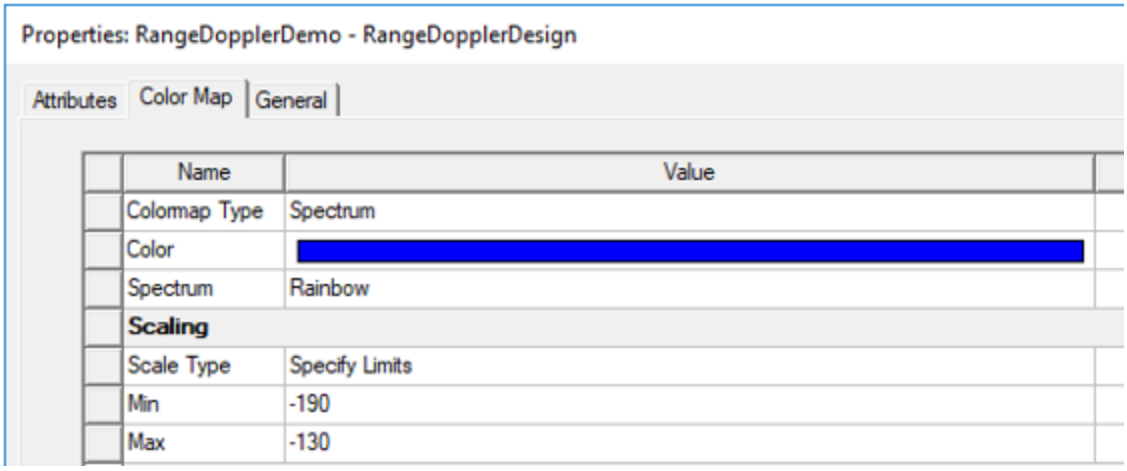
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Because the default color map has a lower extent of -300 dB, while the strongest feature (that of the oncoming car) has a peak strength of ~-129 dB, the descending sidelobes of the Hann window are evident across the range and Doppler velocity extents of the image.

To customize the scale of the color map, double-click on the color map legend.

In the **Properties** dialog that appears, select the **Color Map** tab. Under the Scaling group, change the Scale Type from Auto Limits to Specify Limits and then enter the Min and Max dB values of the color map. In this example, we set Min and Max to -190 and -130 dB, respectively.

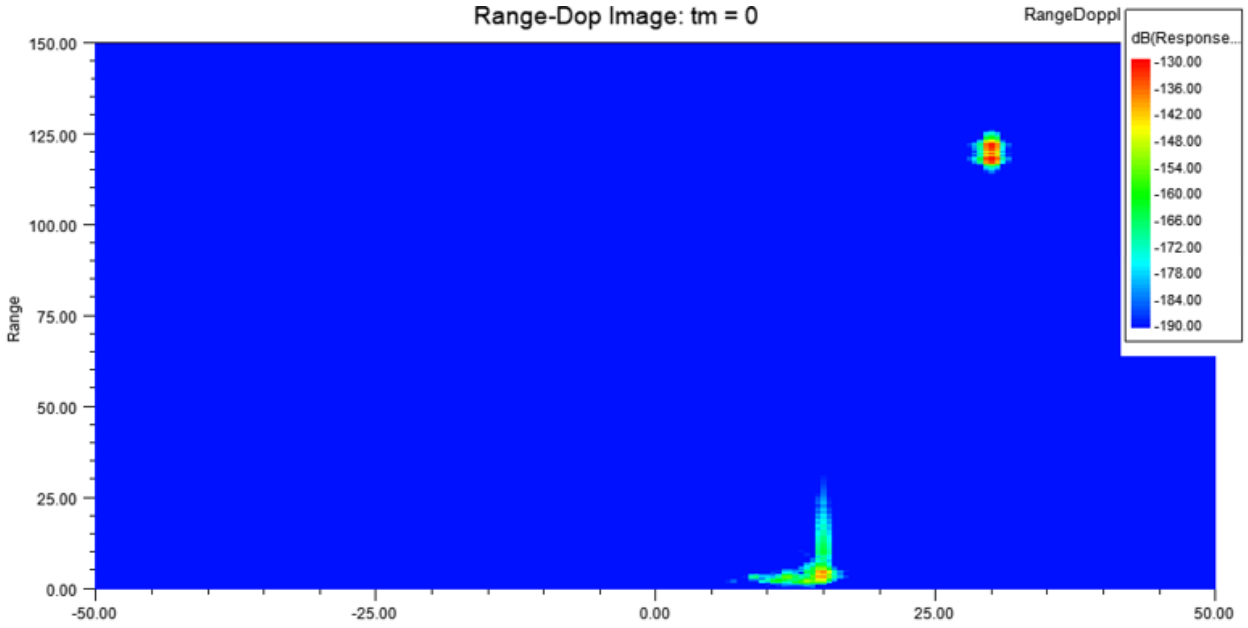


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By default, the number of colors in the color map is 64. To achieve round numbers in the labeling of the color bar, you need to set Value Type to Specify Num. Levels (default) and set Num. Levels (N_{lvl}) according to the dB span (S_{dB}) of the color map. To achieve a fairly continuous color map, N_{lvl} >= 50 is recommended. Beyond that, choose N_{lvl} so that $S_{dB}/(N_{lvl} - 1)$ is a round number like 0.5, 1, or 2 dB. In this example SdB = -130 dB - (-190 dB) = 60 dB. Thus, we choose N_{lvl} = 61.

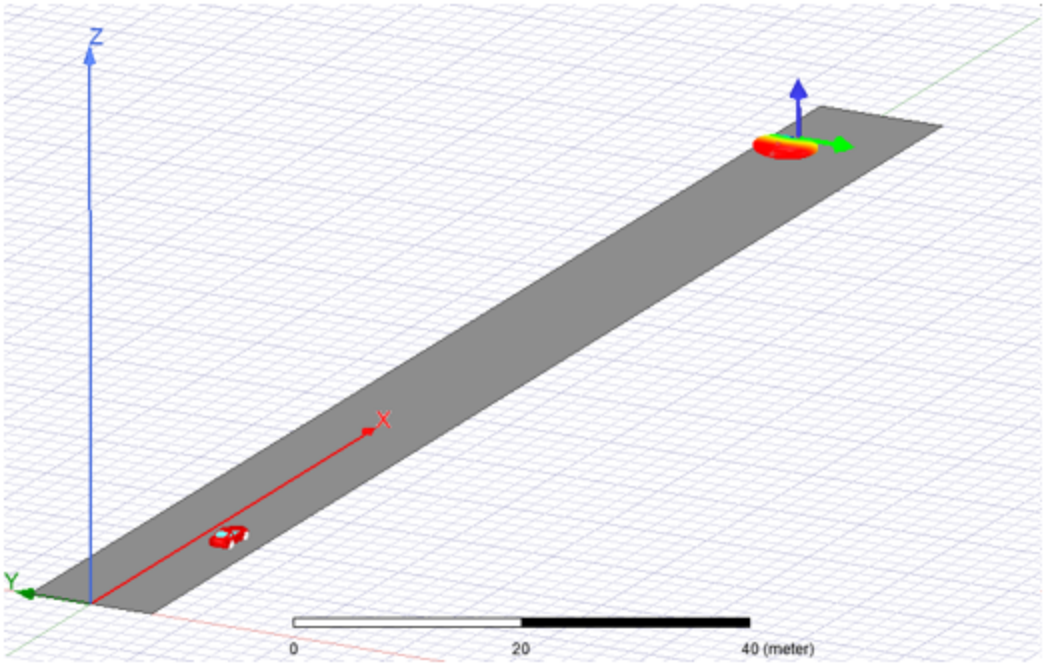
Specify Values	
Value Type	Specify Num. Levels
Num. Levels	61
Spacing	25

This results in a color map with labels at precisely 4 dB intervals.

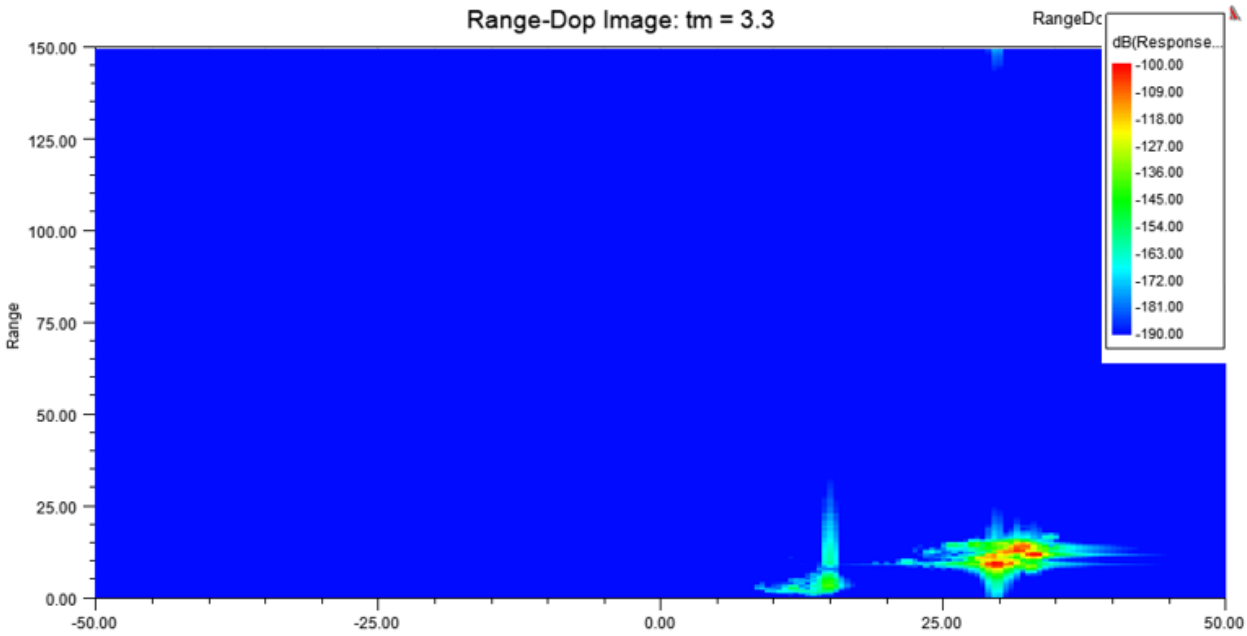


The following image is generated for time tm = 0, when the velocity of the car is 15 m/s in the +X direction and the velocity of the radar is 15 m/s in the -X direction. Also, the position of the car is X = 20 m while the position of the radar is X = 140 m. Thus, we observe a range-Doppler feature at range 140 – 20 m = 120 m and velocity 15 - (-15) m/s = 30 m/s. Notice also that the response of the car is spread out in range because the car is an extended object spanning roughly 5 m. There is also a smeared feature near range = 0 m and velocity = 15 m/s. This represents the return from the road as filtered by the Tx and Rx beam antennas of the radar.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

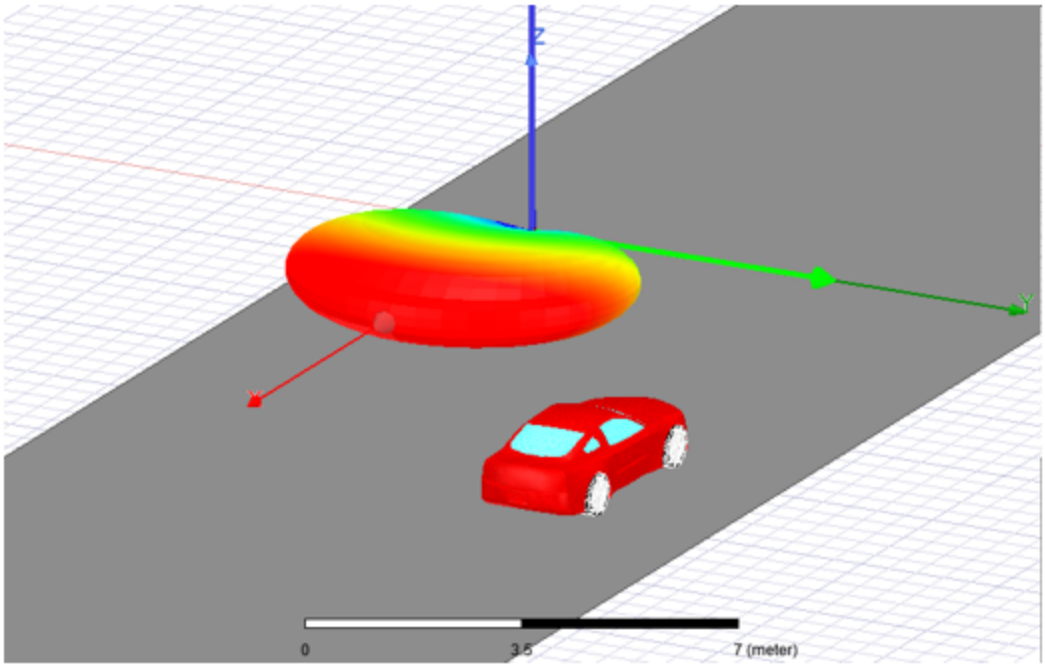


With this setup, you can generate results for any value of time tm. Below we have advanced time to tm = 3.3 s.



At this point in time, the car is about to pass by the radar.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The car is traveling 21.6 m/s in the +X direction while radar remains traveling at 15 m/s in the -X direction. Thus, we might anticipate a Doppler velocity of 36.6 m/s. However, only the radial component of the velocity between radar and the car is captured in Doppler velocity. To determine that, we need to consider the position of the car relative to the radar. At tm = 3.2 s, the center of the car is located at (80.4, -3, 0) m while the radar is located at (90.5, 3, 0.5) m in global coordinates. Relative to the Radar_CS coordinate system, the velocity vector is (-36.6, 0, 0) m/s while the displacement vector from the radar to the car is (10.1, 6, 0) m. Because of the finite vertical extent of the car, we assume the vertical (Z component) displacement from the radar to the car is zero. Thus, we anticipate a feature with a Doppler velocity of 31.5 m/s and a range of 11.7 m. This is consistent with the resulting range-Doppler image shown above (note we have changed the color map to now extend from -190 dB to -100 dB). However, notice that the observed Doppler velocity is spread out between roughly 29 m/s and 34 m/s whereas at tm = 0, the observed Doppler response was more tightly confined about the anticipated Doppler velocity of 30 m/s. At this close proximity to the radar, the front and rear of the car subtend a significant angular span relative to the radar, and this causes a corresponding spread in the observed Doppler velocity response.

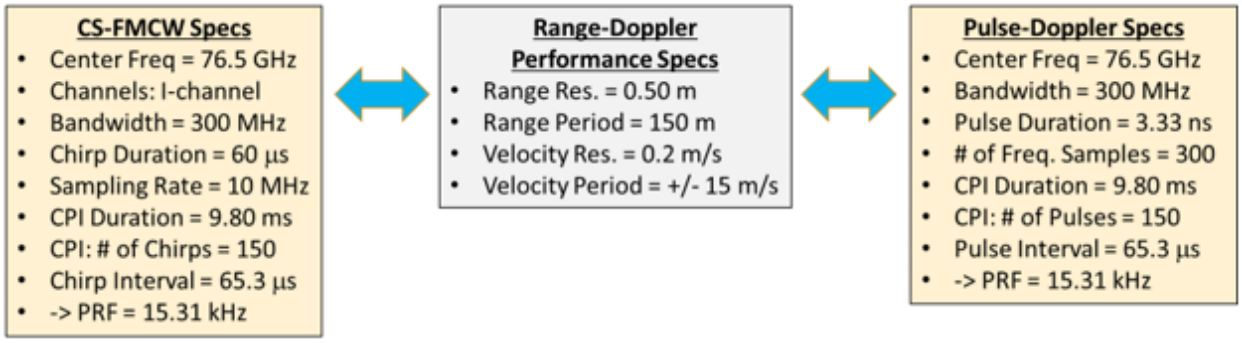
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chirp Sequence FMCW for SBR+ Range Doppler Solution Setup

The Chirp-Sequence FMCW (CS-FMCW) waveform option models radar transmission of a uniform amplitude sinusoid whose frequency continuously rises from a low value to a high value that straddles the center frequency. This is known as a “chirp”. Unlike the pulse-Doppler waveform, where the transmitter issues a very short duration pulse followed by an extended quiet time while one or more receivers listen, the FMCW transmission has orders of magnitude longer duration and corresponding lower peak power, and the receivers listen while the rising-frequency chirp is being broadcast. With the CS-FMCW waveform, the range to each scattering feature is proportional to the difference in the current transmission frequency with that of the received signal component due to the scattering feature. This frequency difference is measured by mixing (multiplying) the current transmission RF signal with the received signal, low-pass filtering it, and FFTing the resulting baseband signal to produce a response vs. beat-frequency. The beat-frequency axis is then rescaled according to the frequency ramp rate and speed of light to yield the response vs. range (distance). Like the pulse-Doppler waveform, with the CS-FMCW waveform, chirps are issued and observed in a sequence over a coherent processing interval (CPI) to yield many range responses, and these are then FFT’d across the sequence to yield the response vs. range vs. Doppler velocity.

CS-FMCW is generally considered more practical to implement than the pulse-Doppler waveform owing to its significantly lower peak-power. Moreover, CS-FMCW requires no more total time (CPI duration) than a corresponding pulse-Doppler waveform to achieve a given level and resolution and ambiguity in range and velocity. Although the chirp duration is much longer than the corresponding pulse, this extended chirp duration is merely replacing the extended quiet time required by the pulse-Doppler waveform. However, FMCW waveforms do introduce an artifact known as range-velocity coupling. Because FMCW radars measure range according to the frequency difference between received signal and the current broadcast shift, and moving scatterers cause a Doppler shift in the received signal, there is a shift in the perceived range to scattering features. The range shift is proportional to the center frequency of the FMCW ramp and each scattering feature’s radial (Doppler) velocity relative to the radar, and it is inversely proportional to the FMCW frequency ramp rate (that is, the bandwidth divided by the chirp duration). The Chirp-Sequence FMCW waveform option in HFSS SBR+ incorporates this range shift into its simulations, but toggling off the **Include Range-Velocity Coupling Effect** will artificially suppress this effect.

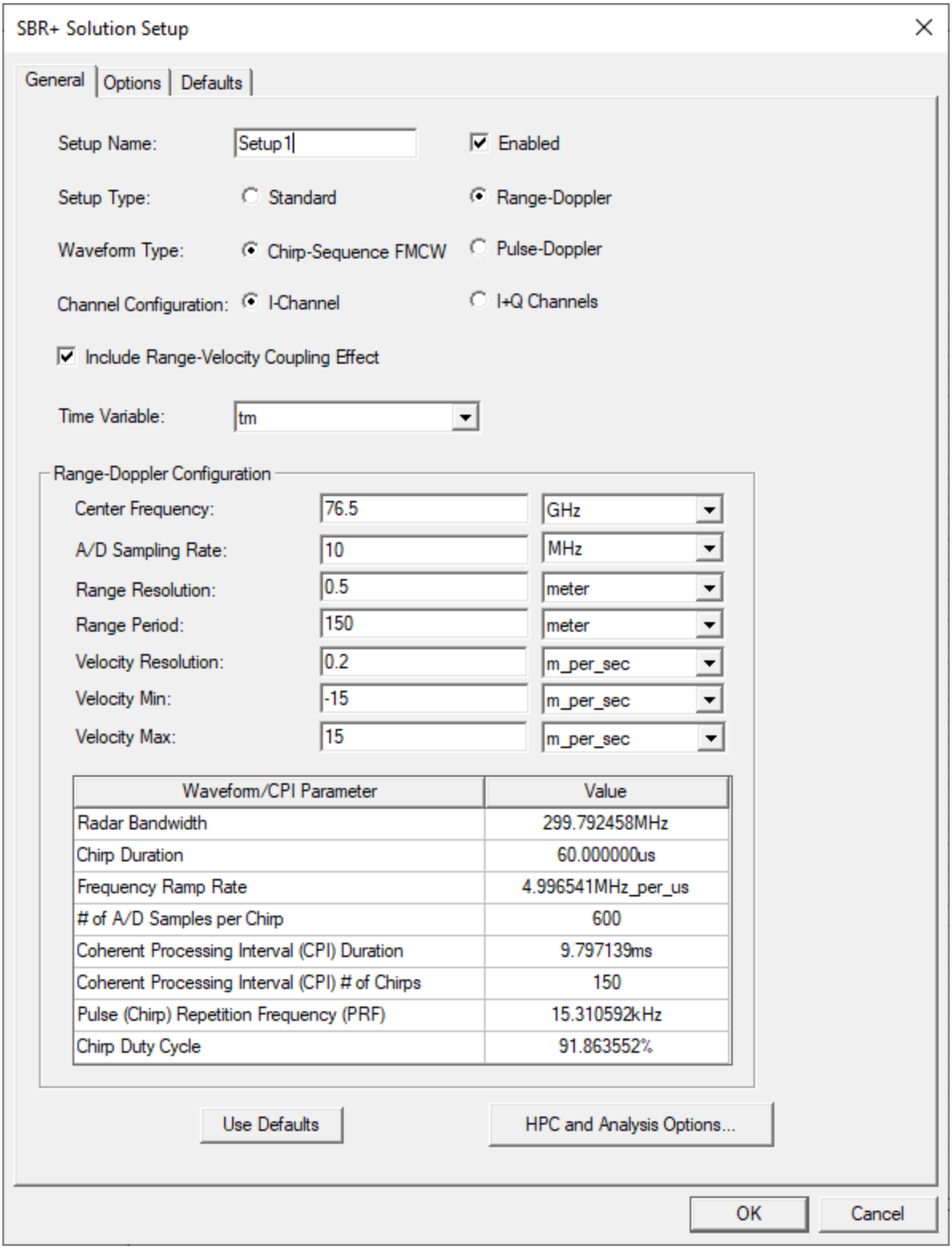
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Waveform/CPI Parameters and Range-Doppler Image Properties are interchangeable.

In the **SBR+ Solution Setup**, setting Waveform Type to Chirp Sequence FMCW (default) exposes additional options and settings relative to the Pulse-Doppler waveform option: Channel Configuration (I-Channel or I+Q Channels), A/D Sampling Rate, and whether the waveform model should Include Range-Velocity Coupling Effect. Additional rows will also appear in the non-editable Waveform/CPI Parameter table located below the waveform input settings: Chirp Duration, Frequency Ramp Rate, # of A/D Samples per Chirp (replacing # of Frequency Samples), and Chirp Duty Cycle.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

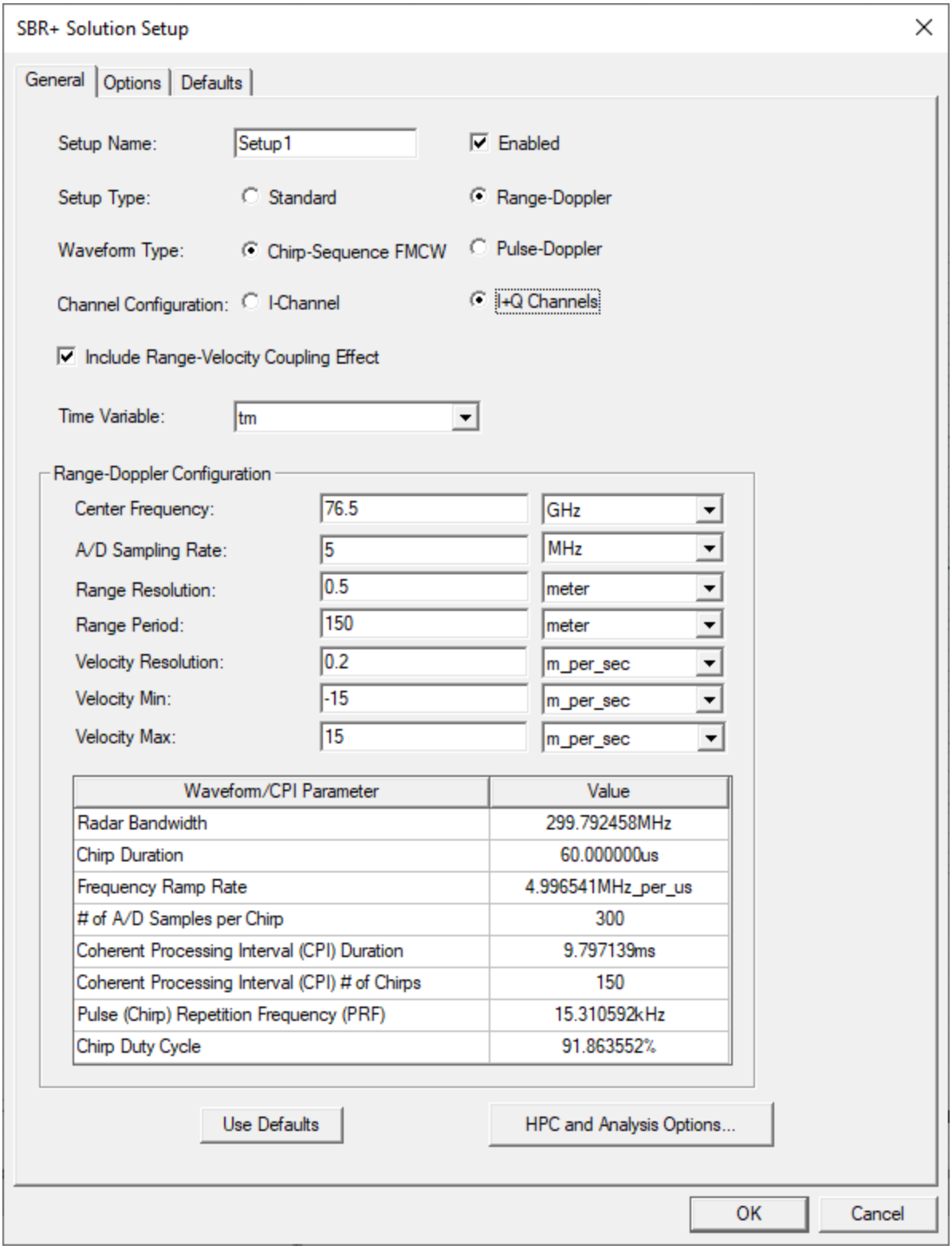
When Waveform Type is set to Chirp-Sequence FMCW, the default Channel Configuration is I-Channel, Include Range-Velocity Coupling Effect is engaged (more accurate), and the default A/D Sampling Rate is 10 MHz. In the CS-FMCW waveform model of HFSS SBR+, the latter setting controls the sampling rate of the analog-to-digital converter that samples the received signal after it has passed through the waveform model's mixer and low-pass filter. If the Channel Configuration is set to I+Q Channels, then this controls the sampling rate for both channels in tandem. For given Range Resolution and Range Period inputs, increasing A/D Sampling Rate reduces the Chirp Duration and Chirp Duty Cycle reported in the Waveform/CPI Parameter table.

The Chirp-Sequence FMCW waveform settings will not be accepted unless the Chirp Duty Cycle is less than 100%, meaning that the Chirp Duration is less than the start-to-start interval between chirps within the CPI, the latter quantity being the reciprocal of Pulse (Chirp) Repetition Frequency (PRF); issued chirps cannot overlap. If the Chirp Duty Cycle reports as exceeding 100%, you must either reduce the Chirp Duration or expand the interval between chirps. You can reduce Chirp Duration by increasing the A/D Sampling Rate, increasing the Range Resolution, or decreasing the Range Period. You can expand the interval between chirps by reducing the difference between Velocity Max and Velocity Min.

As with the case of the Pulse-Doppler waveform, the input dialog for CS-FMCW settings will not permit the product of Range Period (ambiguity) and velocity period (Velocity Max – Velocity Min) to exceed the theoretical limit of c2/(4 fc), where c is the speed of light and fc is the center frequency.

Switching the Channel Configuration from I-Channel to I+Q Channels while holding all other input settings fixed will cause corresponding changes in the non-editable Waveform/CPI Parameter table: Chirp Duration, # of A/D Samples per Chirp, and Chirp Duty Cycle will drop by half, while Frequency Ramp Rate will double. This is because, all other system parameters remaining the same, an I+Q configuration generates twice as much information as its I-Channel counterpart, which will double the range period. But if we hold Range Period fixed while going from I-Channel to I+Q Channels, then the above mentioned quantities must change instead, and in the indicated manner. Alternatively, an equivalence in performance and system specifications between an I-Channel and an I+Q Channels configuration can be achieved by reducing the sampling rate of the I+Q channels configuration by one-half relative to its I-Channel counterpart. However, the results will not be exactly the same, and the I+Q Channels configuration will yield range-Doppler images with reduced signal-processing artifacts.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Roughness for SBR+

HFSS SBR+ provides the option to simulate the scattering impact of surface roughness applied to object boundaries. The surface roughness model includes two additional physical effects into the SBR+ simulation: coherent field attenuation and diffuse incoherent scattering. These effects are wavelength dependent. As the surface height deviations become large relative to the simulation wavelength, coherent specular reflections are converted into a diffuse response that has an incoherent (random) phase.

The SBR+ model characterizes surface roughness as a random process which is defined by two parameters: 1) the standard deviation of surface heights (*H*), and 2) a surface correlation length (*L*).

The surface height deviations are modeled as a Gaussian distribution with mean zero and standard deviation *H*. The surface correlation (i.e., its auto-correlation function) is also assumed to follow a Gaussian distribution with standard deviation $L/\sqrt{2}$. The correlation length *L* may be thought of as the average lateral distance between neighboring peaks on the surface.

For user inputs, the correlation length is replaced with a dimensionless parameter called roughness, which is defined as *R* = *H* / *L*. A roughness value of 0 produces a flat surface. The surface roughness increases with *R* and has typical values in the range [0.,0.5]. As the figure illustrates, the parameter *R* provides an intuitive measure of roughness that is independent of the magnitude of the height fluctuations (*H*).

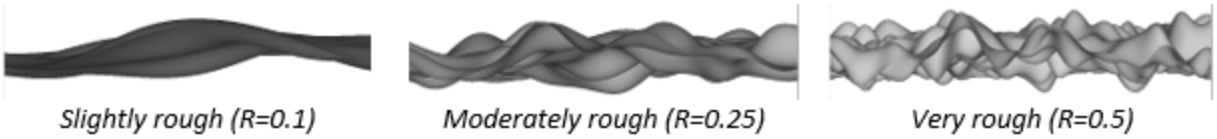
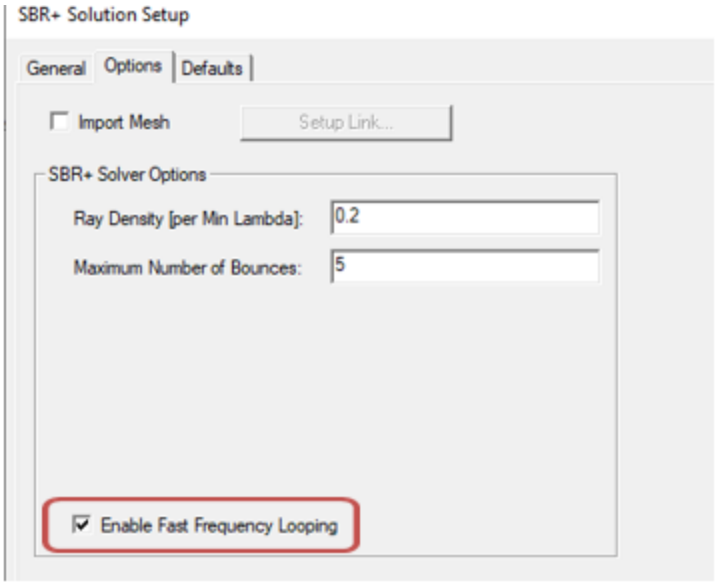


Figure: Explicit rough surfaces with different roughness values R and height standard deviation H=1mm.

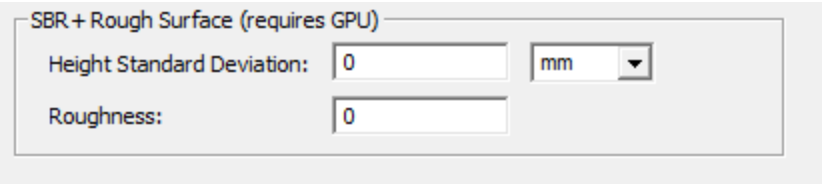
Prerequisites

- The feature is for SBR+ Solution Setups with Fast Frequency Looping (FFL) enabled in the SBR+ Solution Setup Options. If FFL is not enabled, an error message is issued to the Message window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- The roughness model may only be applied to non-penetrable surface boundaries: Perfect E, Perfect H, Layered Impedance (one-sided), Impedance Boundary, and Finite Conductivity.

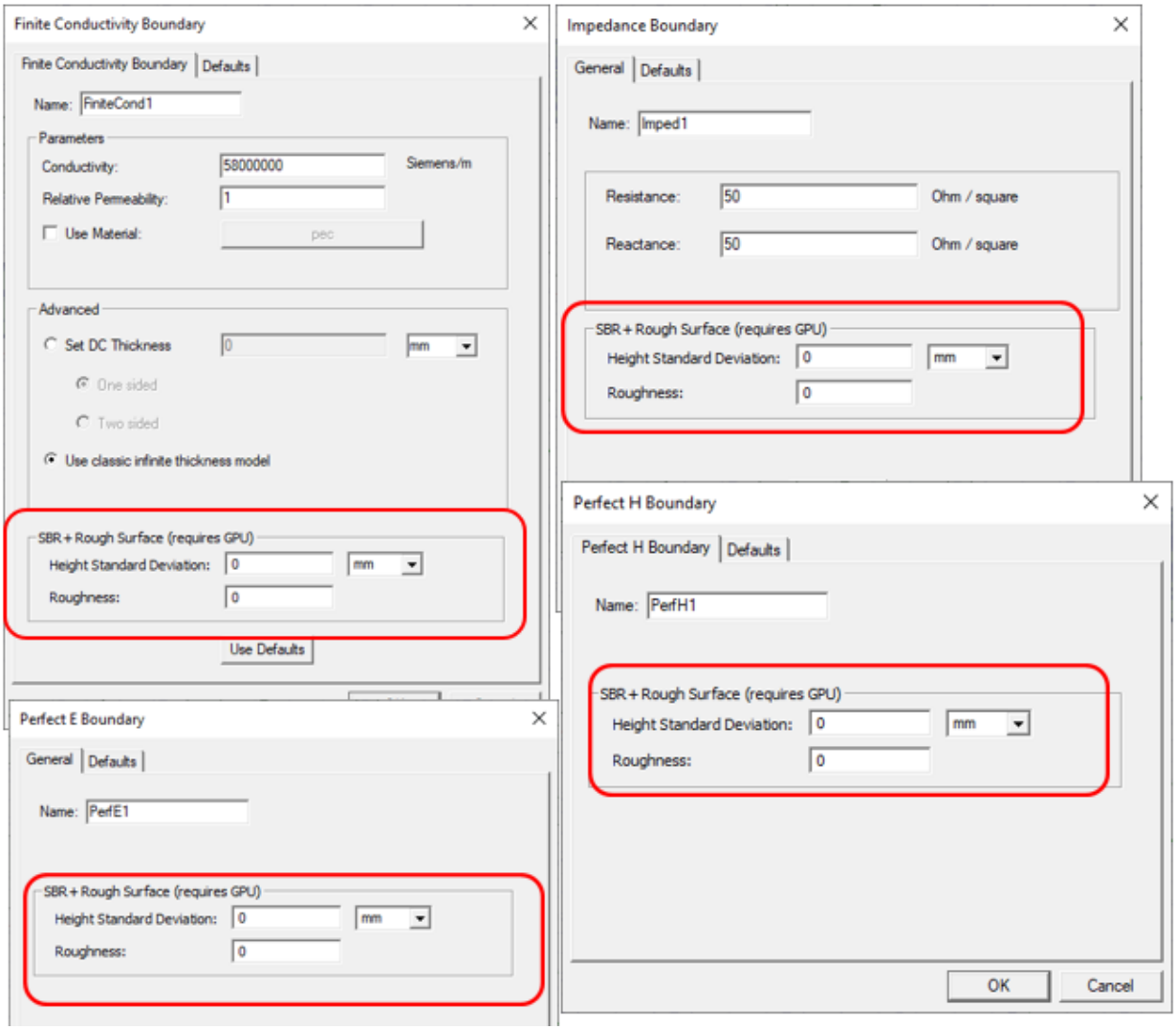


- The SBR+ rough surface model requires GPU acceleration to initialize the model. If this criteria is not met, an error message is issued during SBR+ pre-processing. An HPC license is not required to enable the model.

Setting up an SBR+ Simulation with Surface Roughness

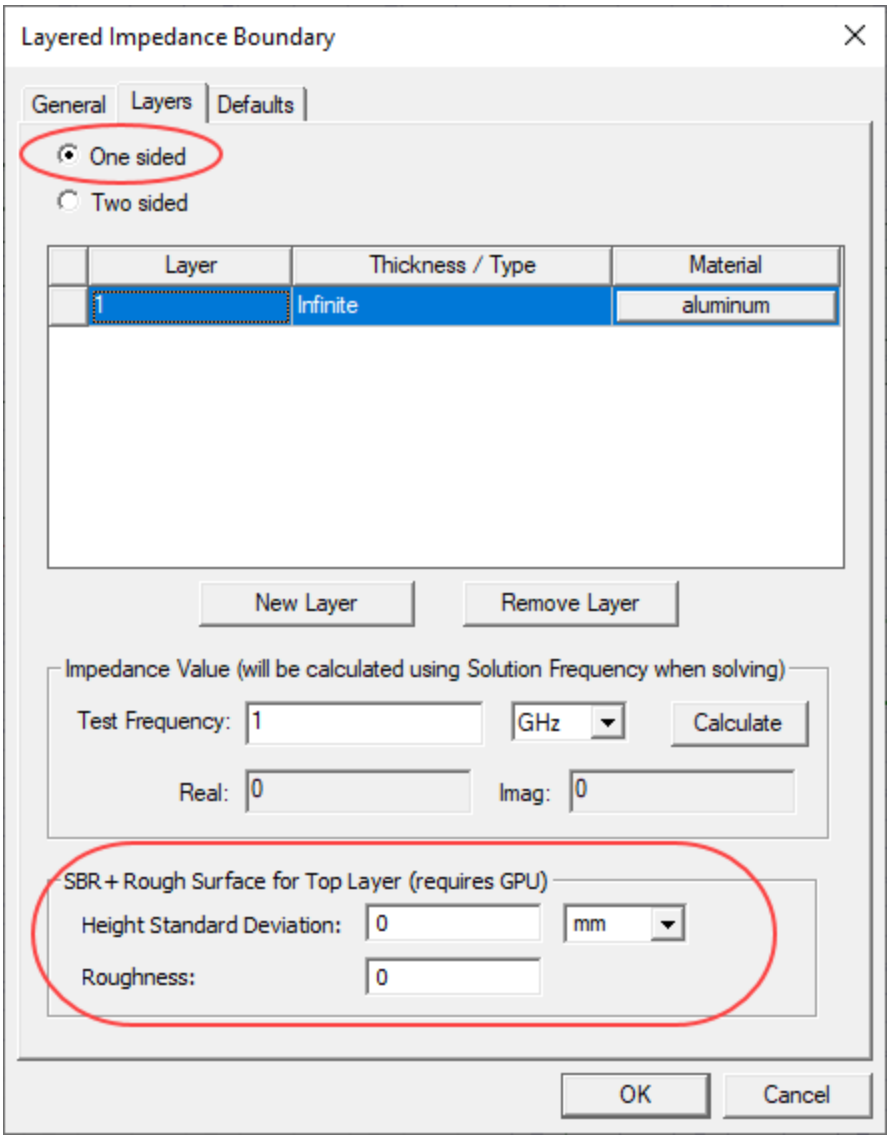
1. Create an HFSS design and set the Solution type to SBR+.
2. Add geometry objects.
3. Assign object boundaries. For the Finite Conductivity Boundary, Impedance Boundary, Perfect E Boundary and Perfect H Boundaries, you can set the SBR+ Rough Surface properties as shown. Typical values for rough asphalt are *H*=1mm for height deviation, with a roughness of *R*=0.2 so the corrolation length *L* = 5mm.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For the Layered Impedance Boundary, select the **Layers** tab and select One sided and edit the properties under the “SBR+ Rough Surface for Top Layer” panel.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. The “SBR+ Rough Surface” properties can also be edited after creating the boundary by selecting the boundary in the project tree and editing values in the property window.
5. Add an excitation, such as Parametric Antenna under 3D components.
6. In the “Options” tab of the “SBR+ Solution Setup” dialog box, select “Enable Fast Frequency Looping”.
7. Run validation
8. Run simulation

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lightweight Geometry for HFSS SBR+ Designs

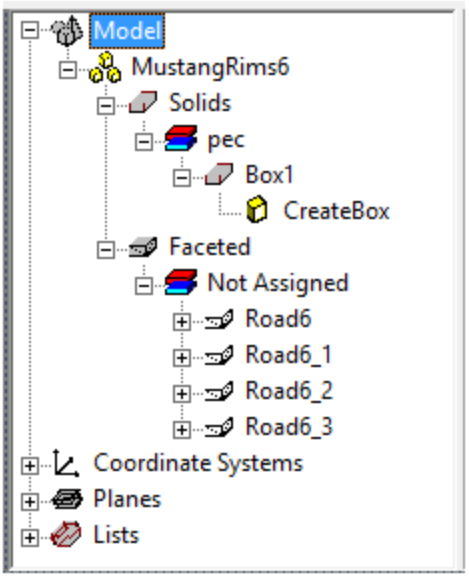
Importing large stl files into Ansys Electronics Desktop can be time consuming. STL files contain triangulations of objects, and importing them to the modeler creates a face out of every triangle, and an edge out of every triangle edge. This creates a large number of faces and edges. which can slow down operations. The lightweight geometry feature gives you the option to import objects as one face, without edges or vertices in them. Internally, the modeler does not create bodies for these objects.

When the solution type is SBR+, the Import File dialog for STL files contains several options.

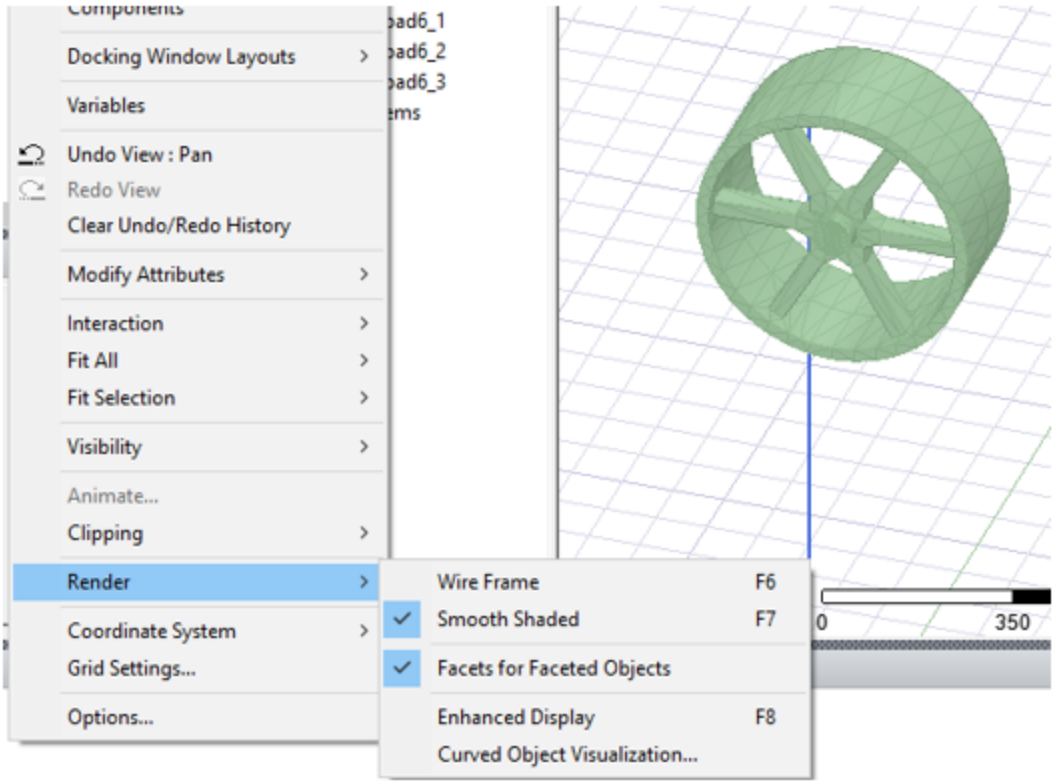
- **Heal** attempts to remove common issues while importing stl files, including closing solids when they are open due to minor cracks in triangulation and removing self-intersections.
- **Import as Lightweight Faceted Body**, if checked, creates faceted bodies from imported objects and enables the Separate Disjoint Lumps option.
- **Separate Disjoint Lumps** option, if checked, create multiple parts if the .stl files have objects having disjoint lumps.
- **Reduce** enables lets you reduce the number of facets in faceted objects while keeping the overall shape of the object. Checking **Reduce** enables fields for specifying Reduce Max Error and Reduce Percentage.

Once imported, faceted objects have distinct icons in history tree. They can also be grouped together, as shown in the following figure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A new option, **Facets for Faceted Objects**, is provided in **View>Render** menu for SBR+ solution types. When this option is check on, the modeler renders edges of triangle contained in the faceted bodies. These edges cannot be selected. See the following figure:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initial Mesh Operations for Lightweight Geometries

Importing a lightweight geometry into an SBR+ design disables editing for the **Use Dynamic Meshing** option for the Initial Mesh Settings. After lightweight geometry import, Use Dynamic Meshing is always On, and eliminates mesh file generation during simulation.

> **Important:**
>
> The consolidation of surfaces into a conformal mesh is skipped for dynamic and tolerant meshing, including for light weight geometries. This can lead to overlapping surfaces in SBR+ simulations. The user should carefully avoid overlapping surfaces or objects as SBR+ can produce unexpected results.

Permitted Modeler Operations on Faceted Objects

- Arrange operations such as Move, Rotate, Mirror.
- Duplicate along line, around axis, with mirror.
- Scale.
- Copy-Paste.
- Create object CS - you can snap to vertices of facets to create object CS.
- Create face CS - you can pick any point, determine the triangle on which the picked point lies, and store the position relative to the triangle vertices. **Note:** this capability is particularly useful for antenna placement on lightweight geometry models in SBR+ designs.
- Create 3D components out of faceted objects.
- Selecting faces for assigning boundaries, coordinate systems, excitations or hybrid regions. **Note:** the entire object boundary is selected.

Other Characteristics of Faceted Objects

- Solid and sheet objects are identified in the History tree.
- Object Measure dialog shows the correct volume, and surface area of faceted objects.
- Faceted objects are skipped during intersection checks and containment checks done during geometry validation.
- Faceted objects cannot be exported from the Modeler.

SBR+ Curvature Extraction for Faceted Surfaces

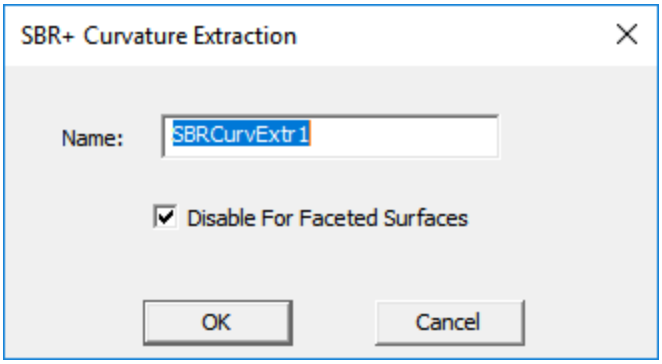
In SBR+, the ray-tube divergence correction and deployment of creeping rays depend on the surface curvature of the geometries. However, faceted CAD models such as STL lack intrinsic

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

curvature information. With these models, a curvature extraction algorithm is used wherever possible to interpolate smooth curved surfaces based on the orientations of neighboring facets. Curvature extraction is enabled by default, and users may disable it on selected faceted faces or objects by using this mesh operation. When disabled, the affected surfaces are assumed to be perfectly flat. This mesh operation can only be applied to faceted or flat surfaces and is ignored on curved surfaces.

1. Select the faceted object or face for which you want to disable SBR+ Curvature Extraction for Faceted Surfaces.
2. Select **HFSS** > **Mesh** > **Assign Mesh Operation** > **SBR+ Curvature Extraction for Faceted Surfaces**, or right-click in the Project Tree and select **Assign Mesh Operation** > **SBR+ Curvature Extraction for Faceted Surfaces**.

   This displays the **SBR+ Curvature Extraction** dialog box.



3. You can accept the default name or provide one of your own.
4. By default, the box is checked to disable curvature extraction on the selected surface. Click OK to apply a mesh operation.
5. Creeping waves will not propagate over faceted surfaces that have curvature extraction disabled.

   The defined Mesh Operation appears in the **Project** tree. Deleting this Mesh operation will invalidate a solution. If applied to a curved surface, it will be ignored.

Assigning SBR+ Hybrid Regions

HFSS can use shooting and bouncing ray (SBR) technology to compute the scattered field contribution of defined platform geometry via one-way coupling. This allows you to efficiently calculate installed near field patterns, far-field patterns and coupling (S-parameters) between antennas in the presence of an electrically large scattering geometry.

Note that the same SBR+ mesh used for both SBR+ simulation and VRT is also used in extracting wedges according to the user-specified criteria. This is done so that the wedge

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

locations extracted are very near the physical edges of the SBR+ geometry model in the simulation, preserving accuracy of the line-of-sight visibility check for wedges.

- An SBR+ Region can only be assigned to no-solve inside object (metal).
- When an object is assigned as an SBR+ region, Boundary conditions (BCs) may be assigned to the geometry inside the SBR+ region(s), but there are some limitations in what boundary conditions are supported. HFSS currently supports assignment of Perfect E, Perfect H, Impedance, Finite Conductivity, and Layered Impedance boundaries to geometry in the SBR+ region. Other boundaries are not yet supported for SBR+, and produce an error. Infinite Ground Planes are not supported with SBR+. Note also that surface roughness must be set to “0” for all BCs assigned to SBR+ region geometry.
- An SBR+ Region cannot be contained by a dielectric cavity.
- Incident plane waves are allowed. If you include an incident plane wave in the design, the Hybrid tab for the Driven Solution Setup dialog includes options for RCS type as Monostatic or Bistatic.
- If an SBR+ Region is assigned, the design becomes one way coupled. (See Assigning Hybrid Regions.) You can use the **Set SBR+ Source Regions** command to group source regions for SBR+ for two-way couplings between the grouped regions.
- Terminal ports with multiple terminals are supported if a an SBR+ region is present.
- Designs with SBR+ regions provide options to view total, scattered, or incident fields.
- SBR+ Region simulations can benefit from GPU Acceleration. See Setting HPC and Analysis Options.
- If your design contains an SBR+ Hybrid Region, you have an option on the Initial Mesh Settings dialog to **Allow tolerant meshing for SBR+ regions**, which can speed up the solve.
- If you want to consider far fields or near fields, the field sample points for SBR+ Hybrid regions must be defined as part of the simulation setup, rather than during post processing as is done for HFSS near and far fields. See the discussion of the **Setup** dialog box, **Hybrid** tab in step 3 below.
- HFSS with Hybrid and Arrays designs that include blockages have an enhanced work flow described.
- When a project contains SBR+ region and the following error appears: "[error] Solving adaptive frequency ... (Domain solver), process hf3d error: Error in using SBR+ region: Each source region should contain at least one excitation."

  "Each source region should contain at least one excitation" means there is at least one source region without an excitation. To work around the issue, please go to Set SBR+ Source Regions and, select all regions and click "**Group**".

You must select an appropriate object in order to enable the menu.

To assign an SBR+ Hybrid Region:

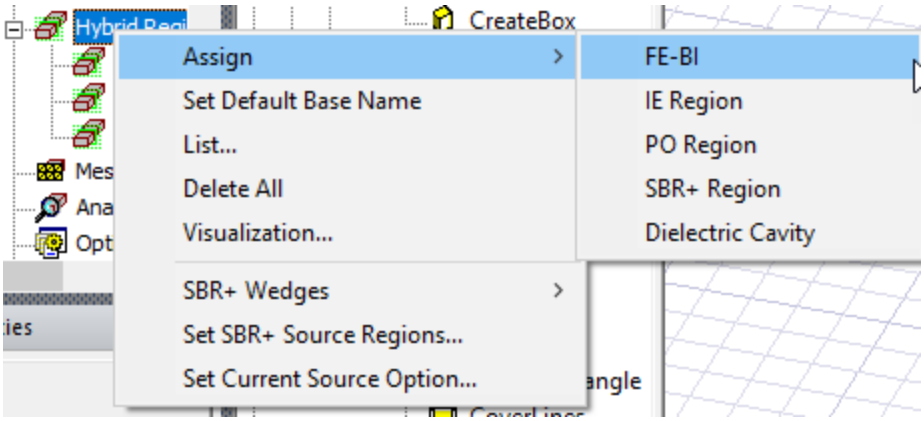
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the object you want to assign as an SBR+ Region, and use one of the following menus:

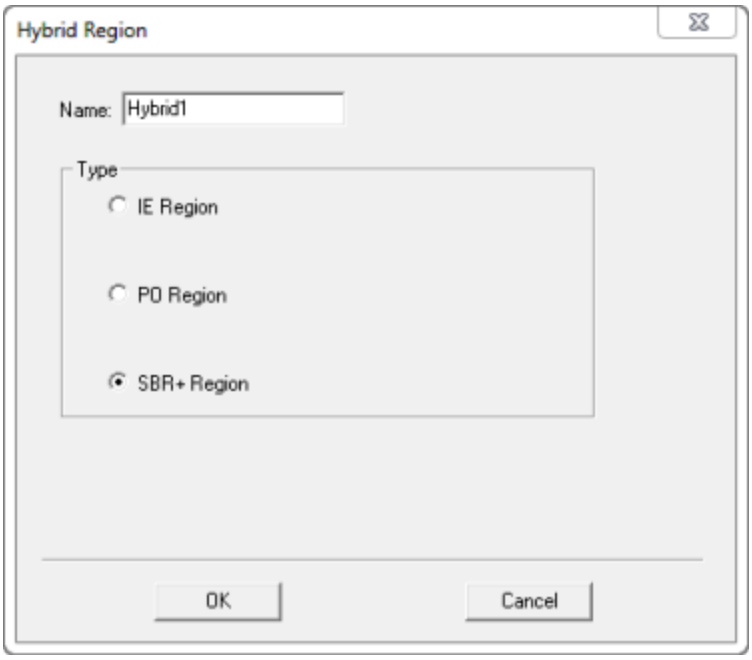
HFSS>Hybrid>Assign Hybrid>SBR+ Region

Right-click in the Modeler window and select **Assign Hybrid>SBR+ Region**.

Select the Hybrid Regions icon in the Project tree, right-click, and click **Assign>SBR+ Region**.

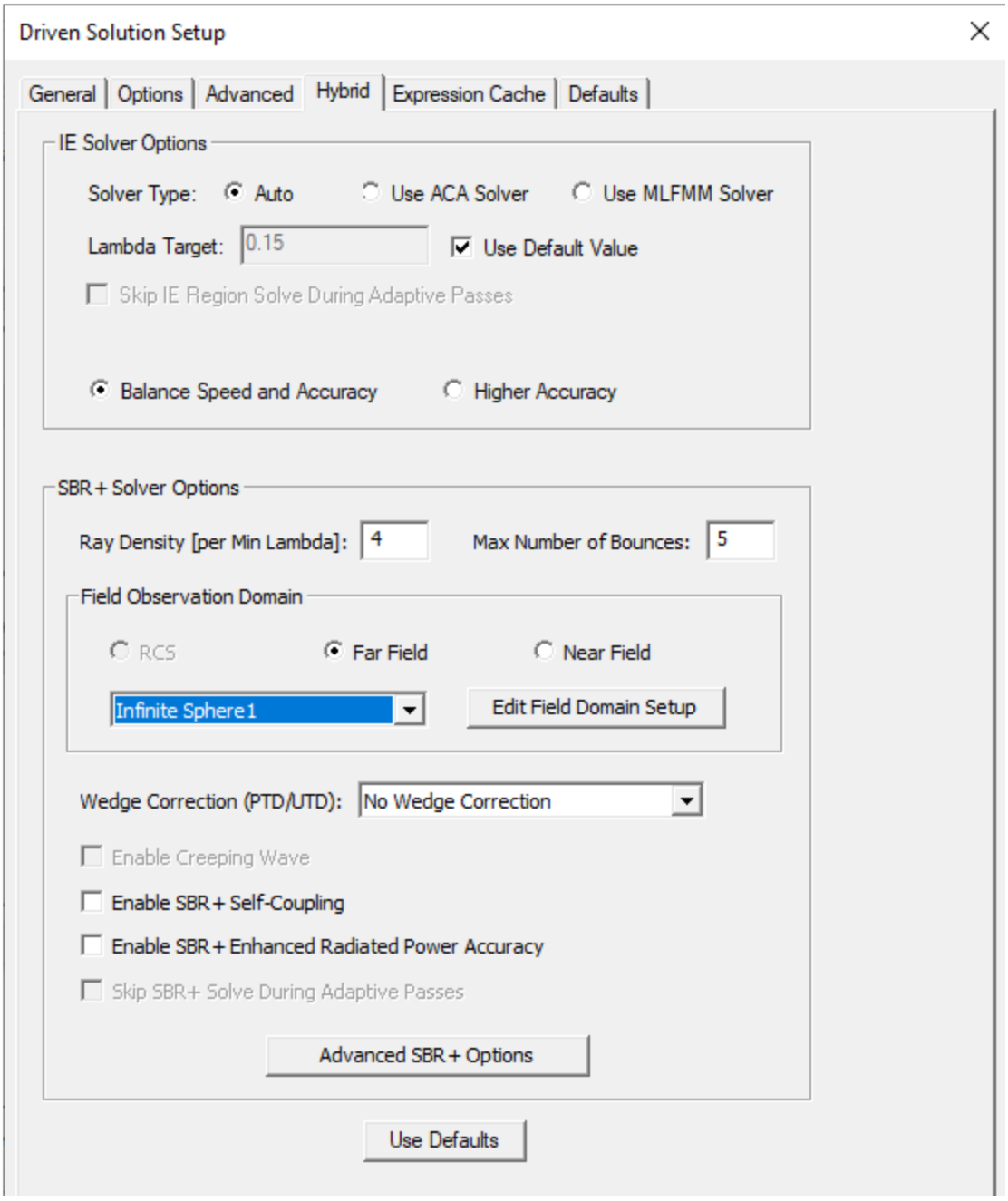


The dialog for **Hybrid Region** opens, showing the default name and type selection as IE Region. Because you may decide to change an assignment for IE Regions, and PO Regions, the dialog shows these types as radio button selections.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. When you **OK** the dialog box, the assignment appears in the Project Tree under Hybrid Regions.
3. SBR+ Solver Options are enabled in Solve setups for HFSS when an SBR+ Region is present.

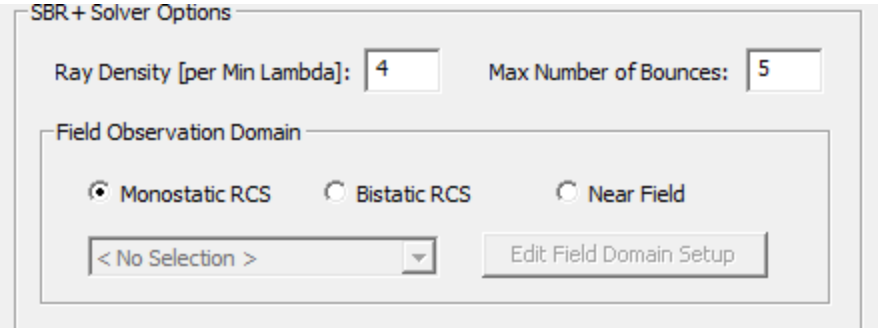


You can set Ray Density Per Wavelength and Maximum Number of Bounces, and if you have not done so, create or edit an infinite sphere setup for Far Field Observation. See Setting Hybrid Region Parameters for HFSS. You can also choose to Skip SBR+ Solve

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

During Adaptive Passes. SBR+ regions are not being mesh adapted and SBR+ solutions have no impact on field solutions on FEM or IE regions. However, SBR+ does impact stopping criteria in some cases such as coupling between two separate source antennas. Therefore, to speed up mesh adaption, you can choose to not solve SBR+ regions until source regions have converged in isolation. However, SBR+ does impact stopping criteria in some cases in the form of cache expression such as coupling between two separate source antennas or far field pattern. In such cases, SBR+ solve could not be skipped. Moreover, SBR+ solve is always launched when the maximum number of passes is reached regardless of source region convergence.

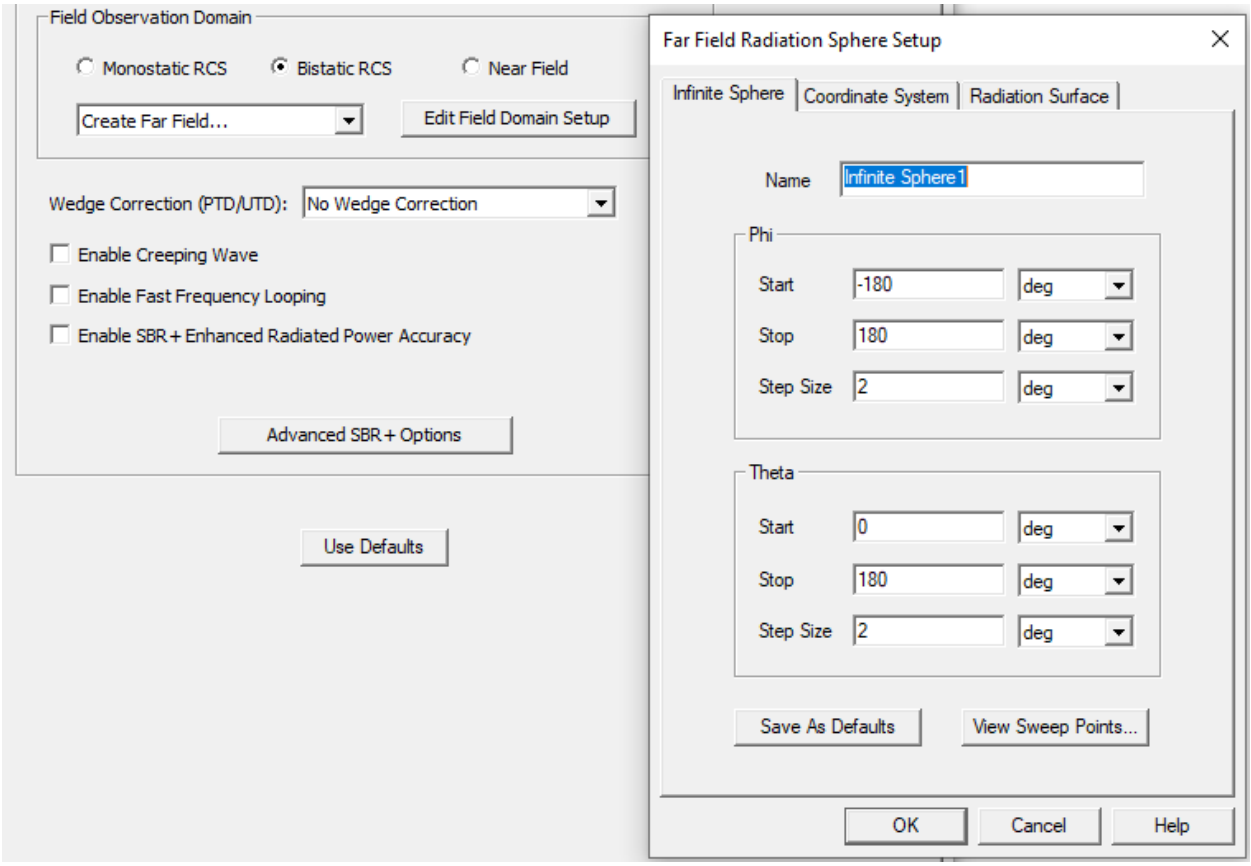
If you created an Incident Plane Wave for the design, the **Hybrid** tab lets you make an RCS Type selection for Monostatic or Bistatic.



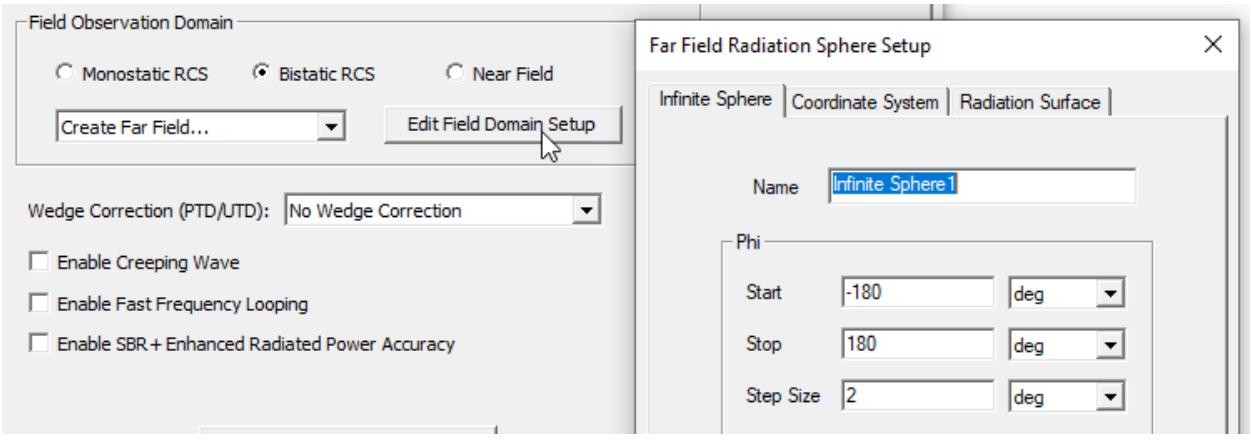
If you select Monostatic, the infinite sphere setup is disabled. In this case, the solver computes the scattered field in the direction of the plane wave. The Reporter will not require a geometry selection, and offers a range of Monostatic quantities for Monostatic RCS traces. See Generating Reports for Monostatic RCS.

If you select have assigned both an SBR+ region and an Incident Plane wave, and select Bistatic as the RCS Type, the Near Field and Far Field Observation domain fields are enabled, and the **Far Field Radiation Sphere Setup** dialog appears as follows, allowing you to create an infinite sphere for a Far Field or a rectangle for examining near fields. You can edit an existing Far Field Infinite Sphere or Near Field Rectangle setup by selecting from the drop down menu and clicking **Edit Field Domain Setup**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you have created one or more field observation domains, these are listed on the dropdown. You can edit an existing Far Field Infinite Sphere or Near Field Rectangle by clicking **Edit Field Domain Setup**.

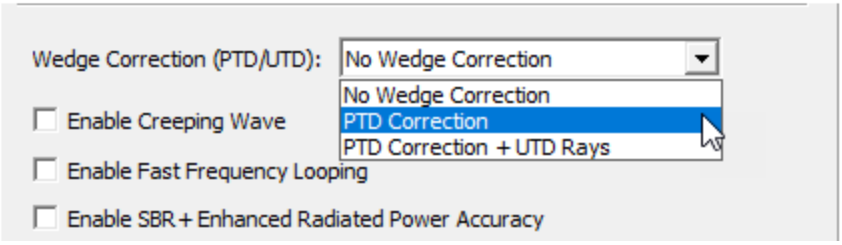


For Near Field rectangle setups, you can use the feature described in Overlay Contour Plot of Near Field on Rectangle.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Enable SBR+ Self Coupling computation capability for driven designs means that the diagonal S-param term takes into consideration the SBR+ scattering effect, and is more accurate. For a discussion of SBR coupling see, Select Tx/Rx Antennas for SBR+ Solutions.

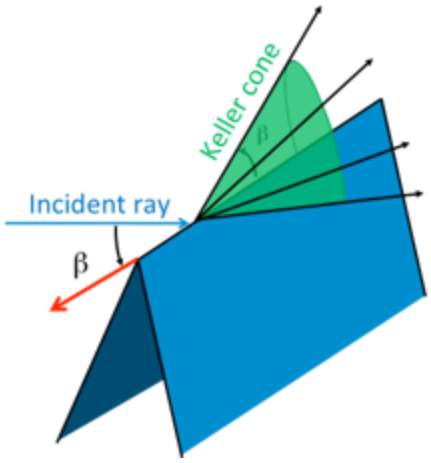
5. The PTD/UTD Simulation Settings allow for the inclusion of additional wedge diffraction phenomenology that can improve the accuracy of SBR+ simulations. You can opt out of using the PTD/UTD settings, or select PTD Correction or PTD Correction + UTD Rays. Selecting one of the PTD Correction options enables a field for specifying PTD Edge Density. This controls how densely wedges are segmented when establishing line of sight from the field source (for example, Tx antenna) to each wedge segment.



For SBR+ simulations, the Physical Theory of Diffraction (PTD) and Uniform Theory of Diffraction (UTD) features can account for additional phenomenology not well predicted by plain SBR due to truncation of uniform Physical Optics (PO) currents at sharp angular discontinuities (“wedges”) on metallic surfaces and blockage of SBR’s Geometrical Optics (GO) rays. PTD is a numeric correction to the scattered fields radiated by PO currents near wedges. UTD launches bundles of edge-diffraction rays from directly illuminated portions of each wedge along the Keller cone. Once launched, the UTD rays behave exactly like regular SBR rays, propagating according to GO and painting PO currents at each bounce that contribute to the scattered field. The UTD rays often illuminate portions of the SBR scattering geometry that are never reached by SBR GO rays launched directly

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

from the field source.



The PTD and UTD wedge features are only deployed for metallic wedges with line-of-sight visibility from the source (Tx) location. If either adjacent surface of the wedge is non-PEC and not within the tolerance for PEC-like, or if the entire edge segment is not visible to the source, the wedge will be ignored in the SBR+ simulation and for visual ray tracing (VRT).

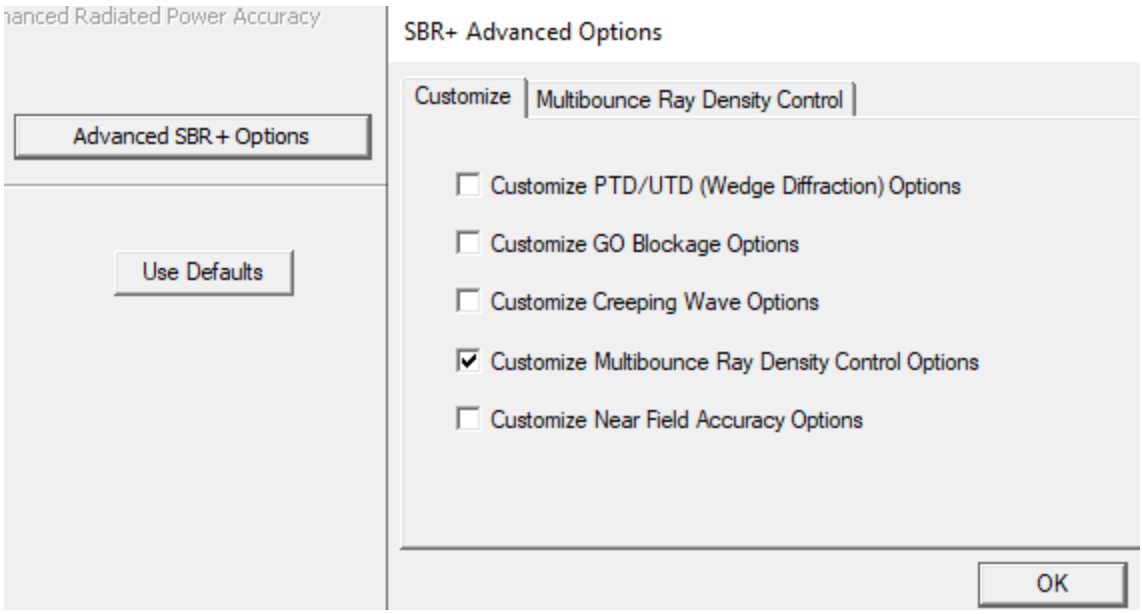
6. Advanced Option for NF Accuracy Settings for SBR+

The NF Accuracy settings (Beta feature, NF Accuracy Controls for SBR+) provide access to methodology settings relating to cases where Tx antennas, Rx antennas, and near-field observation points are in proximity to the scattering geometry (i.e., the platform CAD model). When this feature is switched off, HFSS SBR+ uses default near-field accuracy settings that should be adequate for most situations, including close-proximity Tx/observer conditions that this feature is designed to tune.

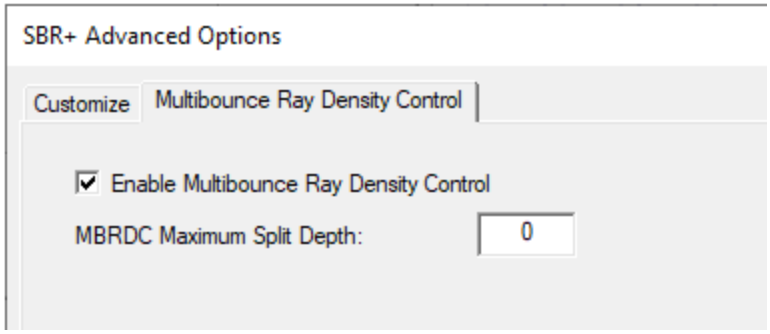
7. Advanced Option for Multibounce Ray Density

By option you can select the **Advanced SBR+ Options** and enable Customize Multibounce Ray Density Control Options.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can then select the **Multibounce Ray Density Control** tab.



Use the checkbox to **Enable Multibounce ray density** and enter the maximum number of subdivisions to be used in the solve. Enabling the feature may increase the solve time.

The **MBRDC Maximum Split Depth** field value increases the number of spawned ray tracks on each ray bounce. The MBRDC algorithm is designed to achieve the ray density per wavelength criterion at any bounce depth. If a footprint at depth *N* in a ray track does not satisfy the ray-density criterion, the MBRDC algorithm attempts a refined ray shoot from an earlier bounce, just for the ray in question. The size of the triggering footprint informs the level of refinement. If the refined shoot is not successful in achieving the desired footprint size, or new footprints are too large, the MBRDC algorithm attempts further, recursive refinements. The **MBRDC Maximum Split Depth** setting limits how aggressively the refinement algorithm is applied by specifying the maximum number of split recursions. For this reason, linear

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

increases in Maximum Split Depth can yield a geometric progression in the total number of rays shot and associated solution time.

When both MBRDC and UTD are enabled, first-bounce UTD rays and bright points on wedges are recomputed during the MBRDC splitting process, but only the original raytrack is rendered up to the 1st bounce. Similarly, length-based ray filters are applied to the original UTD initial ray track up to 1st bounce rather than using the information in the additional MBRDC tracks. This is a known limitation.

8. Advanced Option for Geometrical Optics (GO) Blockage

SBR+ normally applies a physical optics (PO) blockage model where blockage effects emerge from partial cancellation of the incident field by the scattered field. PO blockage not only occurs in connection with the incident field from the Tx antennas, but also within the scattered field itself when reflected fields from the previous bounce are diminished by a blocking surface along the geometrical-optics (GO) reflection path at the current bounce by evaluating the scattering contribution of the incident ray field at that bounce.

In cases of significant blockage by a large obstruction, the PO blockage model requires accurate currents over the extent of the obstruction. Sometimes, this is hard to achieve in the ray tracing approximation. For example, the obstruction may not be well illuminated by direct rays from the Tx antenna or multi-bounce rays after an earlier reflection. An alternative is to use a GO blockage model where scattered field contributions are added to an observation point subject to a line-of-sight (LOS) blockage check performed by the ray tracer. In some cases, this is more accurate than the default PO blockage formulation, while in others it is less accurate. GO blockage also entails a non-trivial computational cost, as the blockage check with the ray tracer must be performed for each observation angle (or point) for each ray hit point.

In addition to GO Blockage for SBR and UTD ray tracks, this feature is also available for PTD and CW.

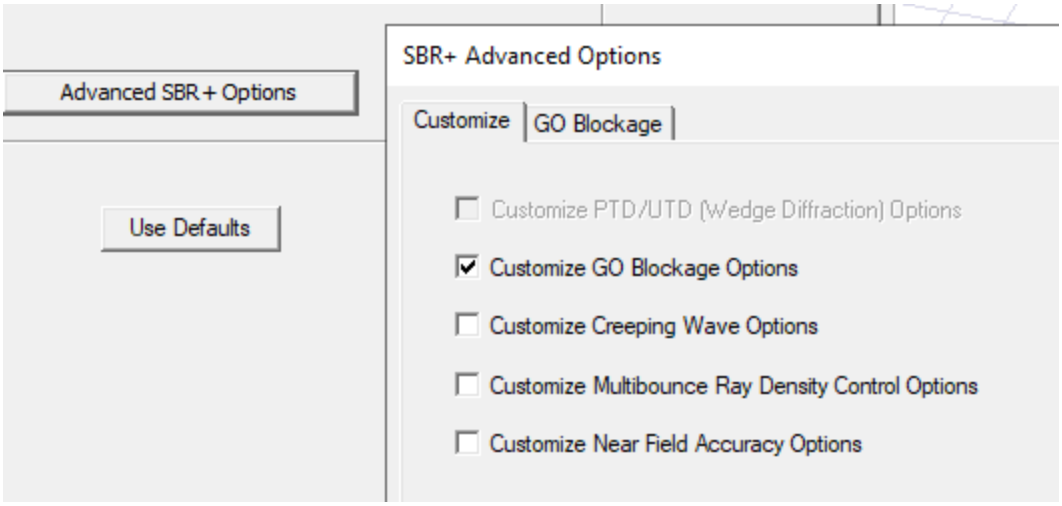
Limitations for GO Blockage:

GO Blockage formulation is suited specifically for SBR+ ray tracks, including UTD ray tracks. While GO Blockage for CW and PTD has been implemented for the first time in this release, results can be surprising and/or inaccurate, as a limitation of the methodology.

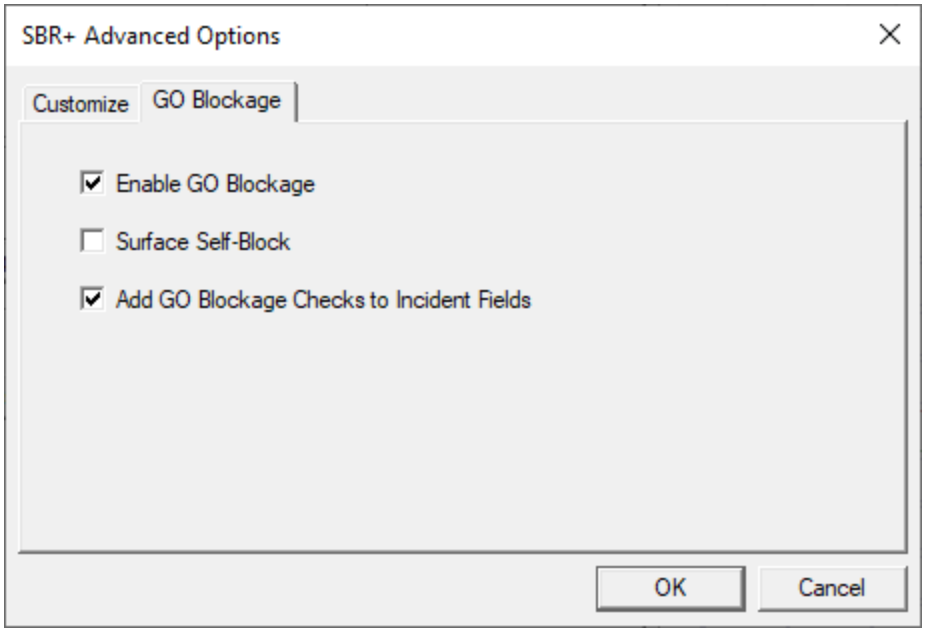
Summary Workflow for GO Blockage:

Click the **Advanced SBR+ Options** button to open the **SBR+ Advanced Options** window. On the **Enable** tab, Check GO Blockage Options to enable the **GO Blockage** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



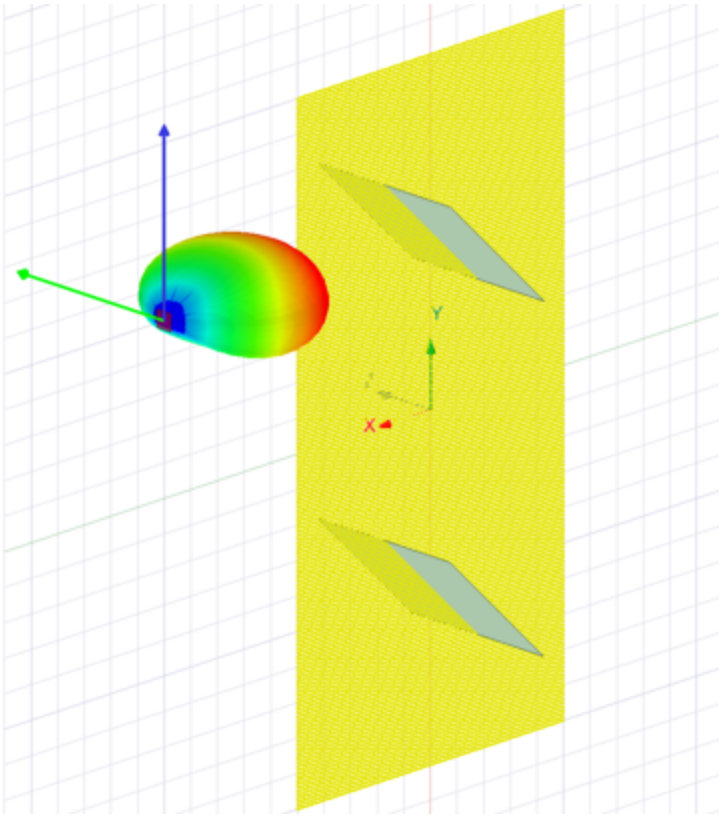
On the **GO Blockage** tab, you have check box options to enable or disable GO Blockage, Surface self-block, and GO Blockage Checks for Incident Fields..



Surface Self-Block: if enabled, the surface where the footprint is radiated from is used for blockage check, that is, anything below the surface with respect to the incoming ray is not in line of sight. If the setting is disabled, the footprint surface is not used for blockage checks (anything below the surface with respect to the incoming ray is in line of sight), but other surfaces are used for blockage.

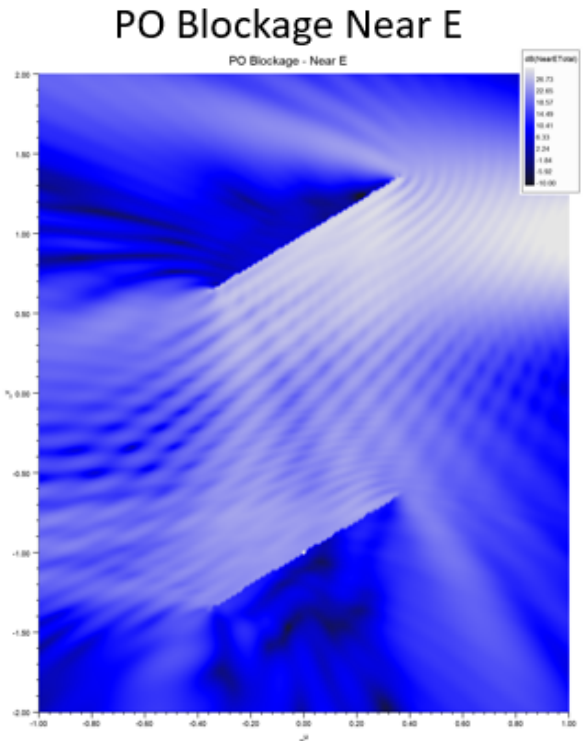
For example, consider a parametric beam antenna illuminating a series of plates.

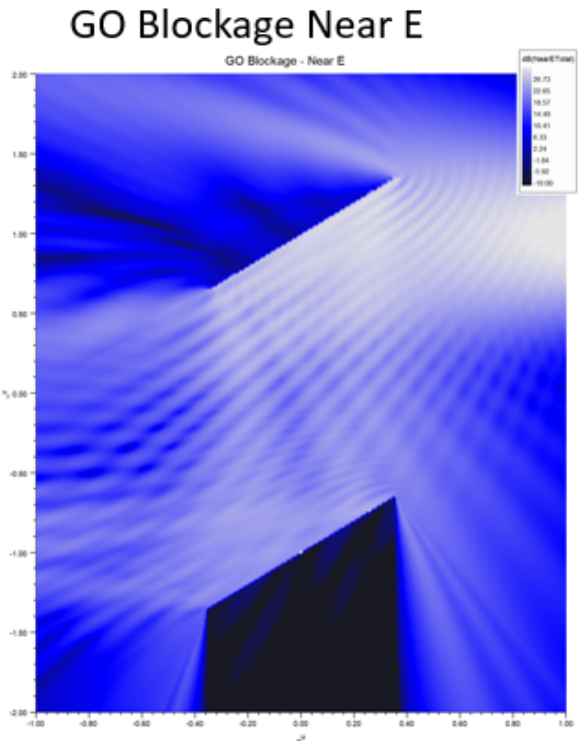
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the plots, the shadows behind the plates are much more pronounced with GO Blockage enabled, but the fields have non-physical discontinuities.

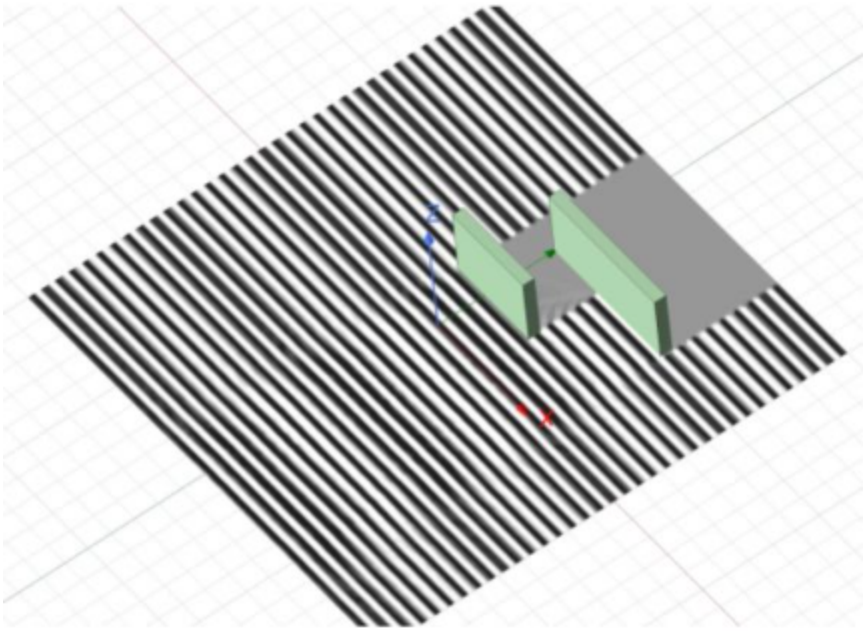
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



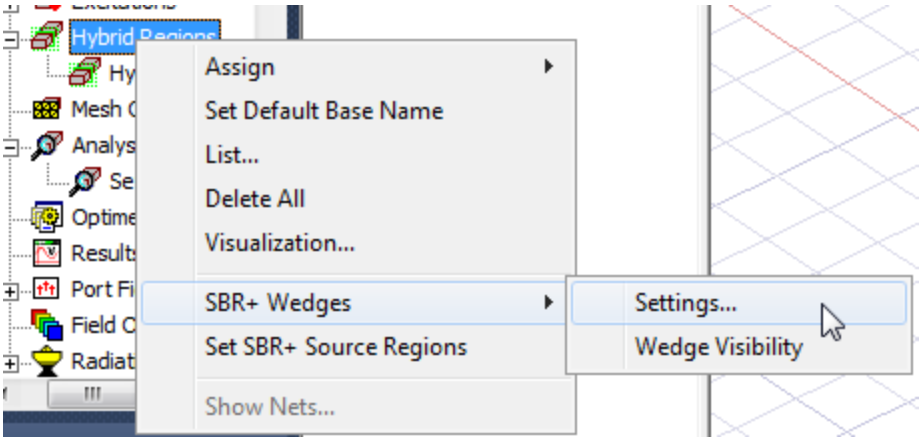


Add GO Blockage Checks to Incident Fields: if enabled, GO blockage effects will also be calculated for the incident field via line-of-sight checks from sources to field points, effectively computing a direct field in the Geometrical Optics sense. For example, the figure below shows the total E-field of a planewave incident on PEC bodies with this option enabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



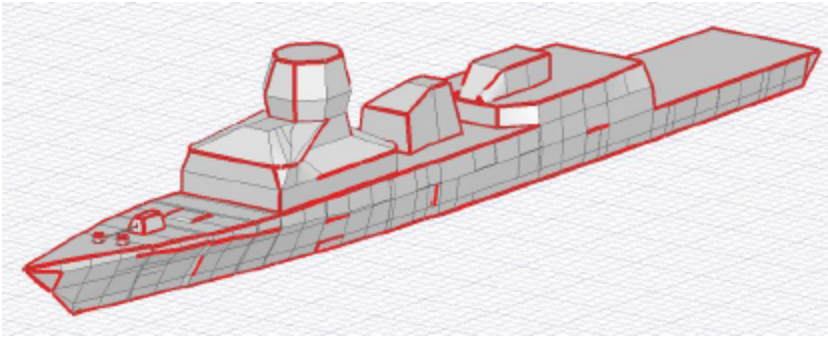
9. Wedge settings for the SBR+ simulation with PTD/UTD are only accessible for an SBR+ design. Wedges are based on theAutomatic Wedge Generation settings, optional Interactive Wedge Editing, and the initial mesh. The initial mesh is automatically generated on launch of the SBR+ Wedge Settings dialog box, if needed. The dialog will not launch if an initial mesh is not available and cannot be generated. The **SBR+ Wedge**s submenu can be accessed through a right-click the Hybrid Regions icon in the Project tree.



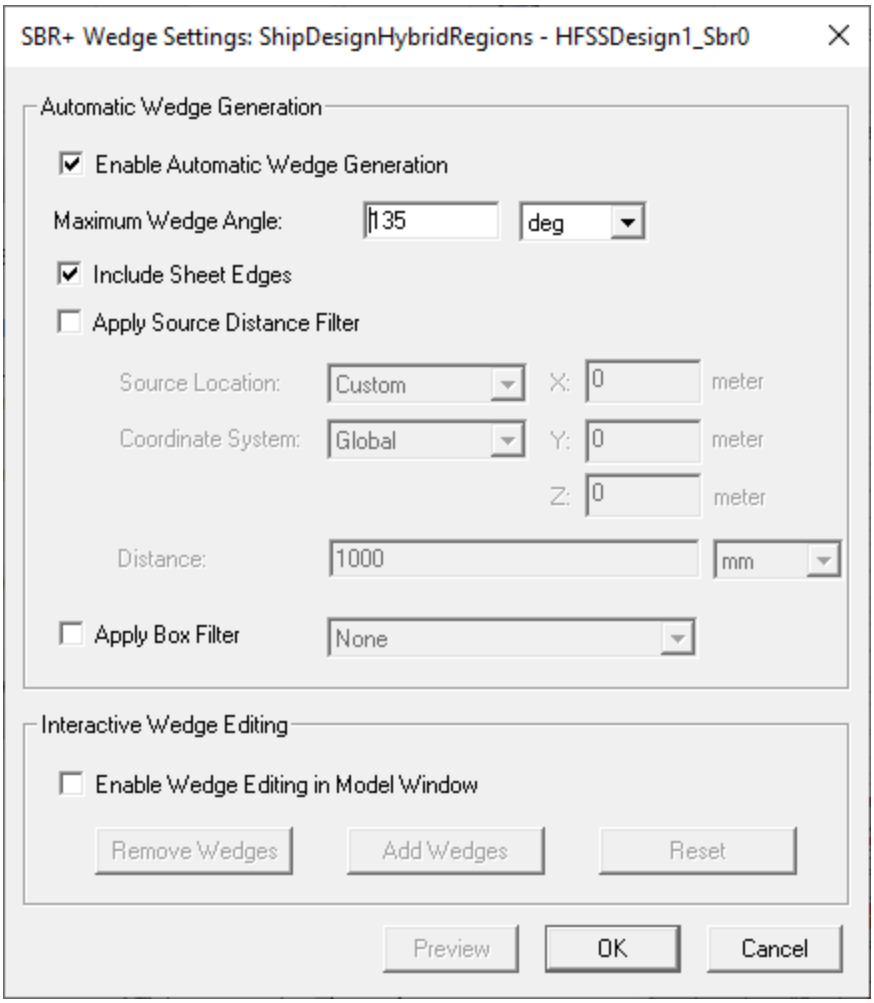
The submenu contains the option for opening the **SBR+ Wedge>Settings** dialog and a checkable option for enabling or disabling **Wedge Visibility**. If visible, wedges are shown as thick red lines in 3D model window for automatically generated wedges, and as blue

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

lines for interactively edited wedges. Visualization is turned off when there is design edits that invalidate the initial mesh.



The **SBR+ Wedge Settings** dialog contains the Automatic Wedge Generation parameters and the Interactive Wedge Editing feature.

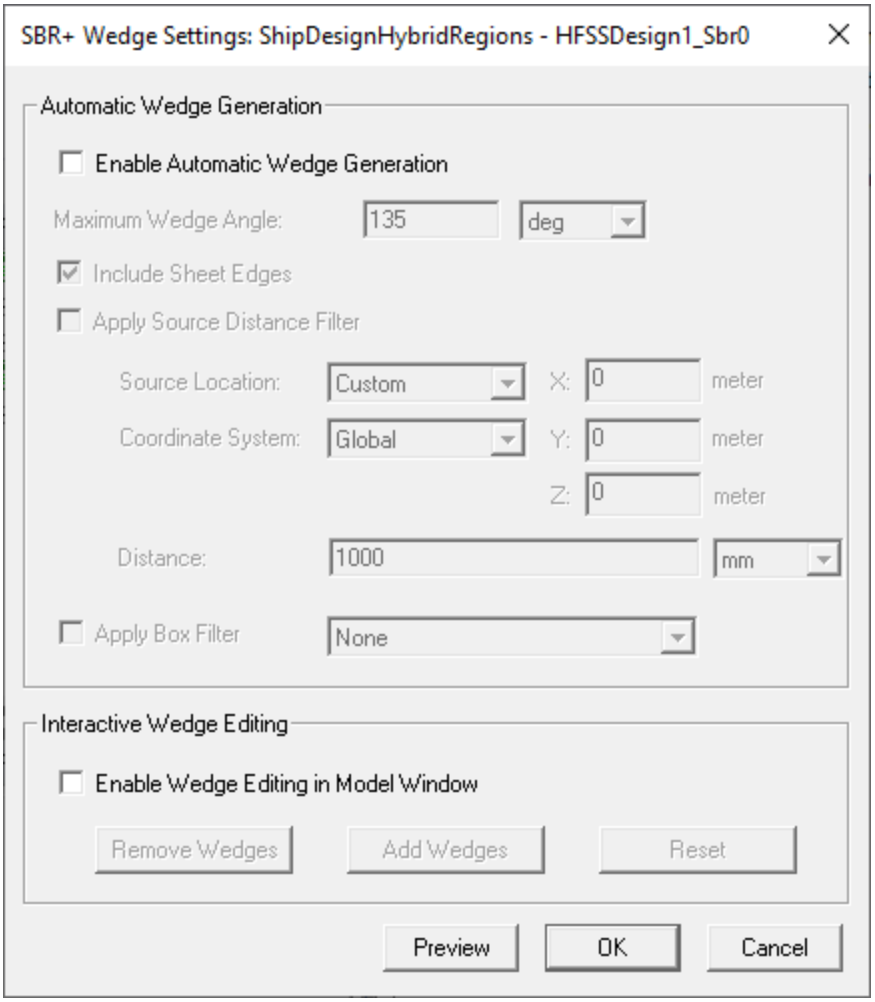


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Wedge Generation parameters are used to determine the candidate set of wedges to be included, with optional filters that can restrict their location on the geometry. The final set of wedges to be used in the SBR+ simulation is the intersection of all the wedge generation criteria, and the effects of any optional Interactive Wedge Editing. complete flexibility in specifying where the wedges are to be used for the simulation. This is important since adding many wedges on very large geometry structures (aircraft, vehicles, etc.) can significantly impact the SBR+ simulation time.

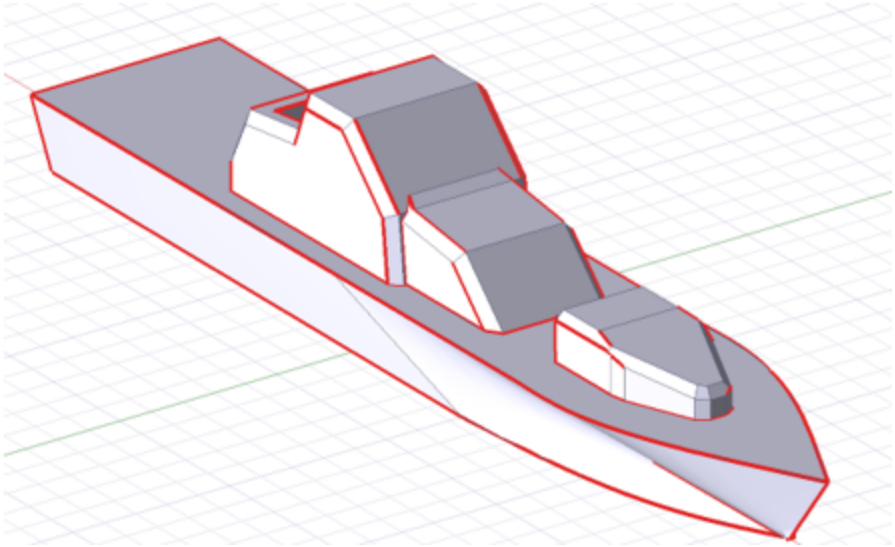
Enable Automatic Wedge Generation.

By default, the checkbox is enabled, allowing access to the automatic features and parameters. If you un-check it, the automatic features are disabled. You can independently enable Wedge Editing.



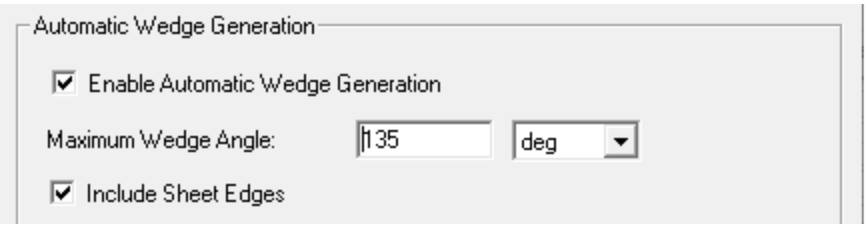
Wedges added through Automatic Wedge Generation are displayed in red.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Maximum Wedge Angle:

An angular criteria can be specified for the maximum wedge angle to help filter the desired set of wedges - default value is 135 deg. This wedge angle ranges from minimum 0 deg to maximum 180 deg. All wedges are included that are less than or equal to the specified maximum angle. An angle equal to 0 deg would be created by a “collapsed” wedge (like a closed hinge), while an angle equal to 180 deg would be created by a completely “open” wedge (like a fully open hinge). You can assign a design variable to Maximum Wedge angle, but you cannot create a design variable within the dialog.



Include Sheet Edges:

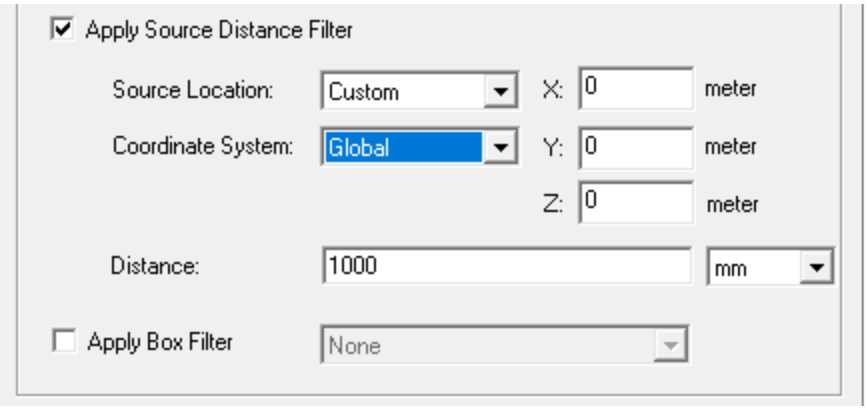
You can choose to include or exclude the edges from sheets (non-connected edges, also known as “knife edges”). The default value is true which includes these edges. Sheet edges are triangle edges with no adjacent/connected triangle along that edge. In the physical object we are modeling these may represent extremely thin surfaces such as metal “fins”, etc.

Optional parameters:

Apply Source Distance filter:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A “source distance filter” allows you to specify that you want to include wedges within a finite 3D distance from a point (the “source”).



You can choose model Points from drop down as source location, or specify the absolute X, Y, Z in model units. X, Y, Z can be specified in global coords or relative to a custom CS. When you select a geometry Point, its coordinate system and X, Y, Z are displayed.

The Distance default is 1 <model unit>. You can assign an existing design variable to the Distance. Default value is false (no Source Distance Filter).

Apply Box Filter:

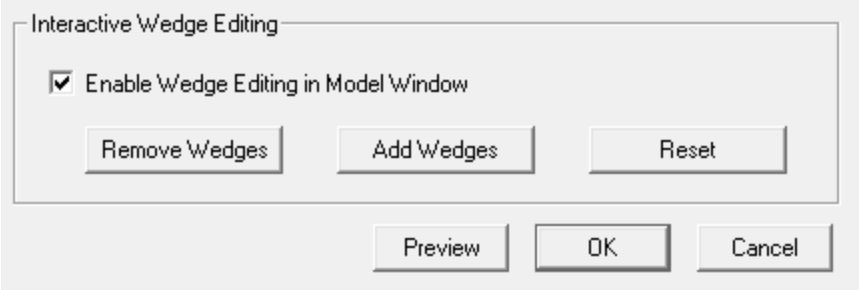
You can select any non-model box that is aligned with the Global Coordinate System axes via a drop-down menu, within which wedges will be included. Default value is false (no Box Filter).

Optional Interactive Wedge Editing

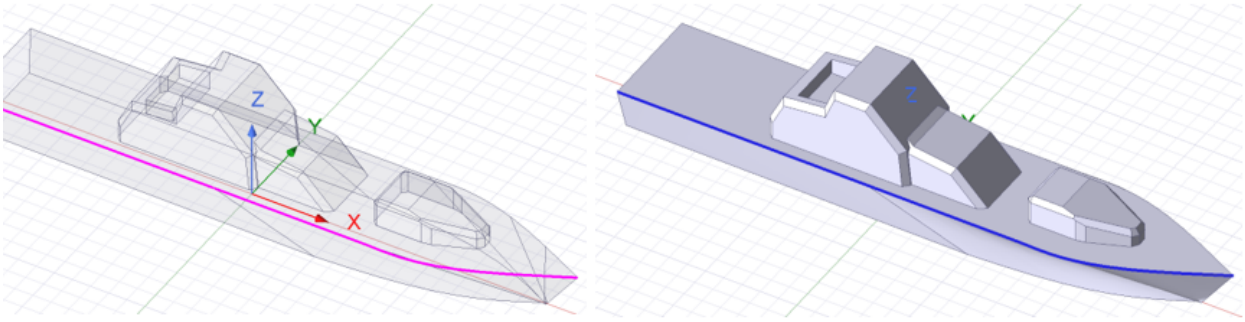
This feature provides complete flexibility in specifying where the wedges are to be used for the simulation. This is important since adding many wedges on very large geometry structures (aircraft, vehicles, etc.) can significantly impact the SBR+ simulation time.

Checking the Enable Wedge Editing the Model Window box enables the buttons for removing wedges from selected geometries (faces, objects, etc.) , adding wedges to selected geometries, and resetting edits.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After you select a geometry and **Add Wedges**, the interactively added wedges appear in blue.



If you select **Remove Wedges**, all wedges on the selected geometry objects are removed. These can be automatically generated wedges or interactively added wedges.

Click **Reset Wedges** to undo all interactive wedge editing operations. A confirmation dialog appears. Press Yes to reset all wedge editing operations. The reset button is disabled if no wedge edit operations are present.

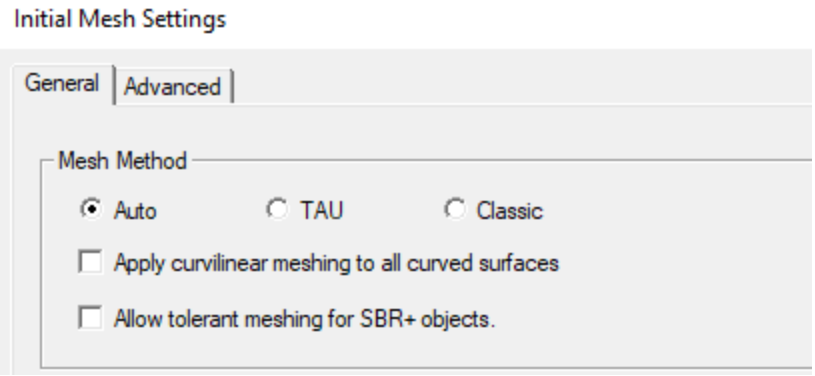
Current Limitations on PTD/UTD Wedge Generation/Editing

Situations where wedges cannot be generated:

Light Weight Imported STL Geometries: STL models imported as light weight geometries are currently not supported for interactive wedge editing (as of R24.1).

In the Initial Mesh Settings for SBR+ Solution type or for HFSS with Hybrid and Arrays that include SBR+ regions, you can turn off tolerant meshing.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



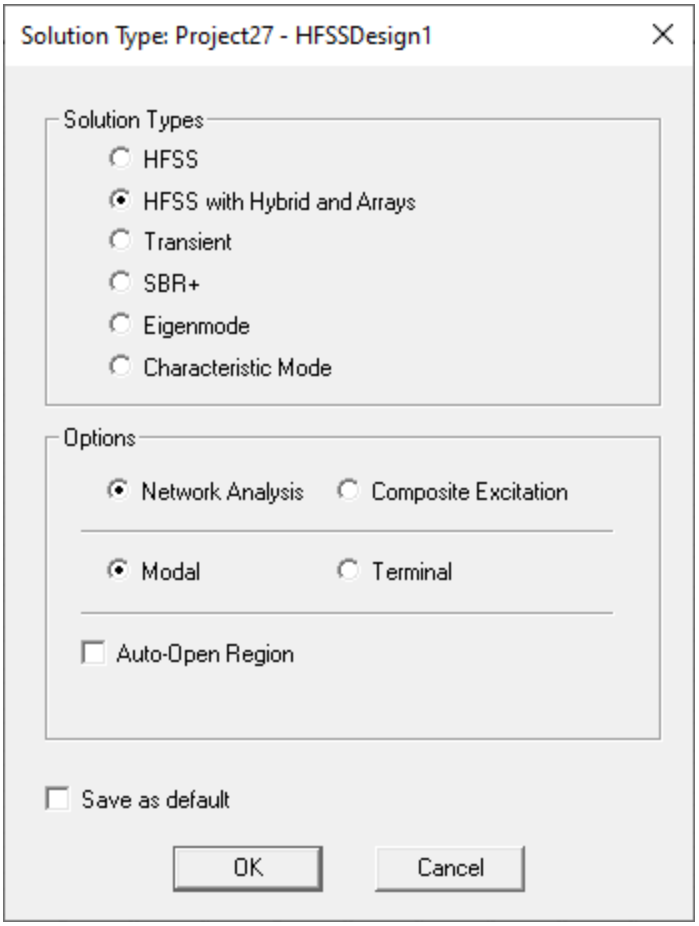
In this case you should consider Current Limitations on PTD/UDT Wedge/Generation Editing.

Configure Simulation with SBR+ Antenna Geometry Blockage (HFSS with Hybrid and Array Solution Type):

The typical configuration begins with a fully integrated model with all geometry. You then select the near antenna region including any nearby geometry interacting with the antenna for analysis by HFSS – either by identifying these objects by hybrid region assignment, or by copying them out to a separate source antenna design to be solved with the HFSS solver, etc. The nearby antenna geometry could even be something different than what is used for the integrated model, such as a flat plate or other structure for considering the nearby antenna interactions. This source antenna design should likely be enclosed in a FE-BI region to allow for easier downstream processing, whether we are in a hybrid or SBR+ solution type. If you use a separate source antenna design, you can re-introduced it to the target design with the full geometry model by creating a 3D component for an antenna that is a linked design to the source design. This means you no longer need to designate any “Model Blockage” when setting up the linked design. You do this as a separate step outside creation of the linked design by selecting geometry from the SBR+ region to be designated as “Model Blockage” associated with a particular source antenna. Note there is no restriction on what objects and/or materials can be designated as “model blockage” geometry. Also note that for either solution type, SBR+ or hybrid HFSS-SBR+, that this selection will be from the available SBR+ region geometry objects, which is different than the previous workflow. Once you select these antenna blockage geometries for each source antenna, HFSS with SBR+ can solve the combined target SBR+ design.The preferred work flow is this.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

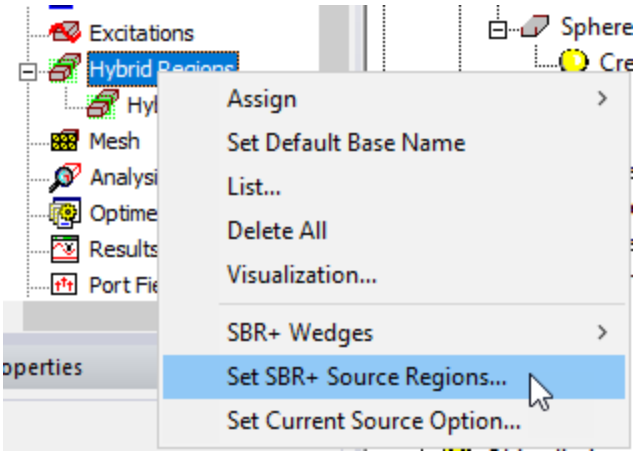
1. Create/open an design with HFSS with Hybrid and Arrays solution type.



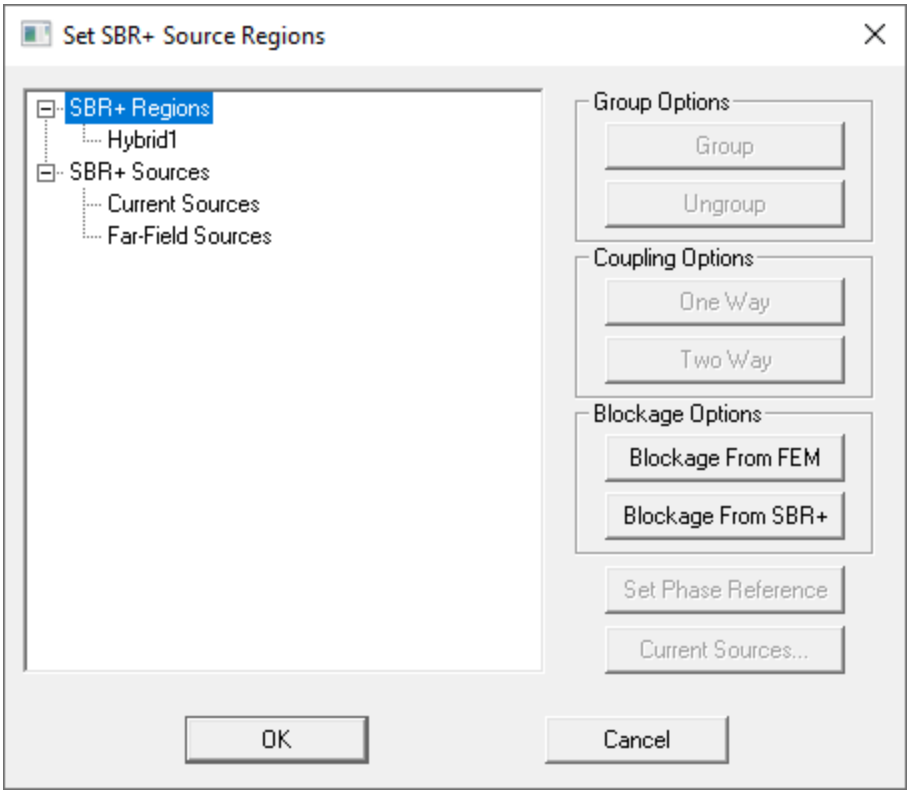
2. Configure the source excitation, including adding or creating any nearby geometry.
3. Create a hybrid FE-BI region to enclose the source excitation and all the nearby geometry objects to be part of the HFSS simulation.
4. Add all the additional geometry objects that will be part of the hybrid analysis, configuring all materials/boundaries as desired.
5. Create/assign a hybrid SBR+ region to all the desired geometry objects to the design.
6. If you want to designate multiple SBR+ geometry objects as blockage, select them in the 3D Modeler window and create an Object List by selecting **Modeler > List > Create > Object List**. You may want to name the list appropriately. If only a single object is to be designated as a geometry blockage, skip this step..

   Only object lists that contain exclusively SBR+ can be selected for antenna blockage.
7. Open the dialog to configure SBR+ source regions (which includes antenna blockage) via the menu option for **Hybrid Regions > Set SBR+ Source Regions...**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

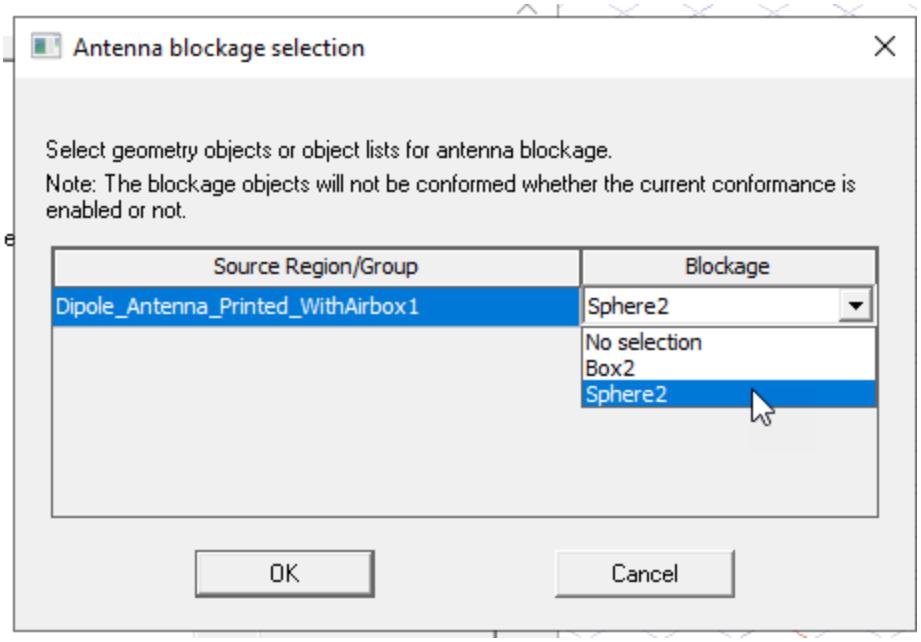


This opens the **Set SBR+ Source Regions** dialog.



Click the **Blockage from SBR+** button to bring up the antenna blockage dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

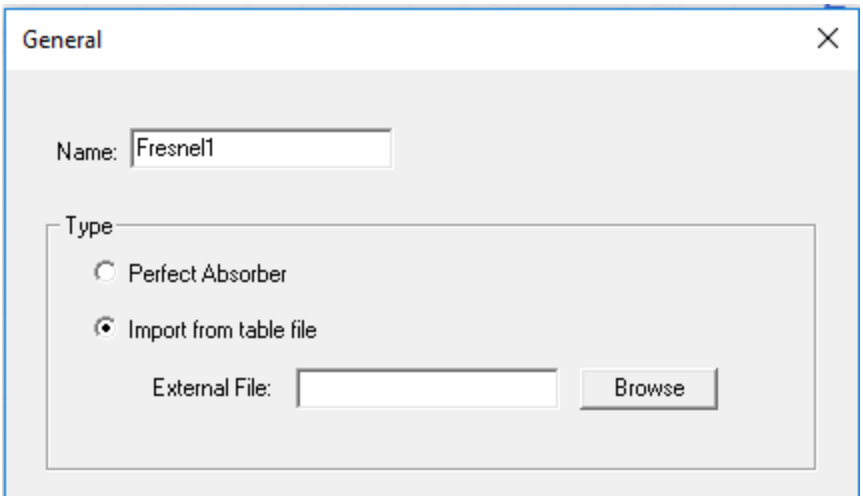


8. For each antenna source, use the “Blockage" column to select the geometry blockage from the dropdown. Each cell in the column contains a dropdown list of all geometry objects and all user created object lists.
9. Click **OK** to save the configuration.
10. Create an **Analysis Setup**.
11. Run the simulation.

Assigning (SBR+) Fresnel Table Boundaries

The **Assign Boundary>Fresnel (SBR+)** command allows you to specify either a perfect absorber or to import a Fresnel table file that describes reflection and transmission coefficients.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Fresnel coefficients are based on the assumption of plane-wave incidence and reflection and represent the ratio of the reflected field to the incident field. Ray bundles can represent complex wavefronts, and each ray represents a sample of that wavefront that is assumed to be locally planar. Provided that the wavefront is not too irregular, and the coatings are not too thick compared to their lateral extent, the Fresnel coefficients applied in conjunction with individual rays provide a good approximation of the reflection of an arbitrary wavefront off a surface

When an SBR+ boundary is configured as an Impedance Boundary or a Layered Impedance boundary, a table of Fresnel coefficients is internally generated for use during the SBR+ ray-tracing simulation. The Fresnel (SBR+) **Import from table file** option lets you directly load the Fresnel tables from an external file. You can develop these from a number of sources, including analytic formulas, specialized codes for simulating the boundary (e.g., HFSS for simulating a frequency selective surface), or measurements. Two types of boundary tables are supported:

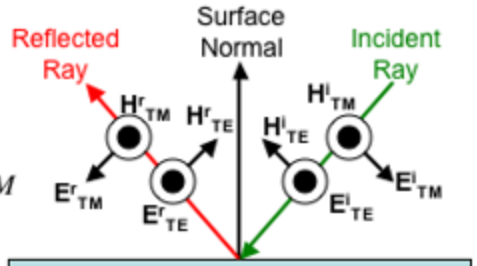
- reflection coefficients
- reflection + transmission coefficients

Tables of reflection + transmission coefficients are explained later.

Fresnel coefficients depend on the polarization of the incident plane wave. There are two unique, orthogonal polarizations: transverse electric (TE) and transverse magnetic (TM). HFSS-SBR+ adopts the convention given in the diagram below. In this convention, PEC metal has a TE reflection coefficient of -1 and a TM reflection coefficient of +1. In general, reflection coefficients are complex quantities that depend on incident angle and frequency

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$E^r_{TE} = R_{TE} E^i_{TE}$$
$$E^r_{TM} = R_{TM} E^i_{TM}$$



Reflective Coating Described by Fresnel Reflection Coefficients

When a ray hits a surface, the solver determines its incidence angle relative to the surface normal. The incident ray tube fields are decomposed into TE and TM components. The solver then looks up the TE and TM reflection coefficients for that angle, whether internally generated or provided by an external table as described here. These are applied to the respective incident ray field components to determine the TE and TM components of the reflected ray field. This, in turn, influences the surface currents and scattered fields generated at the current bounce and subsequent bounces.

Fresnel Reflection Table File Format

The Fresnel reflection table file format is documented in the header comments in the example below. This first example is for a table defined at a single frequency, which means it will be used at all frequencies during a simulation. A multi-frequency example is provided later for a reflection/transmission table, but since their formats are nearly identical, the same encoding can be used for a reflection-only table

```
# Reflection table file.
# Blank lines and lines beginning with are ignored.
# The following key is critical in distinguishing between a
reflection table and a reflection/transmission table.
ReflTab1e
# Incident angle, theta, is measured from the normal to the surface .
# In the table, theta is assumed to vary from C (head-on) to 90 deg
(grazing) .
#The density cf angular sampling is specified by the number cf theta
steps.
# In this example, there is a sample every 1 deg.
# <num theta step> number of theta points - 1
90
# Frequency domain
```

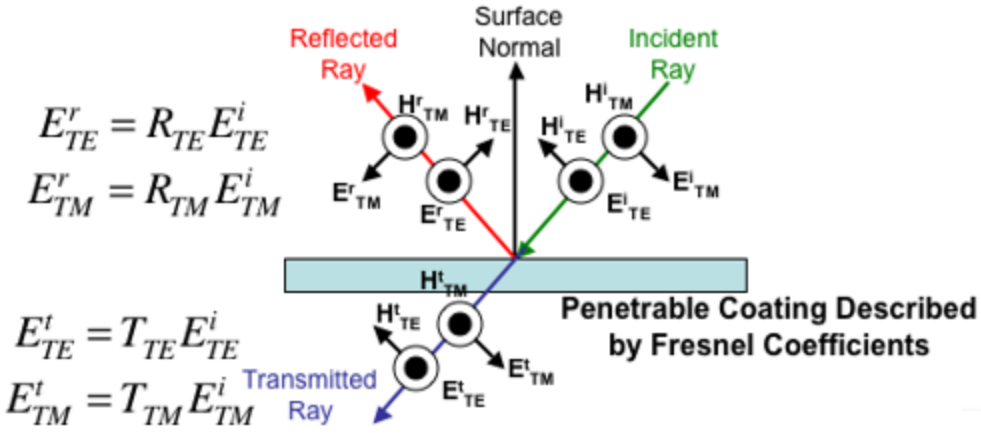

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
# Mono freq. table for 1.000000 GHz data type
# <freq_domain_type> = { MonoFreq, Multi Freq}
MonoFreq
# Data section follows. Frequency loops within theta.
# <rte_rl> <rte_im> <rtm_rl> <rtm_im>
-3.81966e-01 0.00000e+00 3.81966e-01 0.00000e+00
-3.82C18e-01 0.00000e+00 3.81914e-01 0.00000e+00
-3.82174e-01 0.00000e+00 3.81758e-01 0.00000e+00
-3.82435e-01 0.00000e+00 3.81497e-01 0.00000e+00
-3.82799e-01 0.00000e+00 3.81132e-01 0.00000e+00
-3.83269e-01 0.00000e+00 3.80662e-01 0.00000e+00
-3.83843e-01 0.00000e+00 3.80086e-01 0.00000e+00
-3.84523e-01 0.00000e+OC 3.79403e-01 0.00000e+00
-3.8S3C9e-01 0.00000e+0C 3.78613e-01 0.00000e+00
-3.86201e-01 0.00000e+0C 3.77715e-01 0.00000oe+00
-3.87200e-01 0.00000e+0C 3.76708e-01 0.00000e+00
:
:
-9.16560e-01 0.00000e+00 -6.42464e-01 0.00000e+00
-9.32634e-01 0.00000e+00 -7.03164e-01 0.00000e+00
-9.49016e-01 0.00000e+00 -7.68667e-01 0.00000e+00
-9.65704e-01 0.00000e+00 -8.39528e-01 0.00000e+00
-9.82699e-01 0.00000e+00 -9.16389e-01 0.00000e+00
-9.99900e-10 0.00000e+O0 -9.99500e-01 0.00000e+00
```

Penetrable boundaries, as are encountered in the case of canopies and radomes, should be described by both reflection and transmission coefficients. Likewise, arbitrary boundaries that are penetrable can be entered as a table.

When a ray hits a penetrable coatings, it generates both a reflected ray and a transmitted ray, as shown in the diagram below, which also extends the polarization convention of the previous diagram. In the diagram below, it is intentional that the transmitted ray is shown as starting from the top of the surface, rather than the bottom. In the modeler, CAD models can represent infinitely thin surfaces. The effect of boundary thickness is entirely captured in the reflection and transmission coefficients. Hence, in creating a reflection + transmission table input, it is critical to do so such that the phase reference of the transmission coefficient is the surface nearest the incident ray. That way, the transmitted ray will include the correct phase delay given that it is launched from exactly the same point as where the incident ray hits the surface

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following example reflection + transmission table is very similar in structure to the reflection-only table. The only difference is that each line in the table has four extra numbers to represent the complex TE and TM transmission coefficients. This particular example also shows how to define a frequency dependent table of coefficients

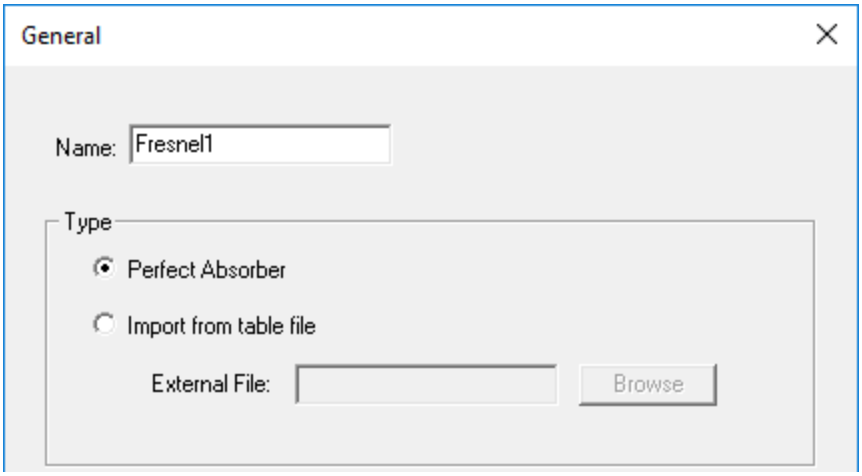
```
# EMA3D native format reflection/transmission table file
# The following key is critical in distinguishing between a
reflection table
# and a reflection/ transmission table.
RTTable
# <num theta step> = number of points-1
90
# Frequency domain
# In this example, the frequency samples are from 0.5 to 4 GHz in
20MHz steps
# MultiFreq <freq_start_ghz> <freq_stop_ghz> <num_freq_steps>
MultiFreg 0.5000000 4.000000 175
# Data section follows. Frequency loops within theta
# <rte_rl> <rte_im> <rtm_rl> <rtm_im> <tte_rl> <tte_im> <ttm_rl>
<ttm_im>
-6.23449e-01 1.00978e-04 6.23449e-01 -1.00978e-04 -5.88718e-01 -
3.66763e-01 -5.88018e-01 -3.66763e-01
-6.12741e-01 7.99650e-02 6.12741e-01 -7,99650e-02 -6.13162e-01 -
3.29510e-01 -6.13162e-01 -3.29510e-01
-5.78215e-01 1.54566e-01 5.78215e-01 -1.54566e-01 -6.44759e-01 -
2.91524e-01 -6.44759e-01 -2.91524e-01
:
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
:
-9.99927e-01 2.21130e-05 -9.99629e-01 9.58350e-05 -1.85062e-05 -
7.37198e-05 -1.07270e-04 -3.64781e-04
-9.99920e-01 2.99358e-05 -9.99596e-01 1.33667e-04 -2.89748e-05 -
7.33898e-05 -1.59538e-04 -3.61020e-04
```

Assigning (SBR+) Perfect Absorber Boundaries

The **Assign Boundary>Fresnel (SBR+)** command allows you to specify a perfect absorber.



Perfect Absorber: Incident SBR+ rays are absorbed by surfaces with this boundary, and no reflection or transmission rays are generated.

This does not imply that there is no radiation from surface currents on perfectly absorbing boundaries. The perfect absorber still generates very strong forward scattering. This scattering is associated with the incident GO ray fields at the surface. These ray fields are converted into equivalent currents at the surface that are radiated to observation angles and receiving antennas in any direction (including the backward direction); it tends to be strongest in the forward direction. At the same time, since the reflected ray has zero field strength (and therefore no reflected ray is traced), the equivalent surface currents associated with the reflected ray are also zero, so scattering in the backward half-space is weak.

The Fresnel (SBR+) **Import from table file** option lets you directly load the Fresnel tables from an external file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning a Dielectric or Conducting Material to Volumetric Region

Both materials and boundaries can be assigned to solid objects in HFSS SBR+. Materials can be dielectric with possibly a frequency-independent dielectric and/or magnetic loss tangent, or conductive, with any conductivity $\sigma > 0$ S/m. Every boundary type already supported in SBR+ is supported for solid objects, too. For one-sided boundaries, including PEC, PMC and Perfect Absorber, the boundary overrides the material assignment. Two-sided boundaries and materials are used together and translated to a proper SBR+ coating.

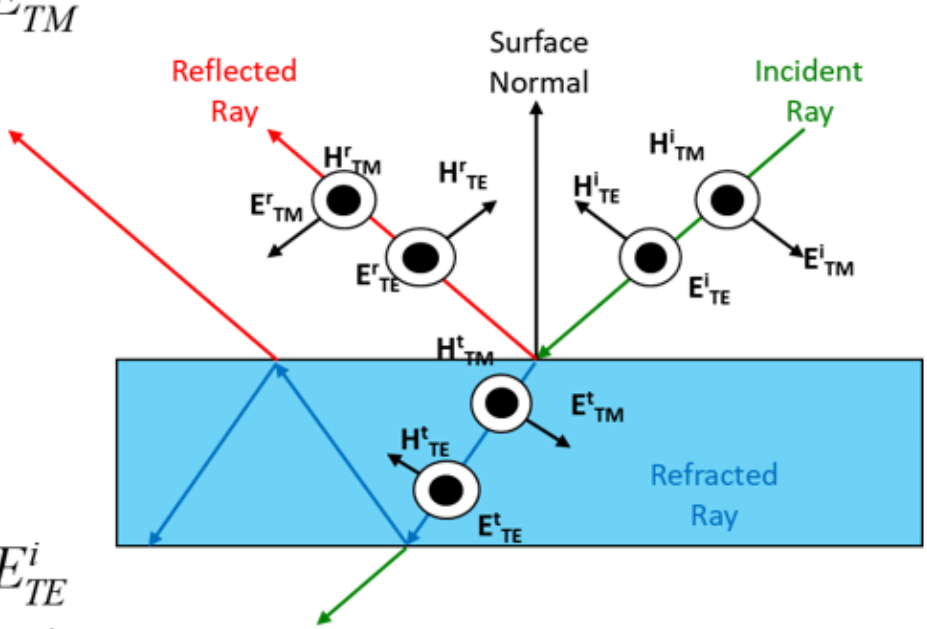
When the material assigned to a solid object is dielectric, and either no boundary is present or the boundary is two-sided, rays propagate inside the dielectric region according to Snell's law of reflection and refraction. SBR+ solid objects can be assigned homogeneous isotropic dielectric materials, making it possible to accurately solve problems involving dielectric regions of non-constant thickness. Dielectric material assignments can include a dielectric and/or magnetic loss tangent. This feature is available for SBR+ solution-type designs and hybrid designs with SBR+ regions.

Dielectric material assignments must be homogeneous, isotropic, and with zero conductivity. Lossy dielectrics can be configured by specifying a frequency-independent dielectric and/or magnetic loss tangent. Dielectric objects can be located in a free-space background, or contained fully or partially within another dielectric region or can be in contact with metal. In SBR+, ray bundles can represent complex wavefronts, and each ray represents a sample of that wavefront that is assumed to be locally planar. Provided that the wavefront is not too irregular, the curvature of the interface varies smoothly and slowly relative to the wavelength scale, such interaction of rays with the boundary of a dielectric region provide a good approximation of the reflection and refraction of an arbitrary wavefront into and out of that region.

Unlike SBR+ Fresnel coefficient boundaries, which can be thought of as a lumped representation of a finite- and uniform-thickness coating (e.g., Layered Impedance Boundary), material assignments are useful for describing irregularly shaped dielectric regions. Rays hitting a Layered Impedance Boundary do not need to be traced through multiple bounces within the layers because this effect is summarized by the Fresnel coefficients of such a boundary. In contrast, rays hitting a dielectric region interface generate a reflection back into the incidence region and, critically, a refraction ray inside the dielectric region on the opposite side of the interface. The refraction ray will be traced through multiple bounces within the dielectric region, and these subsidiary rays can refract back out of the dielectric region or reflect off or refract into dielectric regions that are embedded within. The reflection and refraction coefficients and ray directions at dielectric region interfaces satisfy Snell's laws, and they depend on the polarization of the incident wave (ray).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

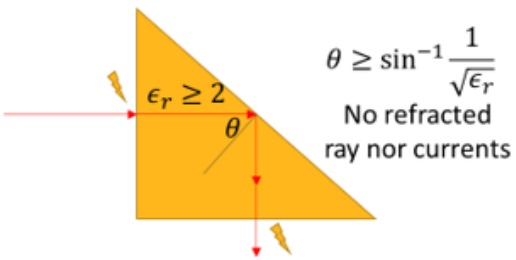
$$E^r_{TE} = R_{TE} E^i_{TE}$$
$$E^r_{TM} = R_{TM} E^i_{TM}$$



$$E^t_{TE} = T_{TE} E^i_{TE}$$
$$E^t_{TM} = T_{TM} E^i_{TM}$$

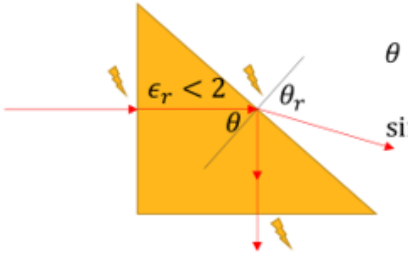
According to the equivalence theorem, only the currents on the free-space side of an interface are radiated to field observers (e.g., far-field angles, other antennas, etc.), with computed fields being valid in free space.

Since rays traveling through dielectric regions follow Snell's laws, interesting phenomena can occur when transitioning between regions of different refraction indexes. Total internal reflection occurs when a ray impinges beyond a critical angle at the interface between a material with higher refraction index and one with lower refraction index. Consider the case in the following picture.



$$\theta \geq \sin^{-1}\frac{1}{\sqrt{\epsilon_r}}$$

No refracted ray nor currents



$$\theta < \sin^{-1}\frac{1}{\sqrt{\epsilon_r}}$$
$$\sin\theta_r = \frac{\sin\theta}{\sqrt{\epsilon_r}}$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The triangular region is an isosceles right triangle with permittivity ϵ_r and set within a free-space background. A ray hits one of the legs, enters the dielectric region, and hits the hypotenuse at a 45^o θ angle with the surface normal. The ray is then reflected and exits the prism from the other leg. According to Snell's law, if $\theta \geq \sin^{-1}\frac{1}{\sqrt{\epsilon_r}}$, energy does not propagate beyond the dielectric interface, but is completely reflected in the dielectric region. A non-propagating evanescent wave is generated in the region of space immediately outside the hypotenuse side.

Since the SBR+ algorithm tracks the history of encounters with dielectric region interfaces in order to establish whether it has entered or left a dielectric region, it is important to make sure that SBR+ excitations are placed in free-space, and not inside the dielectric volume.

Raytracks starting inside a dielectric region, or that encounter incompatible dielectric region coatings, as in the case of Tolerant or Dynamic meshing with intersecting/contained objects, will be flagged as "inconsistent", and result accuracy can't be guaranteed.

When any boundaries are assigned on top of objects, the material assigned to the object will be used to determine the proper coating information passed to the SBR+ solver. Two-sided boundaries can be placed over dielectric or conductive solid material assignments to represent, e.g., a thin layer of paint over a structure.

In addition, SBR+ supports assigning boundaries on sheets inside dielectric regions.

Material/Boundary to SBR+ Coating

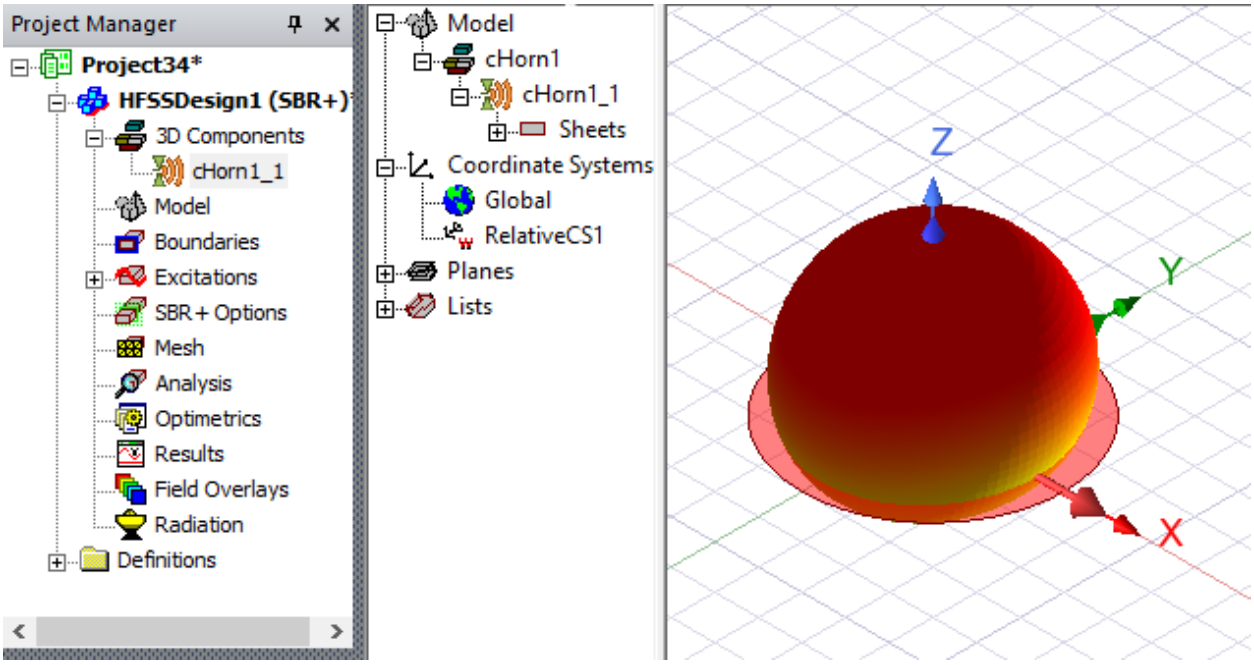
Object type	Boundary	Material inside	SBR+ coating
Solid	One-sided, except finite conductivity with finite thickness	Any	Non-penetrable coating corresponding to HFSS boundary
	One-sided finite conductivity, finite thickness	Any	Layer (same properties as FC boundary) with "material inside" half-space backing
	Two-sided boundary or no boundary	Dielectric, including vacuum	Dielectric volume interface with or without layers
		Conductor	Layers with conductor "material inside" half-space backing
Sheet	Any		Coating corresponding to HFSS boundary

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parametric Antennas for SBR+ Solution Type

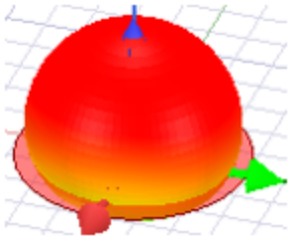
For SBR+ solution type, you can access a range of parametric antennas. Parametric antennas are antenna sources that either have no physical structure definition or a highly idealized physical structure. No model parts are constructed or used in the simulation. They create far-field pattern or current distribution sources that can provide S-parameter outputs and composite far-field radiation patterns. The SBR+ solver will automatically interpolate the source fields from the file-based antenna in order to represent the antenna at all frequencies of the user defined frequency sweep in solution setup. The file-based antenna requires at least two frequencies, and the min/max frequencies of the user defined sweep should fall within the frequency band of the file-based antenna.

Parametric antennas are very quick to define and place, copy and duplicate, and are thus intended for systems studies where the specific antenna structure is either unknown or a simple, topologically representative antenna type is sufficient. The editable parameters depend on the antenna type. In the model window, the antennas display as a scaled far field pattern at the antenna origin. The model is listed under the **Project** tree under **3D Components** and automatically includes a port to support S-parameter output. You can change port post-processing setting from the **Properties** window. Selecting the antenna causes the **Properties** window to display with tabs for **General** antenna **Properties**, and **Visualization** settings.

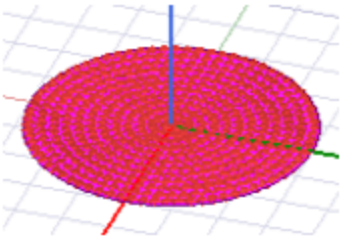


Several of the parametric antennas let you choose Antenna Representation as Far Field Source and/or as Current Source.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

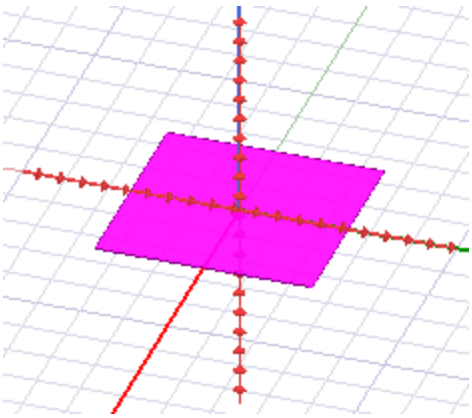


Far Field



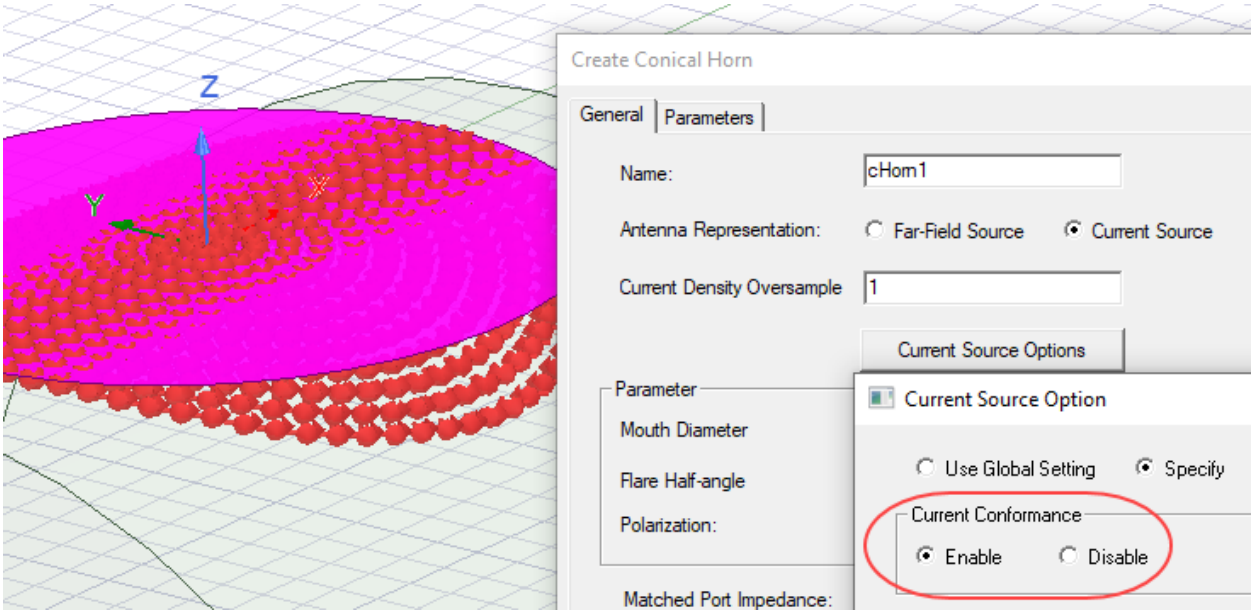
Current Source

Other parametric antennas include only Current Source representation.



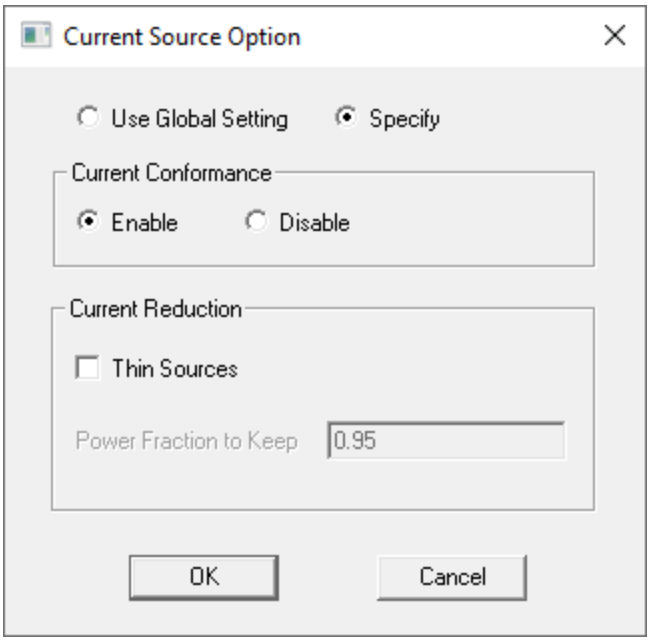
Current sources are visualized in the model window as a set of short lines with arrows to indicate the current direction. When placed near a geometric surface, these current sources can be configured to conform to the nearby surface. The current source conformance can be configured globally, to be enabled/disabled for all antennas in the HFSS design, or enabled/disabled for each parametric antenna or array.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For a technical discussion, see Current Conformance and Array Conformance in SBR+.

The Current Source Option window also includes Current Reduction option.



Enabling Thin Sources lets you specify a Power fraction. It should be greater than 0.1 and less than 1. Solutions are invalidated if you change current source conformance, use Thin Sources, Power Fraction, or change the Global option.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The outputs from SBR+ simulations with parametric antennas include:

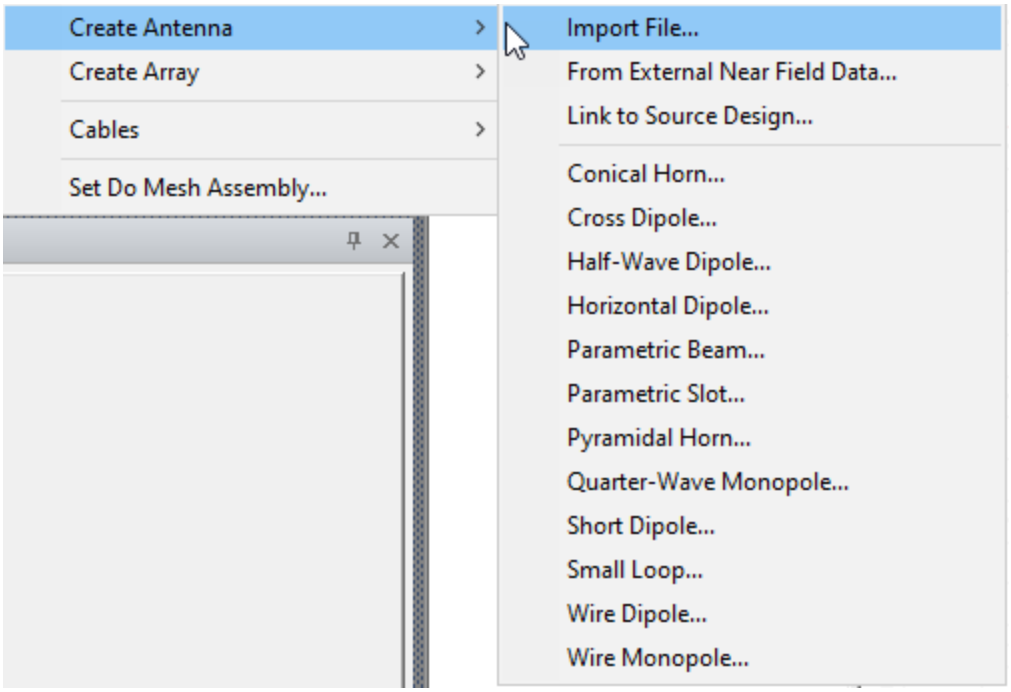
- S-parameters.
- Composite far-field radiation patterns.
- Incident field reporter outputs provide the free space radiation patterns of the antenna.

Prerequisites

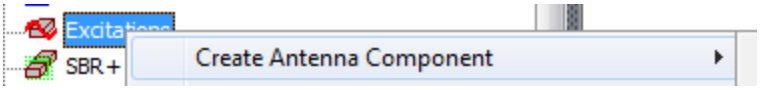
- The Solution Type must be SBR+.
- If you use a File-based parametric antenna, it must be defined with at least two frequencies whose min/max meet or exceed the extents of the simulation frequency sweep.

Adding Parametric Antennas

1. Right-click the 3D Component folder in the Project tree to see the **Create Antenna** menu:

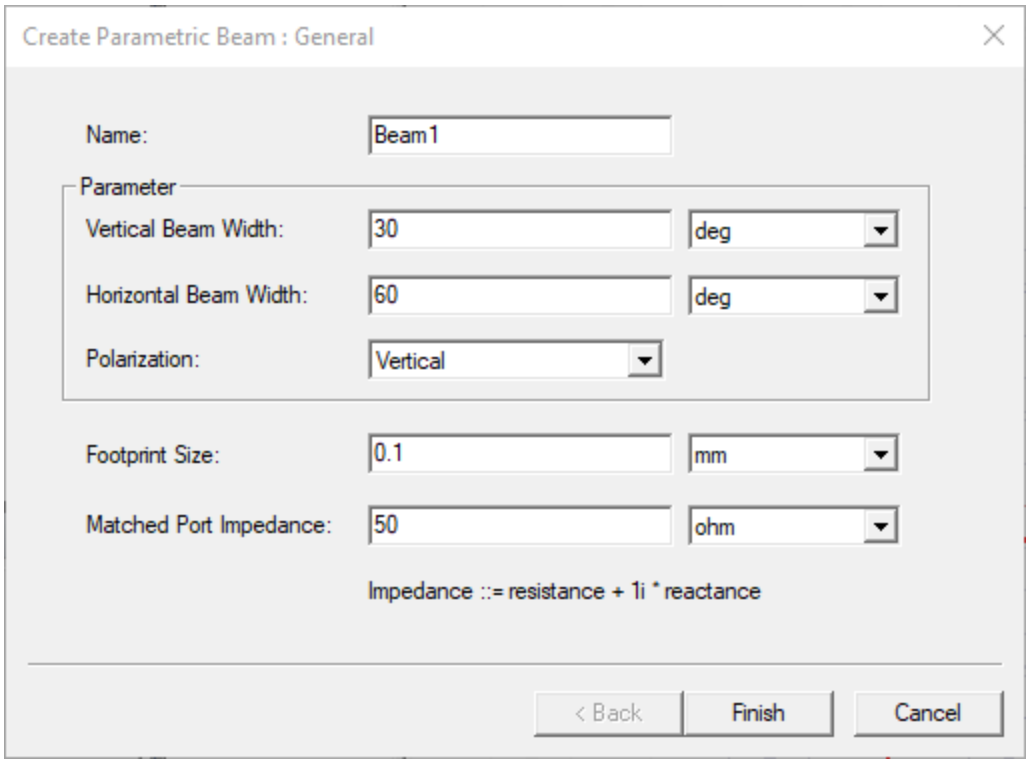


You can also view the parametric antenna menu by clicking **HFSS** > **Excitations** > **Create Antenna Component** or by right-clicking on the **Excitations** icon and selecting the submenu under **Create Antenna Component**.



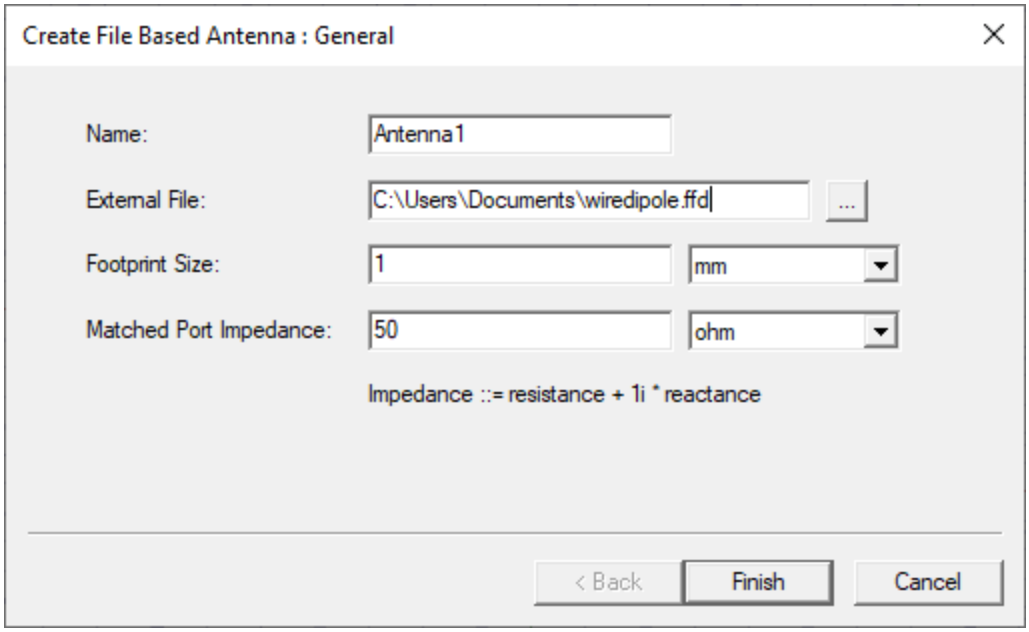
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting any of the predefined antennas causes a dialog to appear, which includes the name, and other parameters. For example, the dialog below shows the **Create Parametric Beam** dialog box.



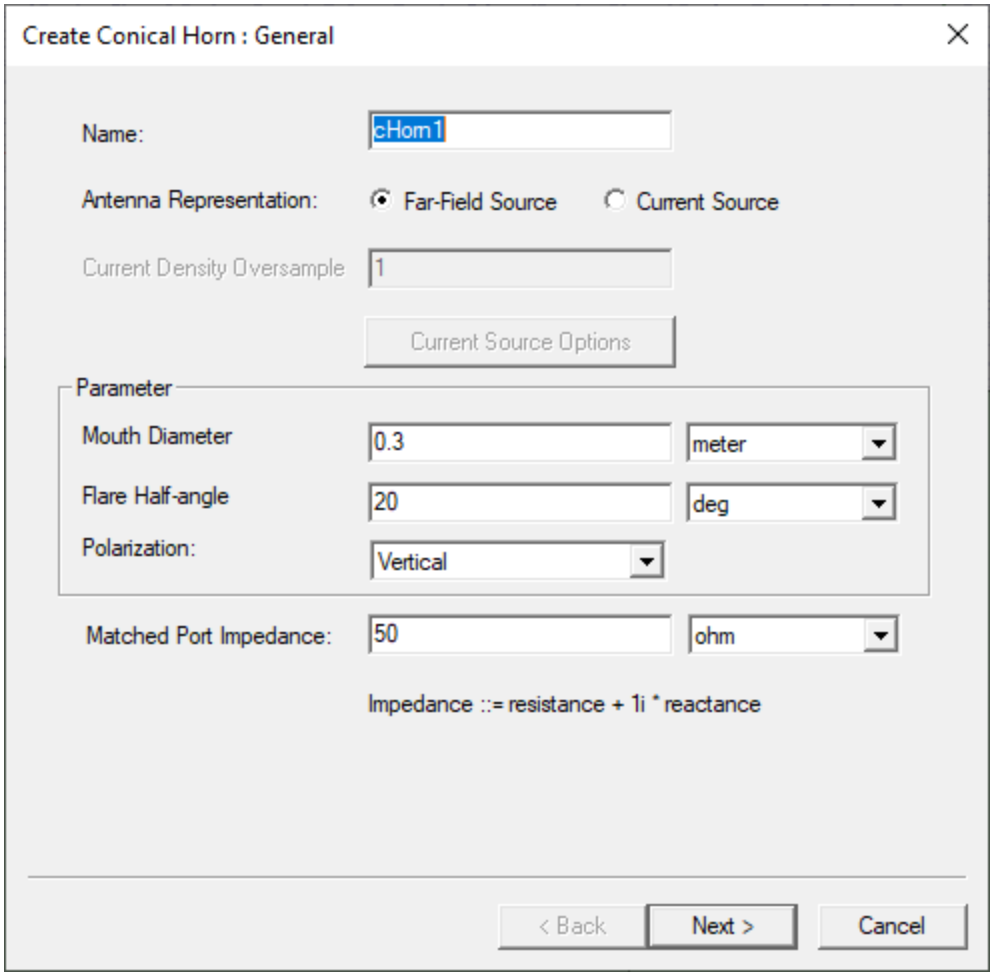
For the **By File** antenna, the Parameters include a path for a Far Field data file (*.ffd format).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

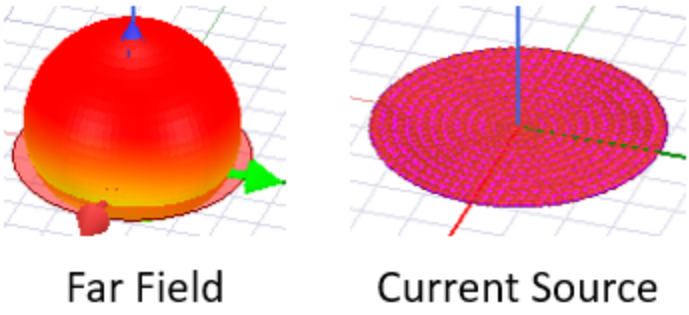


The Conical Horn dialog includes the option for representation as Far Field and/or Current Source.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



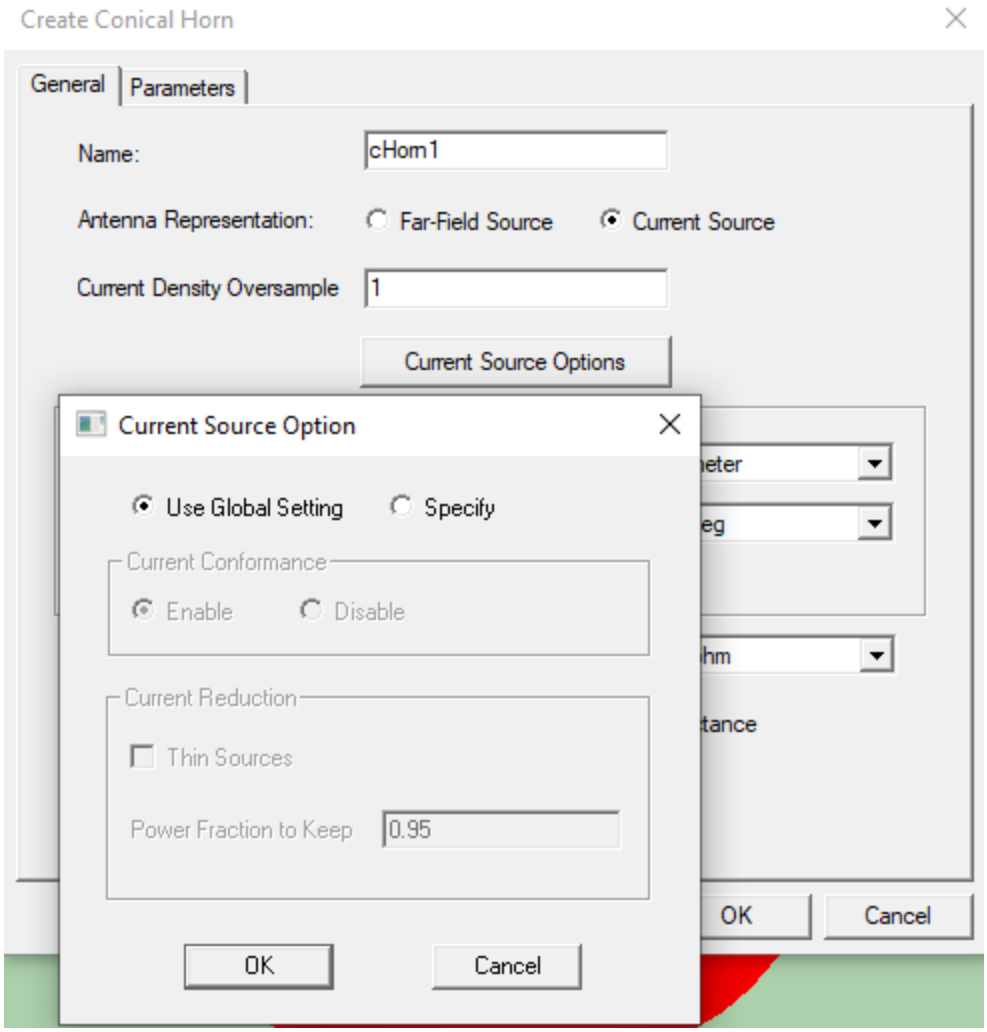
Depending on your selection, the visualization may appear as follows.



Far Field Current Source

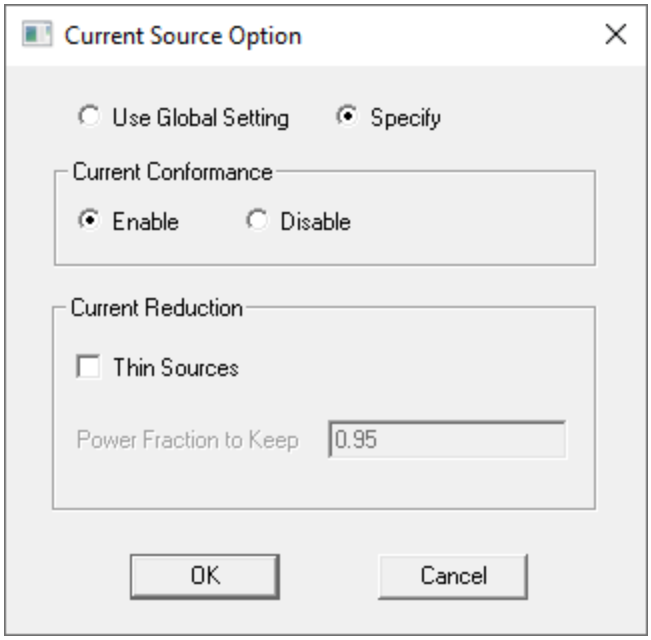
If the antenna offers current source, and you select that option, the **Current Source Options** button is enabled on the **Create** dialog box. Select it to open the **Current Source Option**. dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

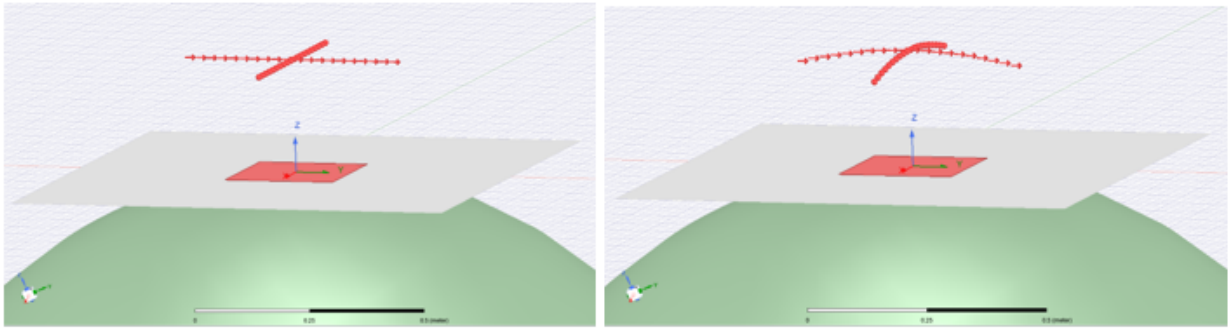


For the global current source options, see Set Current Source Option. To override the global setting, select Specify. This enables Current Conformance and Current Reduction settings for this antenna.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Current conformance controls whether antennas placed near a geometric surface conform to that surface. The figure on the left shows current conformance disabled. The figure on the right shows current conformance enabled.



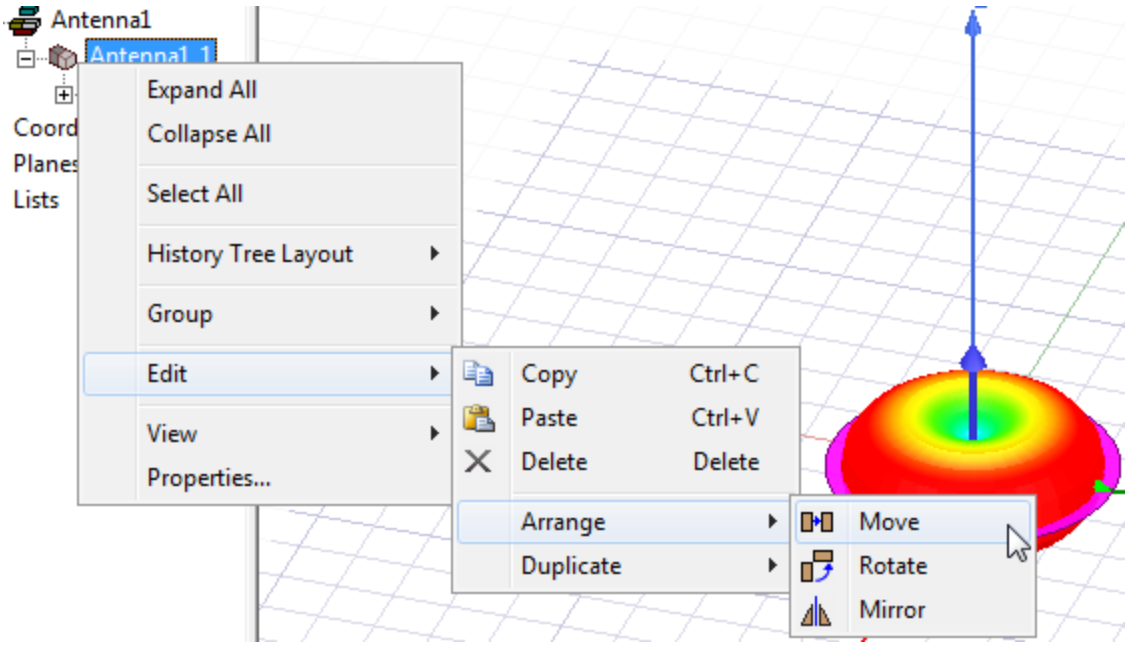
Enabling Thin Sources lets you specify a Power fraction. It should be greater than 0.1 and less than 1. Solutions are invalidated if you change current source conformance, use Thin Sources, Power Fraction, or change the Global option.

2. Specify the parameters as desired. The parametric antenna footprint is a visual to aid in placing and orienting the antenna, but it has no impact on simulation results.

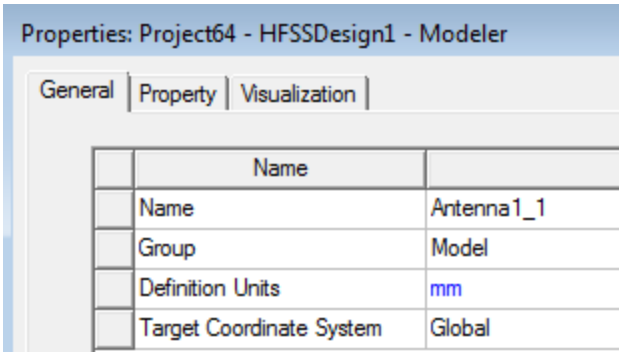
The antenna appears in the Model window.

3. If you want to edit the antenna placement, or orientation, select the model in the History tree, and right-click and select **Edit** > **Arrange**, and select from **Move**, **Rotate**, or **Mirror**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also change or edit the target coordinate system in the instance Property window.

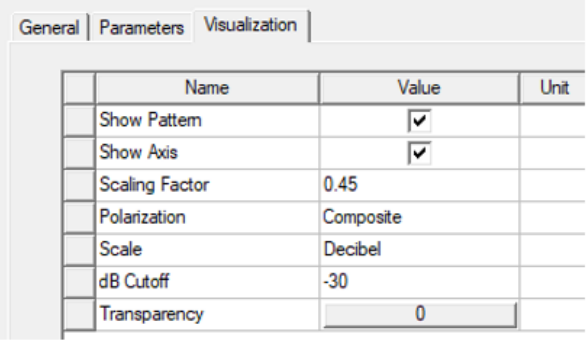


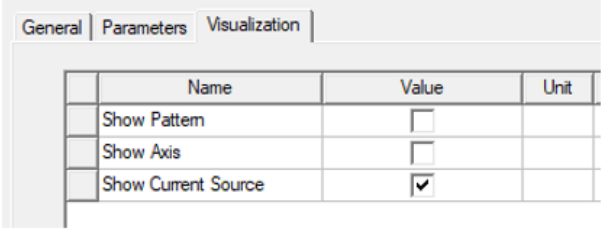
Notice that other edit commands for Copy, Paste, and Duplicate are also available.

4. If you want to edit the Visualization parameters, select the Antenna model icon under the **Project** tree or **History** tree to view the **Properties**, and select the **Visualization** tab. The visualization parameters available can be different for different antenna models. Visualization parameters have no influence on simulation results. They only determine

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

how the parametric antenna is displayed in the model viewer.

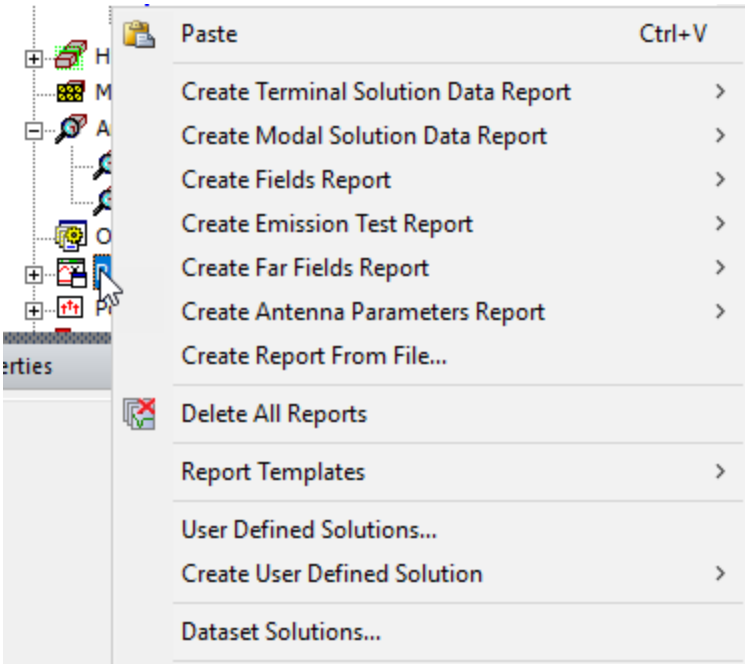




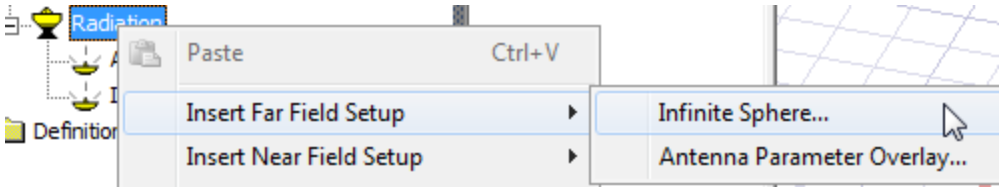
- Choose whether to show the pattern and/or axis. Parametric antennas in a current-sources representation also provide the option to view them according to their far-field Pattern, even though they always launch rays from a distribution of currents sources during SBR+ simulation.
- If the antenna parameters include Show Current Source, you can enable or disable visualization here.
- For Polarization, click on the cell for a drop-down menu selection for Composite, Horizontal, Vertical, LHCP, or RHCP. The Polarization rendering setting only controls which polarization component of the parametric antenna's radiation pattern is currently displayed. It has no impact on the polarization characteristics of the parametric antenna definition and, thus, no impact on simulation results. Some antennas, such as the Parametric Beam, also provide a control for its polarization behavior in the **Property** tab, which does influence simulation results.
- Scale can be Decibel (the default), Linear, or Power.
- dB Cutoff appears when you select Decibel as the Scale. This sets the dB level at the pattern display origin relative to its peak level.
- For Transparency, click on the cell for a dialog with a slider and text field.
- For Scaling Factor, click on the cell to edit the value. The Scaling Factor controls how large the antenna's radiation pattern is displayed relative to its Footprint Size.

5. After you run an SBR+ simulation with parametric antennas, you can create a variety of reports, including S Parameters, Far Fields, and Antenna Parameters.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You must create a Far Field Setup to access the Far Fields and Antenna Parameters reports.



Parametric Antenna Descriptions

When you specify an SBR+ Solution type, you have access to the following sources and parametric antennas:

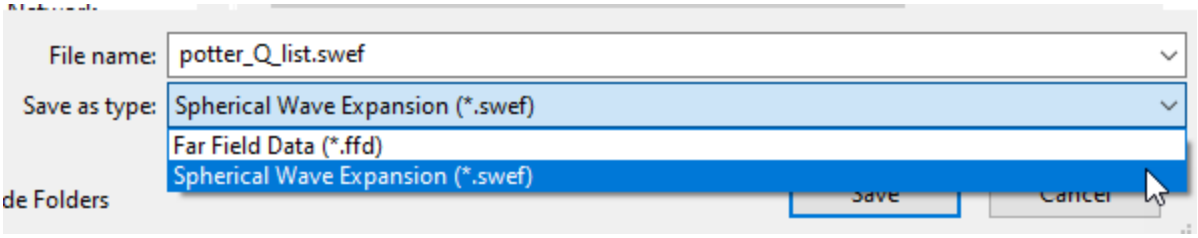
- "By File Antenna for SBR+" on the facing page
- "From External Near Field Data" on page 13-131
- "Link to Source Design" on page 13-137
- "Conical Horn" on page 13-138 (far field or current source option)
- "Cross Dipole" on page 13-140 (current source)
- "Half-Wave Dipole" on page 13-142
- "Horizontal Dipole" on page 13-143 (current source)
- "Parametric Beam" on page 13-145
- "Parametric Slot" on page 13-145 (far field or current source option)
- "Pyramidal Horn" on page 13-147 (far field or current source option)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- "Quarter-Wave Monopole" on page 13-149
- "Short Dipole" on page 13-150
- "Small Loop" on page 13-150
- "Wire Dipole" on page 13-151 (far field or current source option)
- "Wire Monopole" on page 13-152 (far field or current source option)

By File Antenna for SBR+

You define the **By File...** antenna by importing an external file containing far field data in .ffd, or for Spherical Wave Expansion Import, you can import .swef format files.



Pattern samples are entered over a two-dimensional angular sampling domain in spherical coordinates, θ and φ per frequency. The initial or final angle can be arbitrary (that is, no restriction on θ being from 0 to 180 degrees, and φ from 0 to 360 degrees). Nevertheless, the sampling of the θ and φ domain must be such that if adding more points to the domain, 0- and 180-degree samples are exactly captured.

For example, suppose one domain is defined as 1 90 90, that is, from 1 to 90 degrees with 1-degree step. This domain is compatible with the above restriction, because by adding points to the domain every 1 degree the domain can be extended to 0 180 181.

Consider another domain defined as 1 89 45, that is, from 1 to 89 degrees with 45-degree step. The domain is not compatible with the above restriction, because 0 and 180 degrees cannot be exactly sampled by continuing the domain. In other words, the external far-field can be specified over a sector which does not include θ = 0 or 180 degrees and φ = 0 or 180 degrees. However, if the defined sector is extended to cover the entire sphere using the specified sampling, the entire sphere should contain samples at θ = 0 and 180 degrees, φ = 0 and 180 degrees.

The SBR+ solver will automatically interpolate the source fields from the file-based antenna in order to represent the antenna at all frequencies of the user defined frequency sweep in solution setup. The file-based antenna requires at least two frequencies, and the min/max frequencies of the user defined sweep should fall within the frequency band of the file-based antenna.

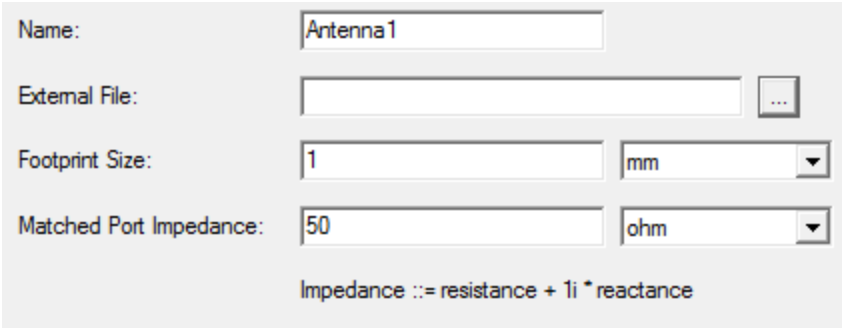
Pattern levels specified in the file are automatically renormalized so the antenna radiates at the power level specified in the **Edit Post Process Sources** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

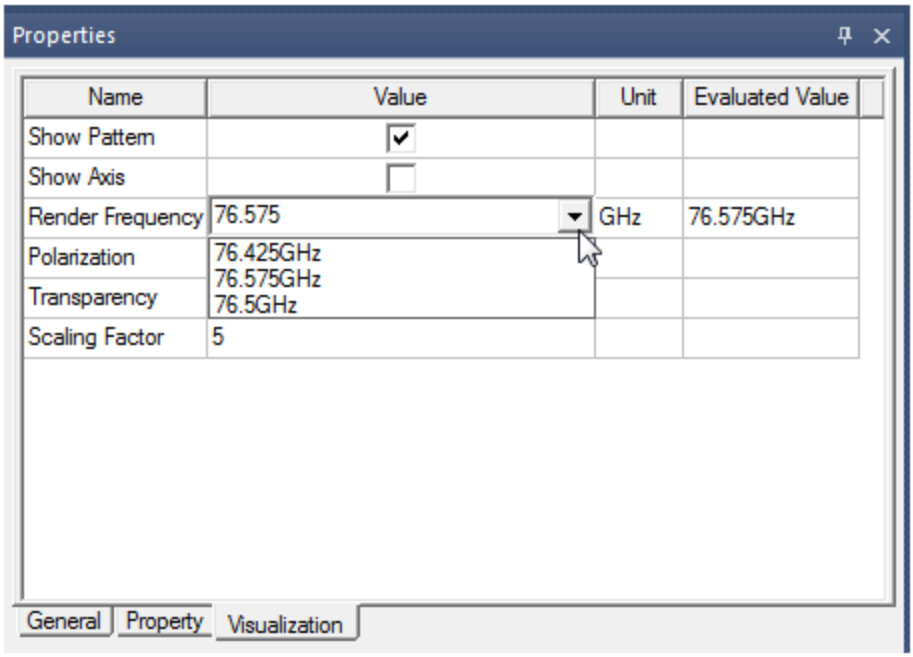
For detailed information on the swef format, and the Parameters area that appears in the import dialog, see Spherical Expansion Wave Source (*.swef).

For detailed information on the ffd format, see External Data File for Far Field Wave.

For editing properties, placement, and visualization, file based antennas resemble parametric antennas.



For the By File antenna, the **Visualization** tab for **Properties** also includes a drop down for **Render Frequency** which lists the available frequencies defined in the Far Field data file (*.ffd).

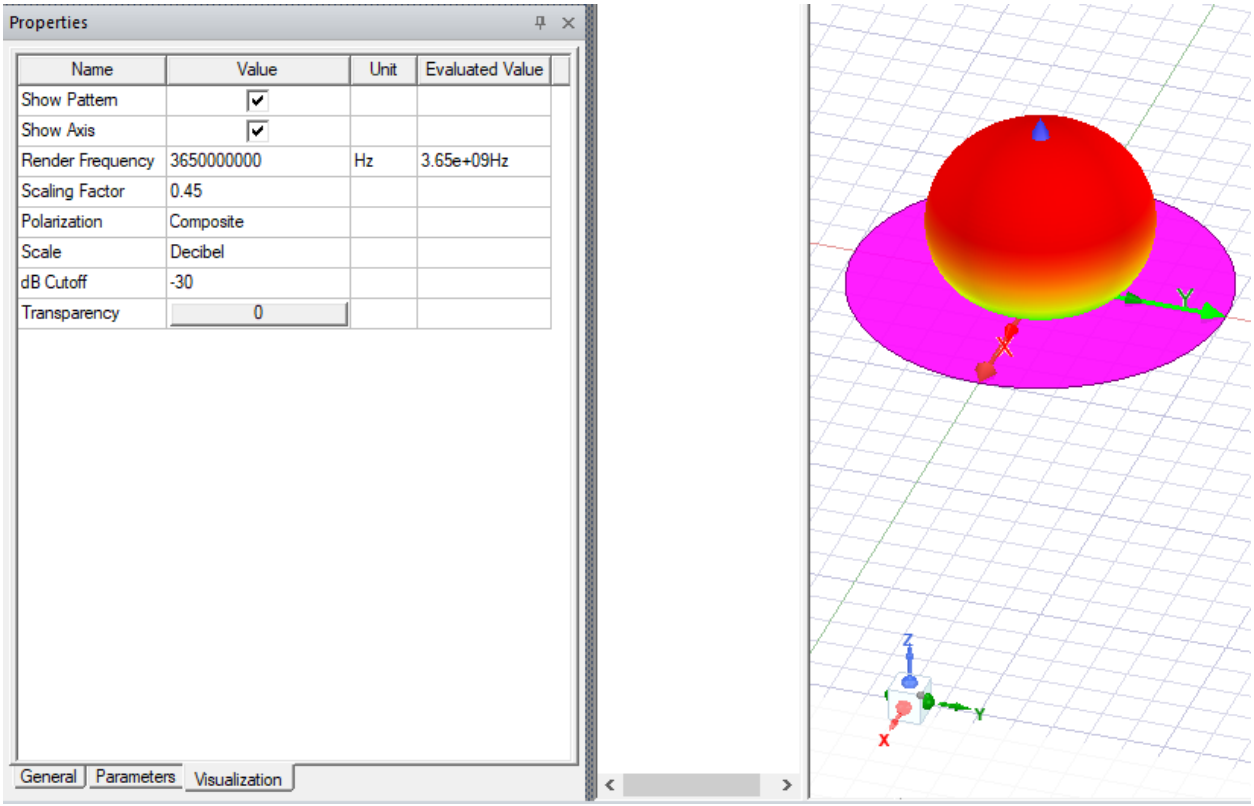


If you enable the **Show Pattern**, or load a design with the option enabled, if the .ffd file is not found, an error message reports this. Validation checks to see if the Render frequency specified is provided in the .ffd file. After you create a File Based Antenna, you have the option to reload

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the *.ffd file. If you archive a project with a File Based antenna, the process automatically includes the *.ffd file and restores it to a relative location for portability.

If you import of Spherical Wave Expansion File import for .swef files, visualization for the antenna is also available. Visualization parameters include Render Frequency, Scaling Factor, Polarization, Scale, dB Cutoff, and Transparency. You can edit fields or select from drop down menus. You can use an SWE antenna to produce S-matrix and far field results.



From External Near Field Data

Near field characteristic of the source antenna, which is saved in .and and .nfd format, can be used to create external near field antenna and act as a virtual port in SBR+ target design. Compared to near field antenna link, this "file-based" near field approach does not need any information of the source antenna design except for the near field on specific sample points, and is therefore convenient to use and easily maintainable and scalable.

For defining simple-primitive near field source, HFSS uses two kinds of external data file. First, the actual location/field data is stored in .nfd format. Second, the Ansys Near field Descriptor file (.and file) stores the general geometry information and paths of .nfd data. Adding a external near field link to HFSS design is therefore a two-steps workflow. During excitation setup, an external near field design (that is, a design with a box representing mesh points) is created based on

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

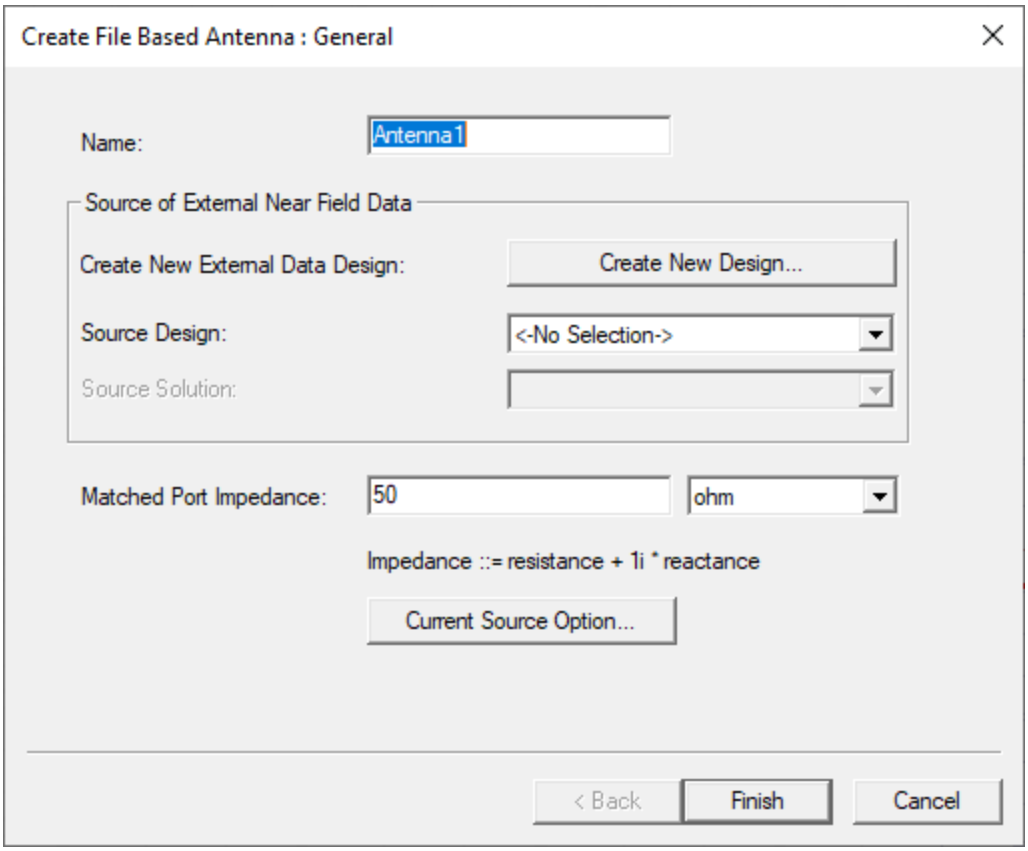
information in a .and file and linked to target design. During simulation, external near field design acquires mesh points and field from .nfd file based on specific frequency the target design asks for.

For HFSS SBR+ design you can also create external near field design based on an .and file or select from existing ones. A virtual port is created for each external near field antenna, and its corresponding footprint is created similar to an HFSS linked antenna.

To create a new Near Field design, based on an .and file:

Open an empty SBR+ design, open external near field antenna dialog by selecting **3D Component** > **Create Antenna** > **From External Near Field Data...**

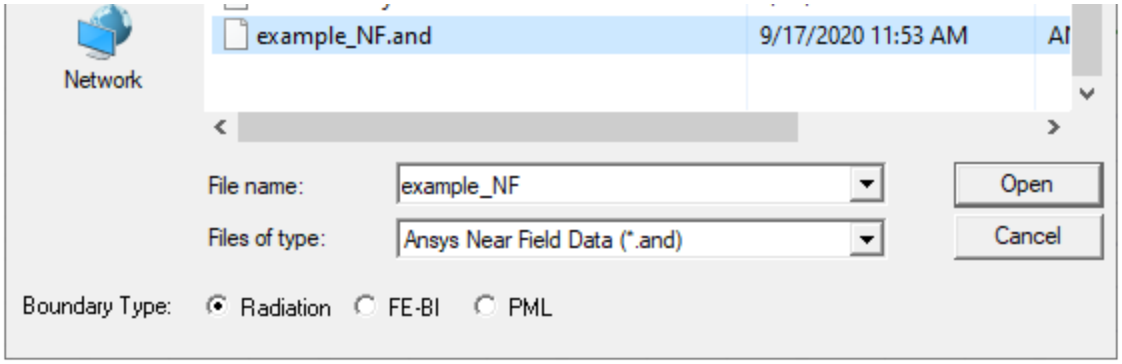
This opens the following **Create File Based Antenna** dialog box:



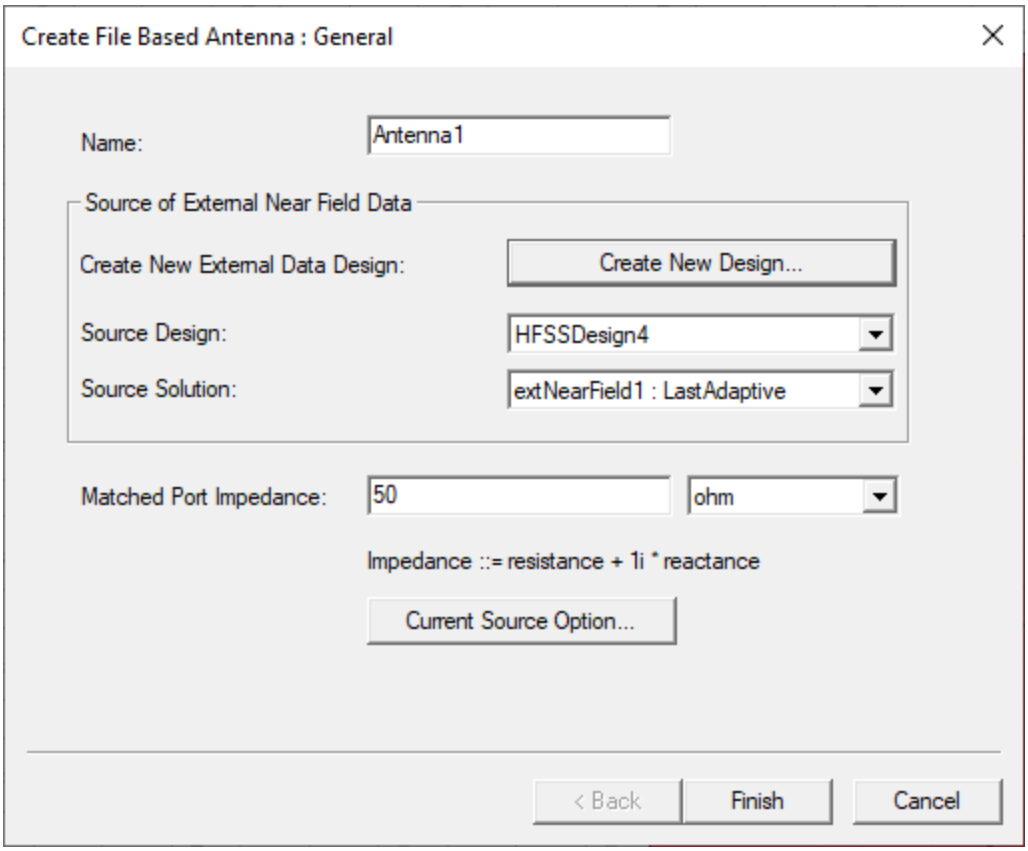
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initially, because there is no external near field design, the “Source Design” combo box shows “No Selection” and “Source Solution” combo box is grayed out.

To create a new external near field design, select the **Create New Design...** button. A file selection dialog box will pop out and boundary type is allowed for selection.

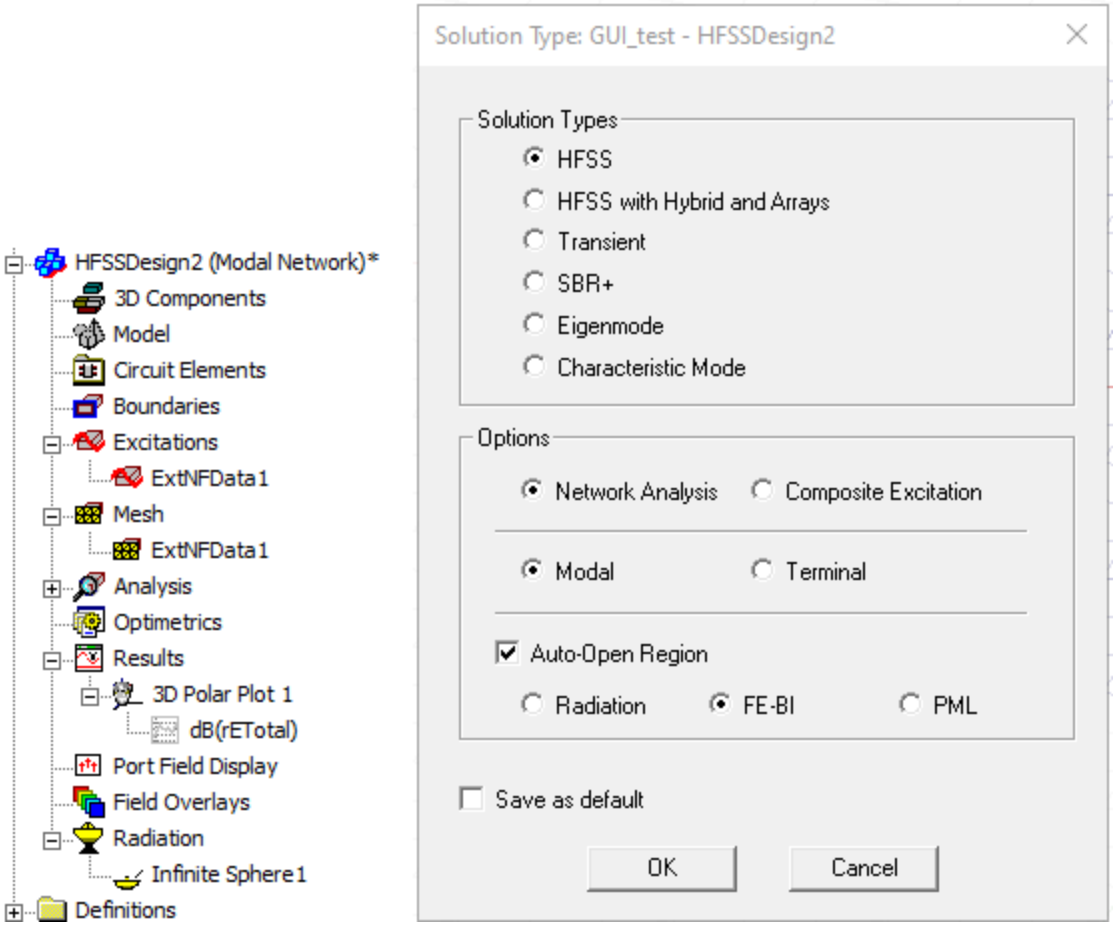


When you select an .and file, make the Boundary Type selection, and click open, the Create File Based Antenna dialog is populated with information about the Source Design and Source Solution.



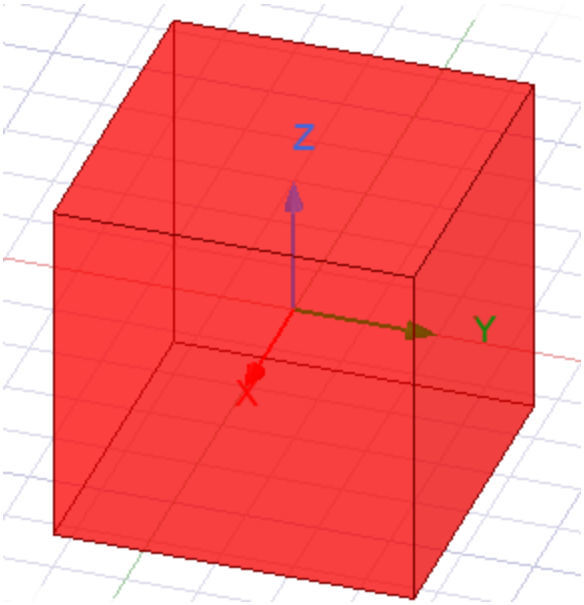
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Additionally, a new external near field design (“HFSSDesign2”) is created in the Project tree. This newly created design automatically creates several setups like mesh, solution, report and radiation. It also creates a native geometry as defined in .and file.

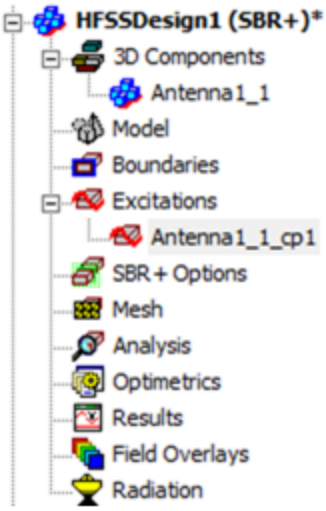


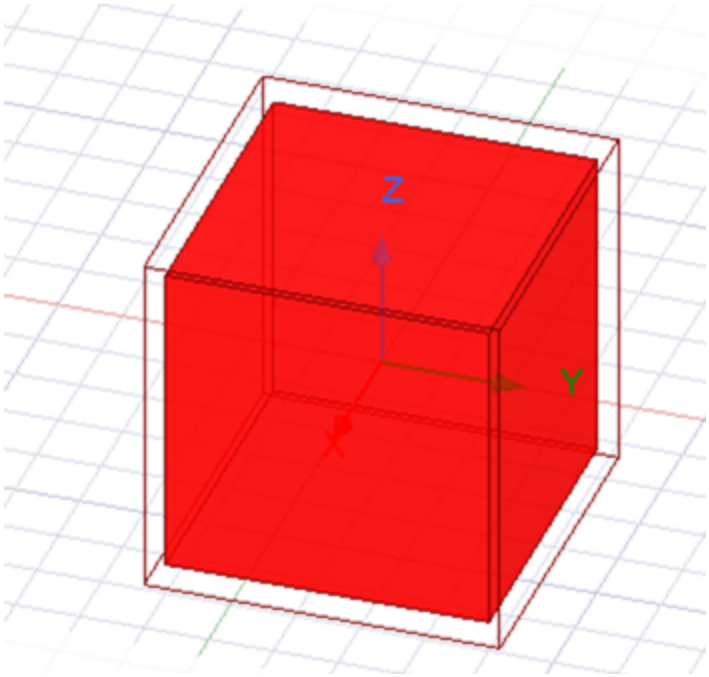
The modeler window shows a native geometry as defined in the .and file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Click **Finish** in the **Create File Based Antenna** dialog and a new external near field antenna is created, together with a virtual port and antenna footprint (including outer bounding box and inner near field geometry).

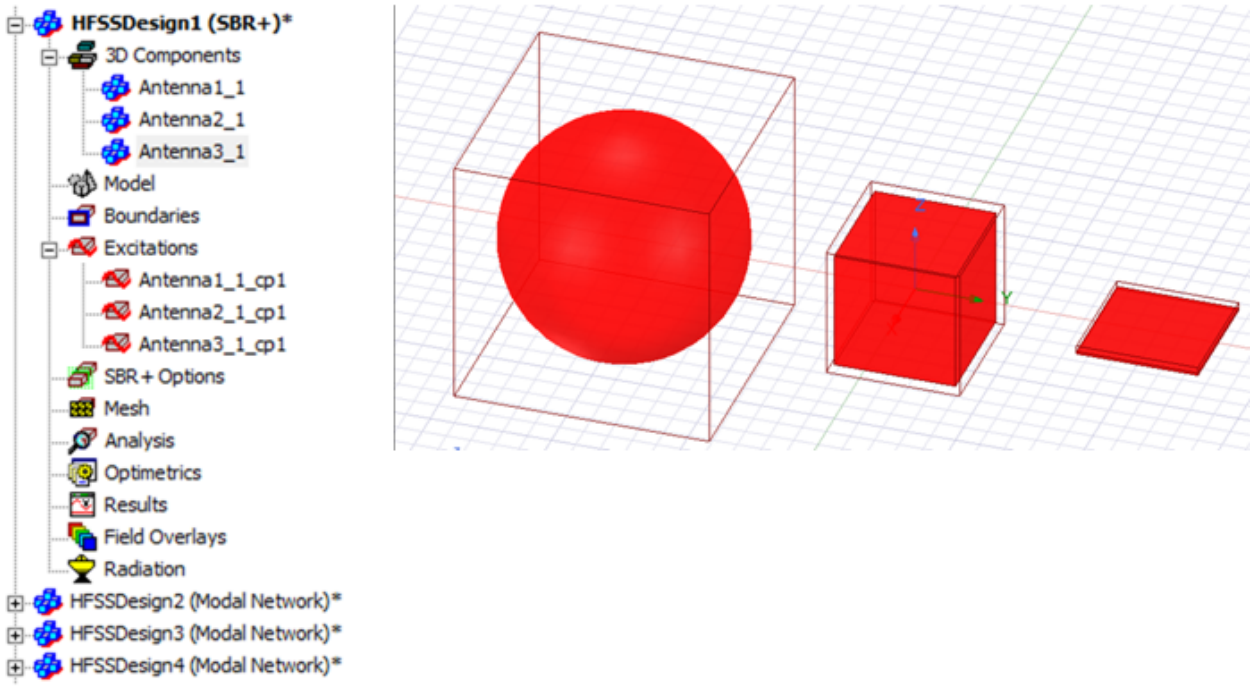




Editing an existing external near field antenna

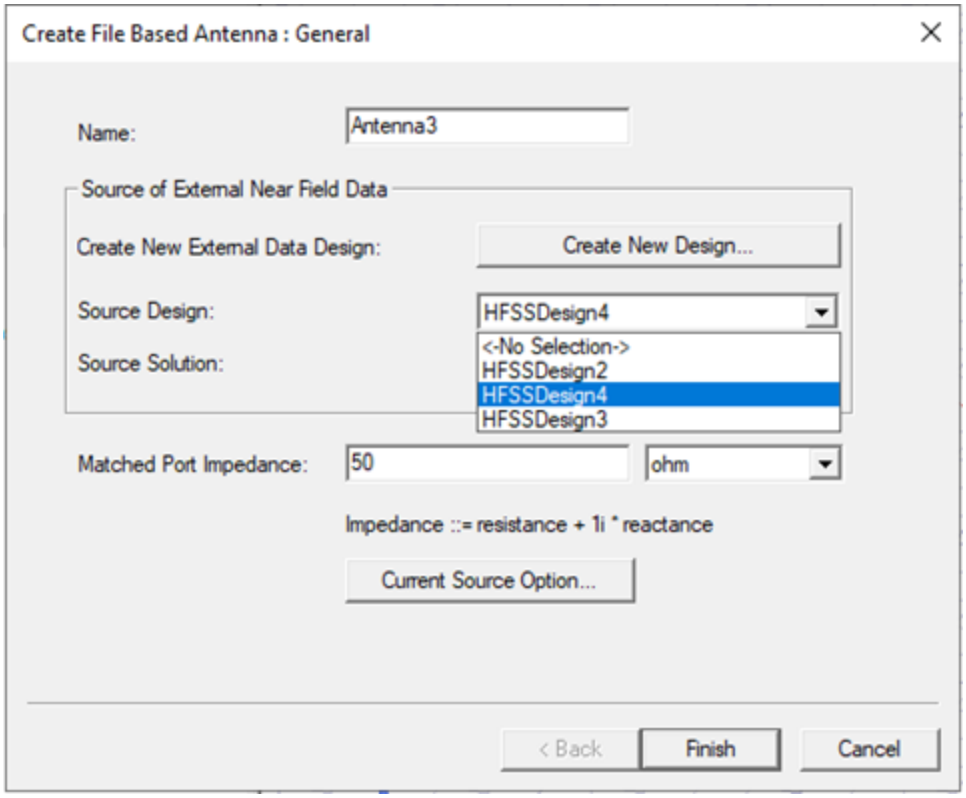
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Let say we already create several external near field antennas and their corresponding external designs using similar approaches.

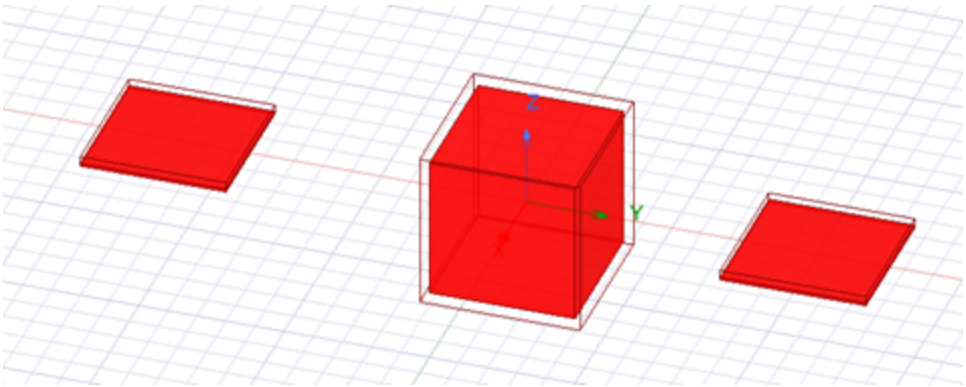


When we edit one of the antennas (in this case Antenna3), the dialog show its corresponding source design name and solution name.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If we change the source design and click **Finish**, its footprint will be updated.



Link to Source Design

HFSS lets you create a one-way link from a Driven Modal or Driven Terminal design to an SBR+ design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

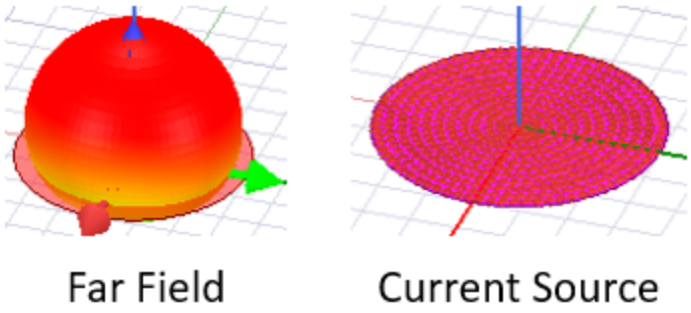
Conical Horn

This frequency-dependent antenna approximates a flared conical horn with user-defined dimensions and polarization. It is based on the approximated field distribution of a circular cross-section conical horn driven in the TE_{11} waveguide mode. The model accounts for phase taper at the edges of the flat aperture (horn mouth) due to the spherical phase front introduced by the walls flaring from the circular waveguide feed at the throat of the horn.

The aperture (horn mouth) sits in the XY plane of the antenna local coordinate system, with boresight along the +Z direction. The antenna is characterized by the following settings: Mouth Diameter, Flare Half-Angle, and Polarization.

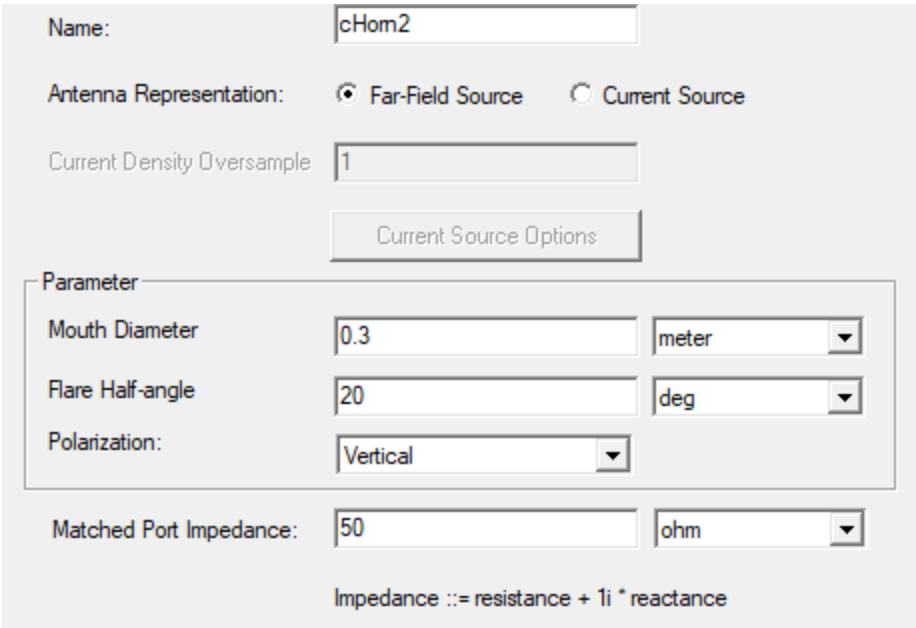
The conical horn will not operate below the TE_{11} cut-off frequency of a circular waveguide, given by $f_{cutoff}> = (x'_{11}\ c)/(\ \pi\ d_{aper})$, where $x'11 \sim= 1.8412$ is the first zero of the derivative of the first-order Bessel function of the first kind, c is the speed of light, and d_{aper} is the mouth (aperture) diameter. Attempts to simulate with a conical horn below this frequency will result in an error. Technically speaking, the cut-off frequency should be based on the smaller diameter of the waveguide, not the diameter of the horn mouth, and it should therefore be higher. However, since the waveguide diameter plays no other role in the conical horn model, it is left out of the specification parameters in the interest of simplicity, and we instead tie the cut-off frequency to the mouth diameter in this SBR+ parametric antenna model.

The dialog also lets you choose Antenna Representation as Far Field Source or as Current Source.

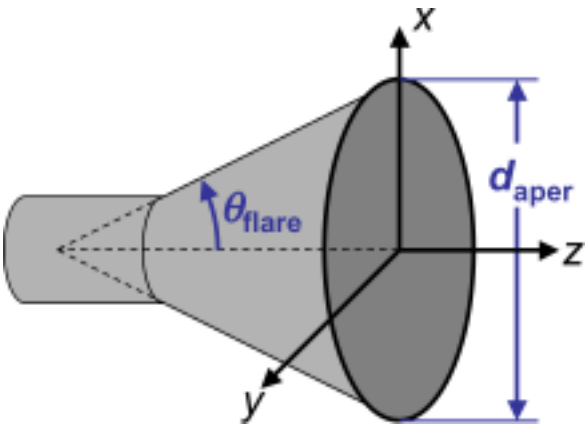


You specify parameters for mouth, flare half-angle, polarization, and matched port impedance.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For conical horn antennas, the following figure explains the meaning of the dimensions.

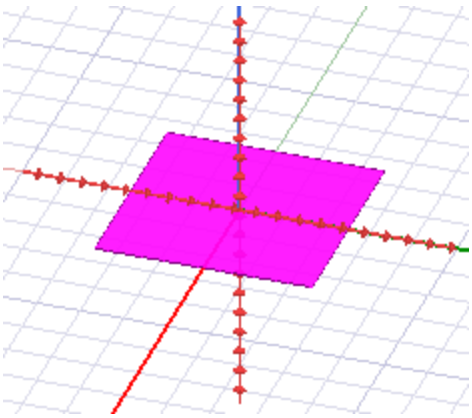


If you select Current Source, the field for Current Density Over sample is enabled and the current source options button is enabled. For Current Source, you also need to specify Current Density Oversample, where the default value of 1.0 corresponds to 4 current-source samples per wavelength. Oversample values less than 1 are usually too course for an accurate result, while those greater than 2 usually do little to improve accuracy while harming run time. The default of 1.0 is the recommended setting for most cases, and we strongly discourage setting Current Density Oversample less than 1.0.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

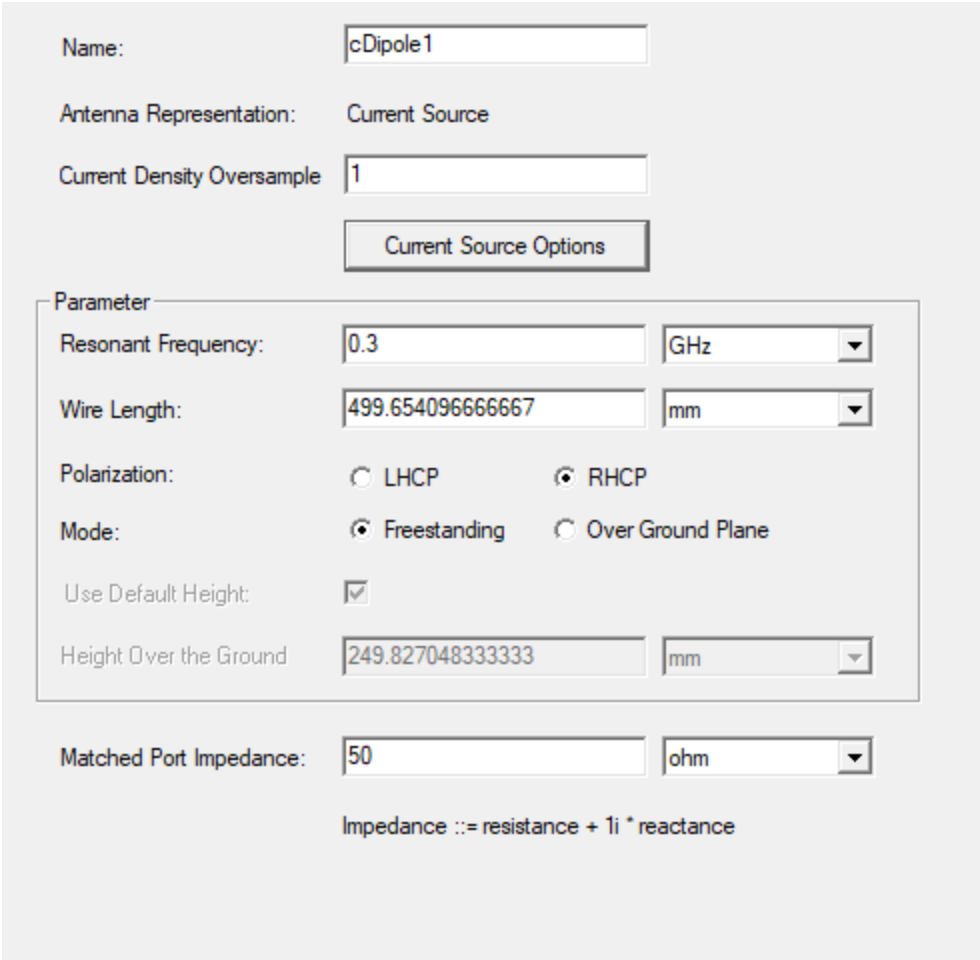
Cross Dipole

This antenna always uses a current source representation. Parameters are resonant frequency, wire length, polarization, and mode as free standing, or over ground plane. In the its Freestanding mode, the electric current sources represent two perpendicular arms of a dipole, one each along the Y and Z axes, driven with a 90 degree phase shift relative to each other to achieve the selected circular Polarization along the +X axis (it will be the opposite polarization along the -X axis). In its Over Ground Plane mode, the antenna arms are rotated to be parallel to the XY plane and at a default height along the Z axis of a quarter-wavelength at the specified Resonant Frequency. This allows the antenna to be placed directly on a metal surface of the SBR+ platform and have its reflected field constructively interfere with the direct incident field from the dipole arms when operating at the Resonant Frequency and in the +Z direction. You can switch off Use Default Height to manually set the height of the current source above the implied ground plane of this antenna if that better fits the design of the cross-dipole antenna to be simulated. You also specify matched port impedance.



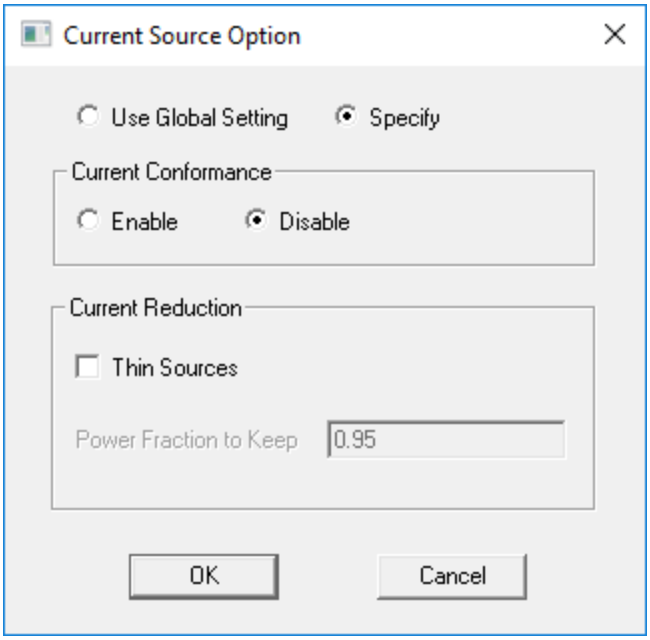
Current Source Options opens a dialog with Use Global Setting (the default) or Specify, which enables the fields for Current Conformance, and Current Reduction settings to allow you to Thin Sources, and to specify a Power Fraction to Keep.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



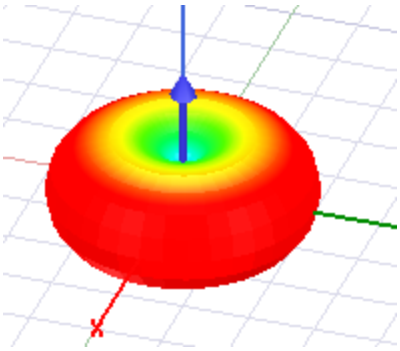
Current Source Options opens a dialog box with Use Global Setting (the default) or Specify, which enables the fields for Current Conformance, and Current Reduction settings to allow you to Thin Sources, and to specify a Power Fraction to Keep.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

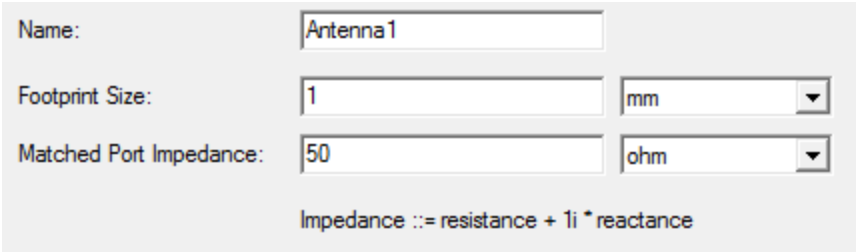


Half-Wave Dipole

This antenna produces a far-field pattern of a center-fed thin dipole with a total length of a half-wavelength, with a peak Directivity of 2.15 dBi. The Half-Wave Dipole has no current-sources representation since it needs to be resonant at all simulation frequencies. For this reason, it is not recommended to locate it within two wavelengths of any SBR+ scattering geometry. If that is needed, one should instead consider to use the Wire Dipole or Horizontal Dipole in their current-sources representation.

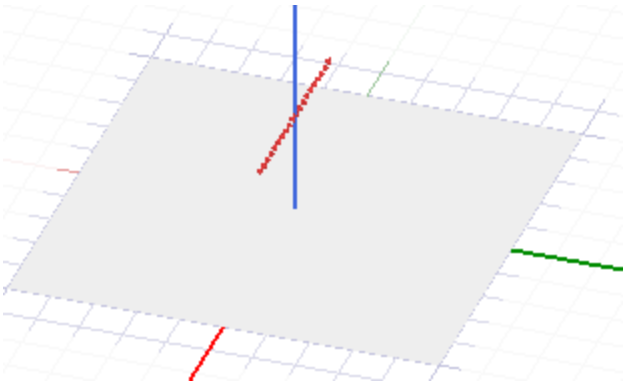


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



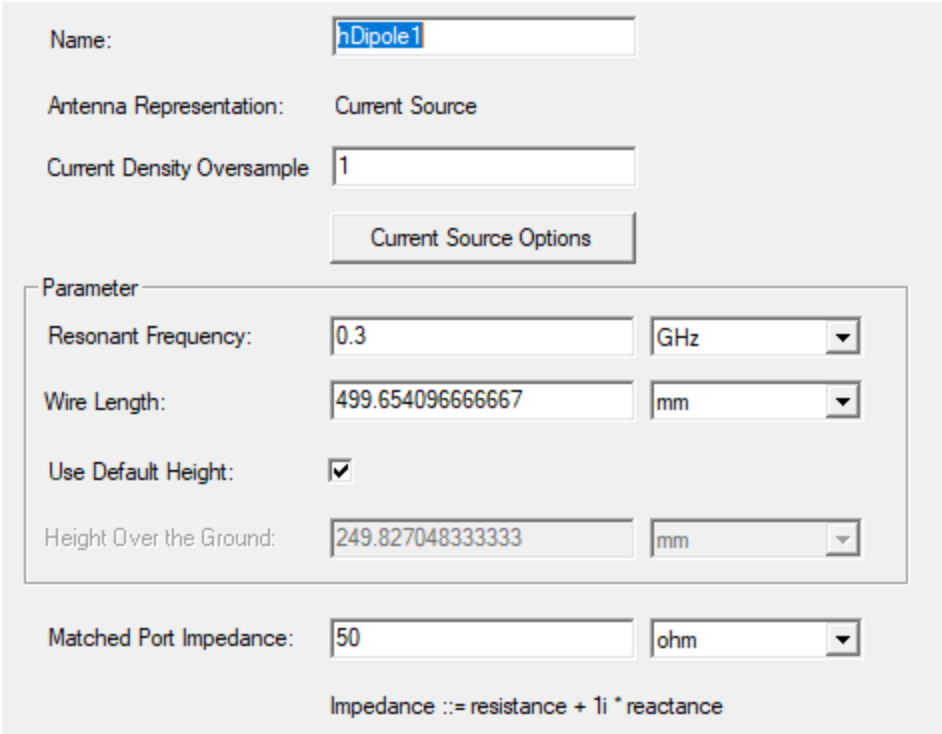
Horizontal Dipole

This antenna always uses a current-sources representation. It is intended for direct placement on a metal surface of the SBR+ platform given its built-in default height of a quarter-wavelength.

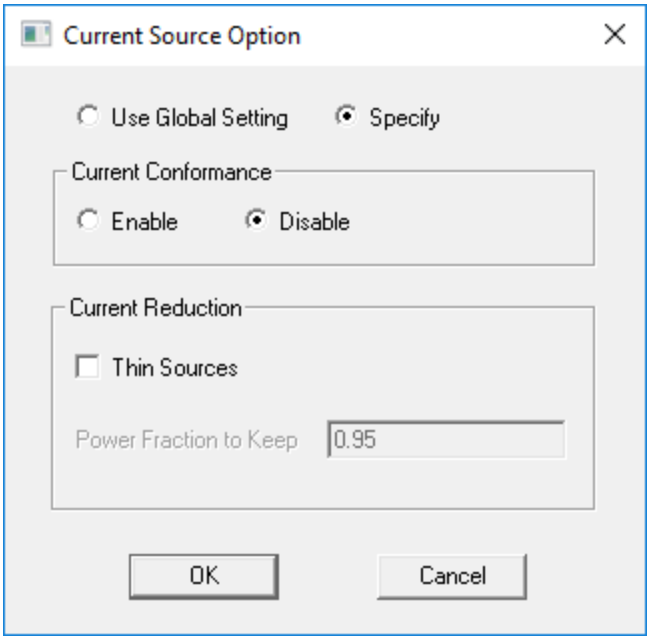


Other parameters include current source options, resonant frequency, wire length, matched port impedance, and whether to use the default height or to override it and manually specify the height of the sources above the implied grounding surface. The default height is a quarter-wavelength at the specified Resonant Frequency. Assuming direct placement on a metal surface of the SBR+ platform, the intended context of this antenna, the reflected field from the platform will constructively re-enforce the direct field from the current sources in the +Z direction of the antenna.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



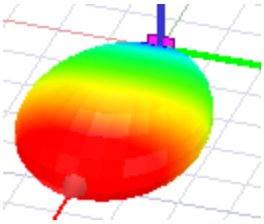
Current Source Options opens a dialog with Use Global Setting (the default) or Specify, which enables the fields for Current Conformance, and Current Reduction settings to allow you to Thin Sources, and to specify a Power Fraction to Keep.

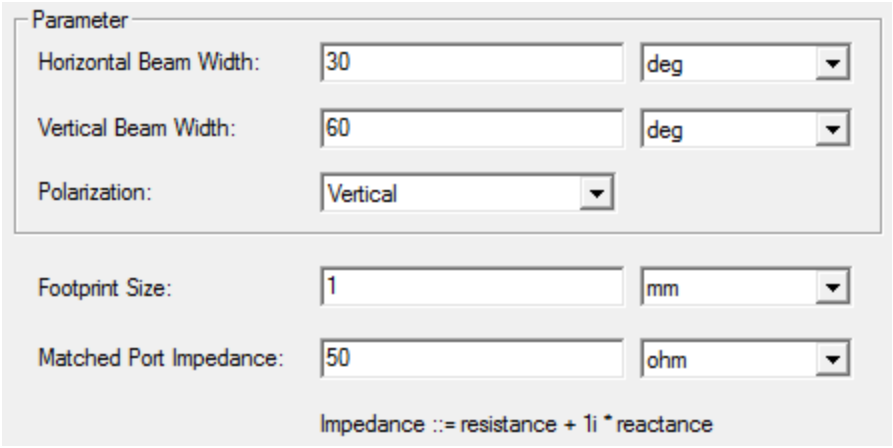


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parametric Beam

The Parametric Beam antenna for SBR+ produces an idealized pencil beam that can substitute as a simplification for a variety of aperture based antennas, including horns and broadside arrays. The main beam is pointed in the X direction of the antenna's local coordinate system and has the mathematical form, $\sin^{Nv}\theta \cos^{Nh}\phi$, where *Nv* and *Nh* are internally determined according to the Vertical and Horizontal Beam Width settings, respectively. The antenna produces an idealized pencil beam ($\cos^{n}\theta$) pattern) with user-defined beam widths and polarization. This antenna produces no side lobes or backlobes and only radiates in the forward half-space. The Parametric Beam antenna has no current-sources representation.





Parametric Slot

This antenna is based on an idealized sinusoidal (magnetic) current distribution for the slot along the X-axis of the coordinate system. The Far-Field pattern will vary according to the slot's electrical length at the visualized frequency. When operated at the user-defined Resonant Frequency, the slot length is 0.5 wavelengths.

Note that changing the Resonant Frequency will update the Slot Length and vice versa.

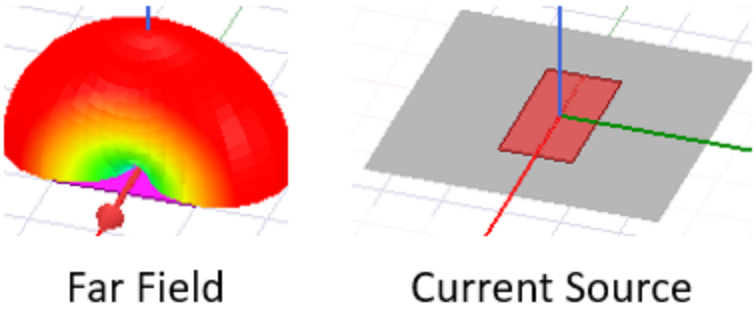
The Parametric Slot is a ground-plane antenna whose intended placement is on a metal surface of the SBR+ scattering geometry or platform. In this context, use of the current-sources

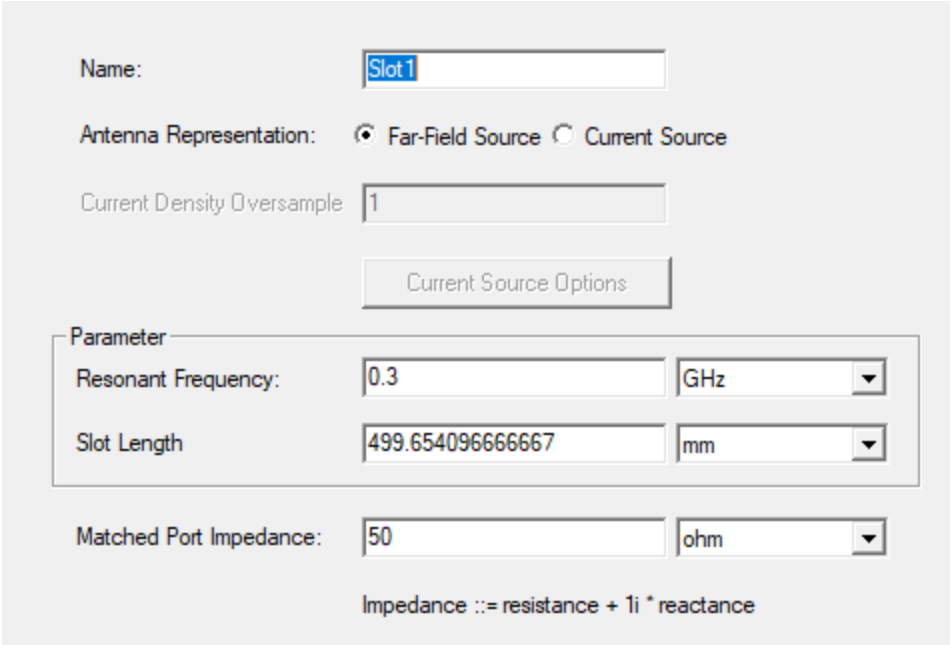
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

representation is strongly recommended. Although notionally located in the z = 0 plane of the antenna local coordinate system, the magnetic sources of the slot will instead be placed at a small offset height of λ/40 measured at the shortest wavelength of the simulation. This allows the parametric slot to be directly placed on a metal surface of the SBR+ platform without the need to manually offset it so that the sources don't inadvertently reside under the placement surface.

One can also select a far-field representation. However, given its assumed ground-plane context, the parametric slot radiates no far fields in its lower half-space, so all direct interaction with the placement platform is ignored during SBR+ simulation with this representation of the slot except for those surfaces of the platform or scattering environment that reside about the z = 0 plane of the slot antenna.

. The dialog also lets you choose Antenna Representation as Far Field Source or as Current Source.





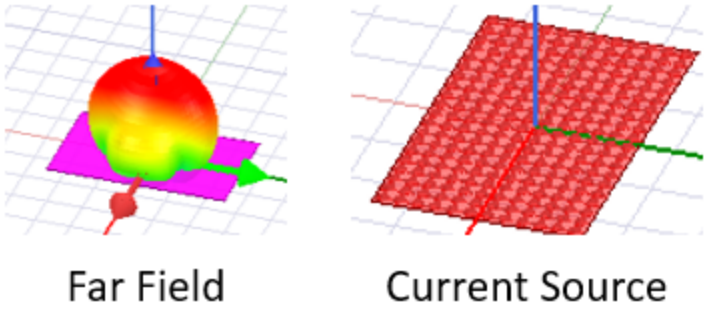
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pyramidal Horn

This frequency-dependent antenna approximates a flared rectangular horn with user-defined dimensions. It is based on the approximated field distribution of a rectangular cross-section pyramidal horn driven in the TE_{10} waveguide mode. The model accounts for phase taper at the edges of the flat aperture (horn mouth) due to the elliptical phase front introduced by walls flaring from the rectangular waveguide feed at the throat of the horn.

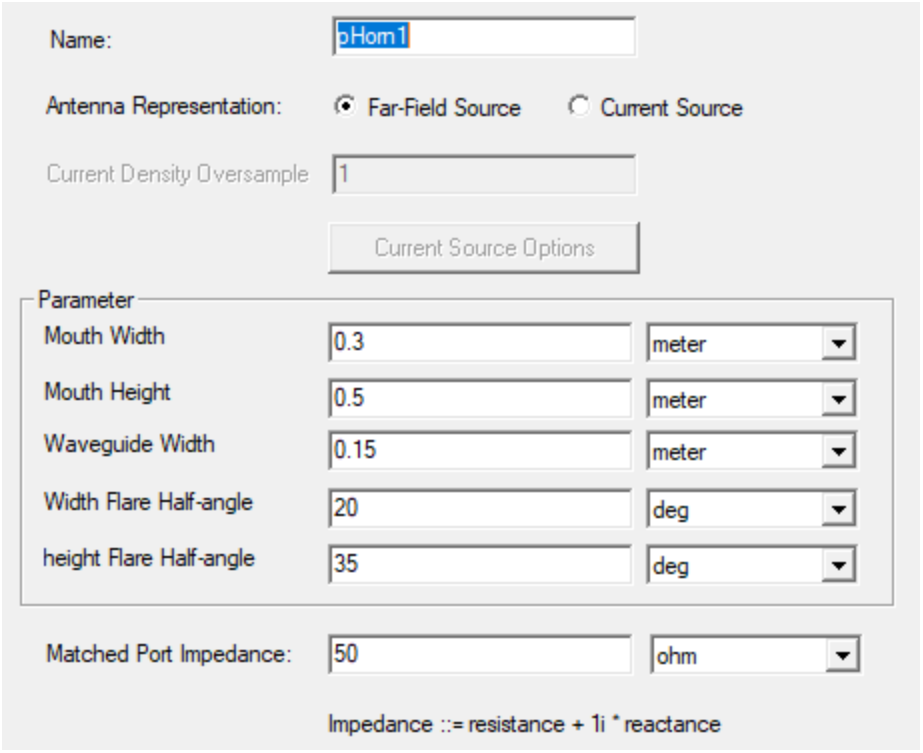
The aperture (horn mouth) sits in the XY plane of the antenna local coordinate system, with boresight along the +Z direction. The antenna is characterized by the following geometric dimensions: Mouth Width, Mouth Height, Waveguide Width, Width Flare Half-Angle, and Height Flare Half-Angle. From these parameters, the waveguide height is implied. If the implied waveguide height is negative, an error report is generated with suggestions for adjusting the input dimensions to resolve the conflict.

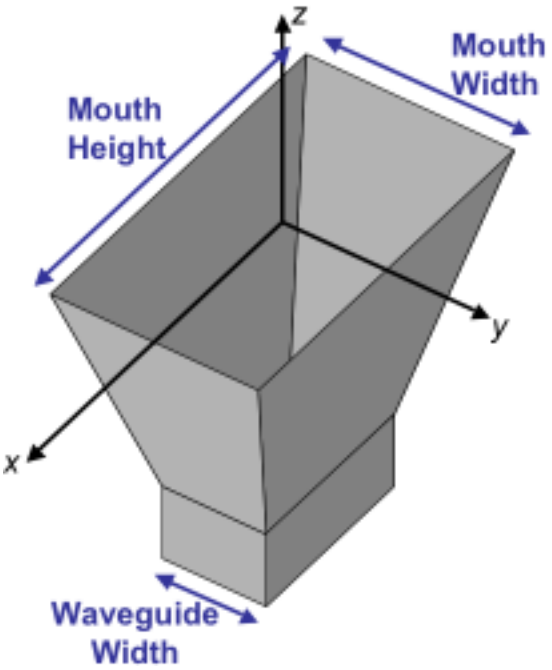
The Pyramidal Horn will not operate below the TE_{10} cut-off frequency of the rectangular waveguide, given by $f_{cutoff} = c/2w$, where w is the waveguide width and c is the speed of light. Attempts to do so will results in an error during simulation.



The dialog also lets you choose Antenna Representation as Far Field Source or as Current Source.

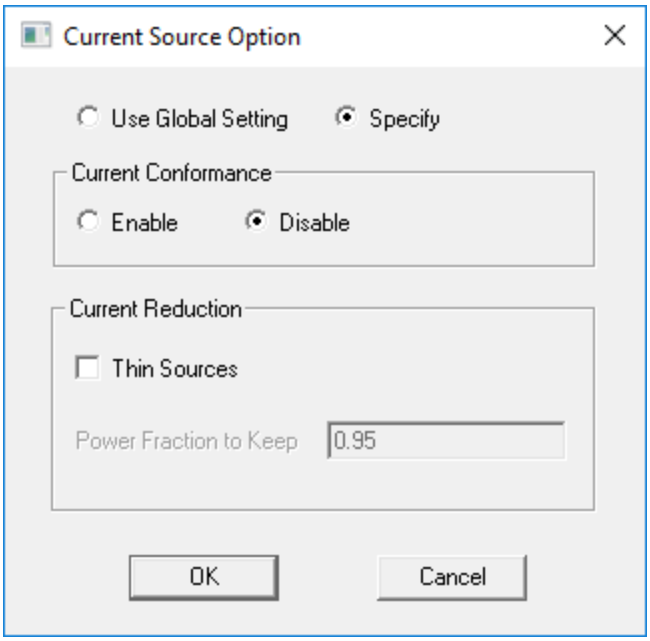
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





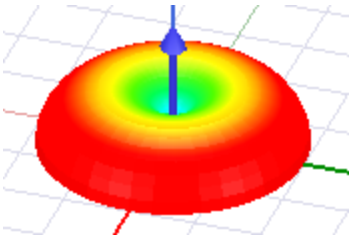
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Current Source Options opens a dialog with Use Global Setting (the default) or Specify, which enables the fields for Current Conformance, and Current Reduction settings to allow you to Thin Sources, and to specify a Power Fraction to Keep.

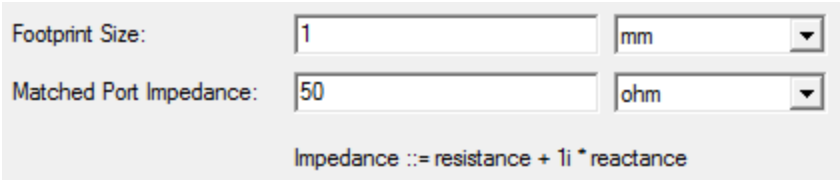


Quarter-Wave Monopole

This antenna produces a far-field radiation pattern defined over a half-space (top half of a torus), consistent with an ideal quarter-wave monopole antenna over an infinite ground plane. It has a Directivity of 5.16 dBi in the upper half-space; no radiation gain in the lower half-space. This antenna only illuminates objects above the monopole and will not capture direct interactions with its placement surface on the SBR+ geometry. To model the interaction of a monopole with its supporting SBR+ platform, users should instead use the Wire Monpole in its current-sources representation.

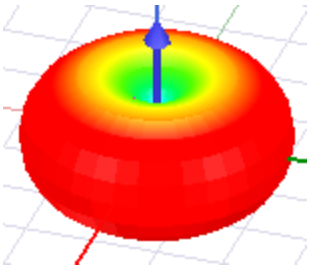


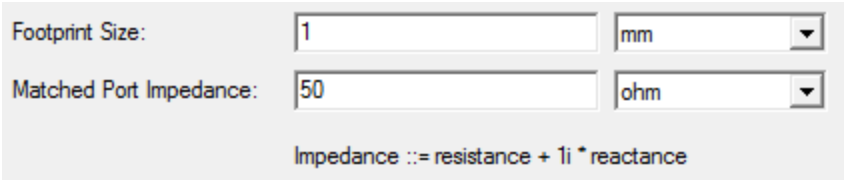
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Short Dipole

This antenna produces a far-field pattern of an ideal Hertzian(electrically short) electric dipole with a peak Directivity of 1.76 dBi. The Short Dipole is suitable as an idealized, yet still physical electromagnetic point source that stands off two or more wavelengths from any scattering geometry. It only has a far-field representation with no higher-order near-field terms. If the antenna is to be placed within one or two wavelengths of the SBR+ scattering geometry, one should consider to instead use the Wire Dipole or Horizontal Dipole in their current-sources representation since these have a valid near-field description.

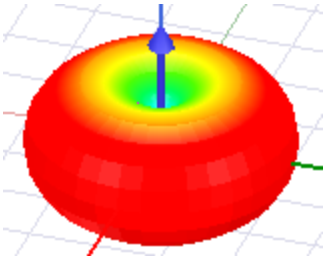


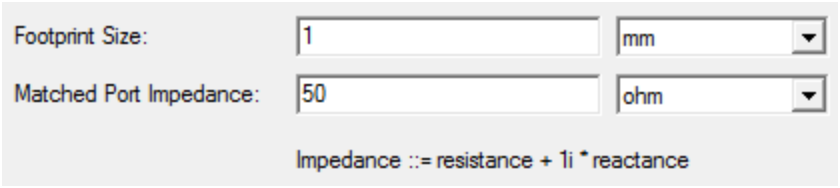


Small Loop

This antenna produces a far-field pattern consistent of an ideal Hertzian loop (electrically short) magnetic dipole with a peak Directivity of 1.76 dBi. Because it only supports a far-field representation, it should only be used when it can be placed two or more wavelengths away from any SBR+ scattering geometry.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

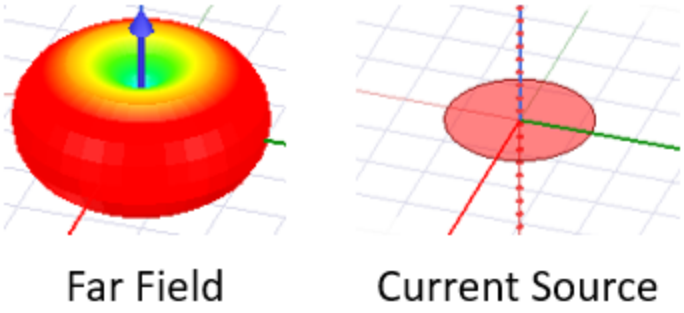




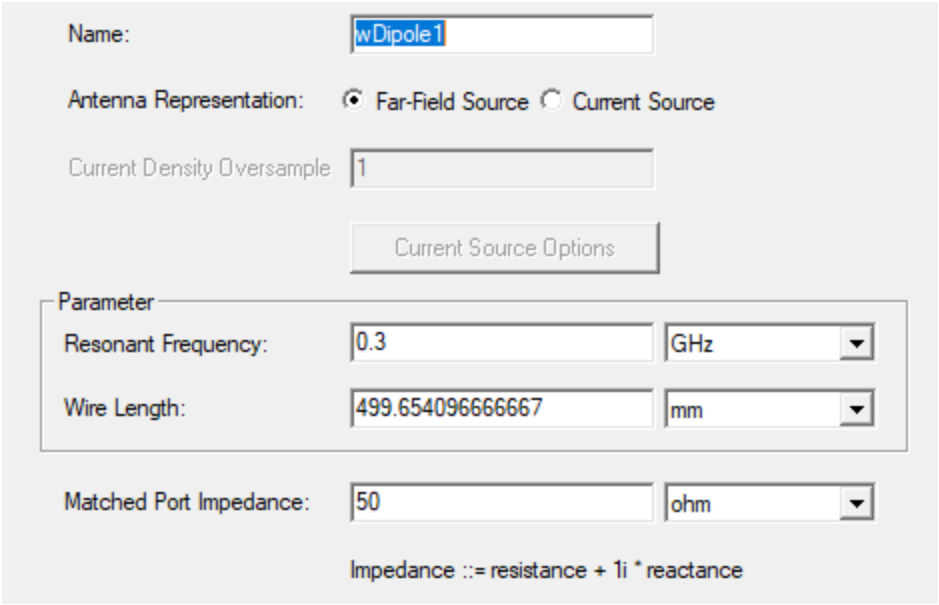
Wire Dipole

This antenna is based on an idealized sinusoidal current distribution of a wire dipole along the Z-axis of the antenna coordinate system. The Far-Field pattern will vary according to the dipole's electrical length at the visualized frequency. When operated at the user-defined Resonant Frequency, the wire is a half-wavelength long. Note that changing the Resonant Frequency will update the Wire Length and vice versa.

The Wire Dipole offers both a far-field and current-sources representation. The former is more computationally efficient to use in an SBR+ simulation, but the current-source representation should be used if the antenna is to be placed within 2 wavelengths of the SBR+ scattering geometry or its Wire Length is a significant fraction of its displacement from the scattering geometry.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



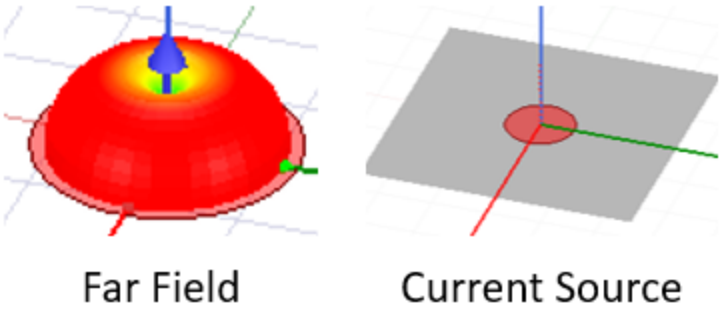
Wire Monopole

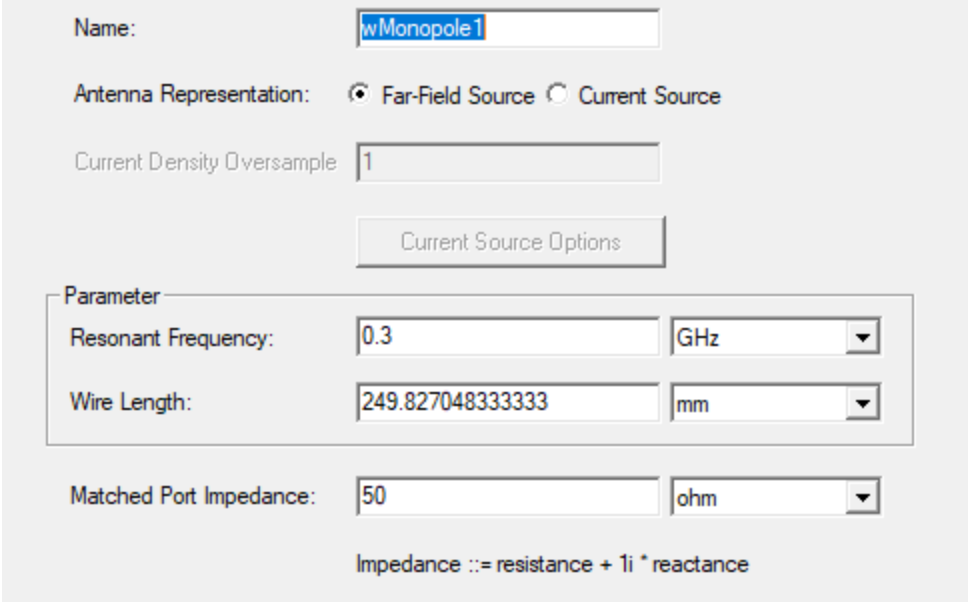
This antenna is based on an idealized sinusoidal current distribution for the monopole along the Z-axis of the antenna coordinate system. The Far-Field pattern will vary according to the monopole's electrical length at the visualized frequency. When operated at the user-defined Resonant Frequency, the monopole is a quarter-wavelength long. Note that changing the Resonant Frequency will update the Wire Length and vice versa.

The Wire Monopole is a ground-plane antenna whose intended placement is on a metal surface of the SBR+ scattering geometry or platform. In this context, use of the current-sources representation is strongly recommended.

One can also select a far-field representation. However, given its assumed ground-plane context, the Wire Monopole radiates no far fields in its lower half-space, so all direct interaction with the placement platform is ignored during SBR+ simulation with this representation of the monopole except for those surfaces of the platform or scattering environment that reside above the z = 0 plane of the monopole antenna.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

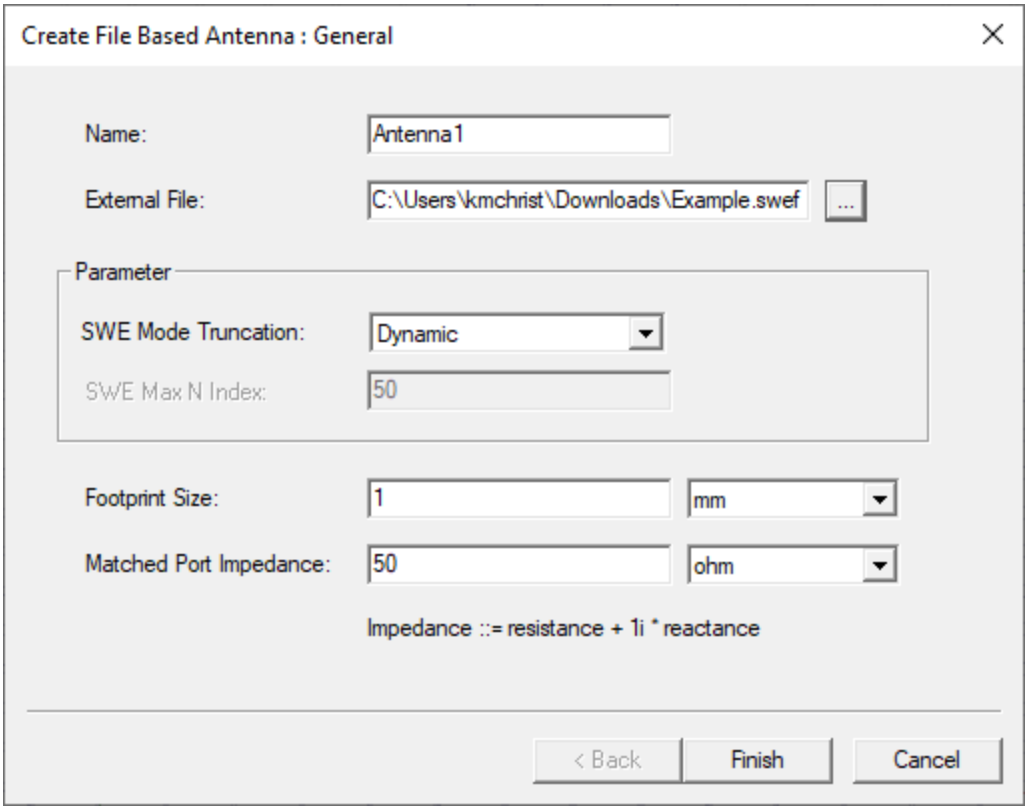




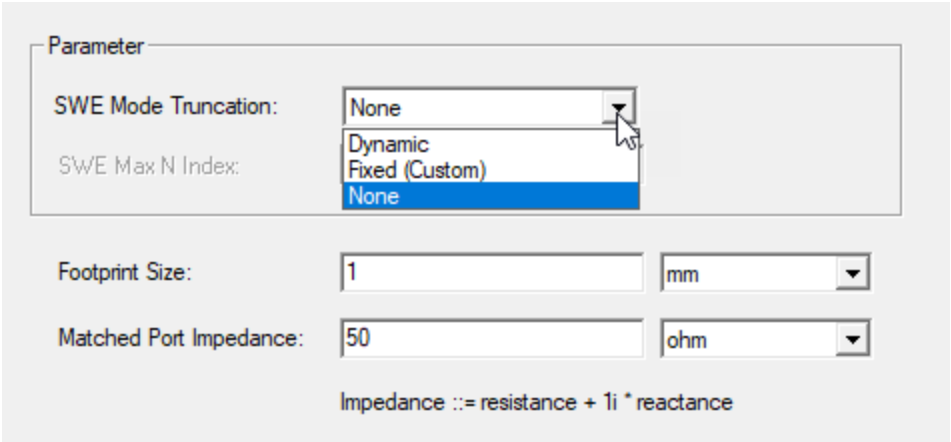
Spherical Wave Expansion Source (*.swef)

HFSS SBR+ Parametric Antennas of By File... type can import single- and multi-frequency spherical wave expansion (SWE) files in a commonly used format exported by several commercial antenna tools (e.g., Ticra Champ). Like current-source list (.csrc) files, and unlike antenna pattern table (.patdat) files, SWE antennas have high-order, near-field radiation properties, making them intrinsically more accurate than antennas imported in far-field pattern formats when illuminating nearby structures. For compatibility with SBR+, the file extension must be set to .swef. When you import a .swef file, the input dialogue includes a parameter area with a drop down menu:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

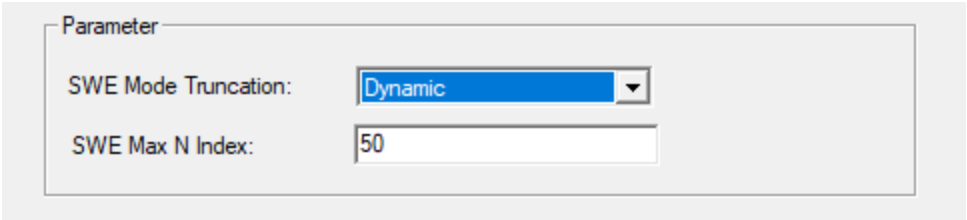


The Parameter area includes a drop down menu to select SWE Mode Truncation. You can select None, Dynamic, or Fixed (Custom).



If you select Custom, the SWE Max N Index field is enabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For editing properties, placement, and visualization, file based antennas resemble parametric antennas.

For .swef files, *you must insert into the header block(s) of the provided SWE file a Frequency key line, as described below*. HFSS SBR+ supports both the Q-coefficients format and the older AB-coefficients format. It can also import .swef files containing multiple expansions (i.e., coefficient blocks), each one representing a different frequency sample. A multi-frequency .swef file is created by concatenating, in increasing frequency order, multiple single-frequency (.swef) files. Mixing AB and Q coefficients in the same file is not permitted.

AB Coefficients

In the case of the AB-coefficients format, the header before each coefficient block must contain the following four lines, maintaining the indicated line order and with no spaces between. Keys in bold must be included as typed.

- Title
- **Frequency** f_hz (extra line relative to SWE standard format)
- MMax (integer)
- MMax+1 integers for NMax coefficients

After the header, 4 real numbers on each line are used to represent the real and imaginary parts of the a_{mn} and b_{mn} coefficients, such that the coefficients follow the sequence below:

```
a0,1 b0,1
a0,2 b0,2
...
a0,N(0) b0,N(0)
a1,1 b1,1
a-1,1 b-1,1
a1,2 b1,2
...
a1,N(1) b1,N(1)
a-1,N(1) b-1,N(1)
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
...
aM,M bM,M
a-M,M b-M,M
aM,M+1 bM,M+1
...
aM,N(M) bM,N(M)
a-M,N(M) b-M,N(M)
```

A 2-frequency AB .swef file example is shown below.

```
High-efficiency Horn
Frequency 11500000000
2
2 2 3
0.115467E-03 0.305491E-03 -0.597125E-03 -0.174065E-03
-0.404947E-04 0.252913E-03 -0.200067E-03 0.152949E-03
0.128725E-03 0.298848E-03 0.204994E-03 0.240563E-03
0.282213E-03 -0.271228E-03 0.713716E-04 0.646115E-04
-0.181136E-03 -0.262236E-03 0.568925E-04 0.129049E-03
-0.711213E-04 0.108705E-03 0.196338E-04 -0.174016E-03
-0.912608E-04 0.378009E-04 0.121696E-04 -0.177426E-04
0.649170E-04 -0.501389E-05 0.446827E-05 0.553745E-04
0.502546E-04 0.664317E-05 -0.214199E-04 -0.201388E-04
-0.256921E-04 -0.297337E-04 0.147851E-04 0.432019E-03
High-efficiency Horn
Frequency 12000000000
2
2 3 3
0.233219E-04 0.609080E-04 0.312162E-03 -0.144566E-03
-0.276714E-03 -0.188706E-03 -0.467364E-03 -0.446705E-04
0.247382E-03 0.211029E-03 -0.132354E-03 0.156872E-03
0.173807E-03 -0.105876E-03 -0.807569E-04 0.175449E-03
-0.610848E-04 -0.452046E-04 0.377000E-04 -0.306724E-03
-0.508518E-04 0.126624E-04 -0.227225E-03 0.462618E-04
-0.164177E-04 0.444172E-04 0.112475E-03 0.291202E-04
0.557026E-04 0.190620E-04 0.115186E-03 -0.238145E-04
0.186932E-04 -0.565990E-04 -0.556033E-04 0.775226E-05
-0.643883E-04 0.521515E-04 0.124740E-04 0.252009E-04
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
0.181098E-04 -0.447097E-04 -0.396963E-04 -0.279198E-04
0.182588E-04 0.209599E-04 0.316011E-04 -0.109181E-04
```

The above example will not produce any meaningful SWE representation. It only serves to illustrate the file format.

Q Coefficients

Q-coefficient files have a different header than AB-coefficient files. Several header lines contain ignored text and dummy data, but all the lines in the header must be present for HFSS SBR+ to parse it correctly. As with AB-coefficient files, an extra line identifying the frequency sample must be added to the header for compatibility with HFSS SBR+ . Keys in bold must be included as typed.

- Text string
- Text string
- 4 integer control numbers, NTHE NPHI NMAX MMAX
- Text string
- 5 real data numbers (dummy data items)
- 5 real data numbers (dummy data items)
- Text string
- Text string
- **Frequency** f_hz (extra line relative to SWE standard format)

The only data used by HFSS SBR+ Parametric antennas are the values of NMAX and MMAX to determine the total number of coefficients to read. For each M mode from 0 to MMAX, the following block of coefficients follows the header:

- M, POWERM (integer, real)
- Q1MN Q2MN (4 real numbers per line)

where that M is the azimuthal mode index |m|, POWERM is the power in all coefficients +/-m, and the Q1MN and Q2MN coefficients are

- for M = 0

```
Q101 Q201
Q102 Q202
Q103 Q203
...
Q10NMAX Q20NMAX
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- for M different than 0, M = m

```
Q1,-m,m Q2,-m,m
Q1,m,m Q2,m,m
Q1,-m,m+1 Q2,-m,m+1
Q1,m,m+1 Q2,m,m+1
...
Q1,-m,NMAX Q2,-m,NMAX
Q1,m,NMAX Q2,m,NMAX
```

A 2-frequency Q .swef file example is shown below.

```
File created by SWEP2 on 2015/ 9/24 at 7: 0:26
SWE
180 8 3 2
Rotation angles (Theta, Phi, Chi)=(0.00000, 0.00000, 0.00000)
0.0000 180.00 0.0000 359.99 0.00000
0.0000 180.00 0.0000 359.99 0.00000
SWEP_DUMMY_FILE_NAME
SWEP_DUMMY_FILE_NAME
Frequency 11603000000
0 3.0082396525702973E-29
-3.1915279054525894E-16 -1.2892349207803435E-16 -3.8509178493100355E-
16 -1.8049288890924806E-16
-5.4172934656081795E-16 -7.3271596763214834E-16 -4.3842840686185935E-
16 -8.8646620346315665E-16
1.1107004432486917E-15 4.3339505229049944E-17 1.1217237521873848E-15
1.2951602867000140E-16
1 0.4999765885681955
1.4793217033613612E-18 -3.1357721221466865E-18 1.5274090844190164E-18
4.2809100101664339E-18
-1.9252328312778913E-01 1.1651586264459781E-02 -2.0192491940241922E-
01 -6.7317664009445416E-05
1.8666136540373346E-18 2.8787677569168861E-19 -3.1057427227729180E-18
3.3082674111624142E-18
9.8138549095459840E-03 -2.4230456057802729E-01 2.4936045948149160E-02
-2.5035917896031079E-01
-4.2987978961492822E-18 1.5321395932099931E-18 -1.7352837081002618E-
18 -6.5033976194751674E-18
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

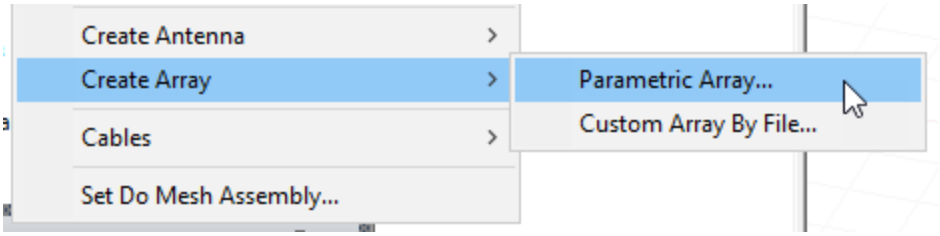
```
2.7508153381481415E-01 2.0893222682079500E-02 2.5940539672443474E-01
2.1630794936559665E-02
2 3.6104890168479111E-33
-1.7470258951002181E-18 -2.0774762527596600E-18 -1.8082213102957466E-
19 -9.1362725841771620E-19
-1.0554789826723718E-18 -1.6897058625735897E-18 3.4220101385728232E-
21 1.9132250922702059E-18
-2.9328385823772792E-18 1.7156784729134458E-19 -3.1341623772531889E-
19 -1.1329591729328374E-18
-2.0100295867262535E-18 -8.8694426018338640E-19 -8.7960399314802765E-
19 1.7312637664486764E-18
File created by SWEP2 on 2015/ 9/24 at 7: 0:26
SWE
180 8 3 3
Rotation angles (Theta, Phi, Chi)=(0.00000, 0.00000, 0.00000)
0.0000 180.00 0.0000 359.99 0.00000
0.0000 180.00 0.0000 359.99 0.00000
SWEP_DUMMY_FILE_NAME
SWEP_DUMMY_FILE_NAME
Frequency 11703000000
0 2.8820236773030870E-29
-4.9205799476449772E-16 -2.4495463494826524E-16 -5.5813128445449028E-
16 -2.7073933336387207E-16
-4.3362371199214033E-16 -9.0688615665946252E-16 -3.7356502612065278E-
16 -9.8376127457496648E-16
7.8526158604867005E-16 3.1041135484342719E-16 7.9694440919737040E-16
4.1316994985027605E-16
1 0.4999777050592423
-5.3472196810921811E-19 -4.3007910149543931E-18 4.1626947467127237E-
19 7.5718687702418272E-18
-1.9072752874637075E-01 1.1766580158570537E-02 -1.9948262852927875E-
01 1.1232179777612324E-03
2.8981563155590727E-18 3.5754557596140601E-19 -5.2146054685044776E-18
2.4253069313164503E-18
1.0945650686875724E-02 -2.4126104316529043E-01 2.4643389614931455E-02
-2.4856384063328121E-01
9.5675273891171790E-19 5.3403417546511193E-18 -5.4286022965855179E-19
-1.1503424690493544E-17
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
2.7311952122770955E-01 2.0597687831541416E-02 2.5840120731726490E-01
2.2084602270527109E-02
2 5.1148291304191848E-33
-6.9532898777596087E-18 6.4300740906263065E-19 3.2739851403263512E-18
5.7078571449140461E-18
-7.0053476956511754E-18 -4.3995227310233528E-19 -5.6372427909811814E-
18 5.8778959090251952E-18
-3.1440268835569142E-19 -3.9850347618643944E-18 2.6093602155778856E-
18 9.1764376219979173E-19
-4.5649017014803075E-19 -7.7811420149503691E-18 -3.8743135072928263E-
18 -2.2314829902343887E-18
3 2.3279879002995588E-32
-1.4634393879898799E-18 -1.4073355212909901E-18 -5.4072438315811004E-
18 3.2319825198116401E-19
-2.5652903303343589E-18 3.1763737384608062E-19 -2.5628199228530487E-
18 3.4049505993075118E-18
```

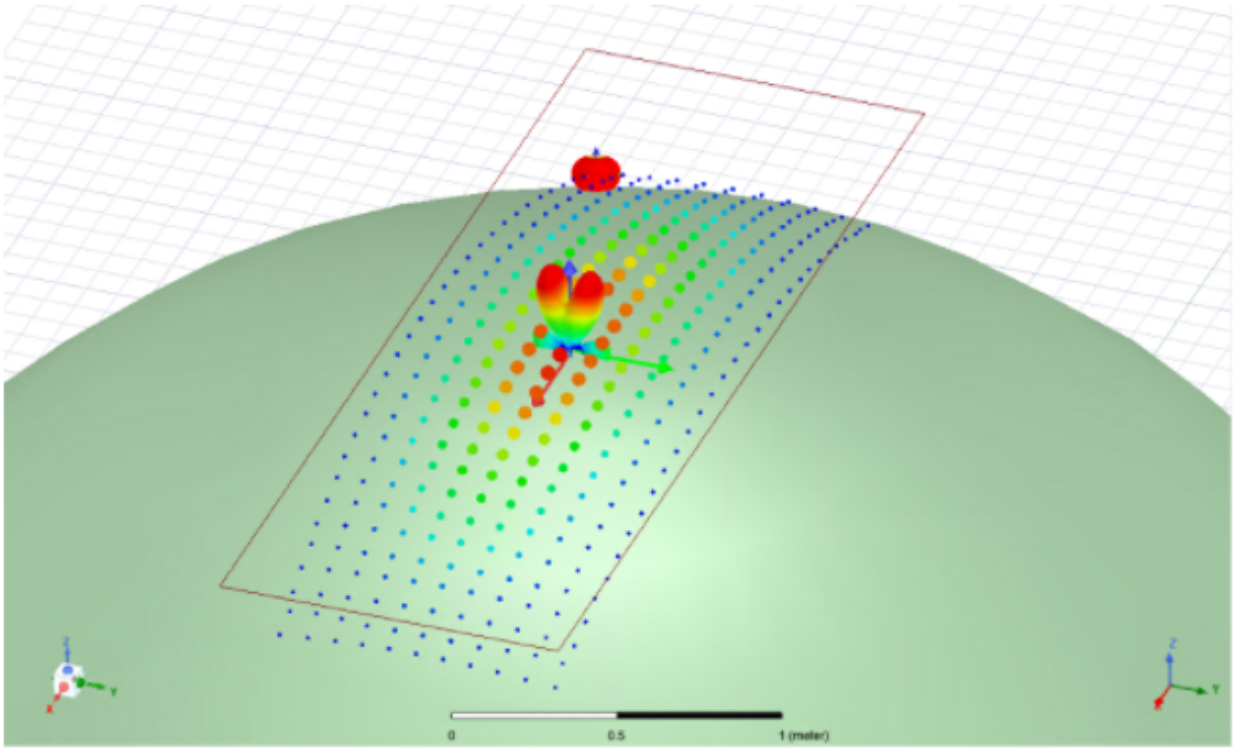
Parametric Array Antennas for SBR+ Solution Type

For SBR+ solution type, you can access a range of parametric array antennas via the **Project** tree **3D Components** icon right-click menu option, under **Create Antenna**.



As with other Parametric antennas Parametric Array antennas are antenna sources that either have no physical structure definition or a highly idealized physical structure. No model parts are constructed or used in the simulation. They create far-field pattern or current distribution sources that can provide S-parameter outputs and composite far-field radiation patterns.

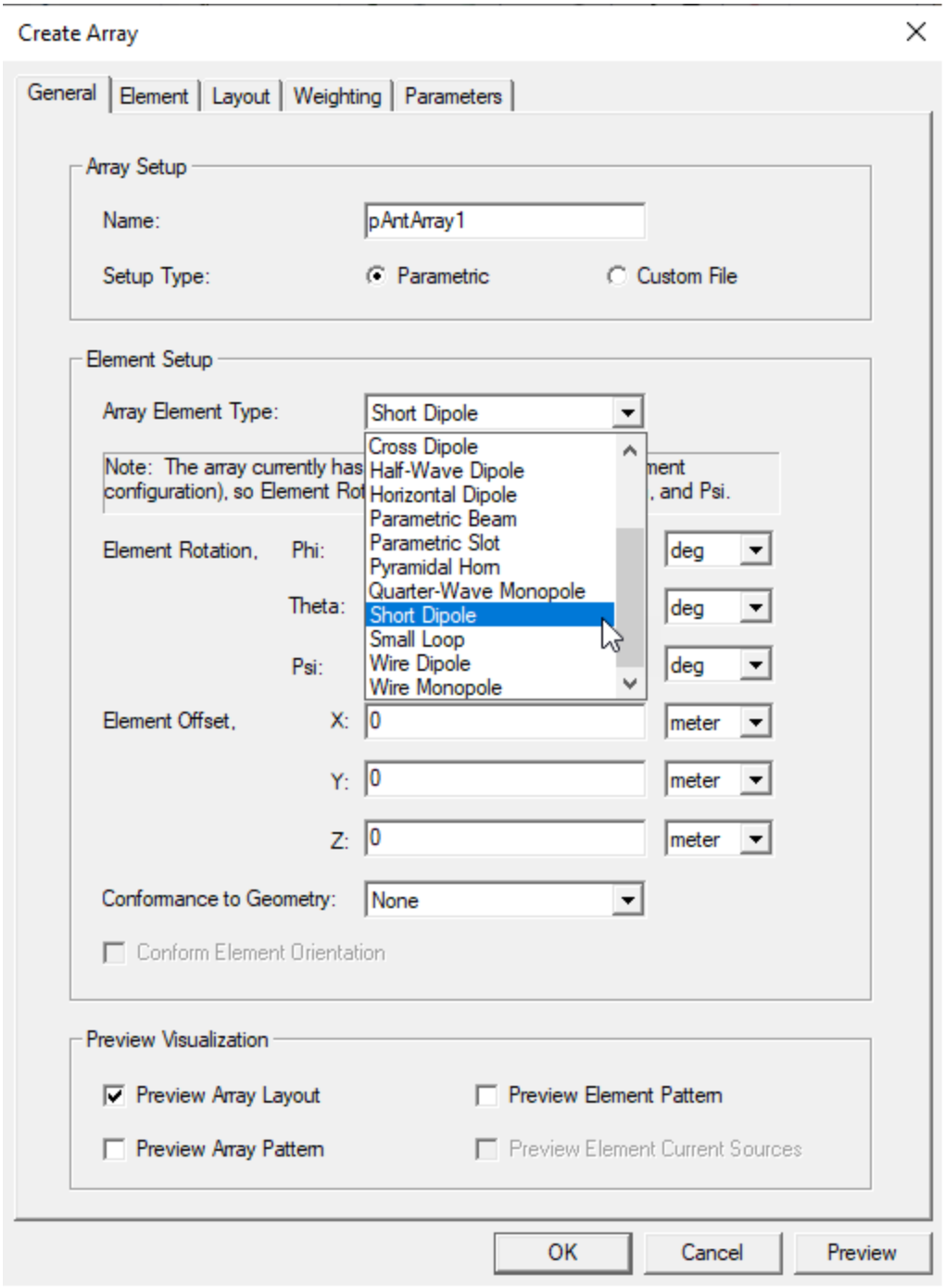
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Parametric Array antennas are very quick to define and place, copy and duplicate, and are thus intended for systems studies where the specific antenna structure is either unknown or a simple, topologically representative antenna type is sufficient. The editable parameters depend on the antenna element type.

When you create a Parametric Array you specify General definition properties for the array, which includes the type of antenna used for the array elements by selecting from the fourteen existing parametric antenna types. You also have options for Element Rotation, Element Offset and Conformance to Geometry. The array definition includes settings to configure the Geometry layout that specifies the overall size of the array and spacing of the array elements, and various other geometry configuration options for how the individual antenna elements are placed within the array. Several Weighting operations and functions for the individual element contributions to the array are also specified with the definition, including beam steering, edge taper functions, sum/difference patterns, etc.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

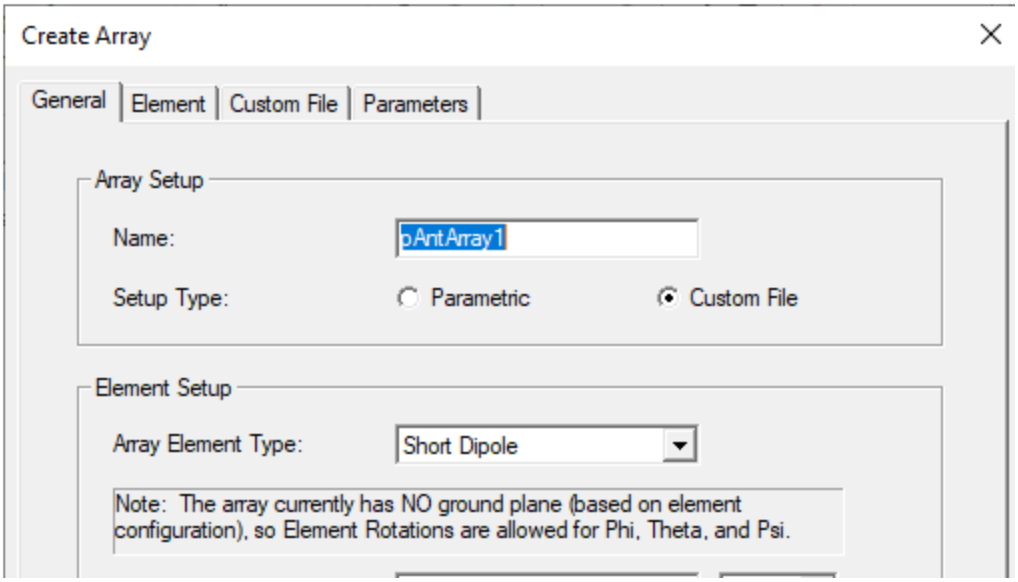


In addition to the 12 predefined parametric antenna types, there are three additional ways to specify the antenna element type for the array: 1) an external file with a far-field pattern, 2) an external file with a near-field pattern, and 3) link to another HFSS source design that provides (either by definition or by solution) a suitable excitation source for SBR+ (radiation or FEBI boundary, etc). Note that in terms of the antenna elements, these parametric antenna arrays are

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

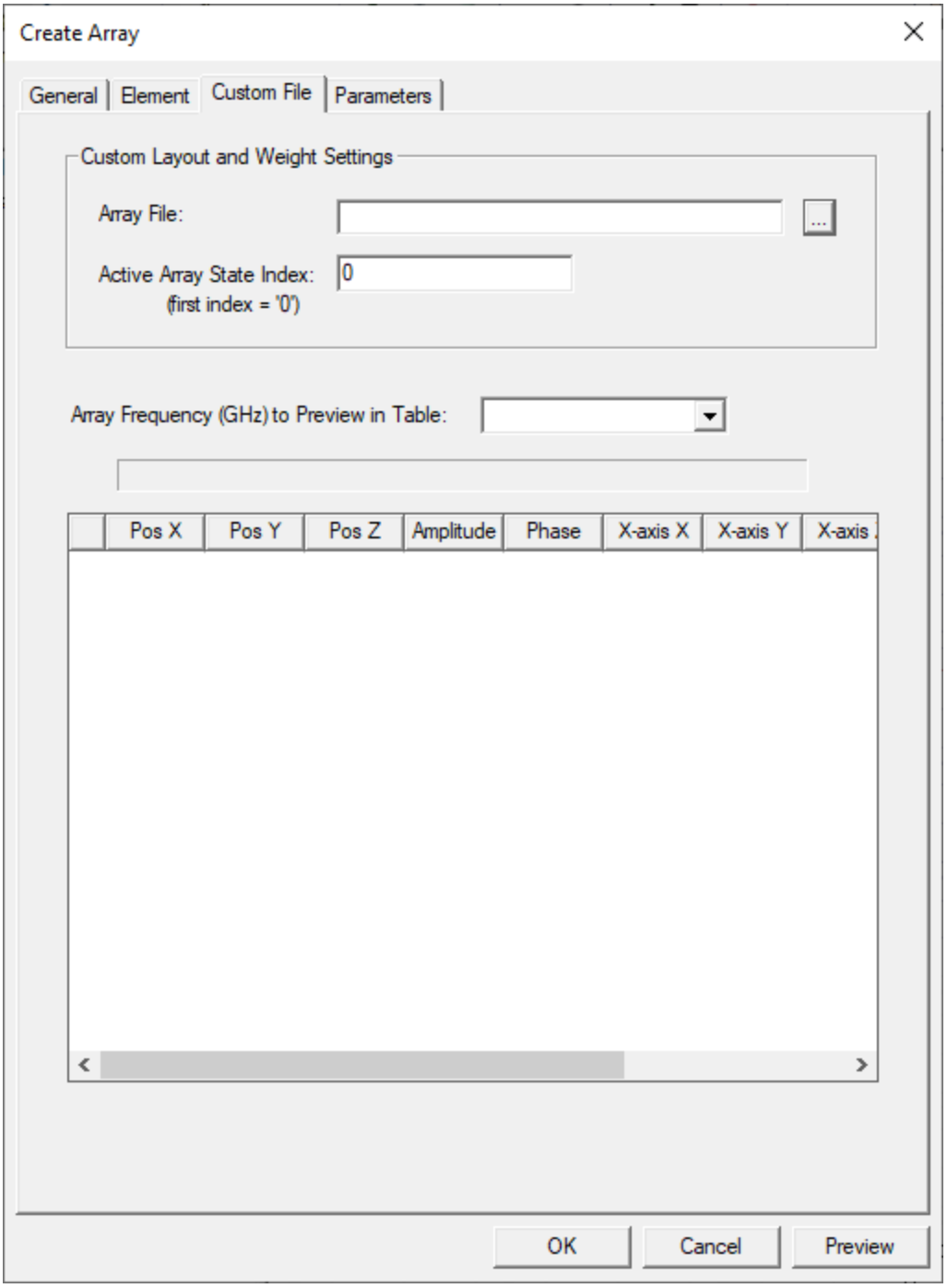
uniform, meaning they are defined with the same antenna type for all elements of the array. When a File Based Antenna is selected as the Array Element Type, the SBR+ solver will automatically interpolate the source fields from the file-based antenna in order to represent the antenna at all frequencies of the user defined frequency sweep in solution setup. The file-based antenna requires at least two frequencies, and the min/max frequencies of the user defined sweep should fall within the frequency band of the file-based antenna.

If you select **Create Array** > **Custom Array by File...** the **Create Array** window opens with **Custom File** selected.



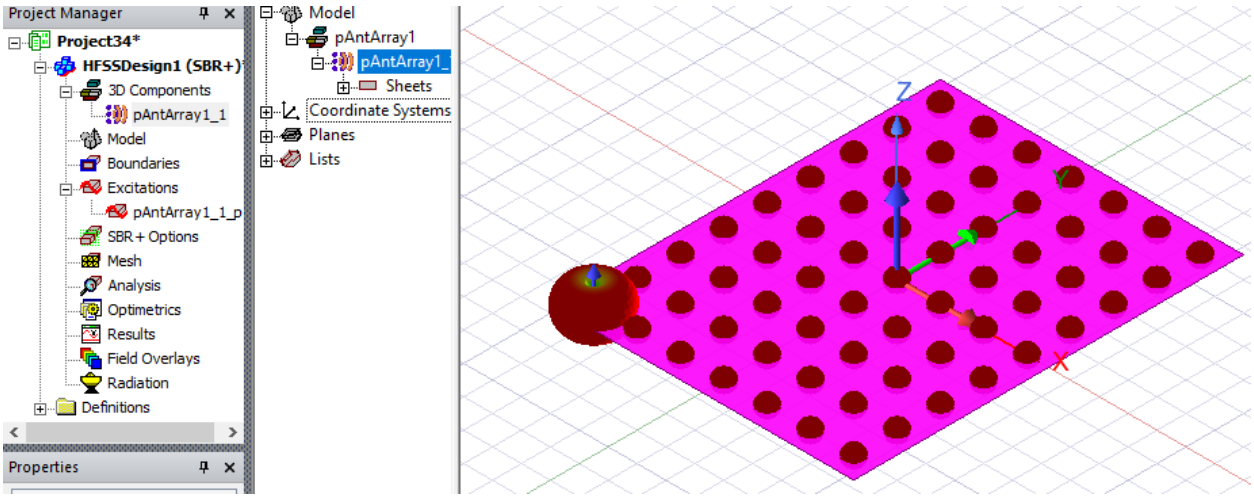
This dialog includes a **Custom File** tab that lets you specify a custom array layout geometry and weighting via an external file format, the “*.sarr” format from EMA3D. The array element positions, orientations, and complex weights are specified in this file format. The complex weights can be specified over a set of frequencies and specified “states” which are indexed across all elements.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After being created, the parametric array model is listed under the **Project** tree under **3D Components** and automatically includes a port to support S-parameter output. The generated port is shown under **Excitations**, and the port post-processing setting can be modified from the **Properties** window when this port is selected.

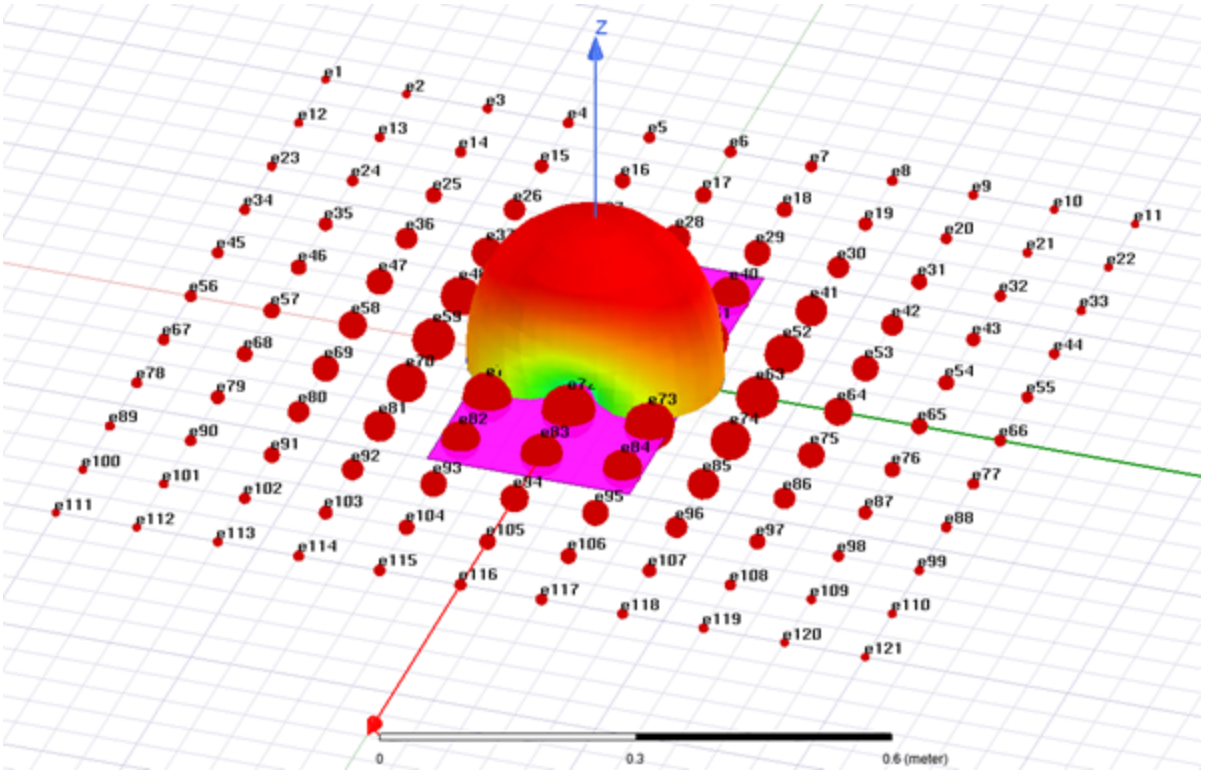
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selecting the antenna under **3D Components** causes the **Properties** window to display with tabs for **General** antenna **Properties** and **Visualization** settings. In the model window, the scaled far field pattern of the antenna array can be displayed at the array origin. In addition, the far field pattern or current source representation (for a near field antenna) of an individual element can be displayed at a specified element number.

In the model window, the scaled far field pattern of the antenna array can be displayed at the array origin.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



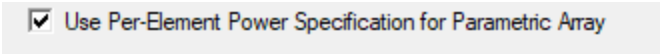
The array definition will consist of a geometry layout that specifies the overall size of the array, type and spacing of the array elements, and various other configuration options for how the individual antenna elements are placed within the array. The array definition also provides methods for weighting the individual element contributions to the array, including beam steering, edge taper functions, sum/difference patterns, etc.

Using with Other SBR+ Features

Power specification

A Parametric Array by default is treated like a single antenna, and Edit Sources defines a total input power as a composite port. All analysis options including coupling, far- and near-field analysis, wedge corrections (PTD/UTD), and Creeping Wave are available. The Parametric Arrays do not analyze their element couplings and are assumed to have efficiencies as same as their element antenna. Results are normalized to radiated power to align with the Antenna Parameters.

On the other hand, you may select Use Per-Element Power Specification for Parametric Array in Edit Sources when only a Parametric Array exists in the scene.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you select the option, Edit Sources defines the input power of an element before applying the Aperture Taper configured in Array Weightings. The Parametric Array with this option gives results similar to HFSS post-processing feature of Antenna Array and coupling analysis and Enhanced Radiated Power Accuracy are not supported.

Visual Ray Trace (VRT) plots for Shooting-Bouncing Rays (SBR) Creeping Wave (CW) rays are available via Field Overlays. Parametric arrays can be specified as the VRT “launch from” object/location for SBR plots. For parametric antenna arrays, the array centroid or an array element # can be specified as the “launch from” location.

There are 2 primary workflows described below: the first is for creating and using parametric antenna arrays in an SBR+ simulation, and the second is for creating visual ray trace (VRT) plots for SBR rays. The summary workflows (no screenshots or detailed explanations) are described first, followed by both of the more detailed workflows.

Limitations:

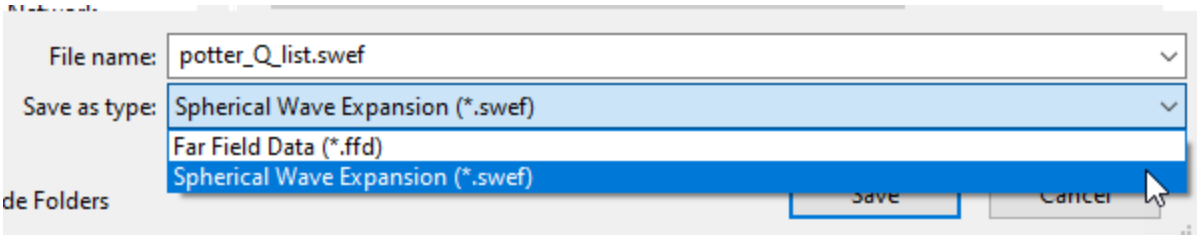
After they are created, parametric antenna arrays appear as a single port in the source excitations. Since the arrays are analytically defined, their individual element ports are not available for separate analysis. This means that in the “Select Tx/Rx” matrix and “Edit Sources” configuration dialogs these parametric antenna arrays appear as a single composite antenna/port for the whole array. In addition, the only source context available for reports and post-processing will be the single composite port for the array.

The outputs from SBR+ simulations with parametric antennas include:

- S-parameters.
- Composite far-field radiation patterns.
- Incident field reporter outputs provide the free space radiation patterns of the antenna.

Prerequisites

- The Solution Type must be SBR+.
- If you use a File-based parametric antenna, it must be defined with at least two frequencies whose min/max meet or exceed the extents of the simulation frequency sweep. A file-based can be in far field wave format, .ffd, or, for Spherical Wave Expansion Import, you can import .swef format files.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Primary Workflows for Parametric Antenna Arrays

There are three primary workflows for using Parametric Antenna Arrays in the SBR+ solution type:

- Create and use Parametric Antenna Arrays in an SBR+ simulation.
- Create and Use Custom Array by File... in SBR+
- Parametric Antenna Arrays for Visual Ray Trace (VRT) plots.

Create and Use Parametric Array in SBR+ Simulation

This chapter provides a summary and a detailed workflow for using Parametric Antenna Arrays for SBR+ simulation.

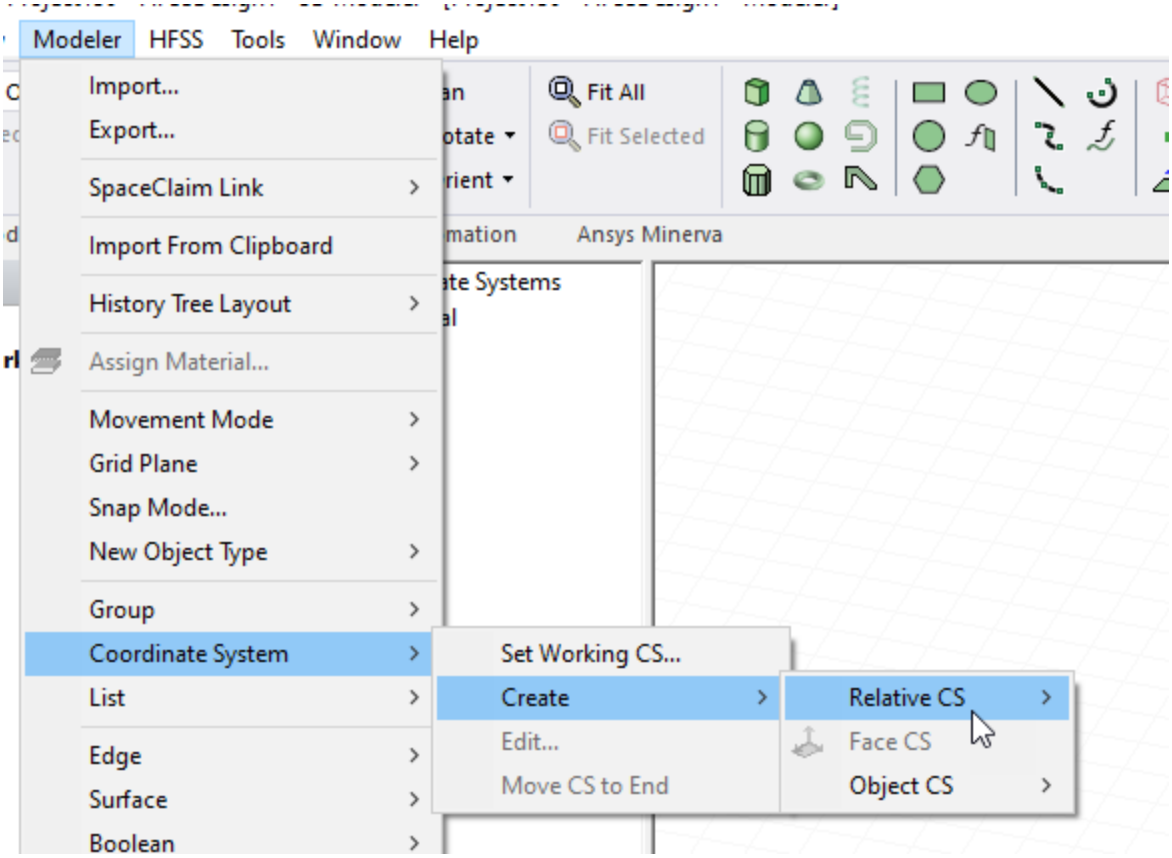
Summary Workflow for Parametric Antenna Arrays

1. Create an HFSS design and select the SBR+ Solution Type.
2. Add model objects, components.
3. Add an antenna local coordinate system .
4. Add the parametric antenna array by right-click on **3D Components** in the **Project** tree and select **Create Antenna>Parametric Array…** or right click on **Excitations** in the **Project** tree and select **Create Antenna Component>Parametric Array...**.
5. In the **Create Parametric Array** dialog
    a. Select the array element type
    b. Configure geometry conformance for the array.
    c. Configure the array element
    d. Configure the array layout geometry
    e. Configure the array weighting
    f. Configure any component instance parameters for the array component
    g. Visualize a preview of the array layout geometry and array pattern
    h. Close the dialog with **OK** to save and create the parametric array 3D component
6. Place and orient (Translate, Rotate) the 3D component at the desired geometry location
7. Optionally modify the 3D component definition in the model tree
8. Optionally modify the 3D component instance in the project tree
9. Add an Analysis Setup
10. Run simulation
11. Generate far field radiation pattern

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

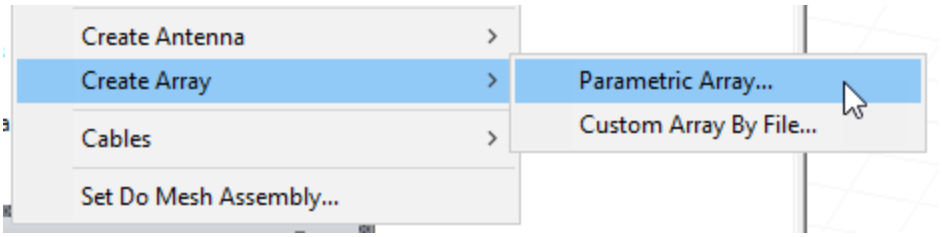
Detailed Workflow for using Parametric Antenna Arrays in SBR+ Simulation

1. Create an HFSS design and select the SBR+ Solution Type.
2. Add model objects, components.
3. Add an antenna local coordinate system. Creation of an antenna local coordinate system (CS) can be very useful in placing and orienting parametric antennas and arrays via 3D components. The CS can be created as a simple relative CS, or as something more advanced and suitable for array placement on a surface, like a face CS . After creating the new CS, make sure it is selected/active when creating the parametric array.



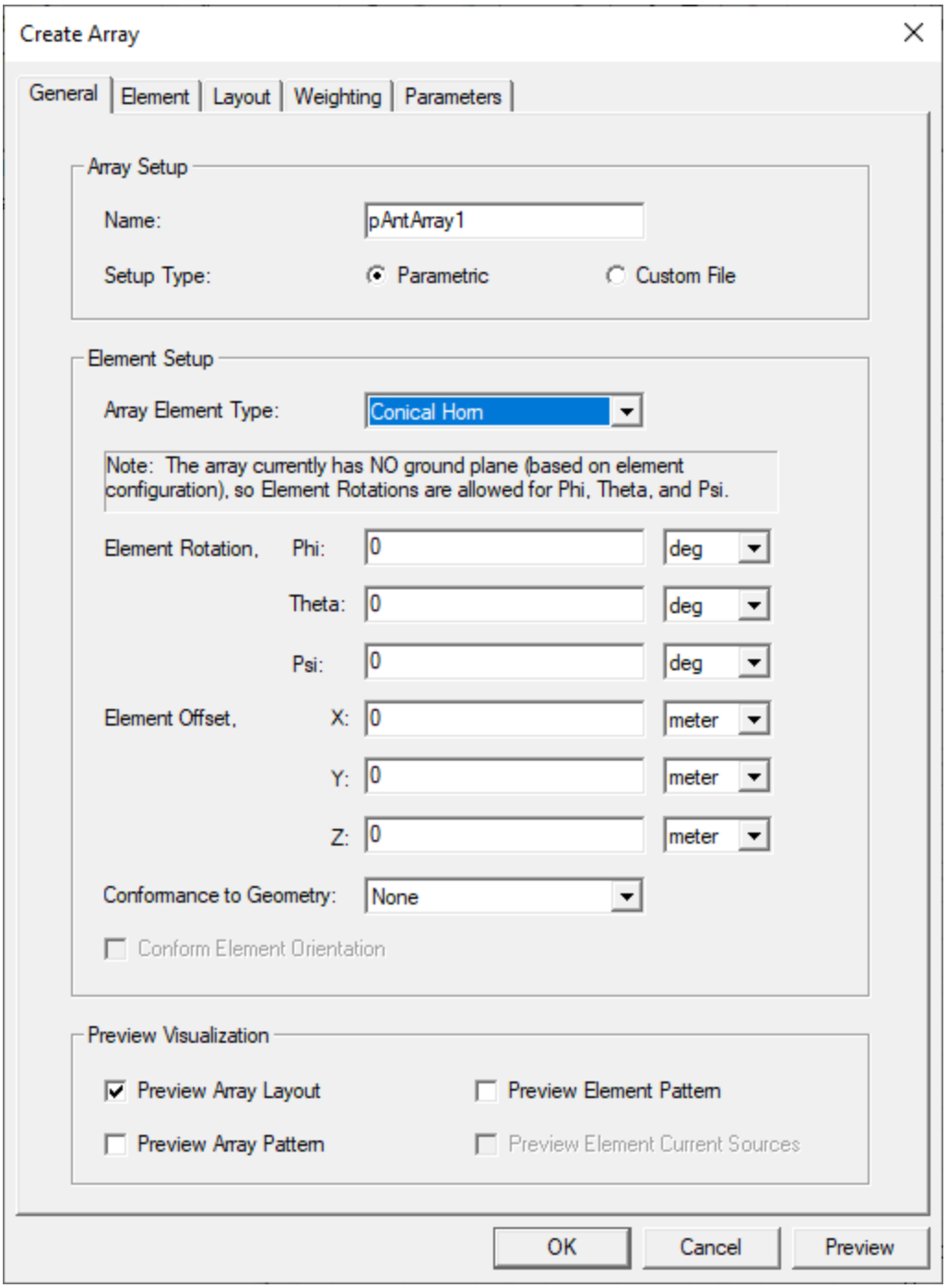
4. Add the parametric antenna array by right-click on **3D Components** in the **Project** tree and select **Create Antenna>Parametric Array…** or right click on **Excitations** in the **Project** tree and select **Create Antenna Component>Parametric Array...**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



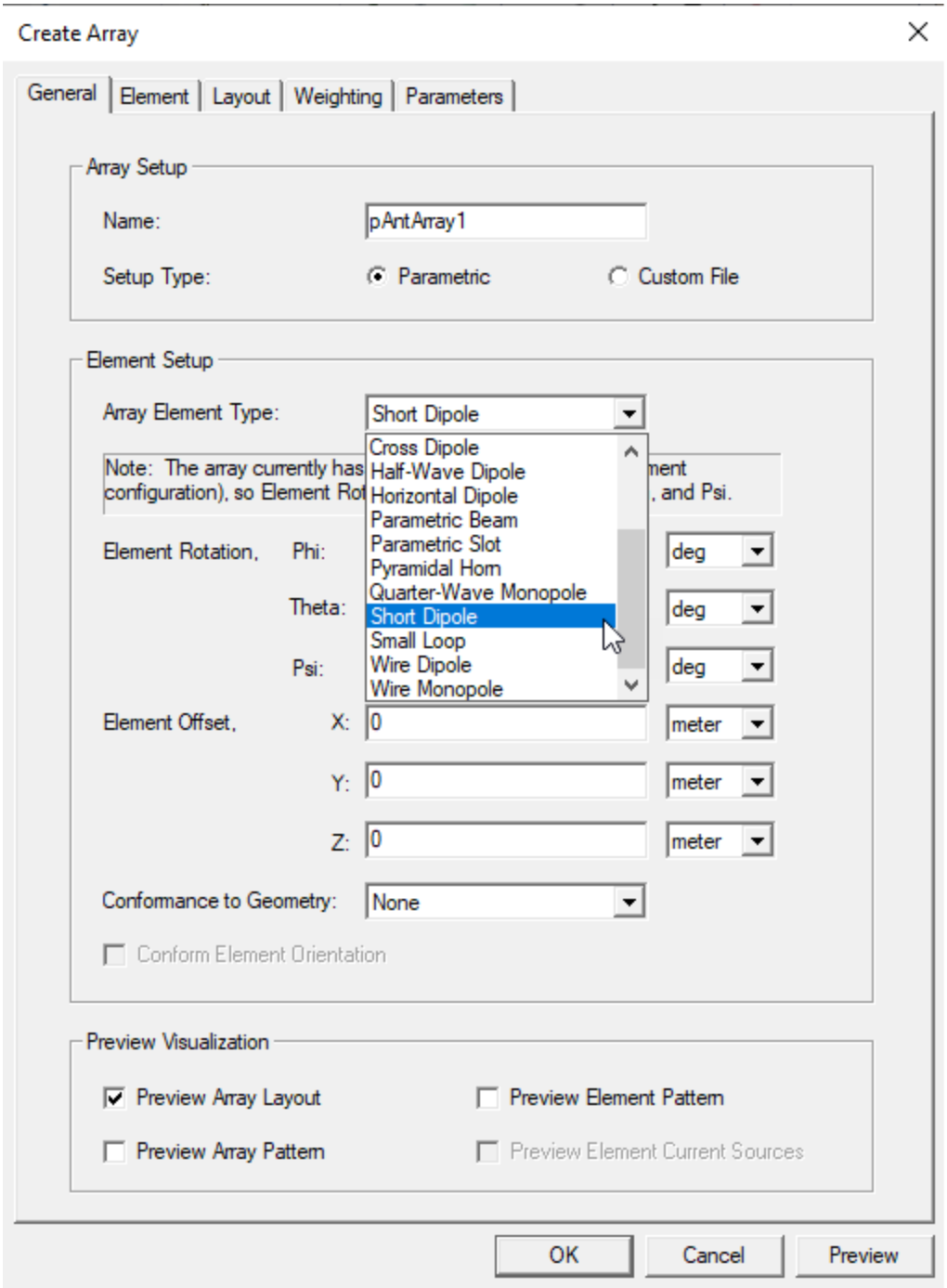
This opens the **Create Parametric Array** dialog box. This figure shows the appearance with **Options>HFSS>Boundary Assignment** Use wizards for data input when creating new boundaries checked. You can choose to view this information as either a wizard or as a multitab dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



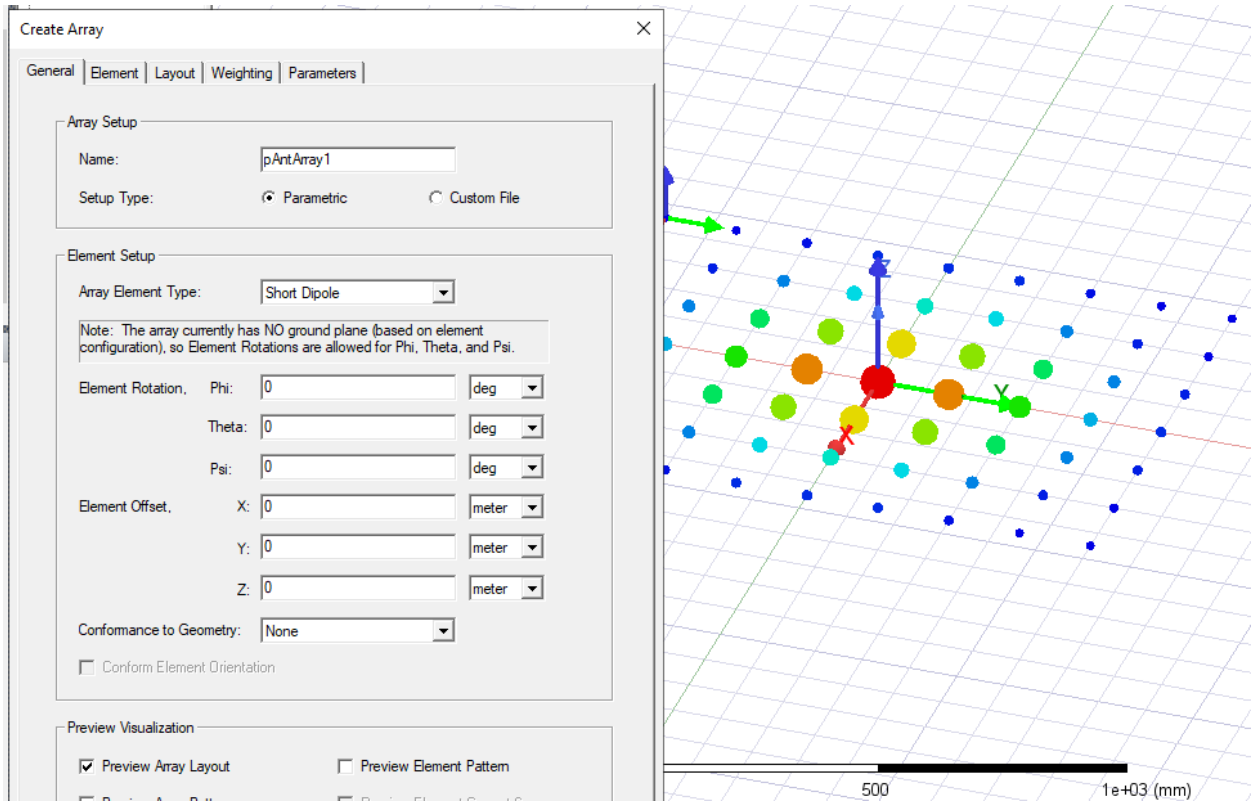
5. In the **Create Parametric Array** dialog
    a. Select the Array Element Type of antenna from the drop-down menu. You can select any of the fourteen parametric antenna types as the Array Element.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Specify the Element Rotation for Theta and Phi and select the units from the menus. Click **Preview** for a display in the Model window.

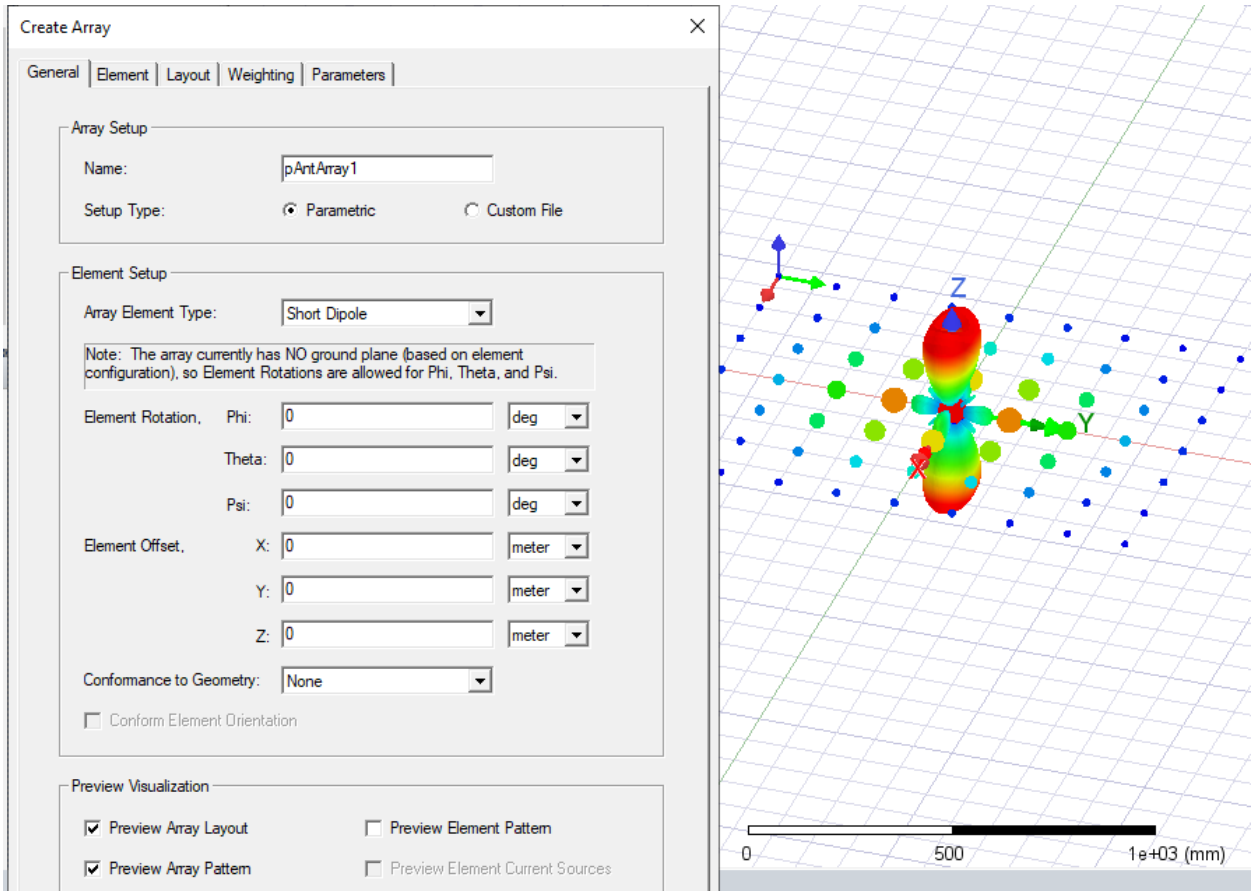
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. You can also specify an Element Offset for X, Y, and Z, and specify Units. After you have specified a value, and the cursor to another text box, the **Preview** button is enabled, allowing you to view the visualization for the values you have specified.

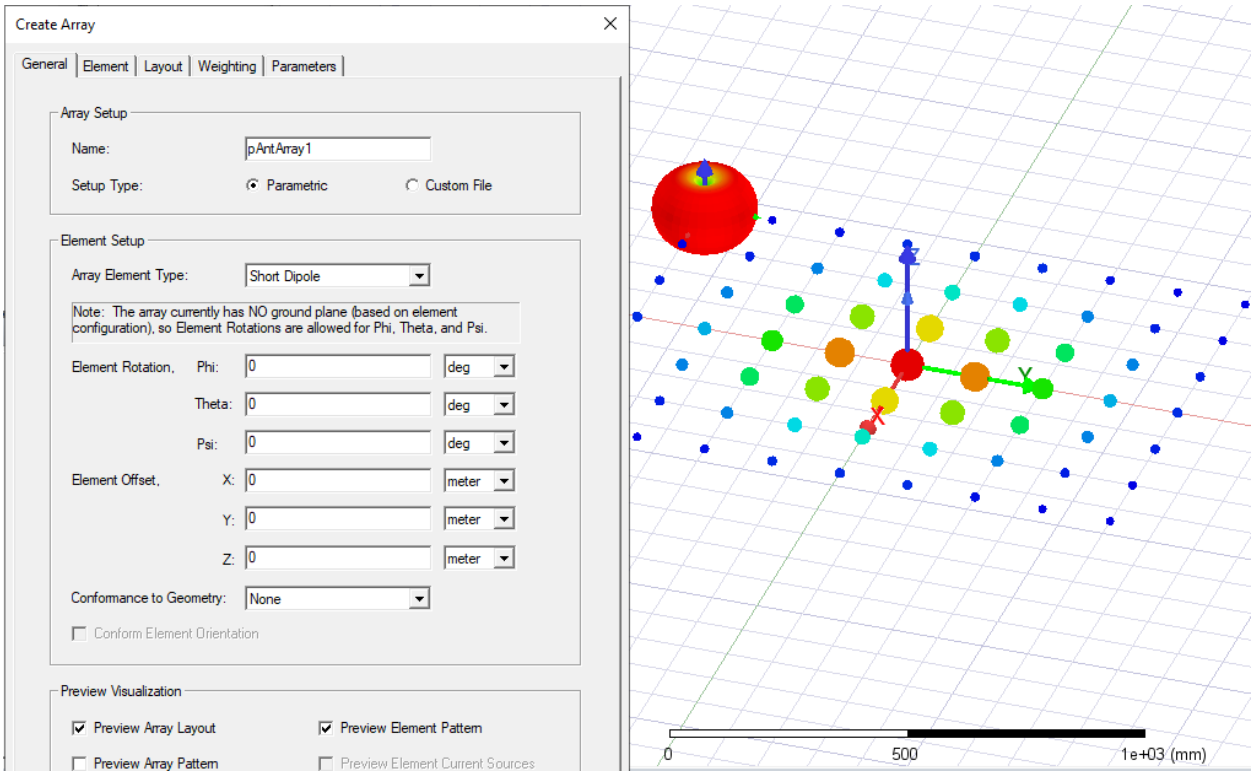
c. Use the checkboxes to Preview Array Layout, Array Pattern, Element Pattern, and Current Sources (when an element type with current sources is selected). The next figure shows both Array Layout and Array Pattern after clicking **Preview.**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

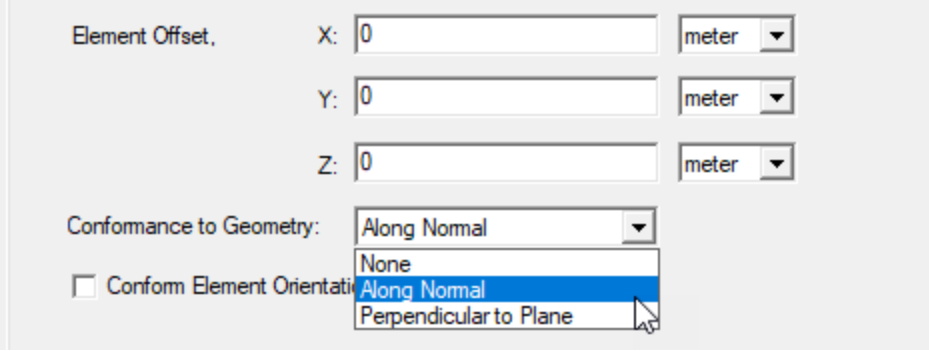


The next figure shows the scaled far field pattern of the antenna array, and then clicking **Preview**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

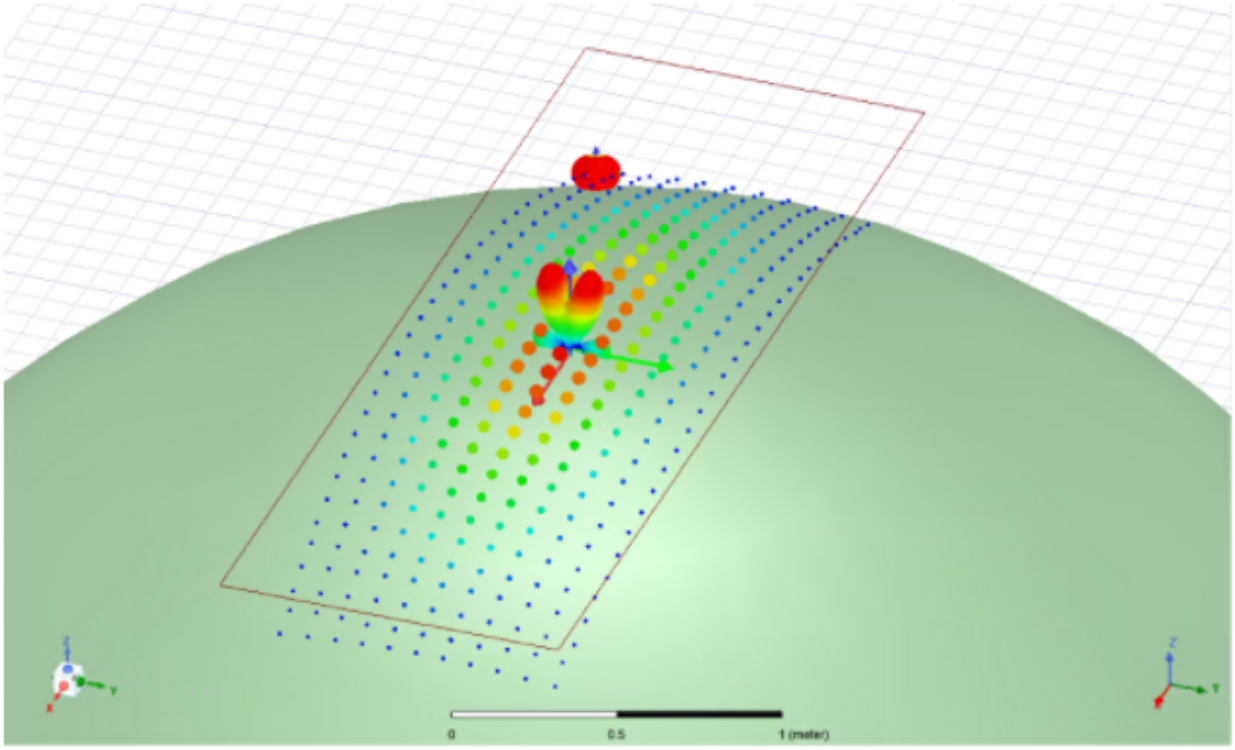


d. You can use the Conformance to Geometry dropdown to select None, Along Normal, or Perpendicular to Plane. Making the selection Along Normal enables the Conform Element Orientation checkbox.



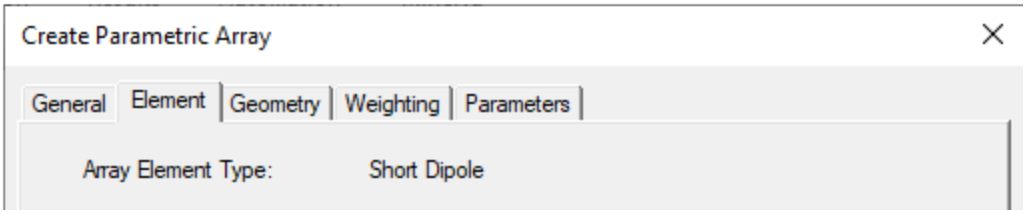
The following figure shows Conformance Along Normal relative to a sphere.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For a technical discussion, see Technical Notes: Current Conformance in SBR+.

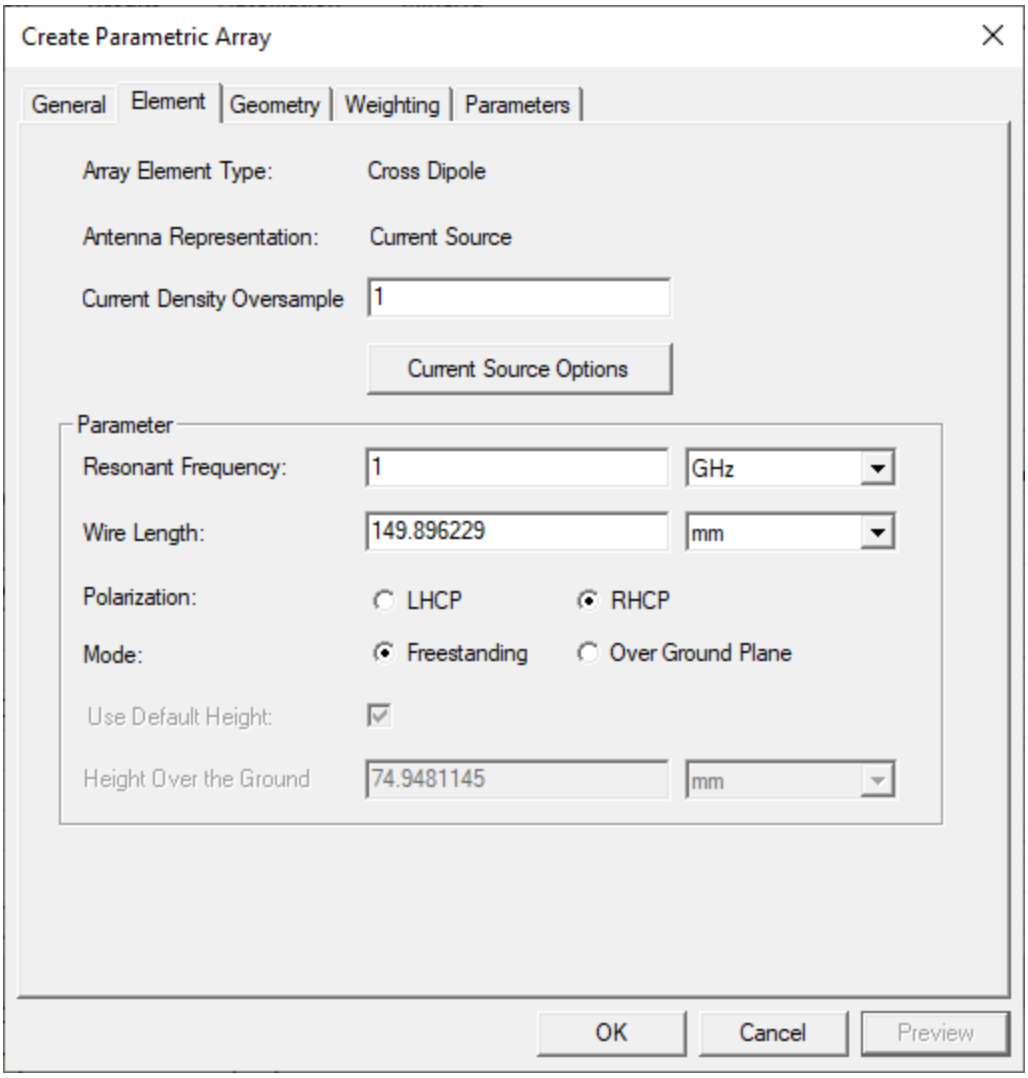
e. Configure the array element. The parameters available on the **Element** tab (or the Element wizard page) will vary depending on the Antenna Element Type chosen. For example, the Short Dipole antenna only shows the Array Element Type as Short Dipole with no additional parameters, since this simple antenna element type has no parameters to configure. Most of the antenna element types will have some additional parameters to configure on this tab/page.



The Cross Dipole **Element** tab shows the same parameters as the Cross Dipole Parametric antenna. Component variables may be defined for many of the antenna array properties. If you specify variables for any of the parameters, these will be

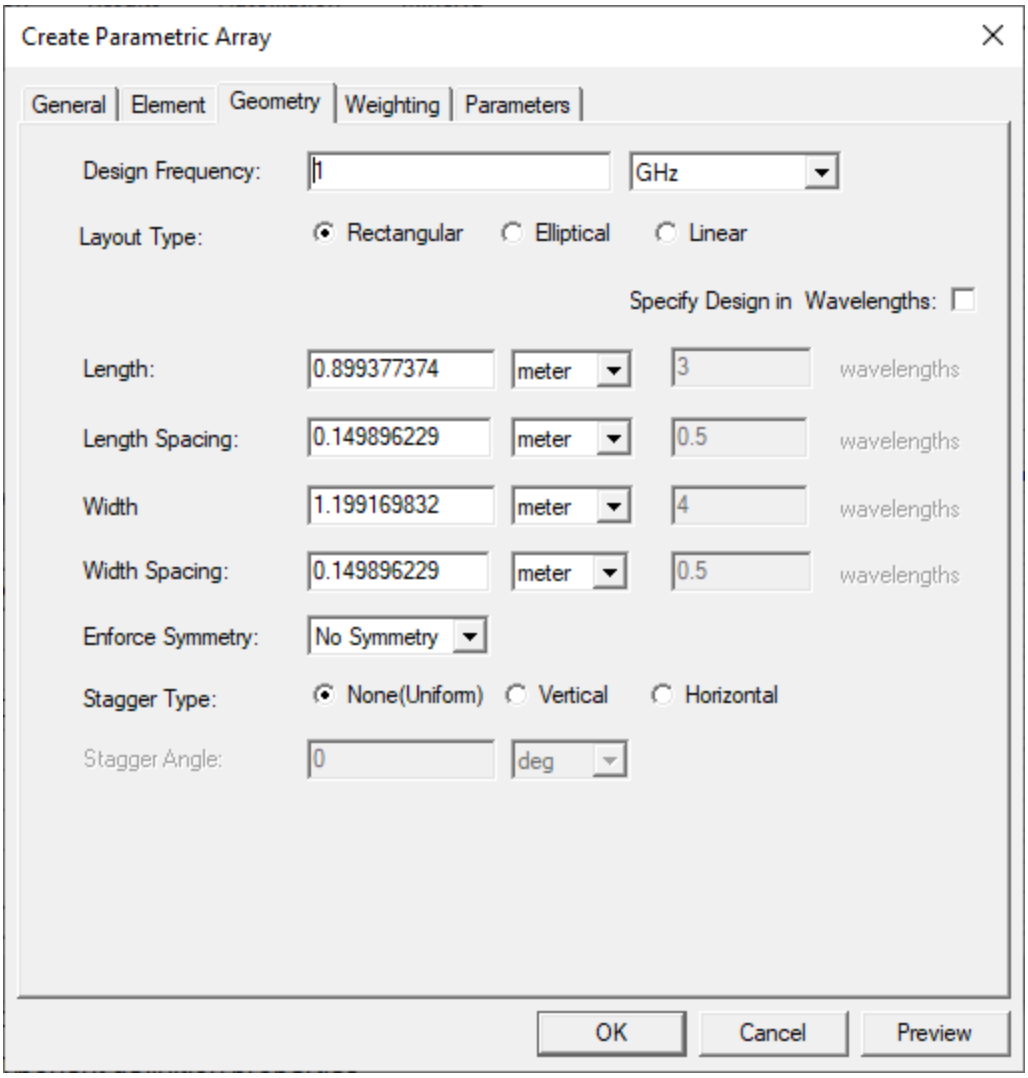
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

displayed in the **Parameters** tab.



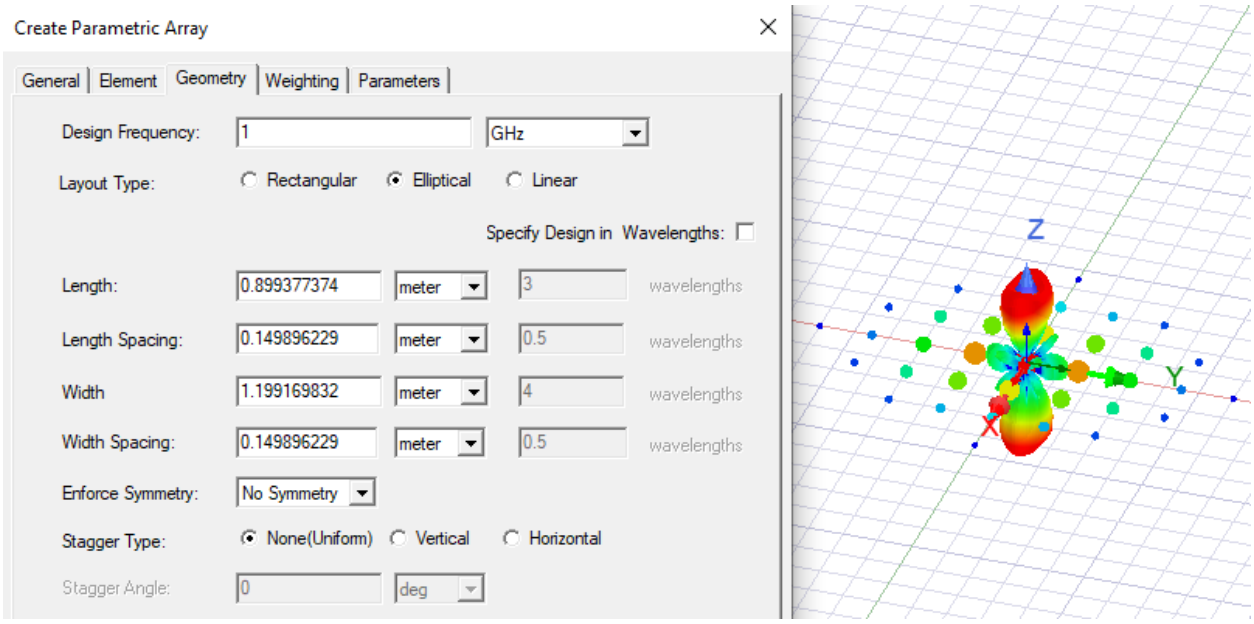
f. Configure the array layout geometry. The array layout geometry is defined in the XY plane of the placement coordinate system as Rectangular, Elliptical, or Linear, and is parametrically defined by array Length, Width, Length/Width Element Spacing, and various other adjustments for the element placement geometry. The geometry layout parameters may be specified in user-selectable units or in units of wavelength, as determined by the Design Frequency. Selecting Enforce Symmetry along either/both axes ensures that no elements are placed directly on the axis of symmetry, essentially constraining the number of element rows or columns to be "even".

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



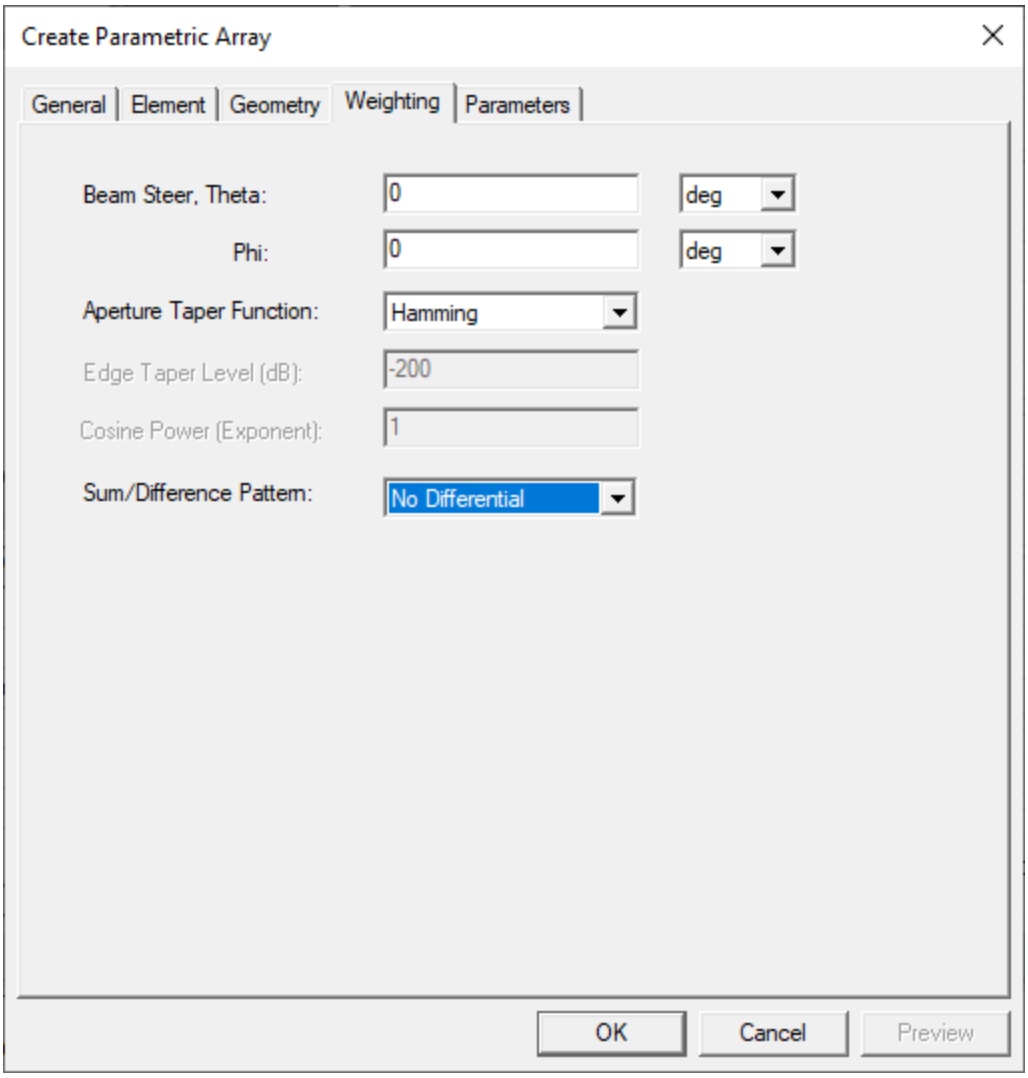
Click the **Preview** button to view the effects of your selections. The following example shows a Preview for selecting the Layout Type as Elliptical.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



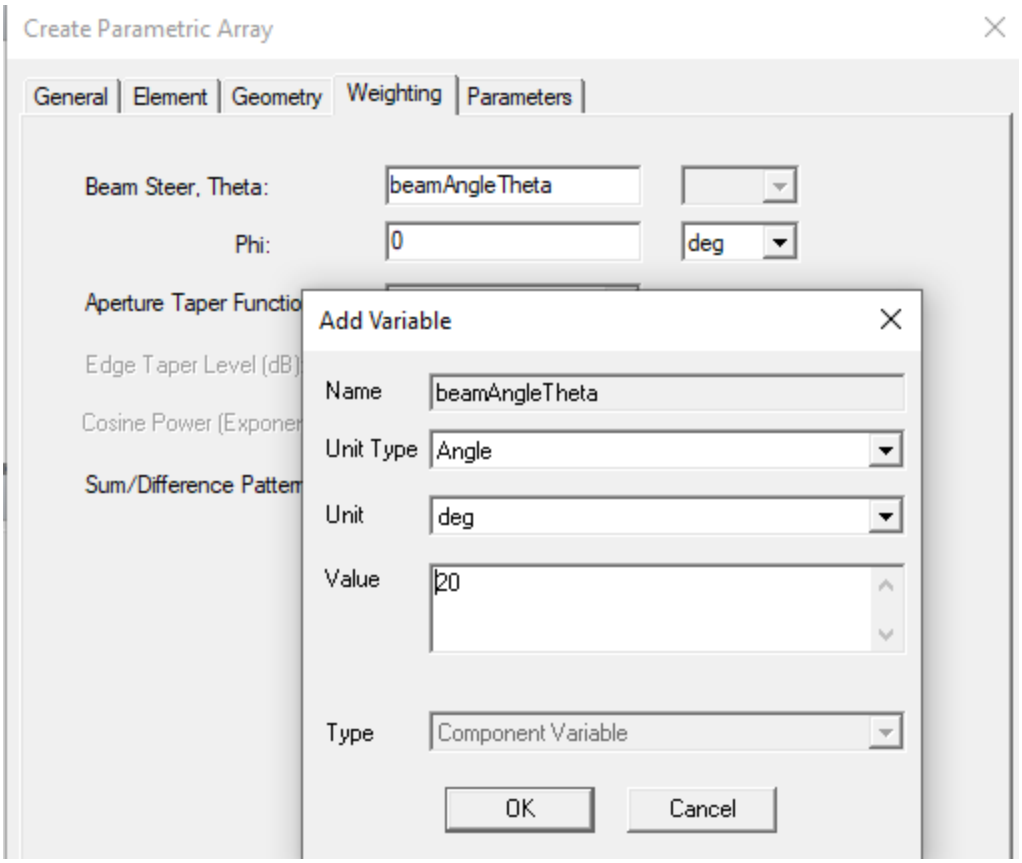
g. Configure the array weighting. You can specify Beam Steer, Aperture Taper and Sum/Difference Pattern.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

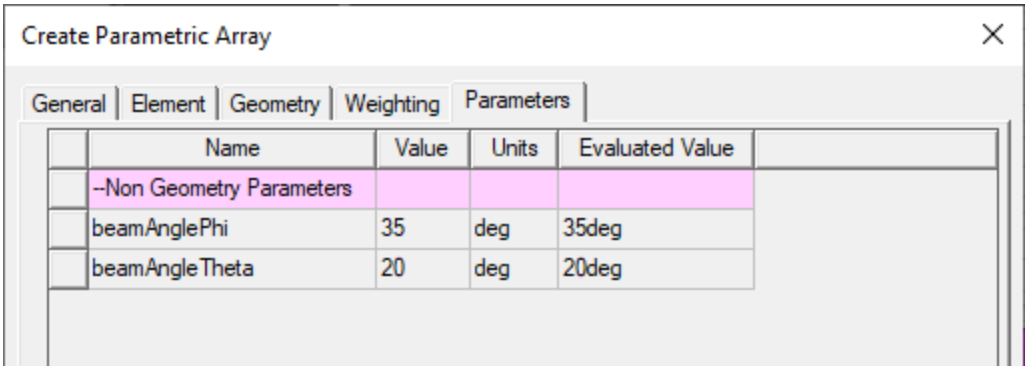


You can specify values as variables. For example.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



h. If you have specified any values for the Parametric Antenna as variables, these appear on the **Parameters** tab. For example, if on the **Weighting** tab you specified beam steer for Theta and Phi as variables, they would appear as shown.

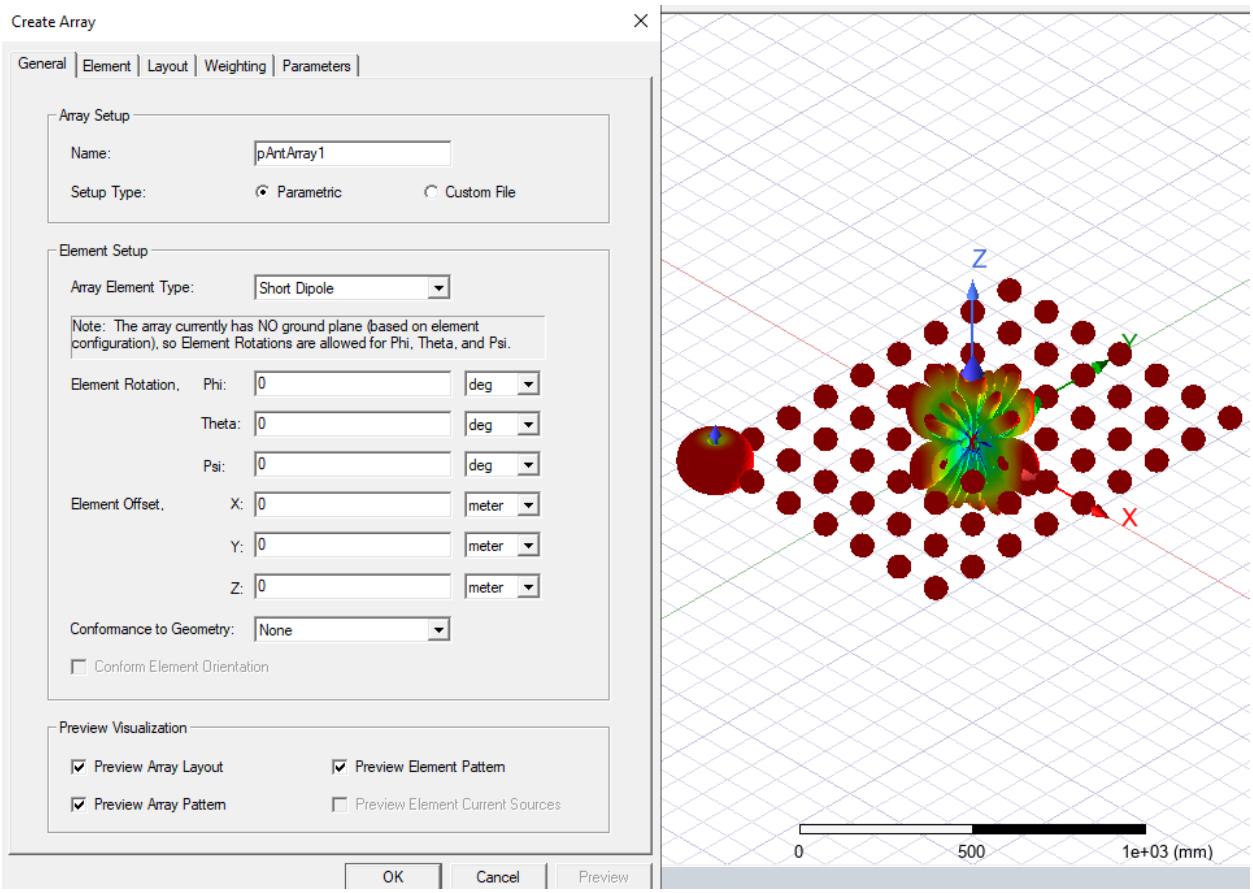


You can configure any component instance parameters for the array component. These can be mapped to design/project variables.

i. As previously referenced, the **Preview** button can be used to visualize the array layout, element pattern, and resulting array antenna pattern, including any ground plane requirement for the selected Array Element Type. Any changes made to the

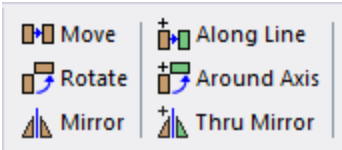
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

array property tabs/pages will enable the **Preview** button. Clicking **Preview** applies the changes in the modeler window and disables the **Preview** button.



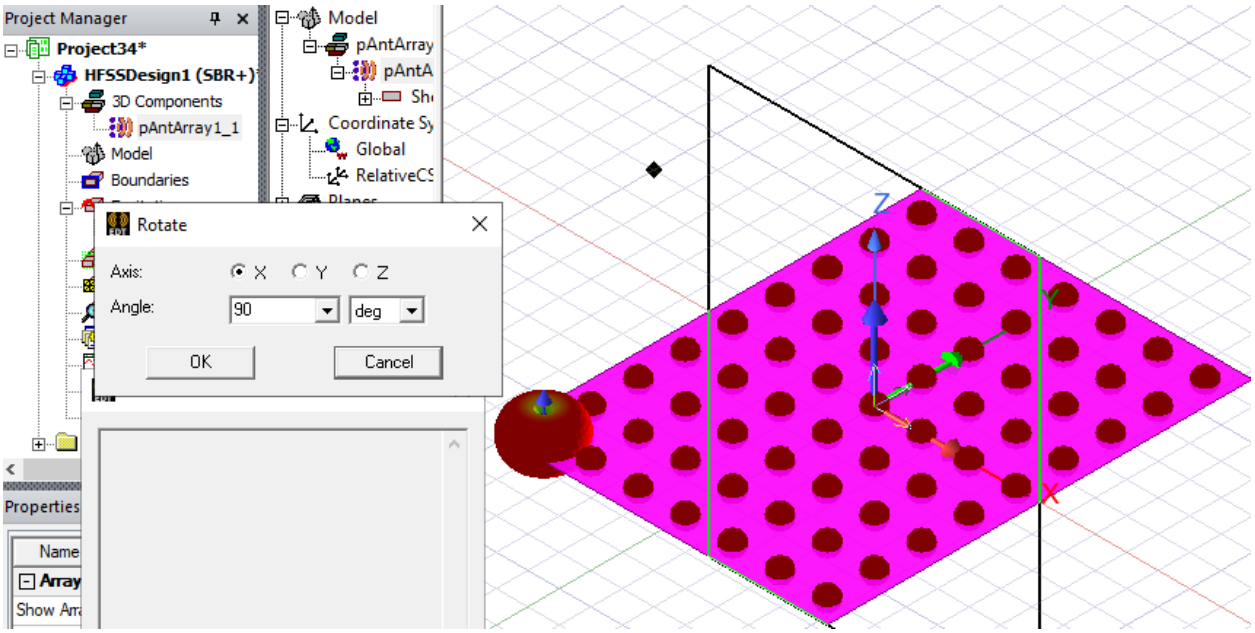
j. Close the dialog with **OK** to save this component definition and create the parametric array 3D component instance.

6. If you want to edit the antenna placement, or orientation, select the model in the History tree, and right-click and select **Edit>Arrange**, and select from **Move**, **Rotate**, or **Mirror**. When you select the Parametric Array, the icons in the ribbon are also enabled.

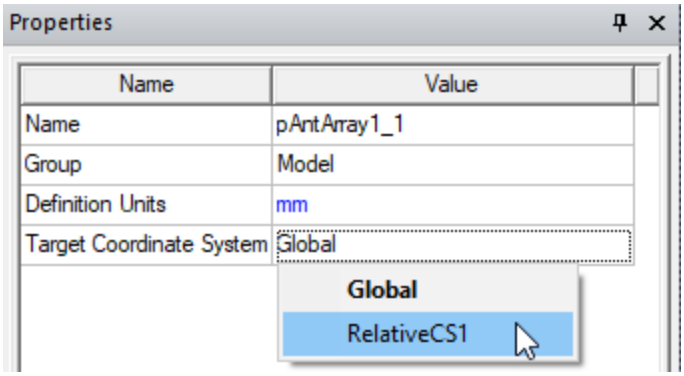


If needed, place and orient (Move, Rotate, Mirror) the 3D component at the desired geometry location.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



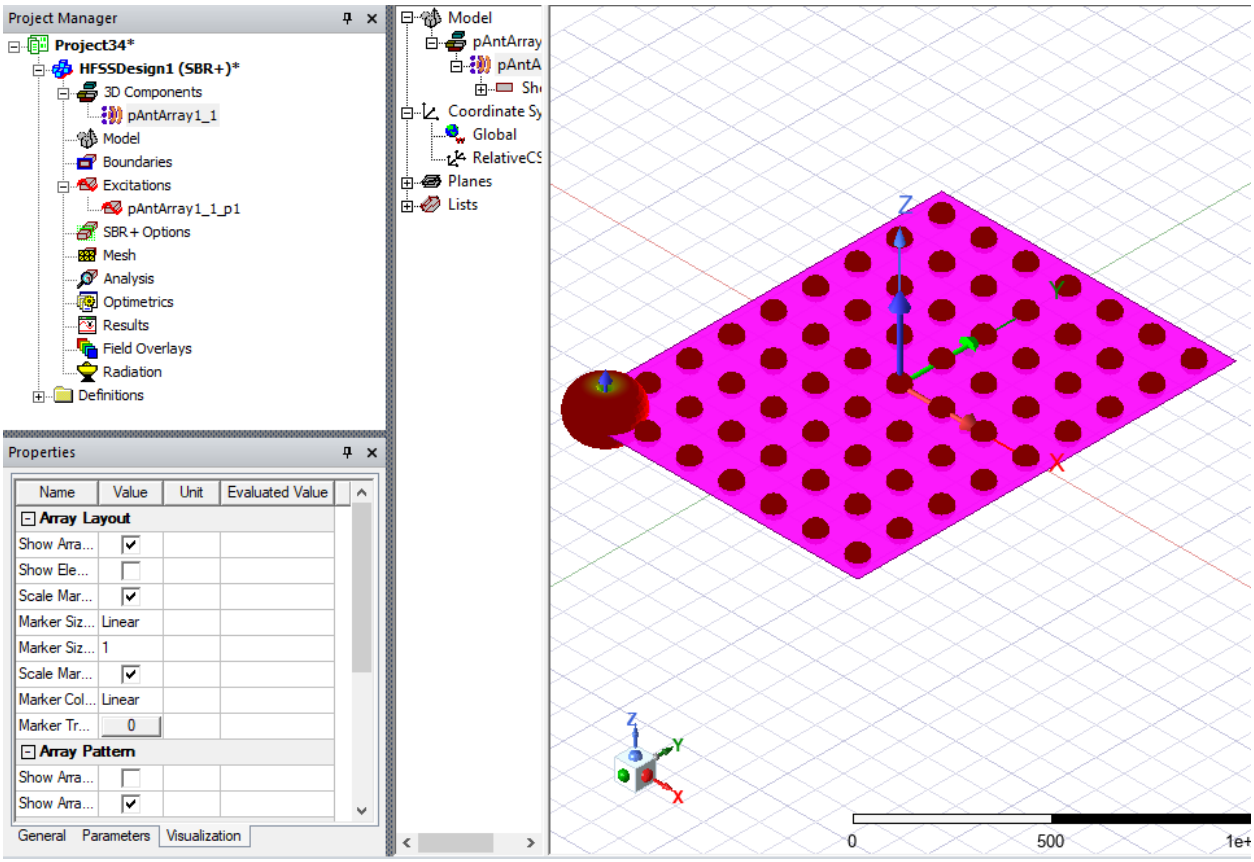
Notice that other edit commands for Copy, Paste, and Duplicate are also available. You can also change or edit the target coordinate system in the instance **Property** window.



7. You can view and edit the 3D component definition in the **History** tree. Selecting the 3D Component and **Edit Properties** opens the **Create Parametric Array** dialog and lets you edit the Properties as you did in creating the Parametric array.
8. You can view and edit the properties of the 3D Component instance listed in the **Project** tree. Select an array antenna instance listed under 3D Components, or in the Model tree listed under one of the created Model component definitions. For each component instance that is created, there are unique instance-level properties separate from the array geometry/weighting definition that can be configured for visualization, etc. They are

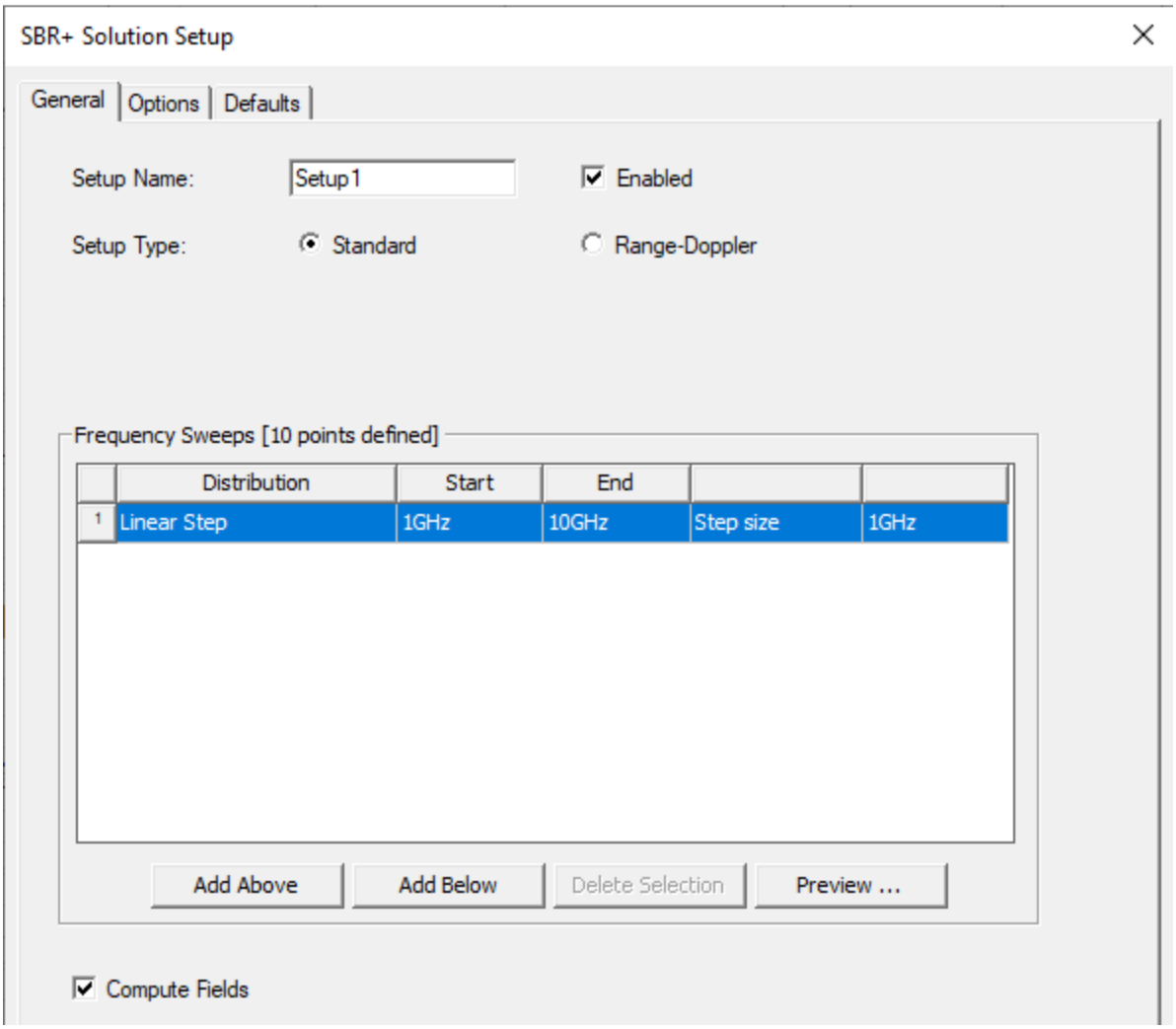
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

grouped into 3 separate tabs for General, Parameters, and Visualization.



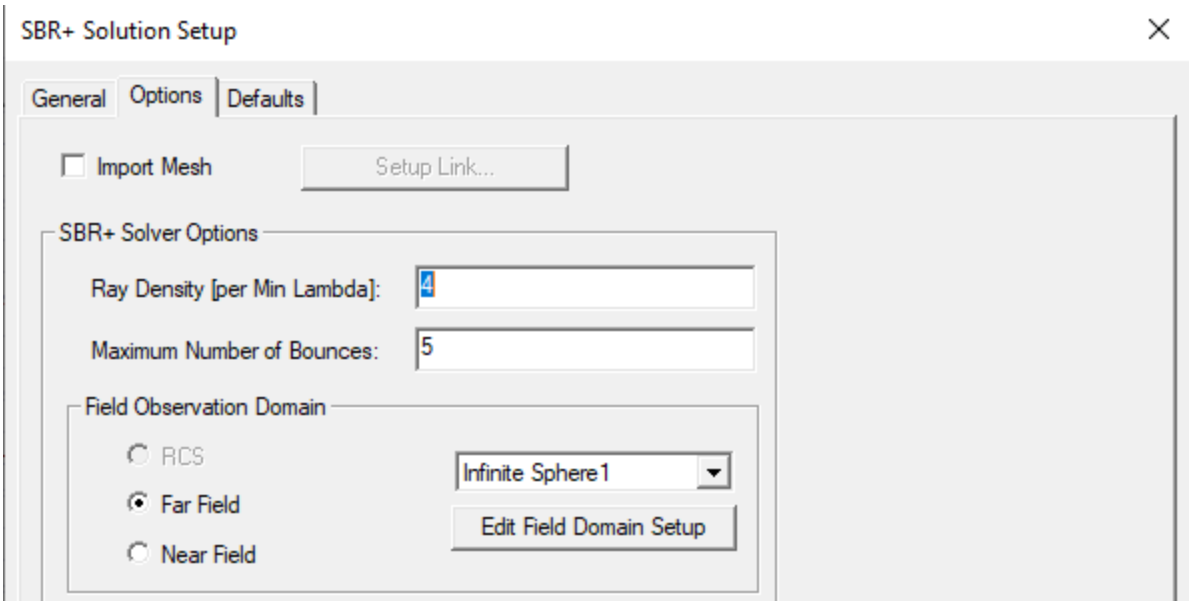
9. Add an **SBR+ Solution Setup**. To generate a far field radiation pattern for this parametric array, be sure Compute Fields is checked on the **General** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

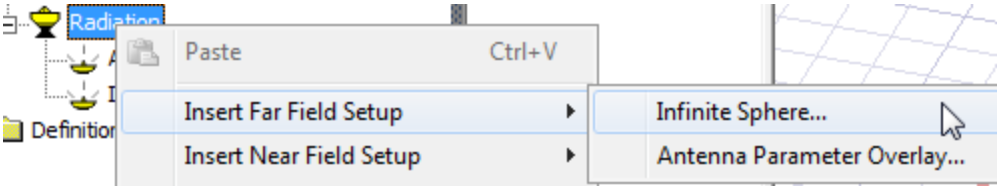


Also ensure that a far field Observation Domain has been correctly configured.

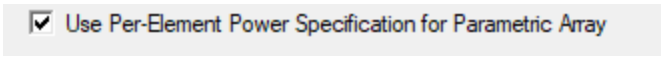
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You must create a Far Field Setup to access the Far Fields and Antenna Parameters reports.



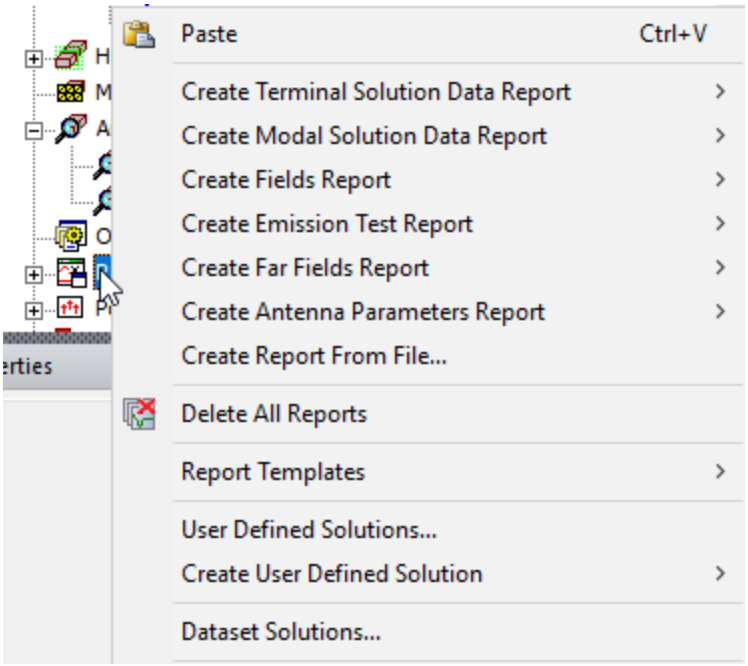
10. You may select **Use Per-Element Power Specification for a Parametric Array** in the Edit Sources dialog when only a Parametric Array exists in the scene. When you select that option, **Edit Sources** defines the input power of an element before applying the Aperture Taper configured in Array Weightings. The Parametric Array with this option gives results similar to HFSS post-processing feature of Antenna Array. Coupling analysis and Enhanced Radiated Power Accuracy are not supported.



11. Run simulation.
12. Generate far field radiation pattern. Right-click on the Result node in the Project Manager and select **Create Far Fields Report > Radiation Pattern** to create a radiation pattern plot for the desired range of theta/phi angles

After you run an SBR+ simulation with parametric antennas, the results available are just like for a single-port parametric antenna. You can create a variety of reports, including S Parameters, Far Fields, and Antenna Parameters.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create and Use Custom Array By File in SBR+ Simulation

This chapter provides a summary and a detailed workflow for using Custom Antenna Arrays by File for SBR+ simulation.

Summary Workflow for Custom Array By File Antenna Arrays

1. Create an HFSS design and select the SBR+ Solution Type.
2. Add model objects, components.
3. Add an antenna local coordinate system .
4. Add the parametric antenna array by right-click on **3D Components** in the **Project** tree and select **Create Array>Custom Array By File…** or right click on **Excitations** in the **Project** tree and select **Create Array Component>Custom Array by File...**.
5. On the **General** tab select the desired array element type from the drop down list.
6. In the **Create Parametric Array** dialog configure any antenna-specific parameters on the “Element” tab
7. On the **Custom File** tab, configure the array
    a. Specify the filename of the *.sarr file to be used. These files provide array element positions, orientations, and complex weighting over a set of specified frequencies and "state". A single state must be selected for the simulation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Select the “Active State Index” to be used for the simulation

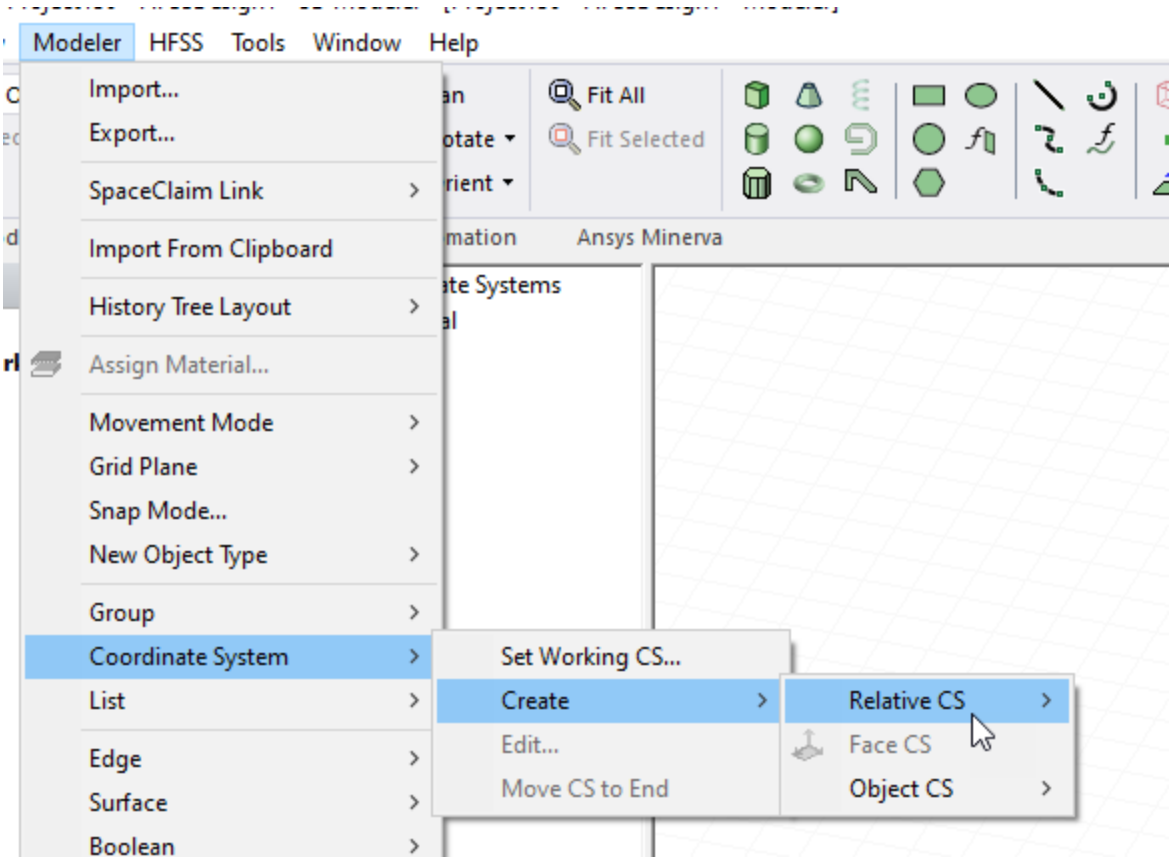
c. Different weights for frequencies specified in the file can be displayed/previewed in the table shown on this tab.

8. Click **Preview** to see a preview visualization of the array in the 3D modeler window
9. Click **OK** to dismiss the dialog and create the 3D component instance
10. Add an Analysis Setup
11. Run simulation
12. Generate far field radiation pattern

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

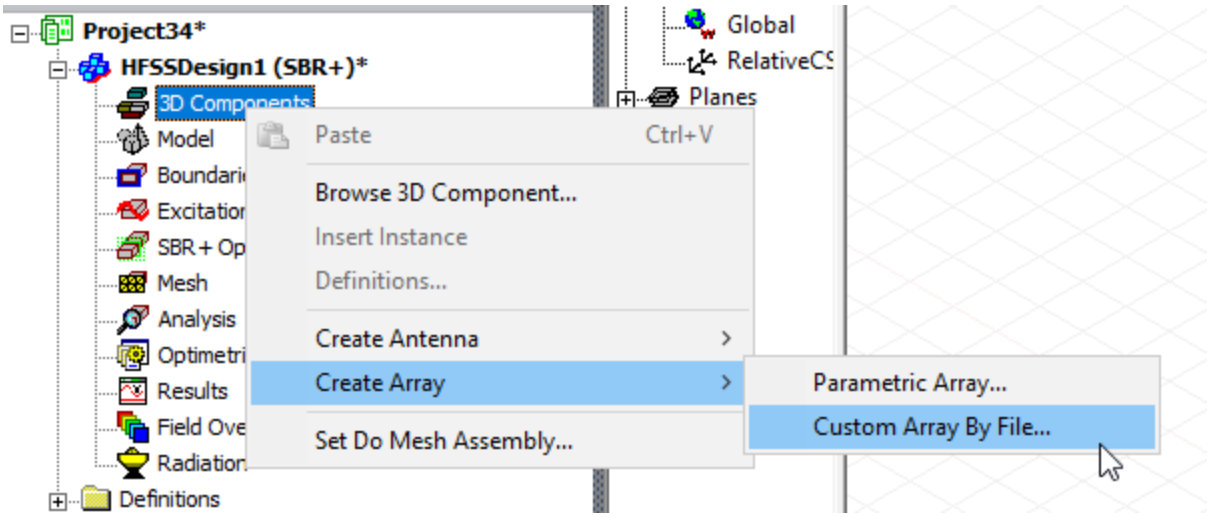
Detailed Workflow for using Custom Array by File... in SBR+ Simulation

1. Create an HFSS design and select the SBR+ Solution Type.
2. Add model objects, components.
3. Add an antenna local coordinate system. Creation of an antenna local coordinate system (CS) can be very useful in placing and orienting parametric antennas and arrays via 3D components. The CS can be created as a simple relative CS, or as something more advanced and suitable for array placement on a surface, like a face CS . After creating the new CS, make sure it is selected/active when creating the parametric array.



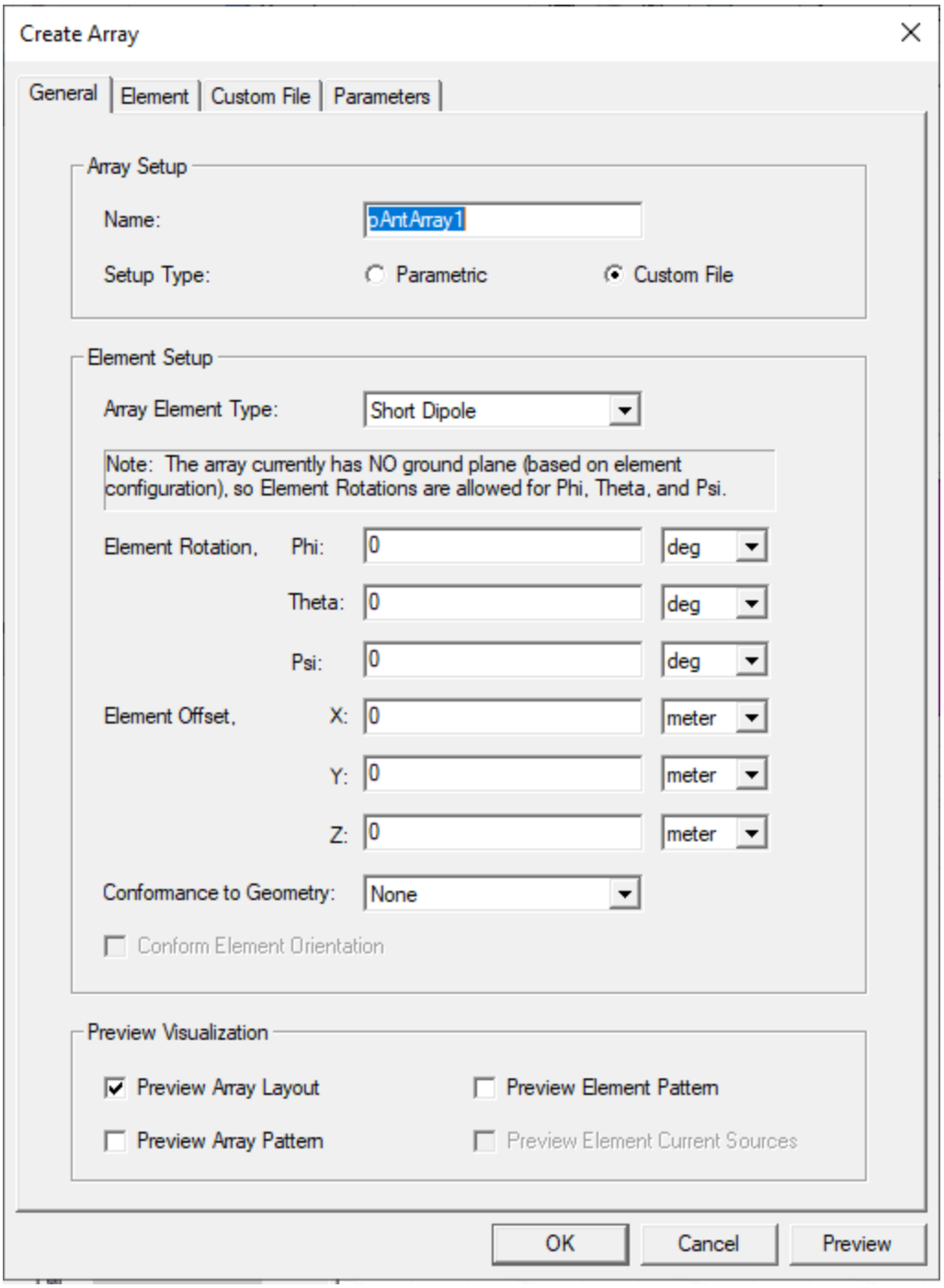
4. Add the parametric antenna array by right-click on **3D Components** in the **Project** tree and select **Create Arrayy>Custom Array By File…** or right click on **Excitations** in the **Project** tree and select **Create Antenna Component>Custom Array by File...**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



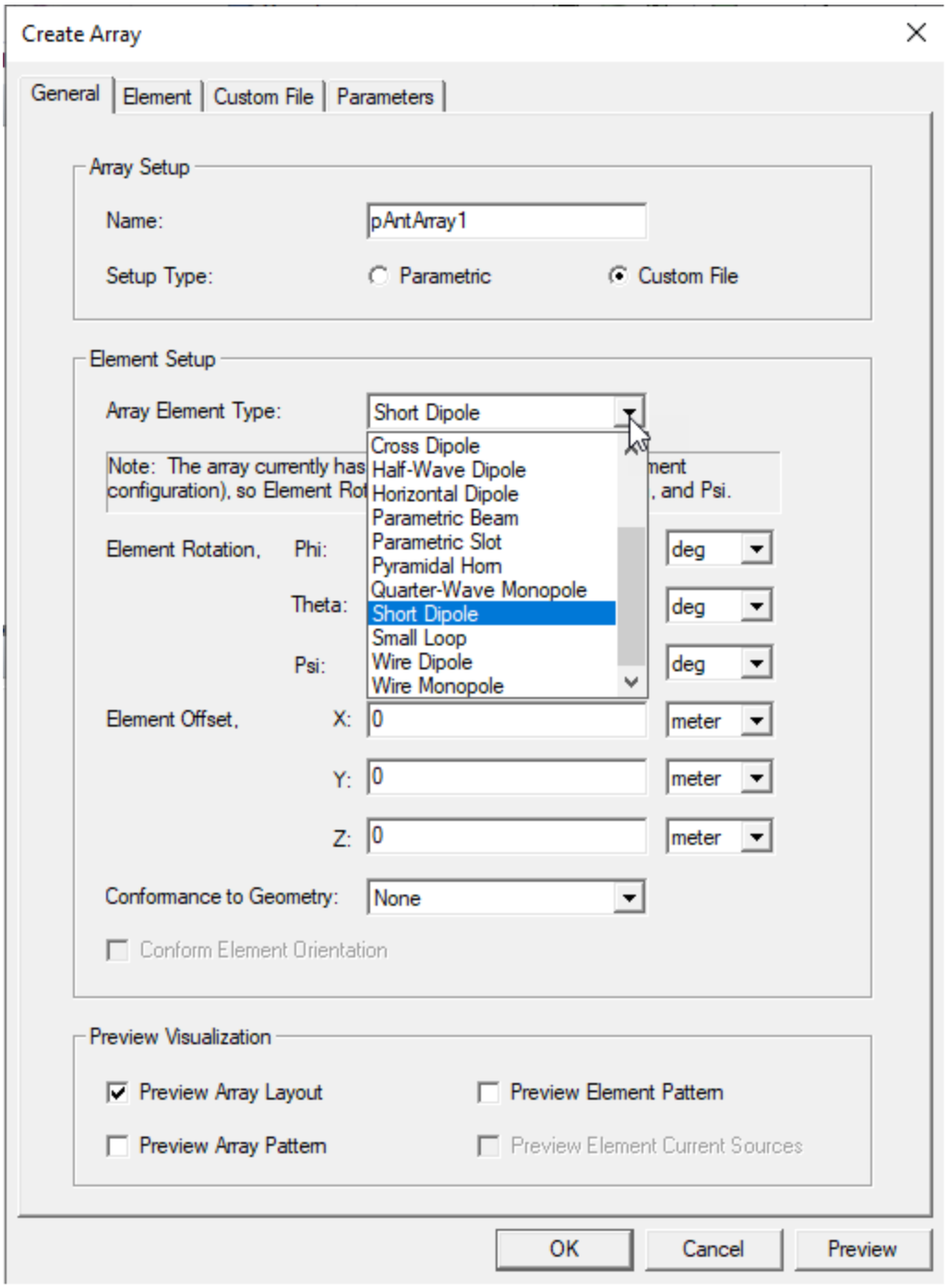
This opens the **Create Array** dialog box with Custom File selected as the Setup Type. For creating parametric arrays, Wizard mode is not used.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. In the **Create Array** dialog
    a. Select the Array Element Type of antenna from the drop-down menu. You can select any of the fifteen parametric antenna types as the Array Element.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



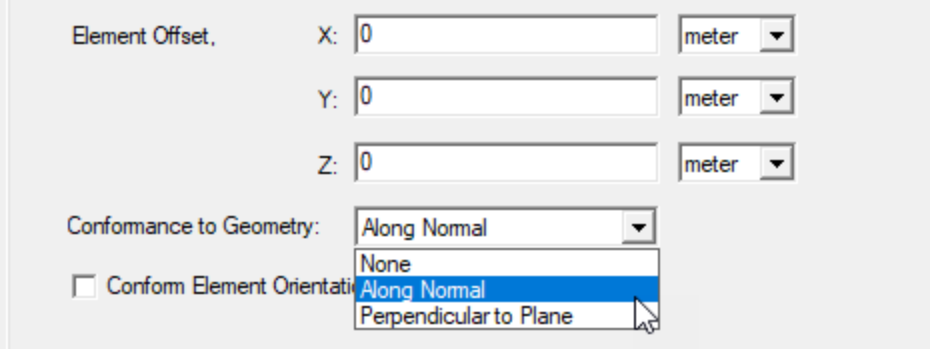
b. Specify the Element Rotation for Theta, Phi and Psi (all 3 quaternion rotations) and select the units from the menus. Click **Preview** for a display in the Model window.

c. You can also specify an Element Offset for X, Y, and Z, and specify Units. After you have specified a value, and the cursor to another text box, the **Preview** button is enabled, allowing you to view the visualization for the values you have specified.

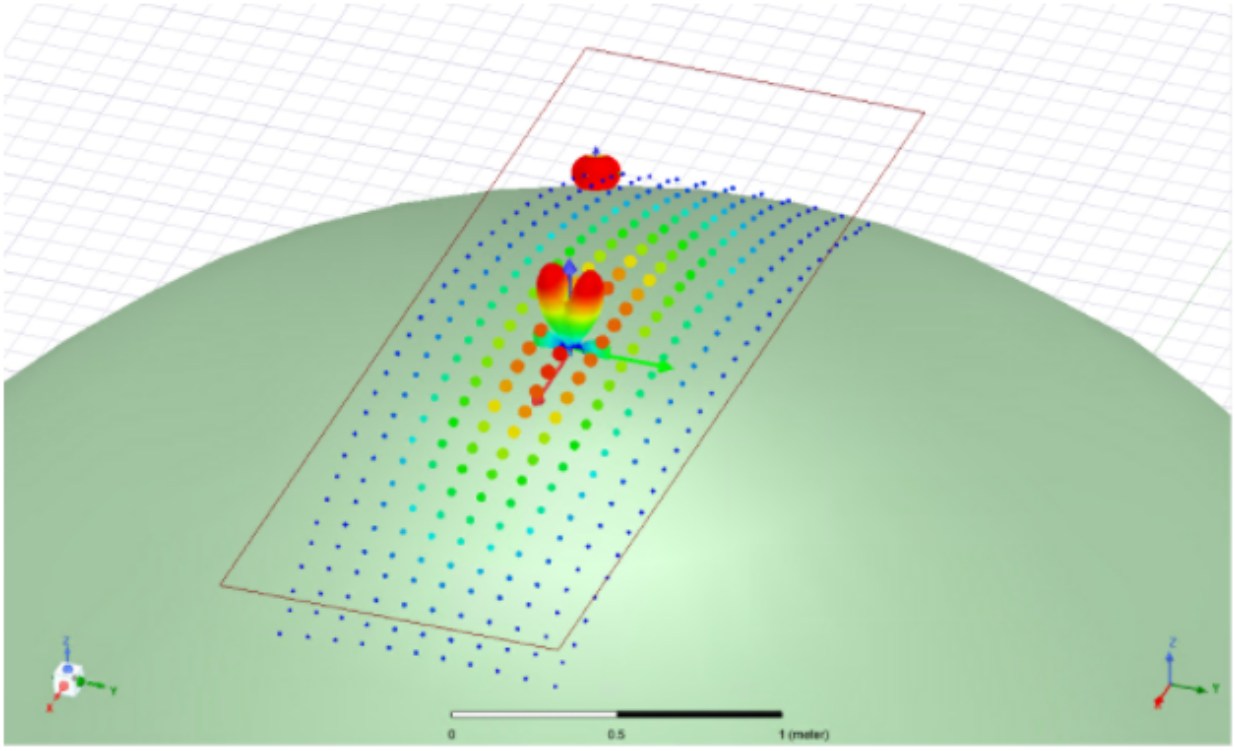
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. Use the checkboxes to Preview Array Layout, Array Pattern, Element Pattern, and Current Sources (when an element type with current sources is selected). The next figure shows both Array Layout and Array Pattern after clicking **Preview.**

e. You can use the Conformance to Geometry dropdown to select None, Along Normal, or Perpendicular to Plane. Making the selection Along Normal enables the Conform Element Orientation checkbox.



The following figure shows Conformance Along Normal relative to a sphere.

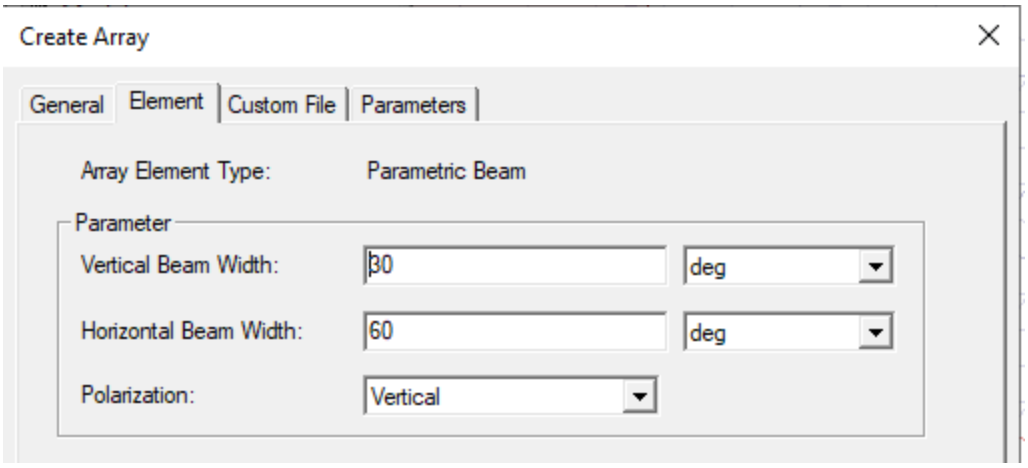


For a technical discussion, see Technical Notes: Current Conformance in SBR+.

f. Configure the array element. The parameters available on the **Element** tab (or the Element wizard page) will vary depending on the Antenna Element Type chosen. For example, the Short Dipole antenna only shows the Array Element Type as Short

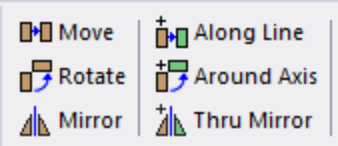
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dipole with no additional parameters, since this simple antenna element type has no parameters to configure. Most of the antenna element types will have some additional parameters to configure on this tab/page.



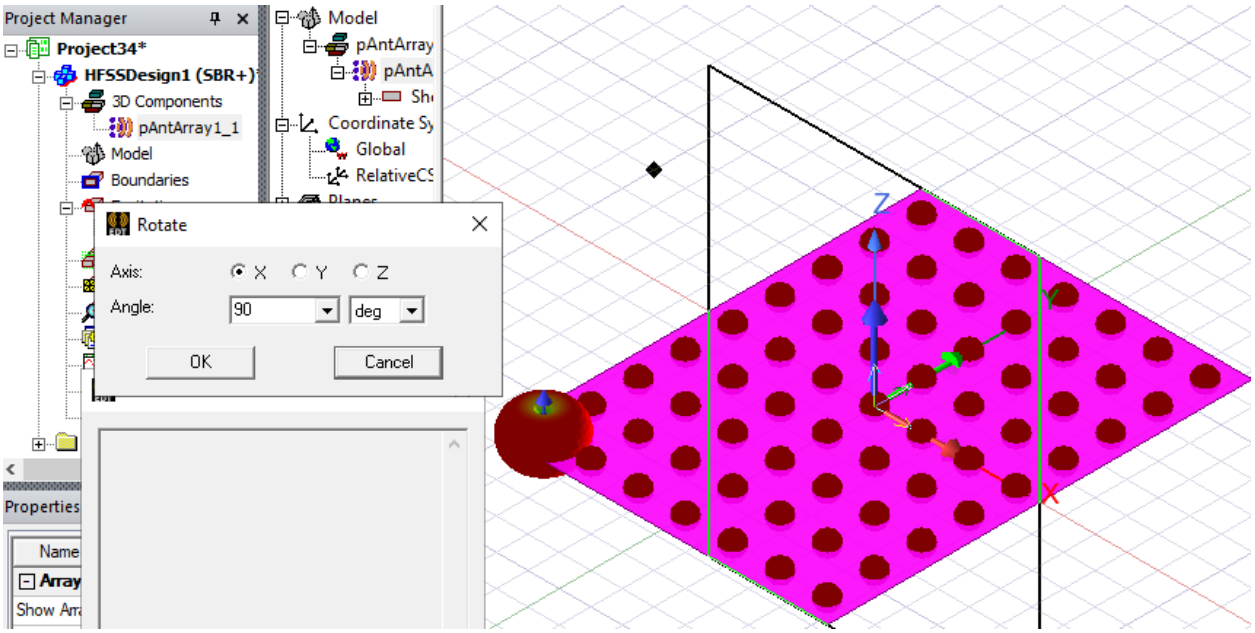
If you specify variables for any of the parameters, these will be displayed in the **Parameters** tab.

6. If you want to edit the antenna placement, or orientation, select the model in the History tree, and right-click and select **Edit>Arrange**, and select from **Move**, **Rotate**, or **Mirror**. When you select the Parametric Array, the icons in the ribbon are also enabled.

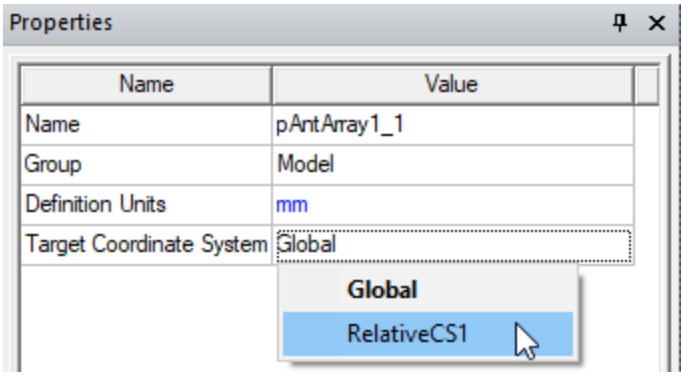


If needed, place and orient (Move, Rotate, Mirror) the 3D component at the desired geometry location.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

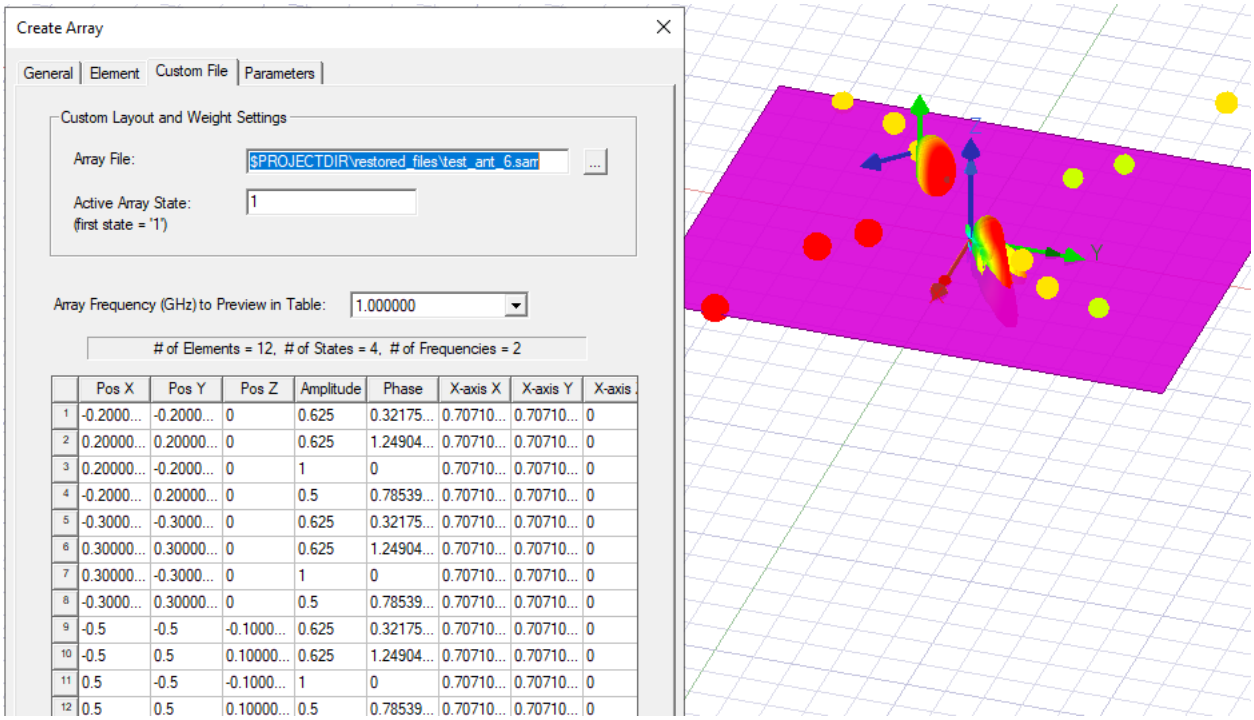


Notice that other edit commands for Copy, Paste, and Duplicate are also available. You can also change or edit the target coordinate system in the instance **Property** window.



7. On the **Custom File** tab, select the Array file to configure the array.
    a. Specify the filename of the *.sarr file to be used. These files provide array element positions, orientations, and complex weighting over a set of specified frequencies and "state". A single state must be selected for the simulation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. Select the “Active State Index” to be used for the simulation

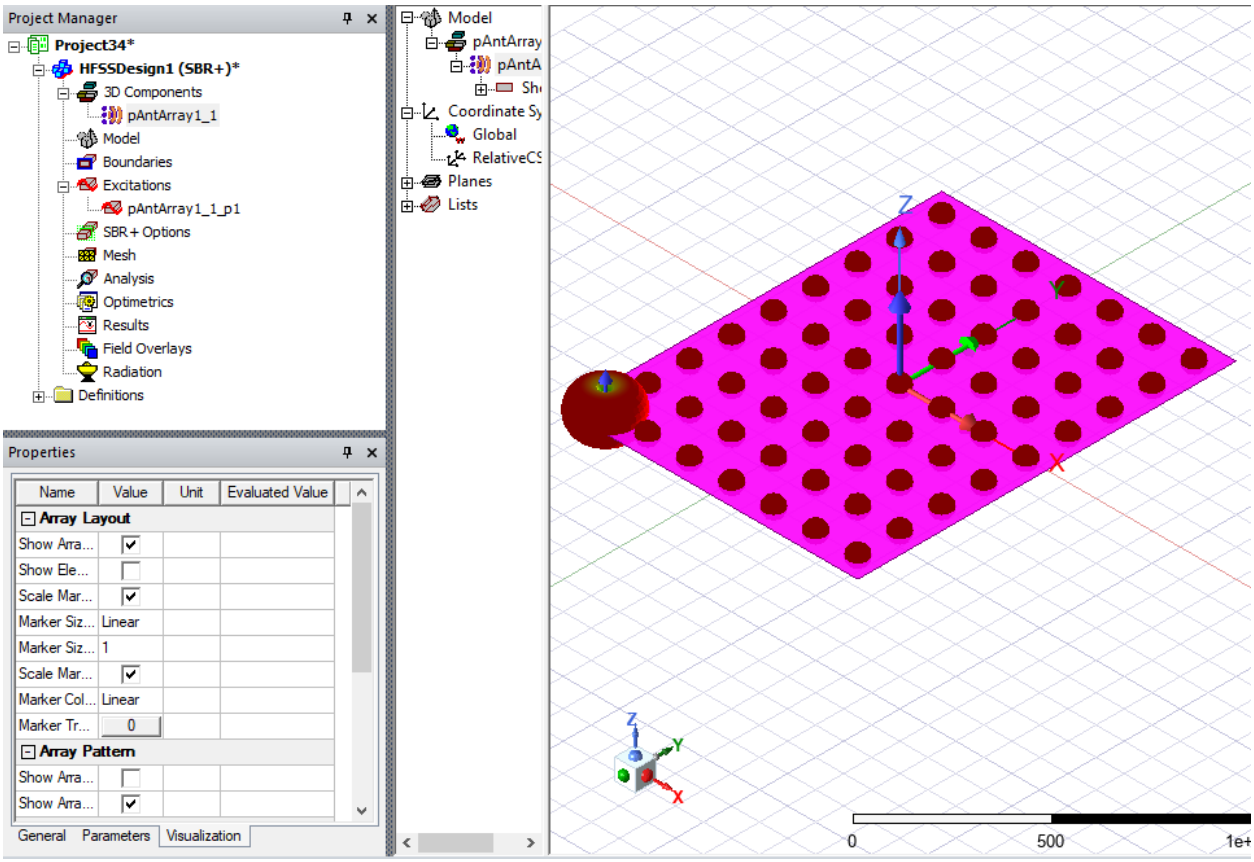
c. Different weights for frequencies specified in the file can be displayed/previewed in the table shown on this tab

d. Close the dialog with **OK** to save this component definition and create the parametric array 3D component instance

8. You can view and edit the 3D component definition in the **History** tree. Selecting the 3D Component and **Edit Properties** opens the **Create Parametric Array** dialog and lets you edit the Properties as you did in creating the Parametric array.
9. You can view and edit the properties of the 3D Component instance listed in the **Project** tree. Select an array antenna instance listed under 3D Components, or in the Model tree listed under one of the created Model component definitions. For each component instance that is created, there are unique instance-level properties separate from the array geometry/weighting definition that can be configured for visualization, etc. They are

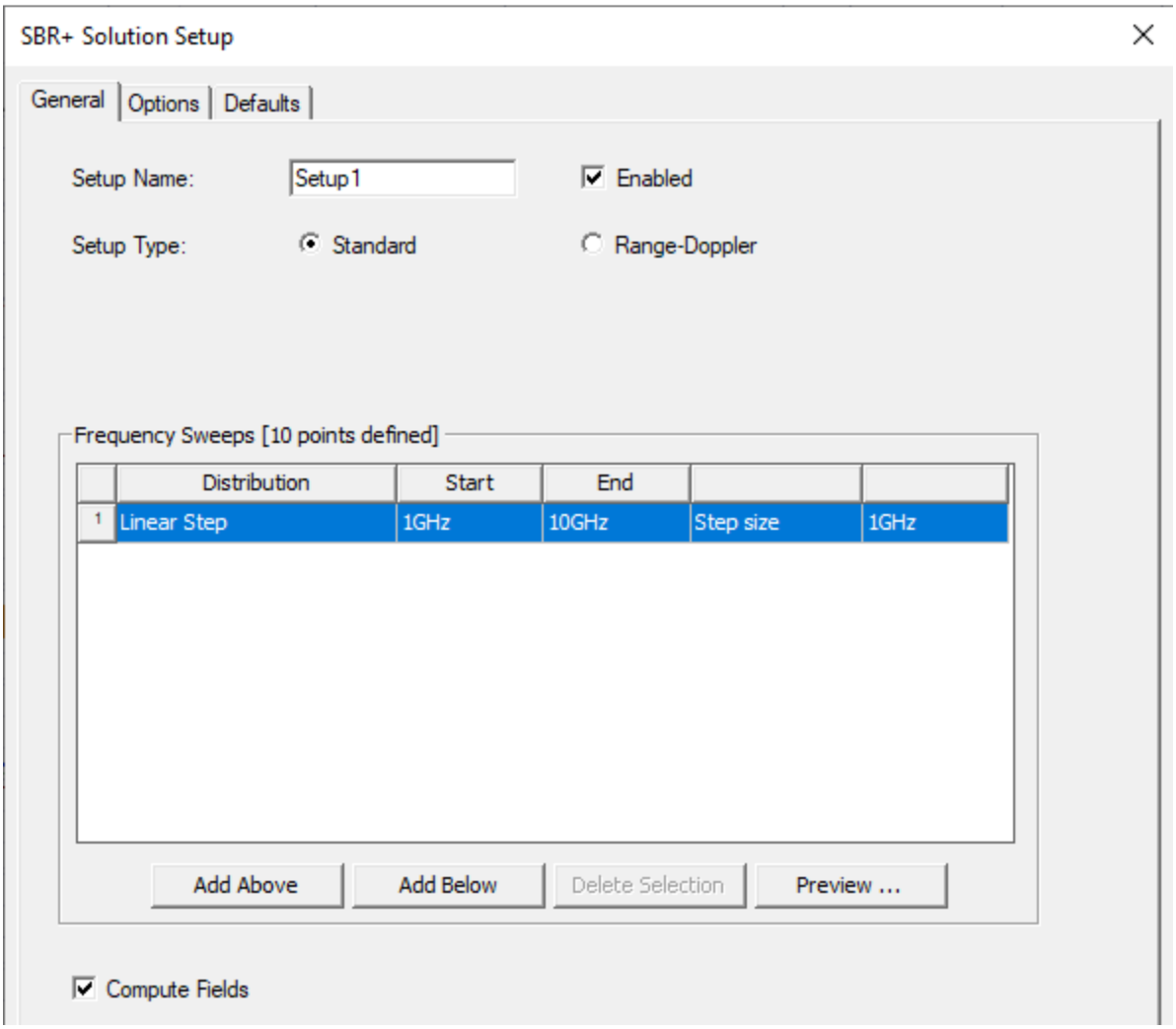
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

grouped into 3 separate tabs for General, Parameters, and Visualization.



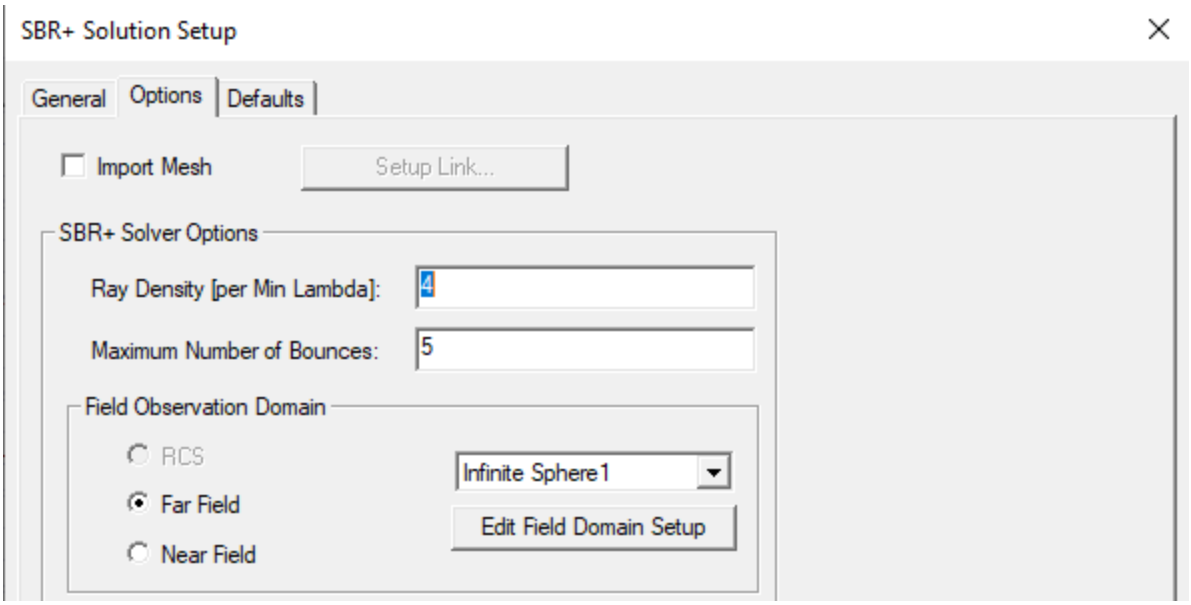
10. Add an **SBR+ Solution Setup**. To generate a far field radiation pattern for this parametric array, be sure Compute Fields is checked on the **General** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

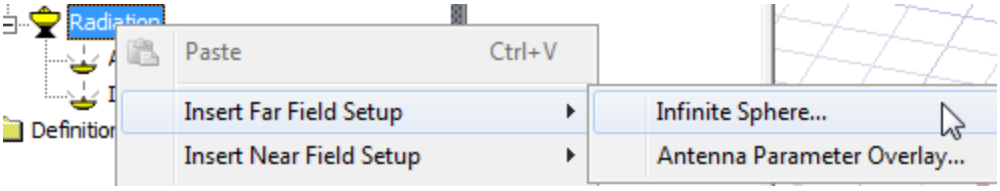


Also ensure that a far field Observation Domain has been correctly configured.

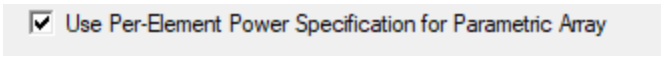
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You must create a Far Field Setup to access the Far Fields and Antenna Parameters reports.



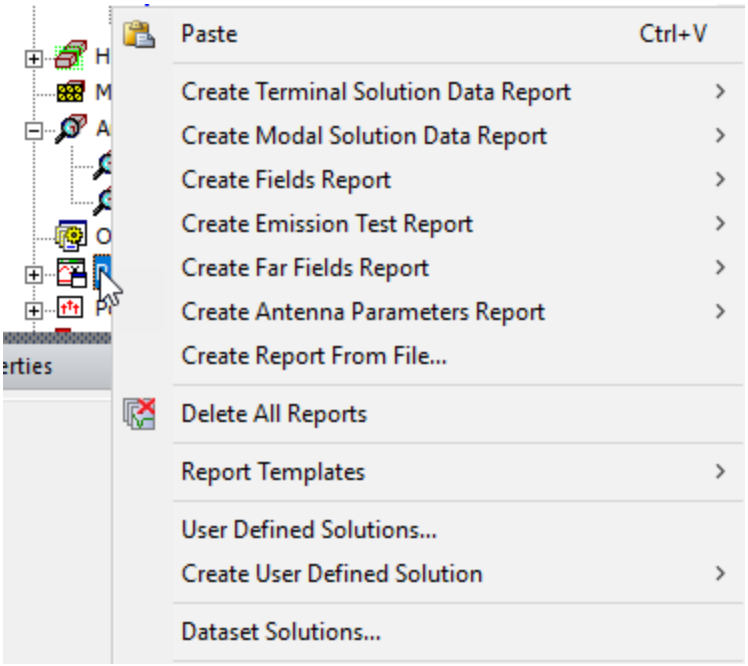
11. You may select **Use Per-Element Power Specification for a Parametric Array** in the Edit Sources dialog when only a Parametric Array exists in the scene. When you select that option, **Edit Sources** defines the input power of an element before applying the Aperture Taper configured in Array Weightings. The Parametric Array with this option gives results similar to HFSS post-processing feature of Antenna Array. Coupling analysis and Enhanced Radiated Power Accuracy are not supported.



12. Run simulation.
13. Generate far field radiation pattern. Right-click on the Result node in the Project Manager and select **Create Far Fields Report > Radiation Pattern** to create a radiation pattern plot for the desired range of theta/phi angles

After you run an SBR+ simulation with parametric antennas, the results available are just like for a single-port parametric antenna. You can create a variety of reports, including S Parameters, Far Fields, and Antenna Parameters.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Array Element List (*.sarr) Format

The array element list (.sarr) file format describes an array as a collection of antenna elements. To use the Custom Array By File... command for an HFSS SBR+ custom array, you must provide a .sarr file that describes the positions, orientations, and weights of array elements. In a .sarr file, each element is defined by its position, orientation, and possibly frequency- and state-dependent weights. The number of weights (Nw) per element is determined by the frequency-domain and num_states specifications in the header; it is the product of the number of frequency samples and the number of states: Nw = Nf Ns. The Ansys Electronics Desktop SBR+ Custom Array feature does not support .sarr files without weights. Therefore, the has_weights header key must be set to true and element weights must be provided.

Multi-state arrays can be configured to capture, for example, multiple electronic beam steering angles or different aperture amplitude tapers in a single .sarr fille. When multi-state results are desired involving an array antenna, using a multi-state array file is more convenient than setting up and running Ns SBR+ designs based on Ns single-state .sarr files.

The file format consists of a header section followed by a data section. The header section is based on keys. Header key lines define what to expect in the data section. The key lines can appear in any order, with the exception of the num_elements key line, which both specifies the number of array elements and marks the end of the header section. The next active (non-comment) line after the num_elements key line is always the first data line of the first array element.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The format supports the liberal use of commenting in both the header and data sections. Blank lines and lines beginning with a pound ('#') sign are ignored. Short comments can also be appended to any key or data line using a pound sign.

Header Keys

has_weights: This is a required key line that indicates whether the array file include element weights. This key must always be set to true, and .sarr files imported into an SBR+ custom array must always contain element weight data.

freq or freq_list: One of these two key lines (but not both) must appear in the header to define the frequency domain of the array. The specified frequency sampling, along with the num_states key, determines the number of complex amplitude weights that will be read for each array element. Use the freq key to configure a uniform frequency sweep. Use the freq_list key if the frequency sampling is nonuniform. In either case, the specified frequencies are always expected in units of GHz. **The lower and upper extents of the array frequency domain must match or exceed the frequency extents of SBR+ solution setups where it is used as part of the HFSS design**. During simulation analysis, an error message will be issued and the analysis aborted if this condition is not satisfied. Beyond that requirement, the sampling density of the array frequency domain only needs to be sufficient for accurate interpolation of the weights and does not need to match the frequency sampling of the solution setup. The example file below shows the precise syntax for encoding the frequency domain using these two keys

num_states: This is an optional key line that specifies the number of array element weight sets (i.e., array states, Ns). This, along with the number of frequency samples (Nf) determines the number of complex amplitude weights (Nw) expected per element: Nw = Nf Ns. When this key line is not included in the header, the number of states is assumed to be one (Ns = 1).

num_elements: This is a required header key that specifies the number of array element data blocks expected after the header. It must be the last key line in the header because it is also used to mark the end of the header section and the start of the element data blocks section of the file.

Following the header are data blocks, one block for each element of the array. Each block specifies the element's position, orientation (relative to the array antenna local coordinates), and frequency-dependent weights.

An simple .sarr file example is given below. It uses header comments to self-document the format.

```
#-----------------------------------------------------------------
-----------
# Template array element list file to define the file format
# Blank lines and lines beginning with '#' are ignored.
#-----------------------------------------------------------------
-----------
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
# Has weights {true} (required line)
# Weight data is required in the element data blocks.
#-----------------------------------------------------------------------
-----------
has_weights true
#-----------------------------------------------------------------------
-----------
# frequency domain (required line)
# freq <start_ghz> <stop_ghz> <nstep>
# where:
# nstep = num freq samples - 1
#-----------------------------------------------------------------------
-----------
# -> Specifies, along with num_states, the number of complex
amplitude weights
# that file reader expects for each element when has_weights = true:
# Nwgt = Nfreq * Nstate
# -> Determines the frequency samples available for array rendering
#-----------------------------------------------------------------------
---
freq 0.15 0.15 0
#-----------------------------------------------------------------------
---
# The frequency domain can alternatively be defined with a list.
# The list must be monotonically increasing but does not need to be
uniformly
# spaced.
#
# freq_list <Nf>
# <freq_ghz[1]>
# <freq_ghz[2]> ( > freq_ghz[1] )
# :
# <freq_ghz[Nf]> ( > freq_ghz[Nf - 1] )
#
# where Nf is the number of frequencies
#-----------------------------------------------------------------------
---
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
# Uncomment these lines and comment the freq key line in order to
switch
# how the element weights frequency domain is defined
#-----------------------------------------------------------------
---
# freq_list 4
# 0.1
# 0.2
# 0.4
# 0.8
#-----------------------------------------------------------------
---
# number of weight states (optional line, default = 1 state)
#
# Multi-state arrays can be configured in this file format.
# This provides a second dimension in addition to frequency along
which
# element weights can be varied. Some examples of weight set
variations:
# - beam steer angle
# - aperture amplitude taper for sidelobe control
# - switching sub-arrays on/off by selectively nulling weights
#
#-----------------------------------------------------------------
---
num_states 1
#-----------------------------------------------------------------
---
# number of elements (required line)
#
# This key line marks the end of the header and the start of the
element
# list data blocks.
#-----------------------------------------------------------------
---
num_elements 2
#-----------------------------------------------------------------
---
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

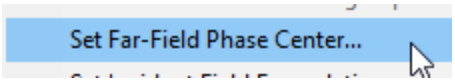
```
# For each element of the array
#-----------------------------------------------------------------------
---
# Array element description (mandatory):
# <pos_x> <pos_y> <pos_z>
# <oriX_x> <oriX_y> <oriX_z>
# <oriY_x> <oriY_y> <oriY_z>
# <wgt_rl[1]> <wgt_im[1]> (Only if has_weights = true)
# <wgt_rl[2]> <wgt_im[2]>
# : : :
# <wgt_rl[NfNs]> <wgt_im[NfNs]>
#
# where:
# <pos_i> - element position in array coords [meters]
# <oriX_i> - local x-axis of element, unit vector
# <oriY_i> - local y-axis of element, unit vector
# <wgt_rl[k]> - real part of kth element weight
# <wgt_im[k]> - imaginary part of kth element weight
# k = ifreq + Nf*(istate - 1), weights are looped with frequency
within state
#-----------------------------------------------------------------------
-----------
0. -0.5 -0.5
1. 0. 0.
0. 1. 0.
1. 0.
0. 0.5 0.5
1. 0. 0.
0. 1. 0.
1. 0.
```

Link from Driven Modal or Terminal Antenna to SBR+ Design

HFSS lets you create a one-way link from a Driven Modal design or a Driven Terminal design with a single terminal to an SBR+ design. The source can be a finite array design, including

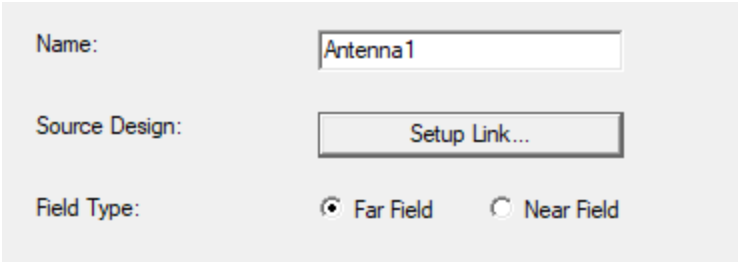
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

designs with blockages. For finite array design, the phase center is configured using the physical cells, and automatically transformed to the virtual cells. The source design **Excitations** menu includes a **Set Far Field Phase Center** command that lets you specify the phase center for each lumped port and wave port it contains. You assign the phase center to a coordinate system. The far-field phase center determines the point from which SBR+ rays are launched when the linked source is configured as far-field. It has no effect for linked sources configured as near-field. It can affect the Export Element Pattern from Antenna Parameters dialog.



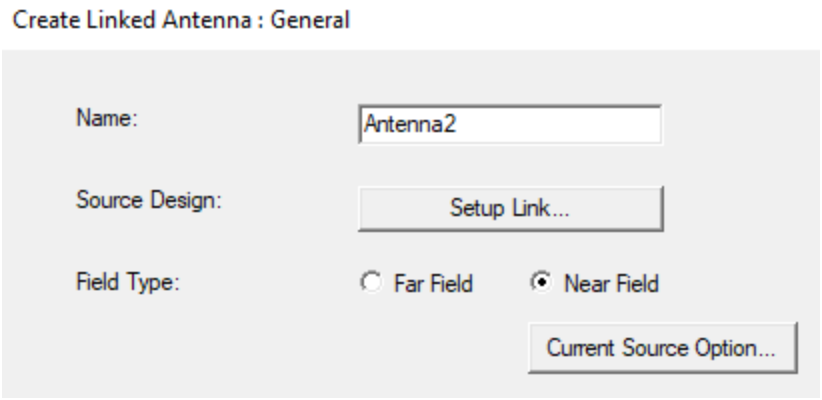
The target SBR+ design includes the associated command **3D Component>Create Antenna>Link to Source Design**.

From the target design, you can select either Far-Field or Near-Field as the linked field style.

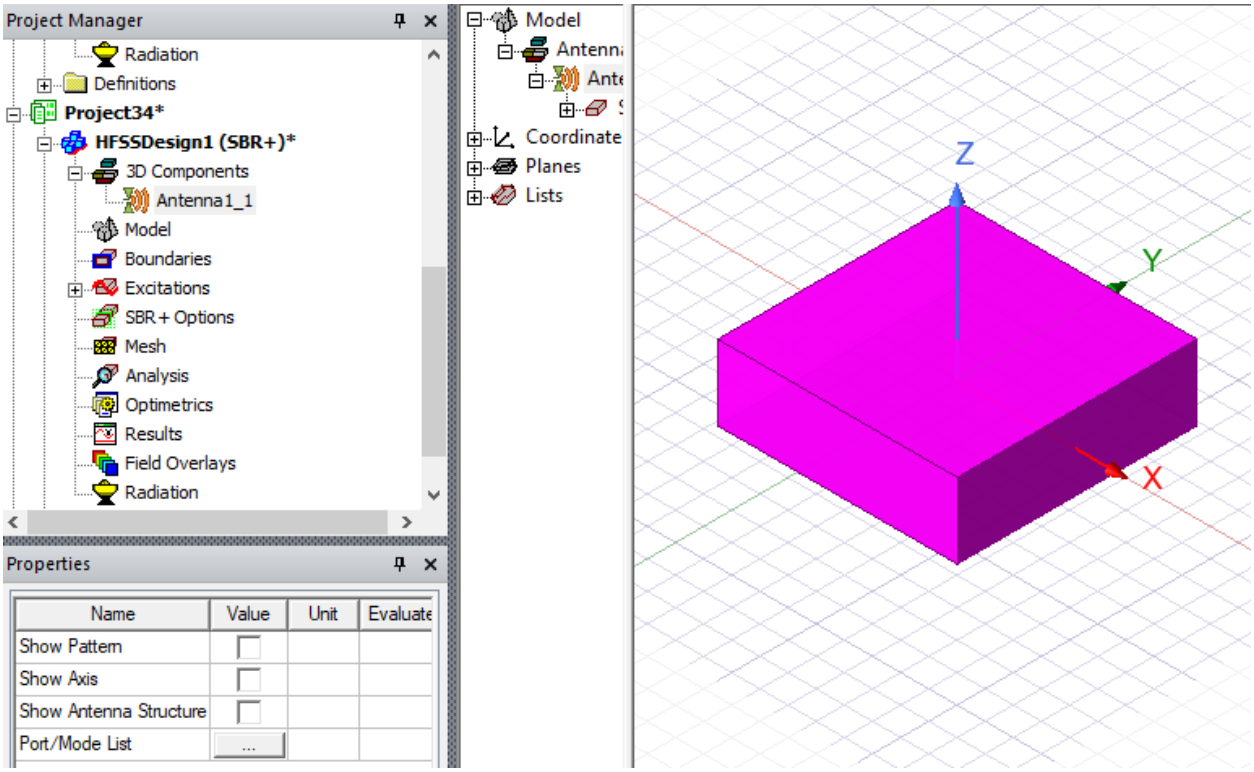


If you select Near Field, the **Create Linked Antenna** dialog includes a button for **Current Source Option...** allowing you to specify a customized power fraction. Use Global is the default. Enabling Thin Sources lets you specify a Power fraction should be greater than 0.1 and less than 1. Solutions are invalidated if you change current source conformance, use Thin Sources, Power Fraction, or change the Global option.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



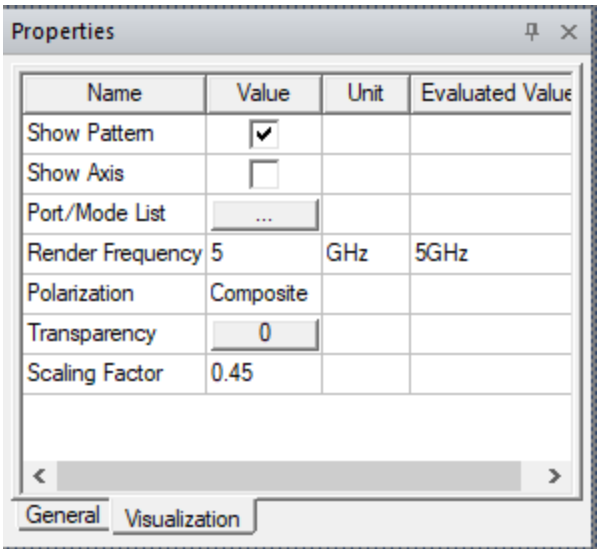
In the target design, the antenna link brings over the source design model and includes visualization options. The selected geometry is shown in its shaded view in the SBR+ target design with its designated color and transparency..



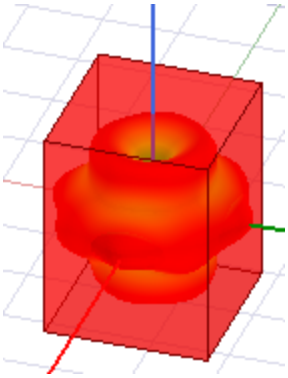
The linked source design appears in the target design under 3D Components. The source design ports appear in the target design under the excitation folder. You have the option when linking to a source project to Link as Composite Port, to simplify and speed up source projects with many excitations. However, you cannot edit those imported ports because they are part of the linked antenna component. You can turn on Field visualization of the source design by

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

checking **Show Pattern** on the **Visualization** tab of the 3D component property in the target design.



With **Show Pattern** checked, the source pattern is visible from the linked 3D Component bounding box in the target SBR+ design.



Prerequisites

- Set the phase center in the source design. The source design requires the presence of at least one wave port or lumped port to activate the **Set Far-Field Phase Center...** command.
- The source design must be solved before you can **Show Pattern** for the 3D Component Instance in the target SBR+ design.
- For simulation, the target design SBR+ solution frequency range must be in the range of source design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To support directivity and gain in the SBR+ design with the link, you must use an interpolating sweep with the 3D Fields Save Option selected in the linked design, either **Save Fields (At Basis Freqs)** or **Save radiated fields only**. This provides interpolatable S-parameters from the source design for the SBR+ target design.
- To model EM scattering effects of antennas in SBR+ simulations, you can select geometries from the source design to represent blockages. These can be model, or non-model objects. For this feature the SBR+ solver is passed a CAD representation of the antenna and it ignores this CAD representation when launching the rays from the antenna source but includes it when tracing the reflected rays from the platform.

Limitations

When the source design is in Composite-Driven Modal solution type or Composite-Driver Terminal with a single terminal:

- You can only choose one single phase center as the global phase center for all ports.
- Composite excitation becomes an incident wave in the target design. Therefore, we naturally should not have the port/mode option anymore.
- For Terminal designs, a single terminal is permitted.

When you select **Show Pattern**, the pattern data is cached for future use. The cached data is available until the design is closed. If you update the source design, **Clear Linked Data** is available for you to clear the cached data. If you clear the Linked data, field pattern is re-computed when you select **Show Pattern** again.

To model blockage structures, the source design can be Driven Modal or Driven terminal or a finite array. If you use a finite array, the **Model Blockage** tab on the **Setup Link** dialog shows up if the array has model / non-model blockage objects. The GUI operations for array blockages are same as non-array projects. See steps 4 to 7 below.

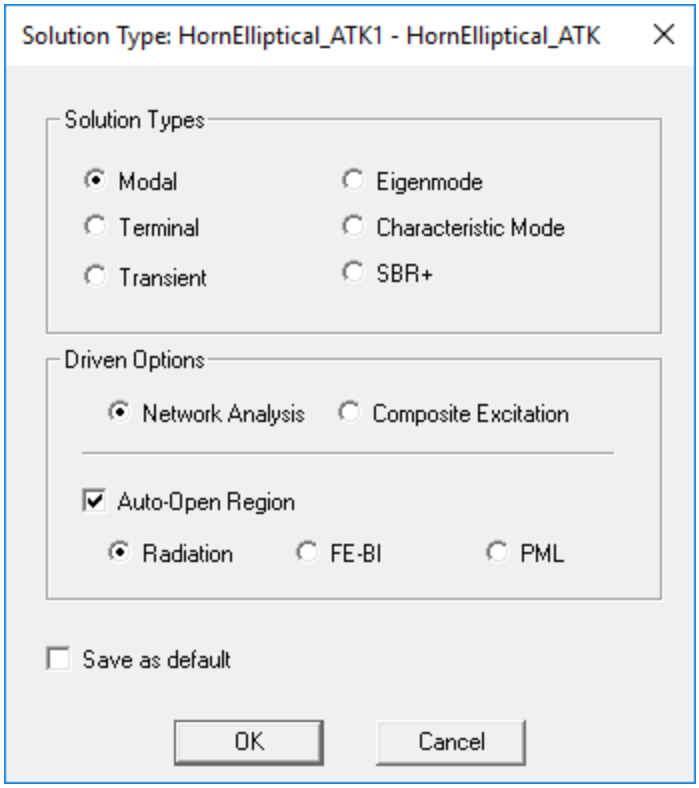
Creating a Link from Driven Modal or Driven Terminal Source to SBR+ Target

To create a link from a Driven Model Source or Driven Terminal Source to an SBR+ target:

1. Set up a driven Modal solution or Driven Terminal as the source.

   You can choose any **Modal** options, although if you choose **Composite Excitation**, you must choose a single phase center for all ports. You can use **Auto-Assign Region** options if convenient.

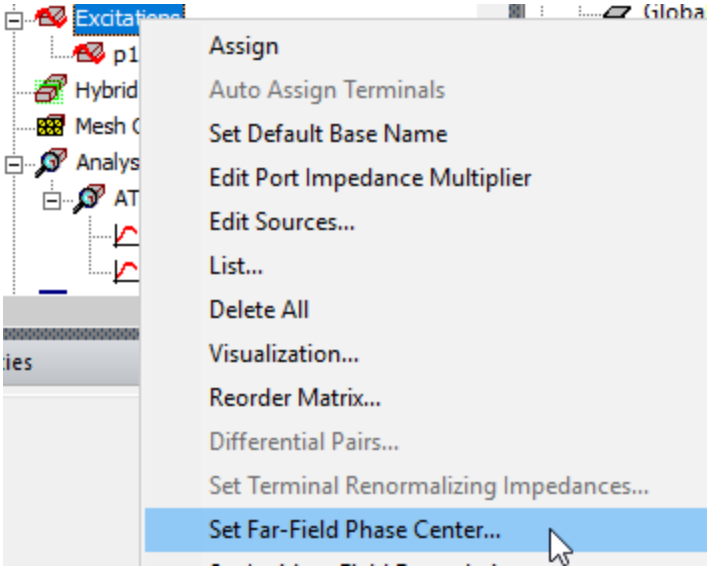
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



All ports in the source design must be either wave ports or lump ports.

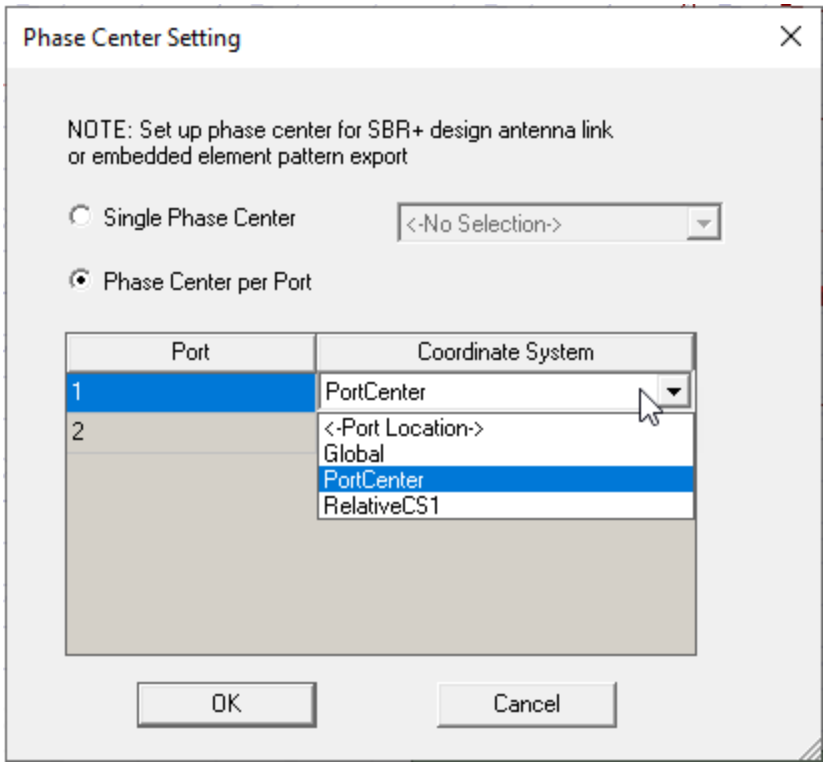
2. Set the Phase Center in the Source design. The phase center(s) determine the point(s) from which SBR+ rays are launched when the linked source is configured as far-field. It is important for accurate results to position the phase center as near as possible to the physical phase center of the antenna for the selected port, especially if the source is placed nearby to SBR+ scattering structures. Otherwise, the accuracy of ray fields illuminating nearby structures will be compromised, leading to inaccurate scattered fields and final results. Either right-click **Excitations** in the Project tree and select **Set Far Field Phase Center...** or click **HFSS>Excitations>Set Far Field Phase Center...**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



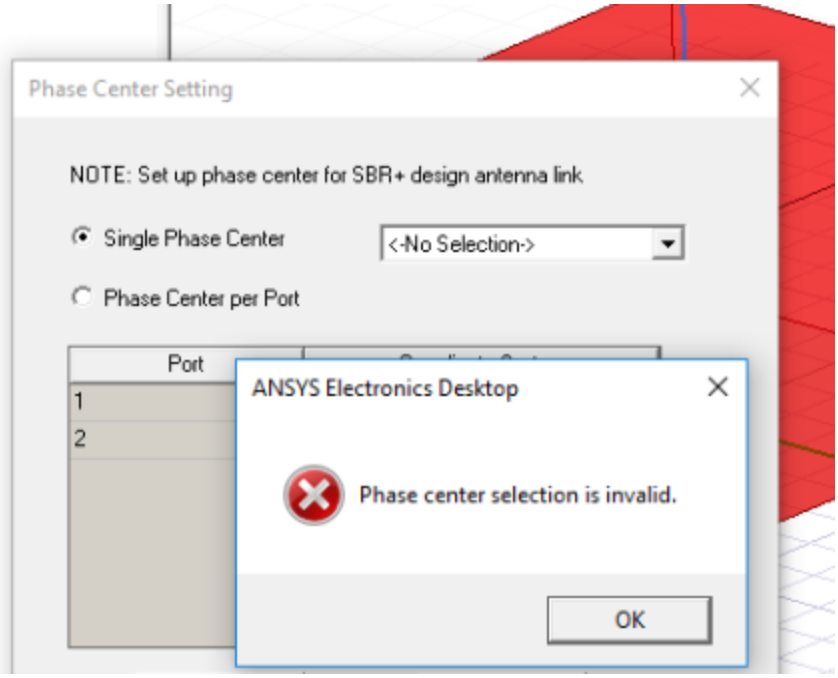
The **Phase Center Setting** dialog appears, with **Phase Center per Port** selected as default with the mode <- Port Location->. By using **Phase center per Port**, you have more fine-grained control over each phase center. This feature provides a convenient option to assign phase center at port location (port geometrical center), which is a common selection for Phase Center per Port mode. If you have created additional Coordinate Systems, these appear in the drop-down menu as options for the phase center.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



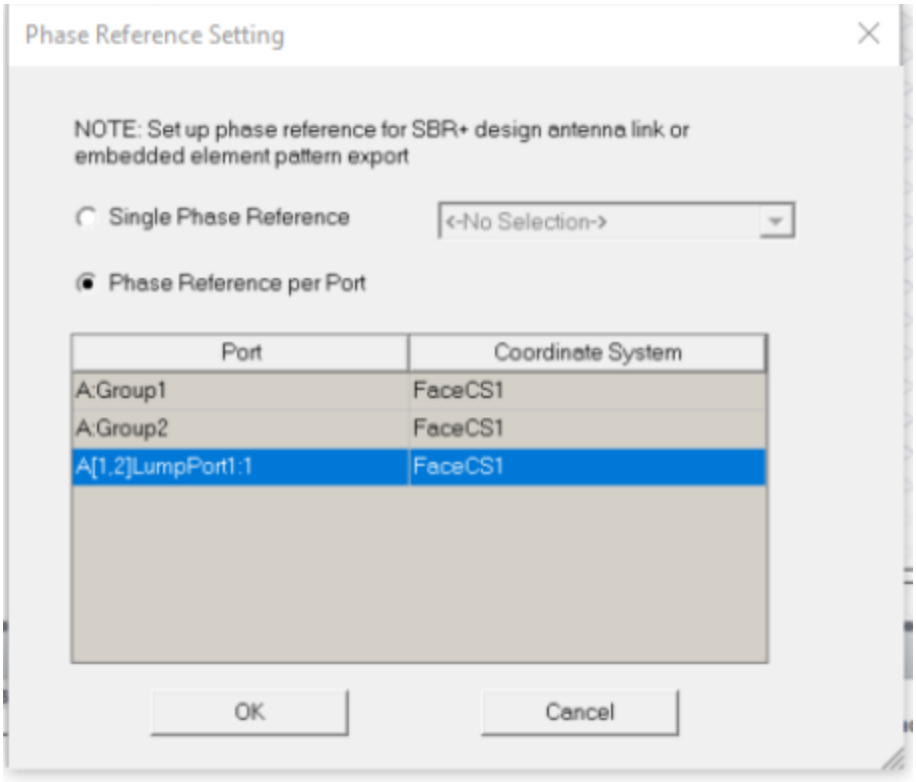
Single Phase Center is to assign the global option to all the ports (one port or multiple ports with the same phase center option). You also have the option to select Link as Composite Port when you setup up the link from HFSS to SBR+ design. Phase Center Per Port lets you specify different phase centers for different ports. Validation checks to ensure that each port is assigned a valid phase center. You must not leave any ports with <No Selection>.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

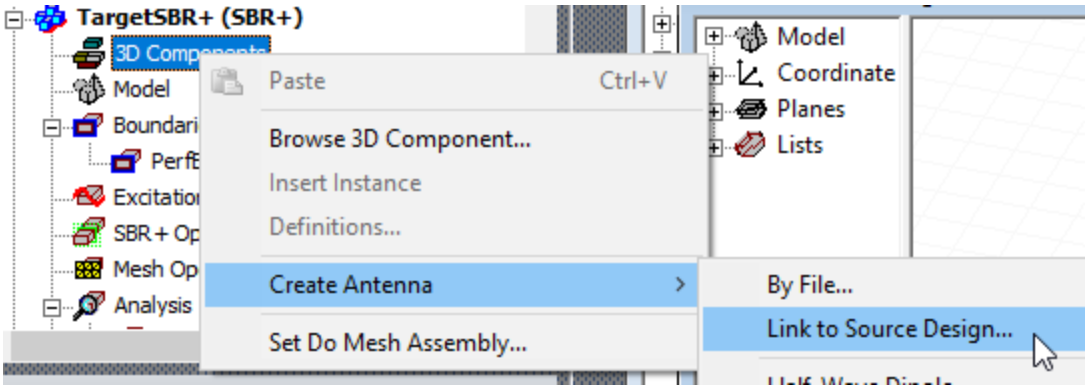


In the case of a design using the Beta feature for subarrays, the **Phase Reference Setting** dialog shows composite groups if you want to specify unique phase reference for each group. In this case, the “Port Location” selection is not available, and you must select an actual reference coordinate system.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

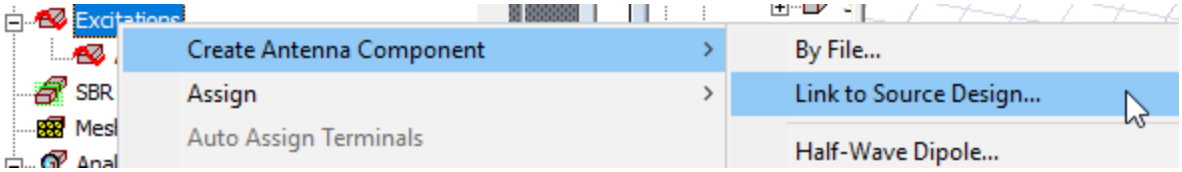


3. When you have completed the source design setting (phase centers, solve setups, etc) and solved it, create an SBR+ design as the target design.
4. Right-click **3D Components** in the Project tree and select **Create Antenna>Link to Source Design...**

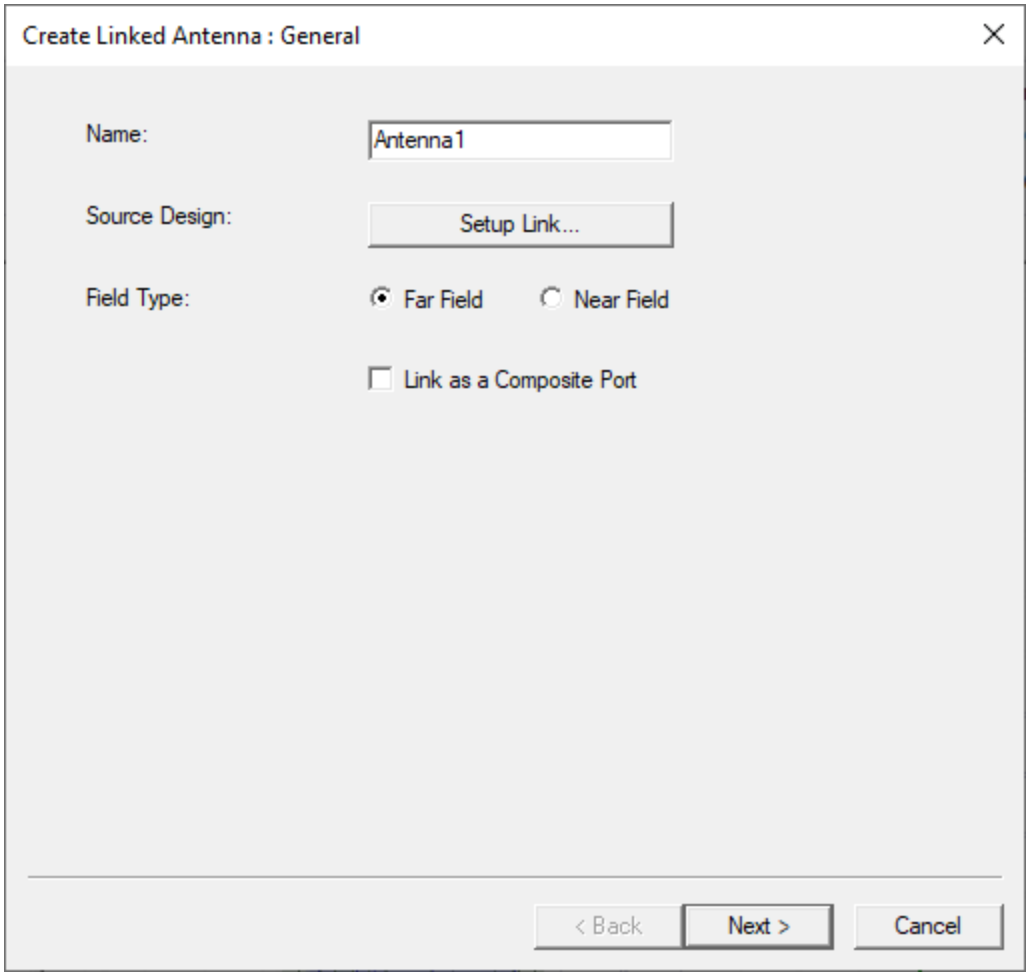


You can also right-click **Excitations** in the Project tree to select **Create Antenna>Link to Source Design...**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Create Linked Antenna** dialog appears.



Select **Far Field** or **Near Field** as appropriate.

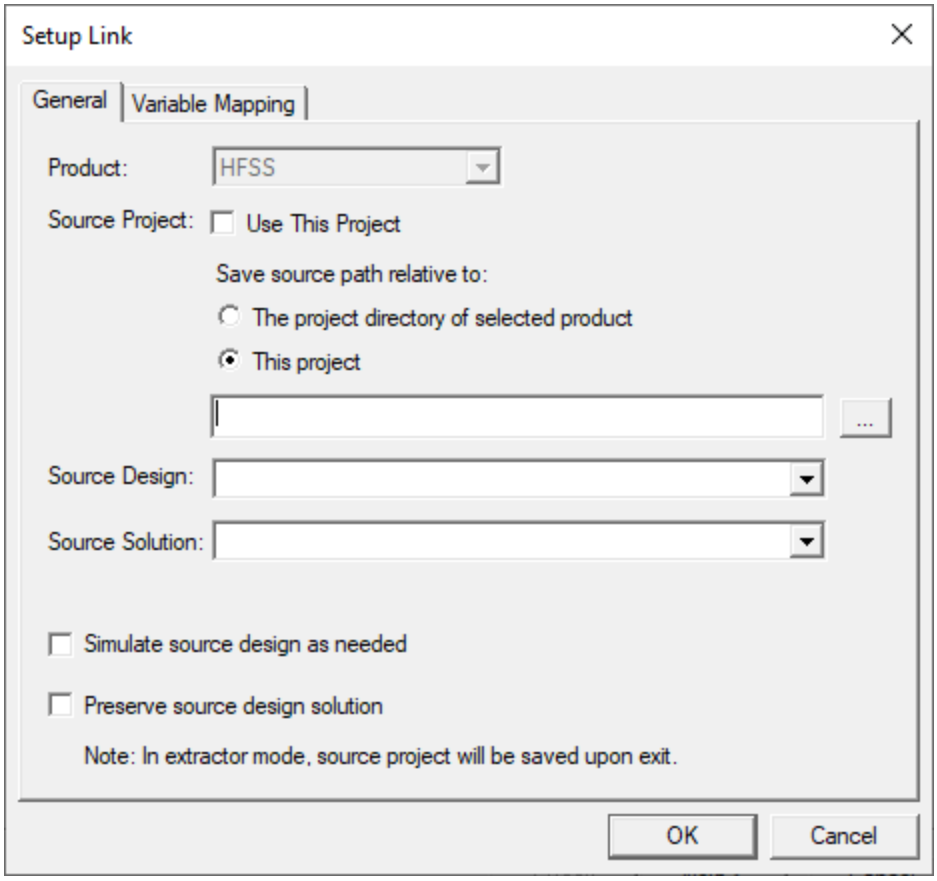
The **Link as Composite Port** option addresses the situation where an imported design contains many ports. By default, the model window displays the same number of virtual ports in the SBR+ target design. However, if the source design contains many virtual ports, they will look cluttered and require significant computational resources to solve. You create a single composite port in the SBR+ target design to represent all the ports in the HFSS source design. During computation, the SBR+ target design sends the information about a single

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

composite port to the solver, rather than information about all the ports. This saves a significant amount of resources. For further discussion, see Link as Composite Port.

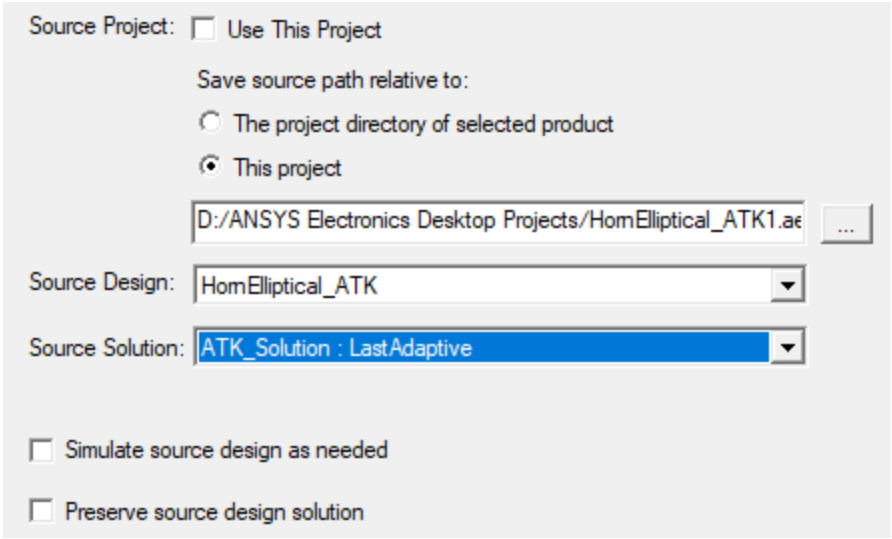
5. Click **Setup Link...**

The **Setup Link** dialog appears.



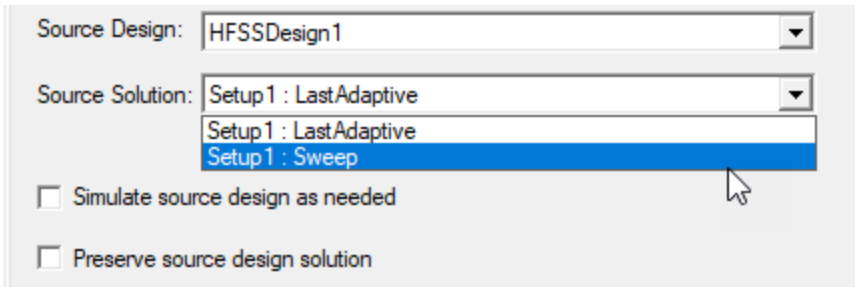
If the source design is in a different project, you can click the ellipsis button **[...]** to open a file browser to select the project of interest. When you select the project and OK the browser, the fields are filled in.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the source design is in the same project, you can check **Use this Project** and select from available Source Designs.

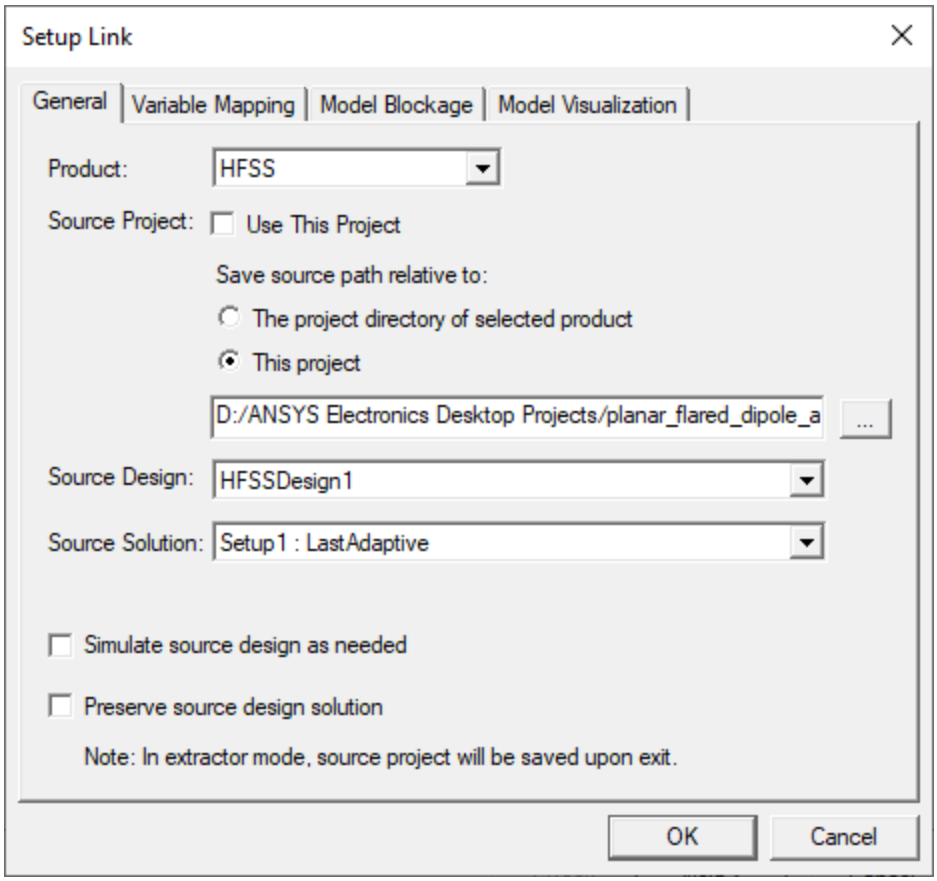
If you have defined an interpolating frequency sweep in the source design, you can select it on the drop-down for the source solution. (This feature supports directivity and gain in the SBR+ design with the link. (See "Requirements for Creating a Gain Report for SBR+ Design with Driven Modal or Terminal Source" on page 13-228). It provides the S-Parametersfrom the source to the target. In the source sweep setup, you must have selected a 3D Fields Save Option.)



For this release Model blockages can handled as a separate step outside creation of the linked design, by selecting geometry from the SBR+ region to be designated as “Model Blockage” associated with a particular source antenna, as described in . Note there is no longer any restriction on what objects and/or materials can be designated as “model blockage” geometry. For either solution type, SBR+ or hybrid HFSS-SBR+, this selection is from the available SBR+ region geometry objects. Once you specify antenna blockage geometries for each source antenna, we can then solve the combined target SBR+ design. For details see Design Flow for SBR+ Solution Type.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

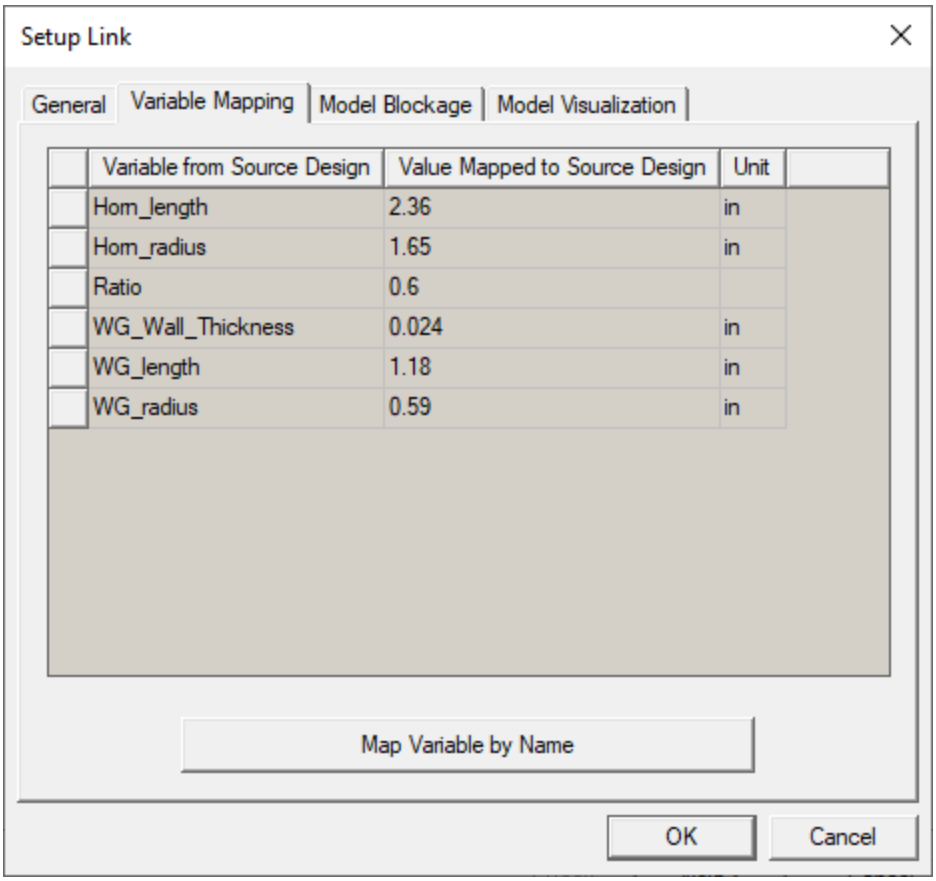
If you use the older method for designating blockages, once you select the source design, additional tabs for **Model Blockage** (if model or non-model blockages exist in the source) and **Model Visualization** appear.



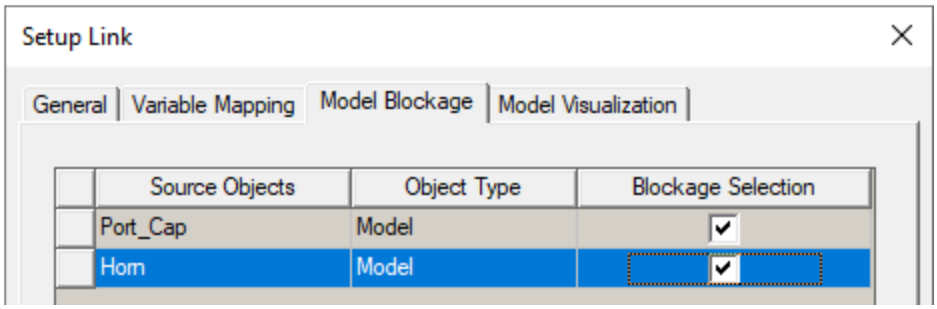
The **Visualization** tab lets you specify how the linked model is displayed in the SBR+ target design.

6. The **Variable Mapping** tab lists variables and lets you define mappings between the driven modal source design and the SBR+ target design, if necessary.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

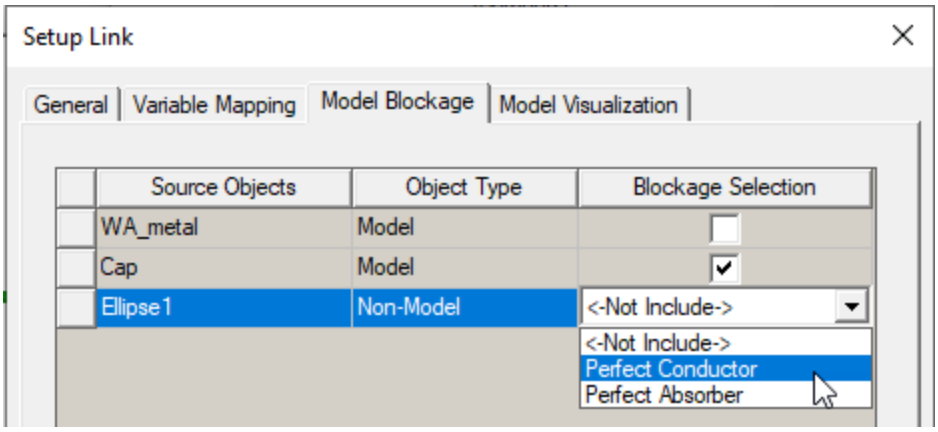


7. The **Model Blockage** tab lets you select geometries from source design to represent the blockage structure. Non-solve inside, and non-model objects are available for selection.



For Non-model objects, you can select <Not Include> Perfect Conductor, or Perfect Absorber.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

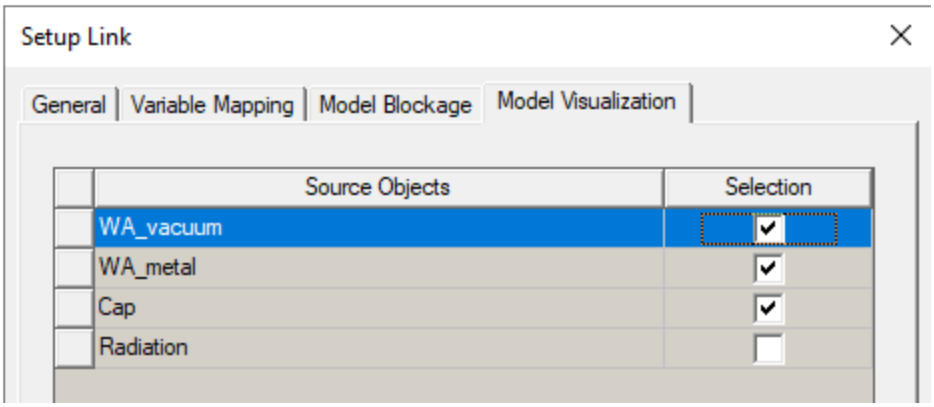


In many cases, it is sufficient to represent SBR+ sources entirely in terms of their far-field or near-field radiation characteristics, and there is no need to configure Model Blockage. However, in some cases, the antenna structure can generate significant secondary scattering after rays have first reflected off the main scattering geometry that the source illuminates. An example would be a feed antenna blocking part of the main beam of a reflector antenna, where the reflector surface is simulated using SBR+. Model and non-model structures selected under Blockage Selection will have special behavior relative to SBR+ ray-tracing. These surfaces will be invisible to SBR+ rays when they are first launched from the source. However, after the first bounce, they will interact with SBR+ rays just like any other surfaces in the scattering geometry (with some further qualification, described shortly). For example, to capture reduced reflector antenna gain due to blockage by the feed horn, you may select the elements representing the body of the horn under Blockage Selection. In some cases, the antenna structure may be too detailed to effectively interact with SBR+ rays, recognizing that ray-tracing is a high-frequency (short wavelength) approximation. In those situations, one may instead create a simplified blocking structure, such as a rectangular or circular plate, to stand in for the physical antenna structure. To do this, create the simplified blocking structure as one or more non-model objects in the source design and then select them under Blockage Selection, configuring them as either Perfect Conductor or Perfect Absorber.

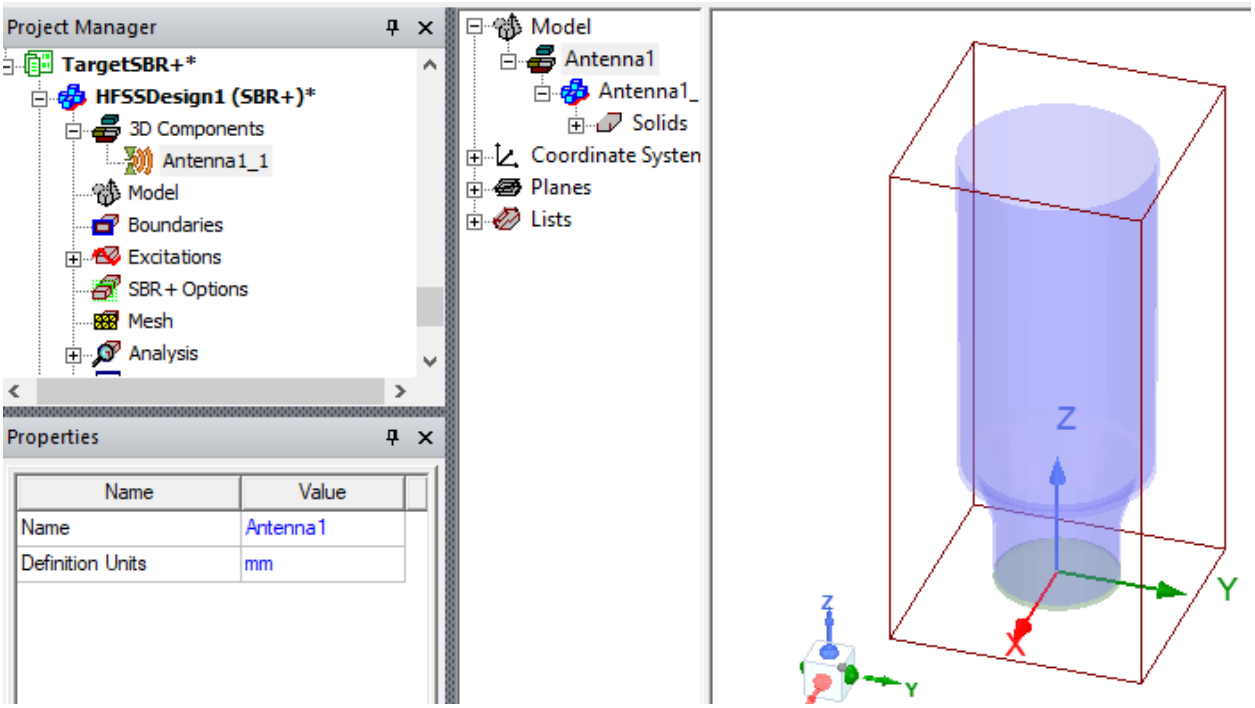
The behavior of model and non-model objects selected under Blockage Selection have some further qualifications in the context of SBR+ with multiple source antennas. Suppose there are two antennas A and B, and both have Model Blockage objects selected, designated as S_A and S_B, respectively. S_A will be invisible to first-bounce SBR+ rays launched from antenna A, and then become visible on subsequent bounces. However, S_B will always be visible to SBR+ rays launched from antenna A, even on the first bounce. A related point is that when rays hit S_A, they will generate scattered fields to every antenna accept to antenna A. Likewise, when rays hit S_B, they generate scattered fields to every antenna accept to antenna B.

8. The **Visualization** tab lets you specify how the linked model is displayed in the SBR+ target design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

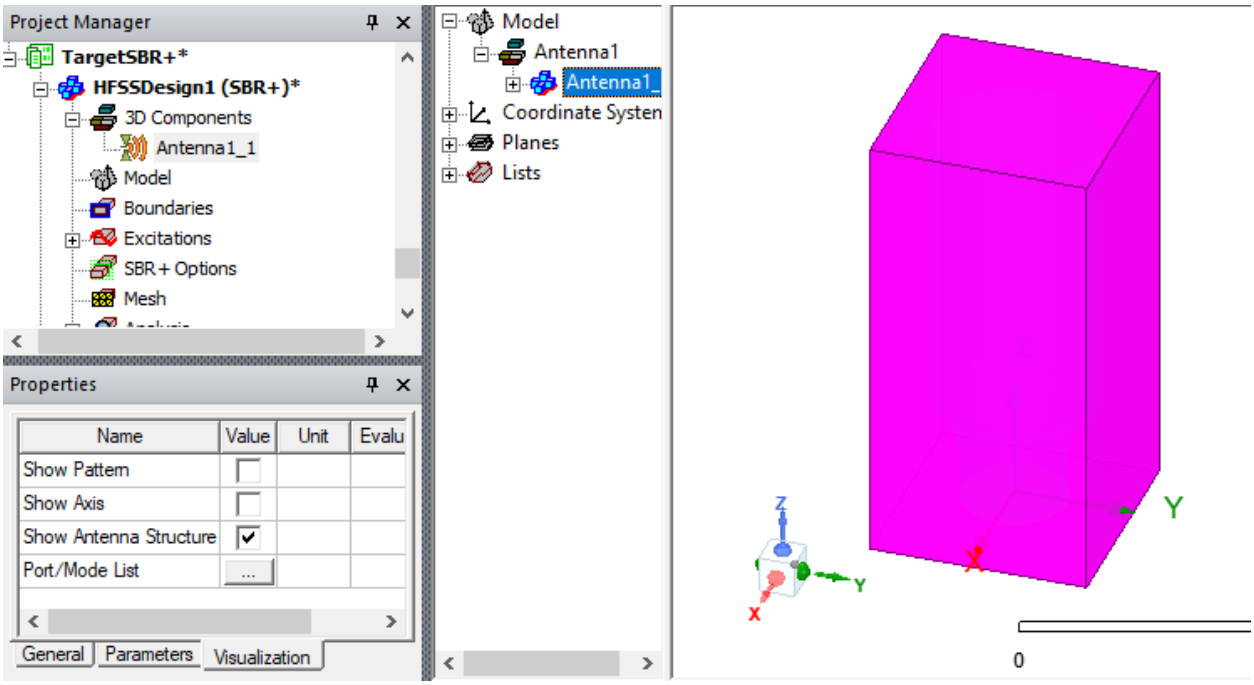


For example:

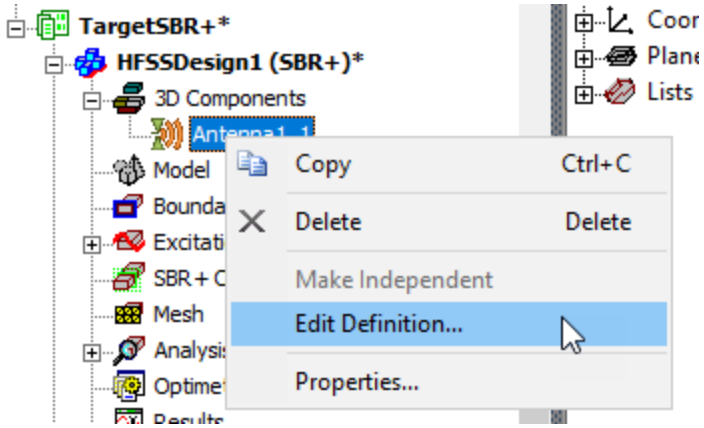


Selecting the linked 3D Component also provides a **Visualization** tab in the properties with additional control over how a linked project displays in the SBR+ target design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also gain access to the Setup Link **Visualization** tab by Editing the definition of the 3D Component.

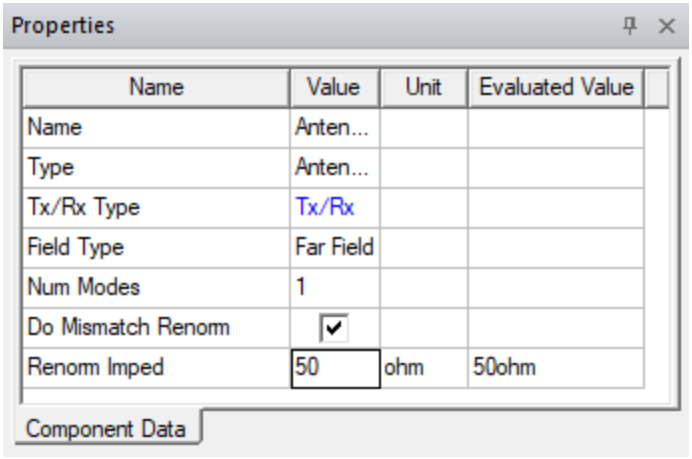


This opens the **Setup Link: General** window, where you can click the **Setup Link...** button.

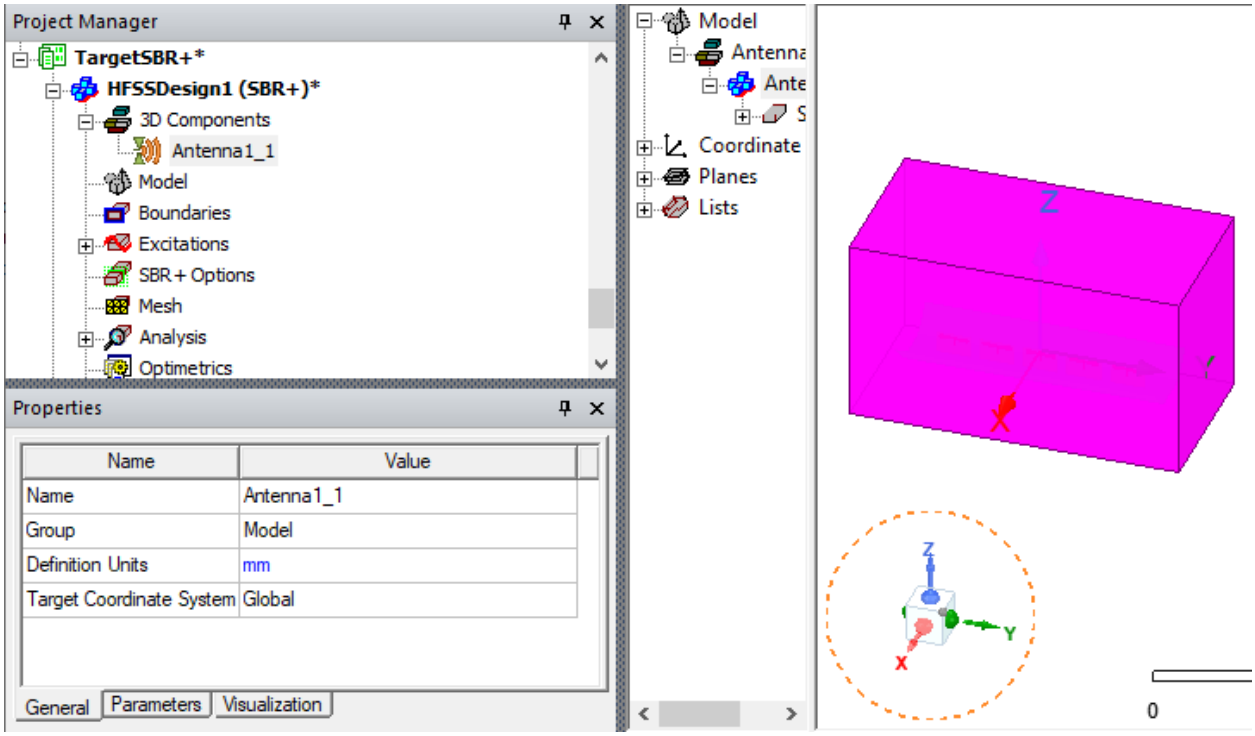
9. Click **OK** to finish importing a linked antenna to the target SBR+ Design. The target design displays the linked antenna as a 3D Model in the Project tree and in the Model window as a model based on how you specify the visualization.
10. Ports appear in the **Project** tree under **Excitations**. If you right click on the **Excitations** icon, you can view and edit Tx/Rx selection for all fields, including coupling between ports within the same antenna and self coupling. The properties for each excitation shows the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

current Tx/Rx type. If you select do **Mismatch Renorm**, an editable **Renorm Imped** property appears which lets you Renormalize linked ports.

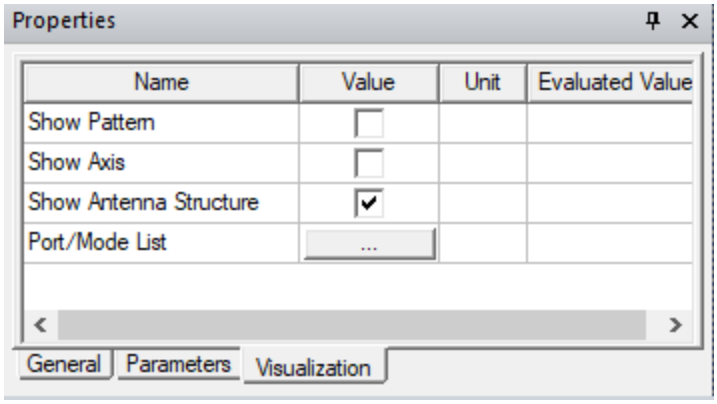


11. Select the 3D Component for the source antenna in the target **Project** tree to view the properties.

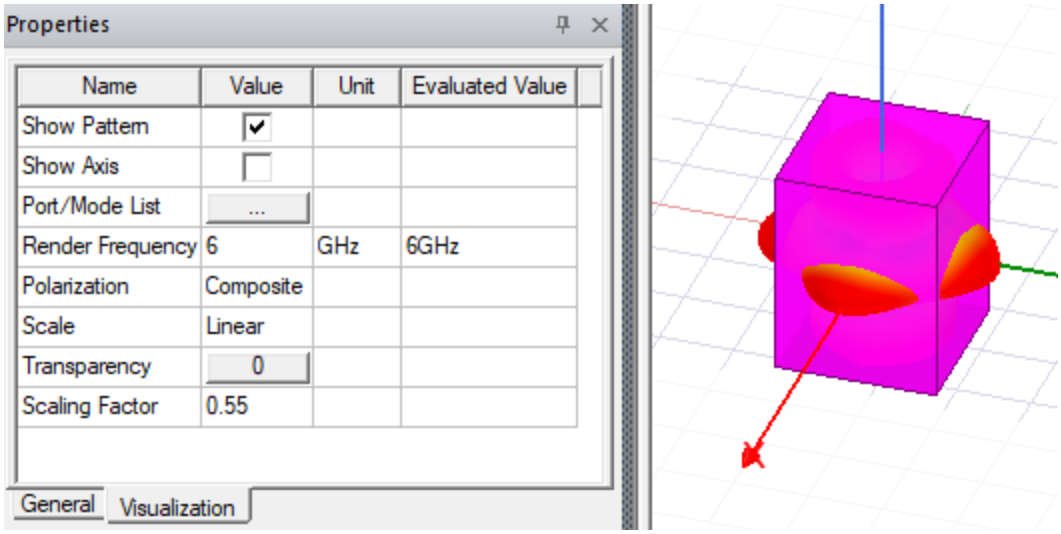


If you select the **Visualization** tab, you see the following properties.

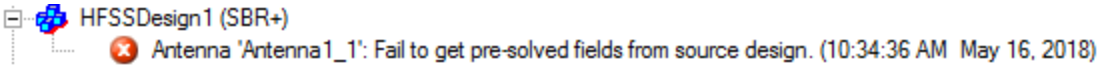
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you have previously solved the source design, and correctly assigned valid phase centers, you can check **Show Pattern**, and see additional display properties. The antenna pattern is computed and displayed in the bounding box in the Model window.



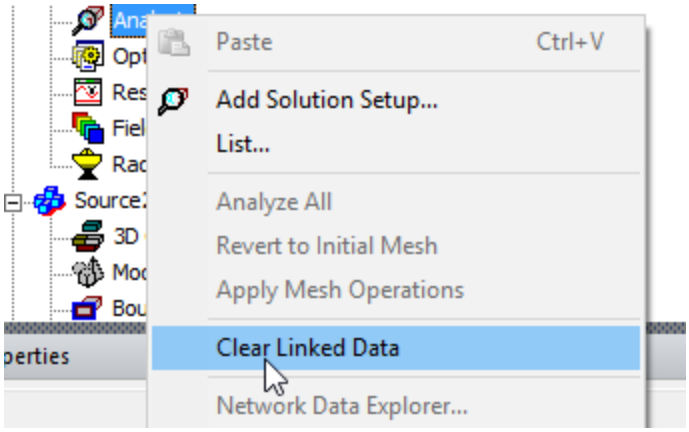
If the source design has not been solved, the **Message** window will display an error message.



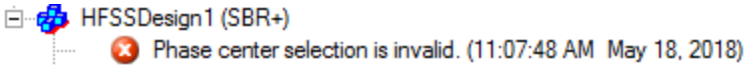
If you update the source design, the **Clear Linked Data** command becomes available for you to clear the cached data.

If you clear the linked data, the field pattern is re-computed when you check **Show Pattern** again.

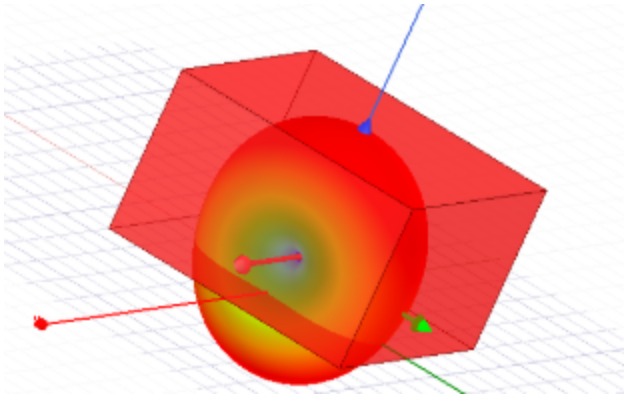
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you made changes to the source design that conflict with the link or phase center configuration, an error message is displayed.

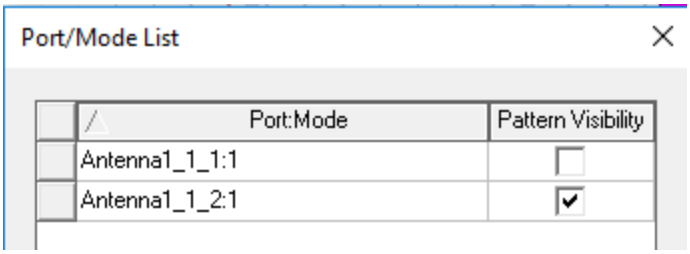


If you check the **Show Axis** property, the phase center axis is shown at the antenna pattern.

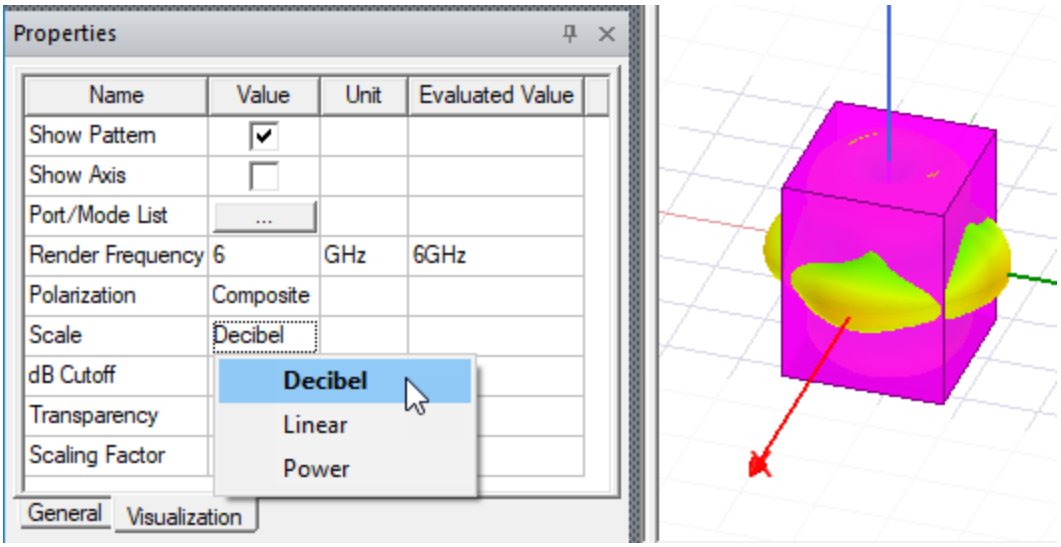


The **Port/Mode List** is available unless the source design is driven modal composite. If you click the ellipsis [...] button for **Port/Mode List** on the **Visualization** tab properties, a dialog displays that lets you select which ports to show:

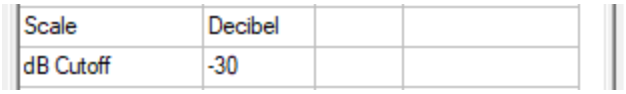
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Scale** property can be Decible, Linear, or Power.



If you select Decibel, an additional property for dB Cutoff appears.

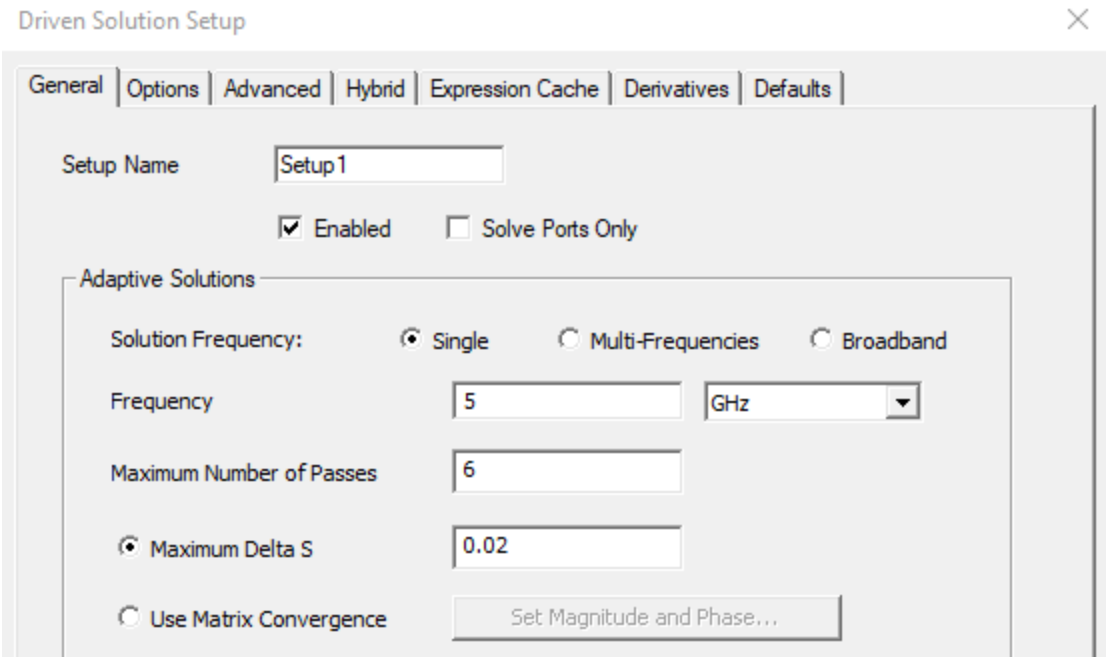


Simulation Setup for Driven Modal Antenna to SBR+ Design Link

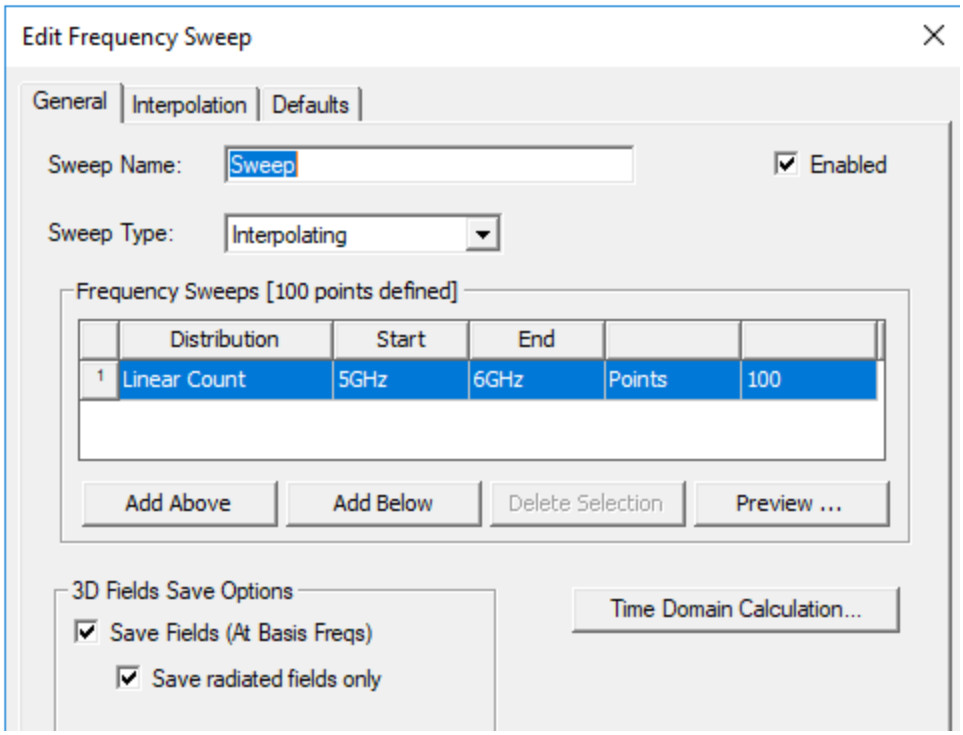
Once you have configured the SBR+ design with an Linked Antenna, you set up a model object and **Solution Setup**. The simulation frequencies in target design must fall in the range of the source design frequencies. Below is an example of source design setting and target design setting, using compatible frequencies. and Save Far fields so that Gain can be plotted.

Example Source Design Setup

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



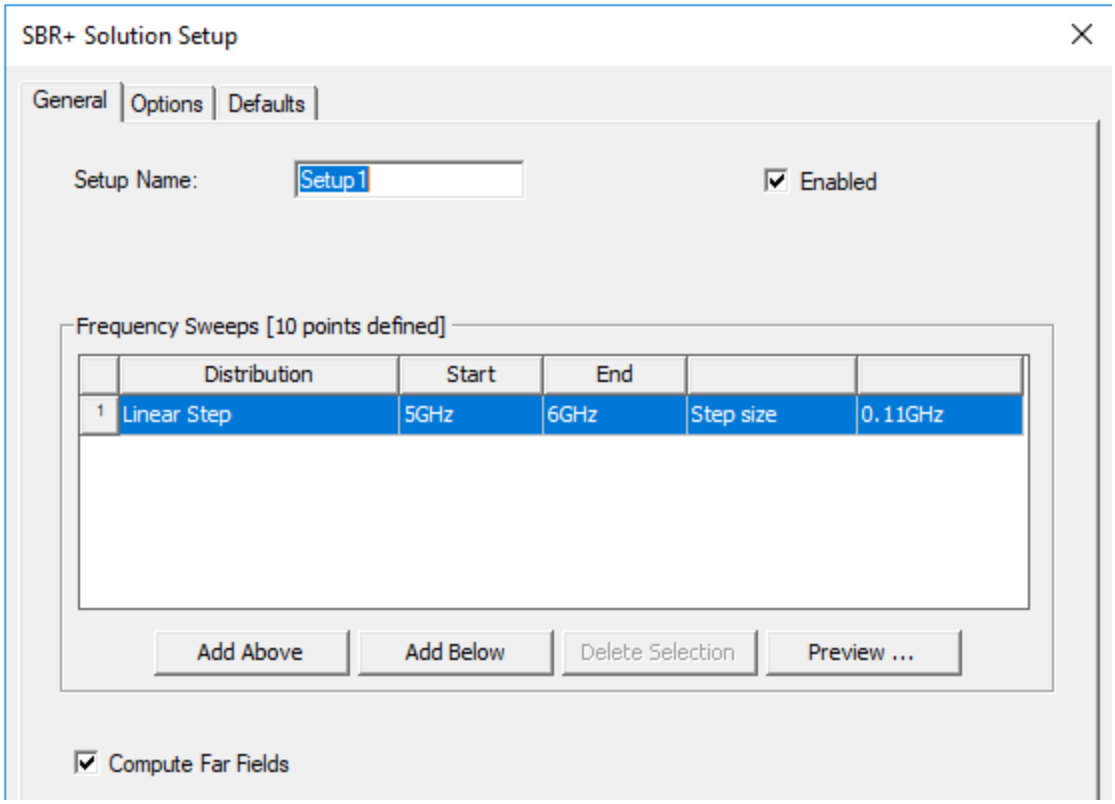
Source Design Frequency Sweep



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The sweep will be selectable in the Source link, and the **3D Fields Save Options** permit the Target SRB+ design to compute far fields and plot gain.

Example Target Design Solution Setup

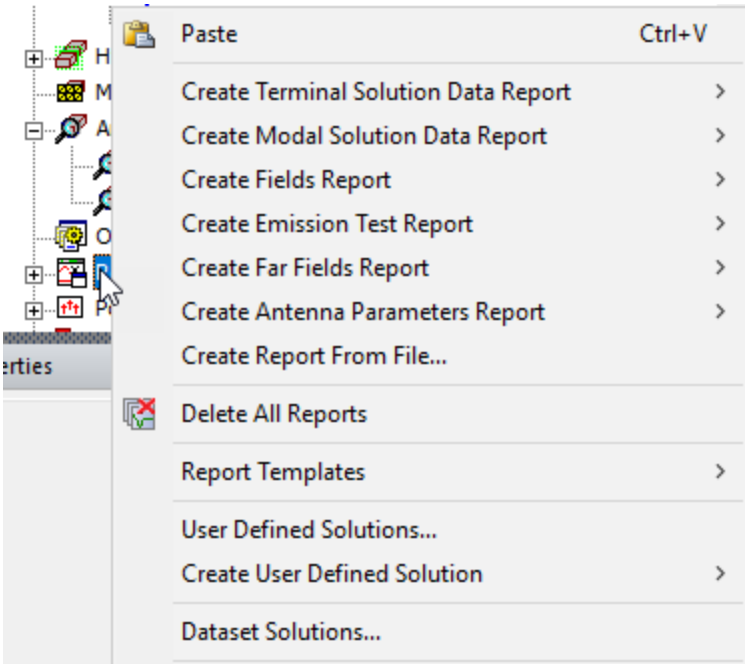


Compute Far fields is so that a Gain Report can be plotted based on the fields saved from an interpolating sweep in the linked source design..

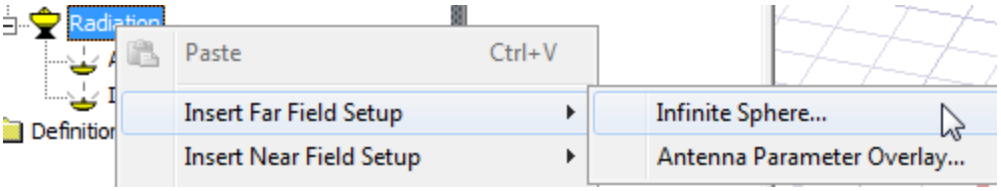
Viewing Results in the SBR+ Simulation

After you run an SBR+ simulation with linked Model antenna, you can create a variety of reports, including S Parameters, Far Fields, Gain, and Antenna Parameters.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You must create a Far Field Setup to access the Far Fields and Antenna Parameters reports.

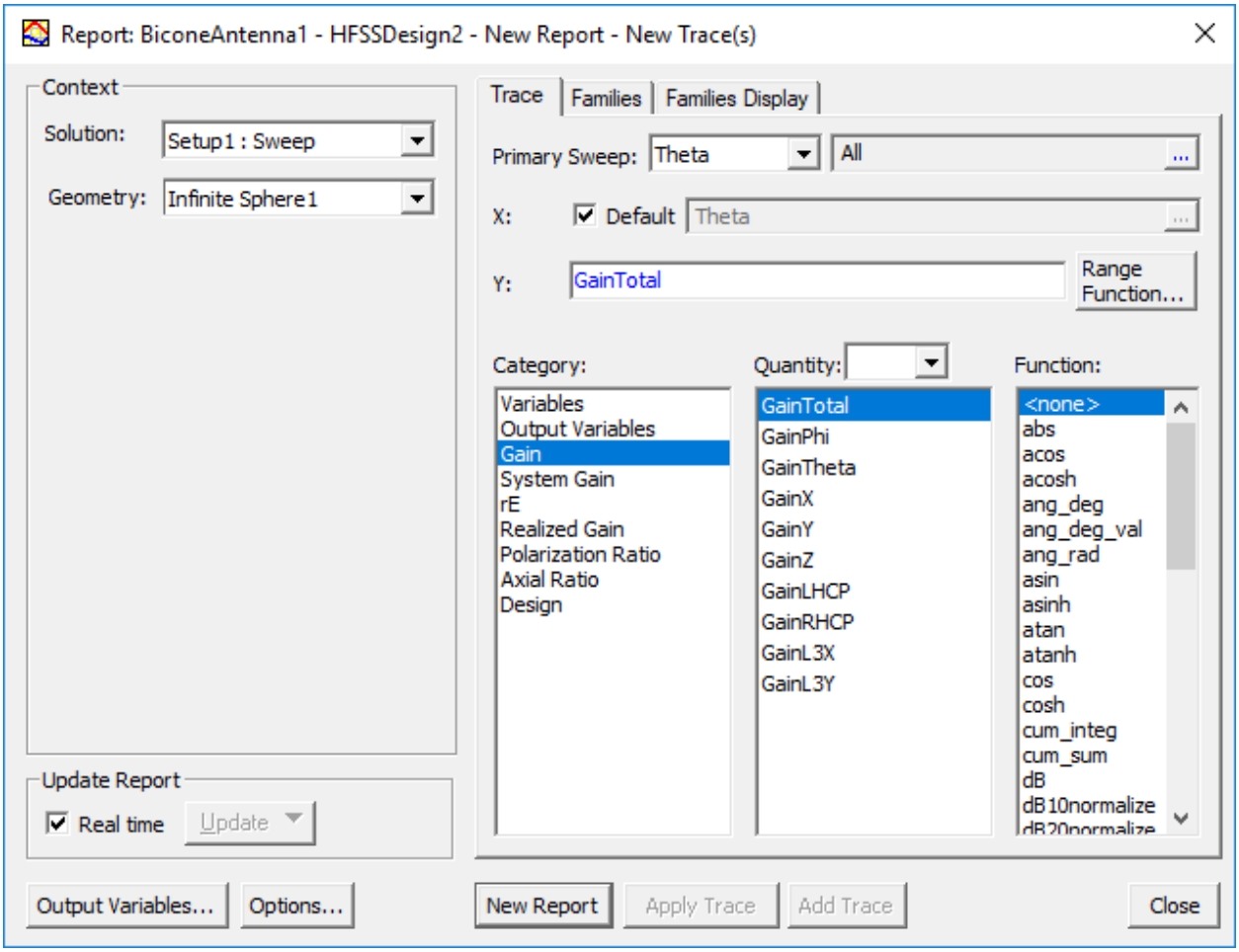


When a driven modal or driven terminal antenna is linked to an SBR+ target, an additional SBR+ power loss is considered for radiated power calculation. See *Technical Notes*: Radiated Power.

Requirements for Creating a Gain Report for SBR+ Design with Driven Modal or Terminal Source

If you have created an Infinite Sphere, solved an Interpolating Frequency sweep in the Source design and saved the 3D fields, and selected Compute Far Fields in the Source setup, you can create a Gain Report.

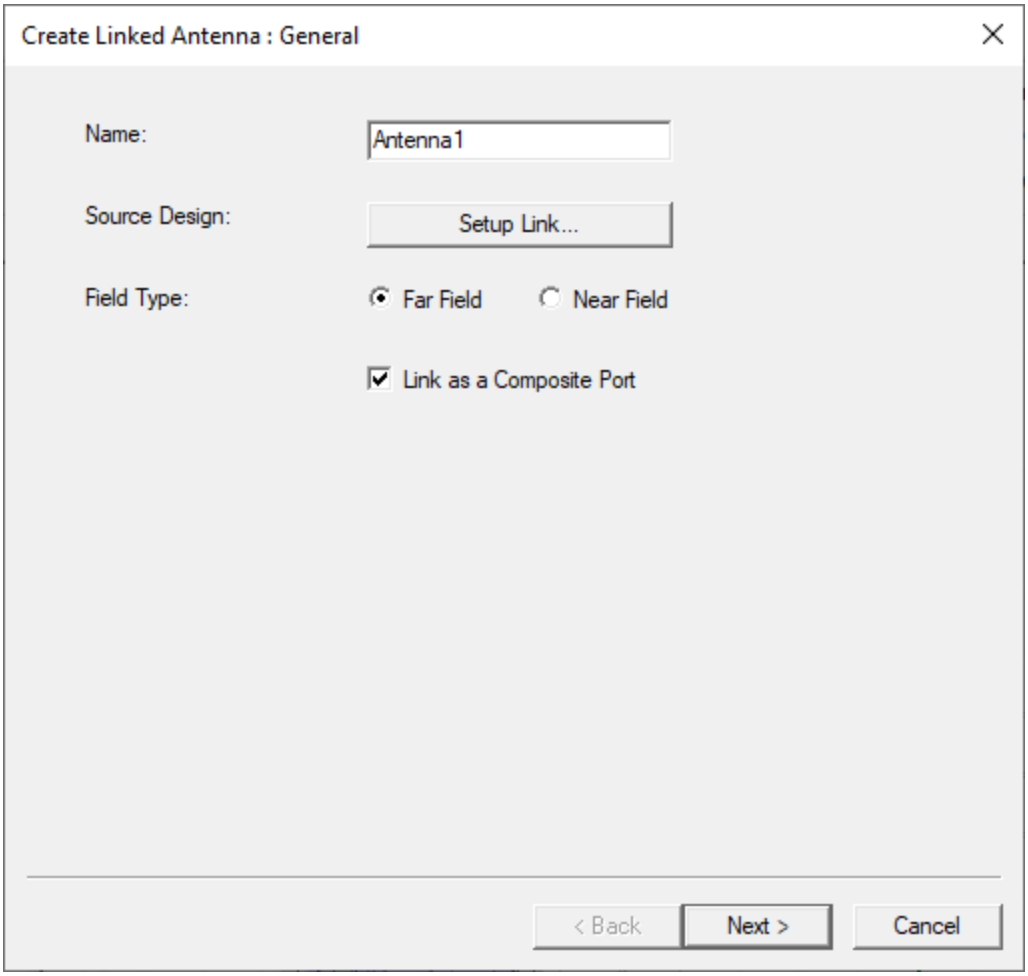
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Link as Composite Port

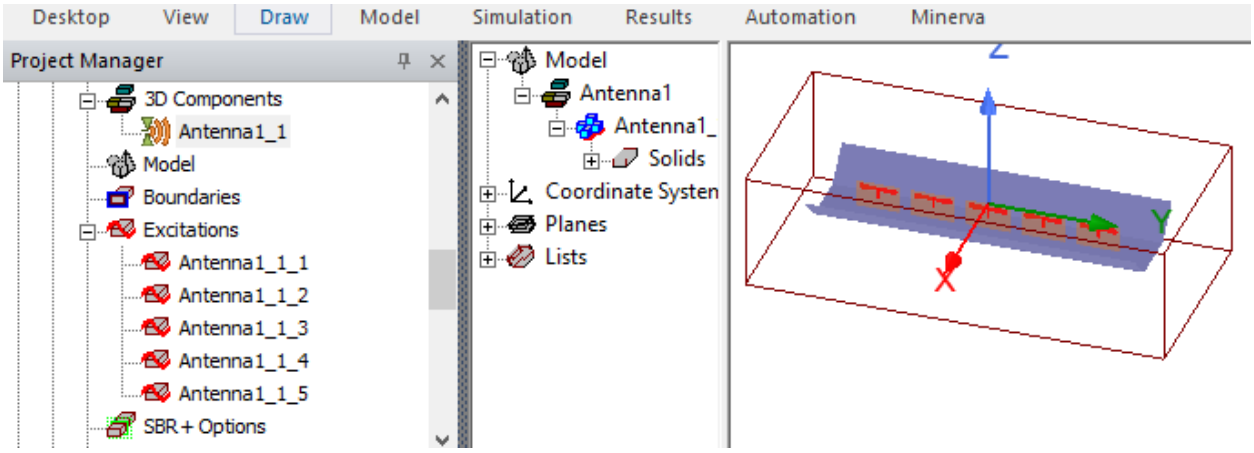
HFSS lets you create a one-way link from a Driven Modal design or a Driven Terminal design with a single terminal to an SBR+ design, as discussed in detail here. The source can be a finite array design, including designs with blockages. The **Link as Composite Port** option in the **Setup Link** dialog addresses the situation where an imported design contains many ports.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

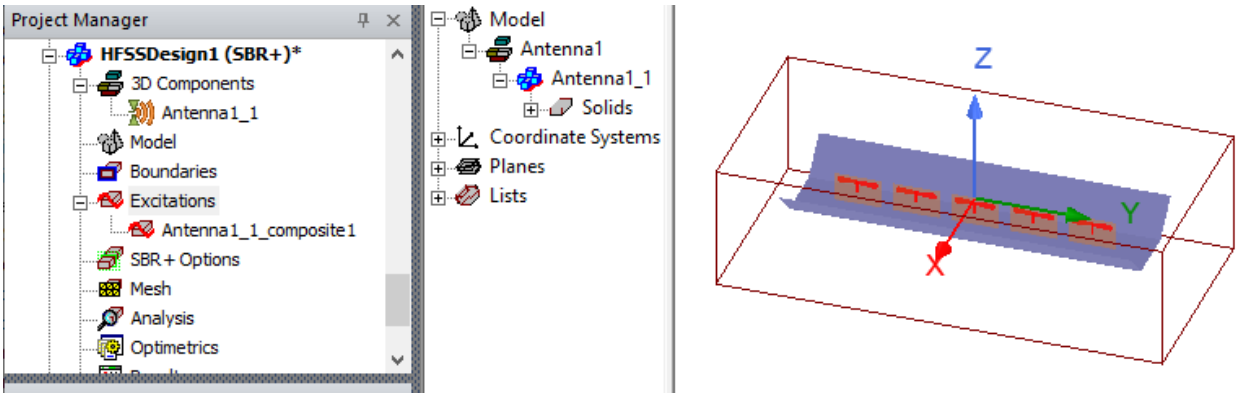


By default, the model window displays the same number of virtual ports in the SBR+ target design. However, if the source design contains many virtual ports, they will look cluttered and require significant computational resources to solve. For example, if you used the planar flared dipole array array example as a linked antenna, the excitations in the SBR+ target design shows this.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



By checking **Link as a Composite Port**, you create a single composite port in the SBR+ target design to represent all the ports in the HFSS source design. During computation, the SBR+ target design sends the information about a single composite port to the solver, rather than information about all the ports. This saves a significant amount of resources.



A composite port requires a single phase center, whereas each regular port requires its own phase center.

In order to reduce the N original ports of the source design to a single port in the target SBR+ design, **Link As Composite Port** implicitly (mathematically, not physically) incorporates a 1-to-N feed network that provides a perfect match to the linked multi-port antenna. At the same time, the composite-port link is formulated in such a way that some antenna pattern metrics and results, such as peak realized gain and radiated near- and far-fields, remain consistent between equivalent composite and regularly linked antenna configurations. As a result, when linking an antenna as a composite port into an SBR+ design, whether as a near-field or far-field link, you should expect the following results and behaviors:

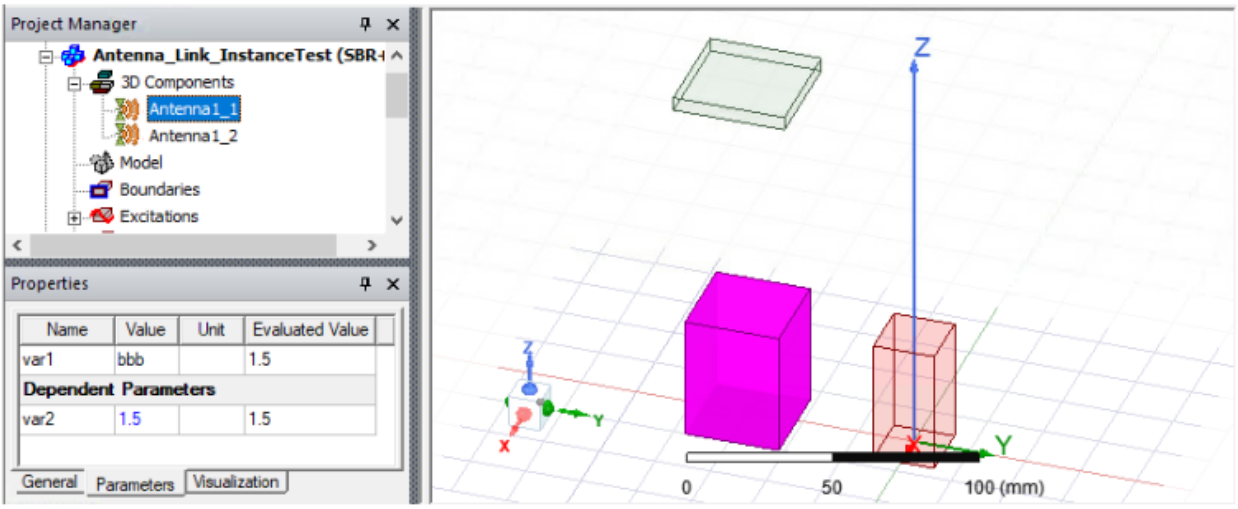
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Self-coupling: $S_{ii} = 0$ due to the matched network assumption.
- Cross-coupling: magnitude of S_{ji} will be larger when one or both of the referenced ports (*j*,*i*) are compositely linked than in the case of a regular link due to the matched network assumption.
- Accepted power: the accepted power reported in the Antenna Parameters table can be significantly different, especially if the magnitudes of *Sji* are large in the case of composite port link(s), due to the definition of accepted power including the effects of cross-coupling in determining the amount of incident power returned to each port in the target design. This effect can be verified (by eliminating it) by disabling SBR+ couplings in the Select Tx/Rx Antennas dialog.
- (Peak) Gain: the reported gain and peak gain can be significantly different owing to the inverse relationship of regular gain and accepted power (see above)
- (Peak) Realized Gain and Max U: these reported antenna parameters will match between composite and regular link configurations owing to the design and formulation of the **Link as Composite Port** feature.

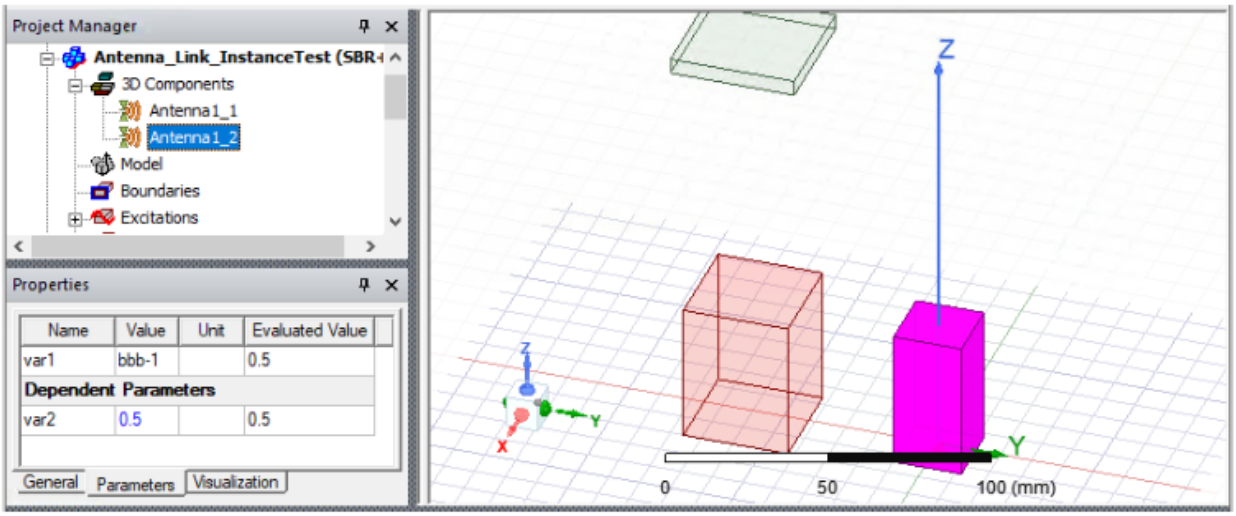
Linked Antenna Variable Support

Linked antenna components can be assigned with both component variables and design variables. The Model window updates bounding box size if the such variables are changed. This allows you to create multiple components instances that share the same component definition but have different variable values. You can then generate different analysis results and far field patterns for components with different variable values.

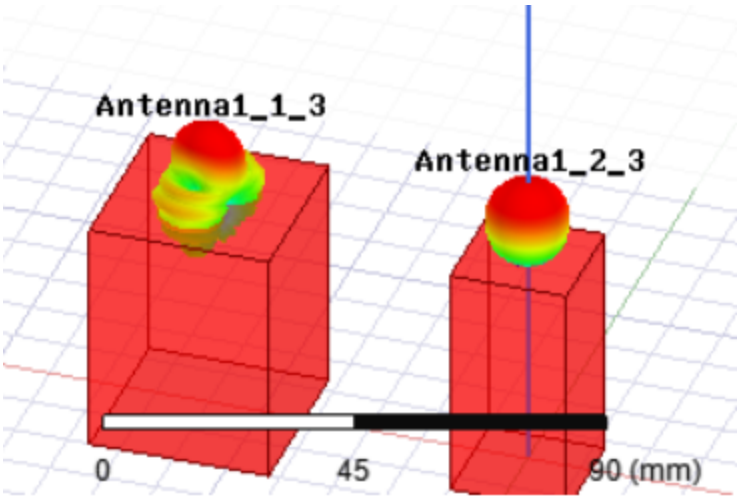
The following design has 2 linked antennas, which are 2 instances of the same component definition. Var2 affects its box size. Var2 is equal to Var1, Var1 is equal to bbb, and bbb is a design variable.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following figure shows the Far Field patterns of the two instances.



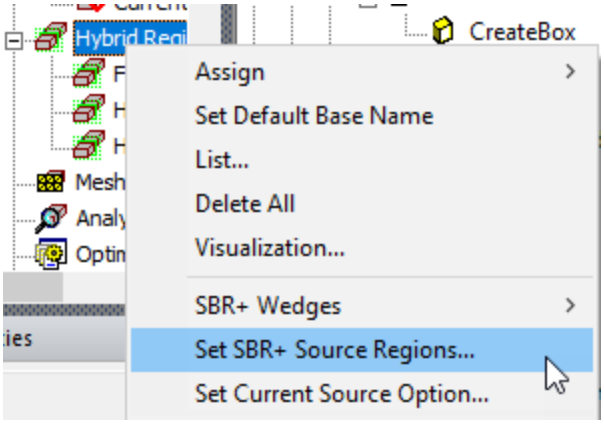
Set SBR+ Source Regions

The **Set SBR+ Source Regions** Command for Hybrid Regions opens a dialog that lets you group source regions for SBR+. Designs containing SBR+ regions will always be set to All One-Way and the **Set Coupling** menu is replaced by the **Set SBR+ Source Regions** command. Non-SBR+ regions will serve as sources, and interact with each other through the SBR+ regions. All source regions interact as one-way links to SBR+ and coupling between the source regions are handled by SBR+ using reciprocity theory.

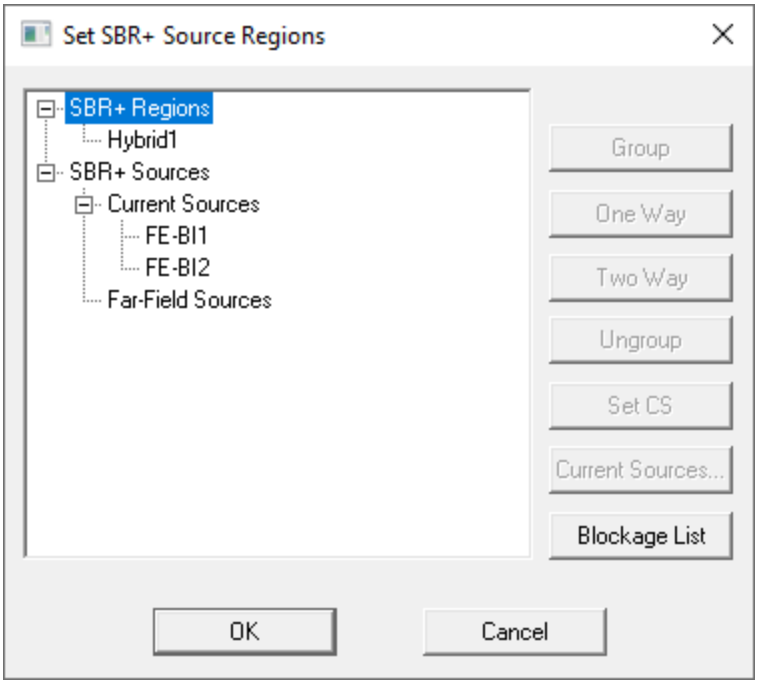
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** The result using far field source option for SBR+ region is sensitive to the phase reference defined when the excitation is close to the SBR+ Region. Phase reference needs to be carefully adjusted to obtain reasonable result.

The **Set SBR+ Source Regions** command appears in the right-click menu of the Hybrid folder at the project tree and at the **HFSS>Hybrid** menu if an SBR+ Region is assigned.

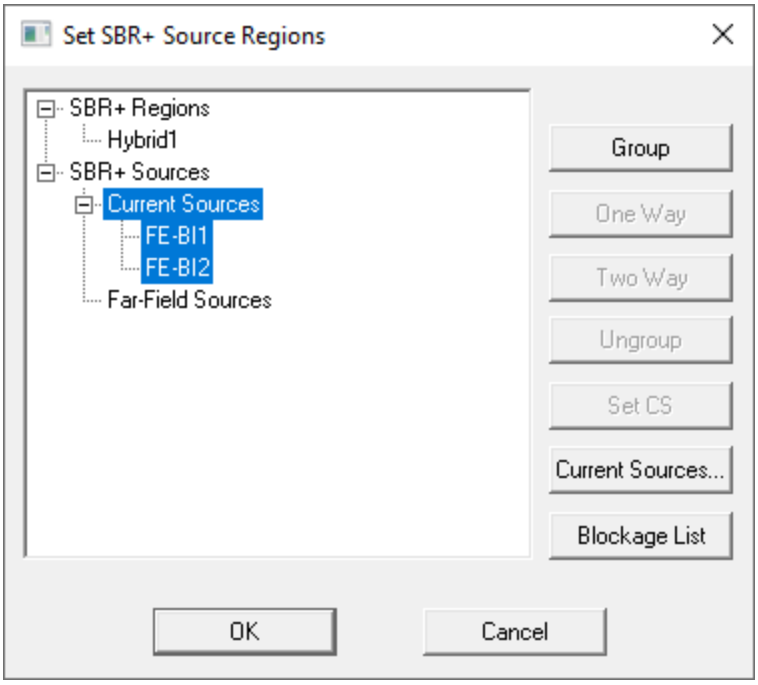


Selecting **Set SBR+ Source Regions** opens a **Set SBR+ Source Regions** dialog that lists the SBR+ Regions and the SBR+ Sources in the design. If the design includes an Incident Wave excitation, you see a message that such designs do not support advanced coupling. However, designs with Current Sources and Far Field Sources will be listed in the dialog box.



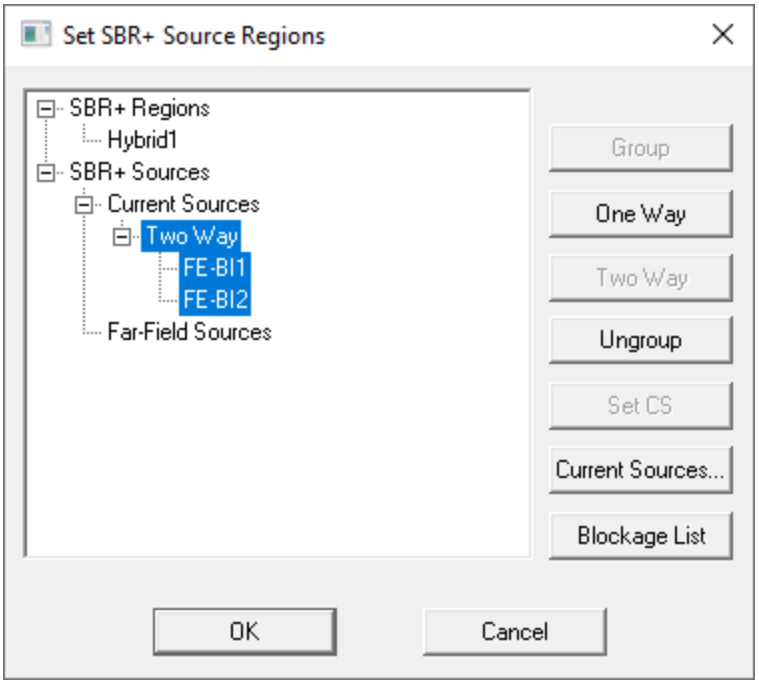
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Any Selected regions in this dialog are highlighted in the view window. You can edit groupings (as shown below) but remember that editing groups invalidates existing solutions.

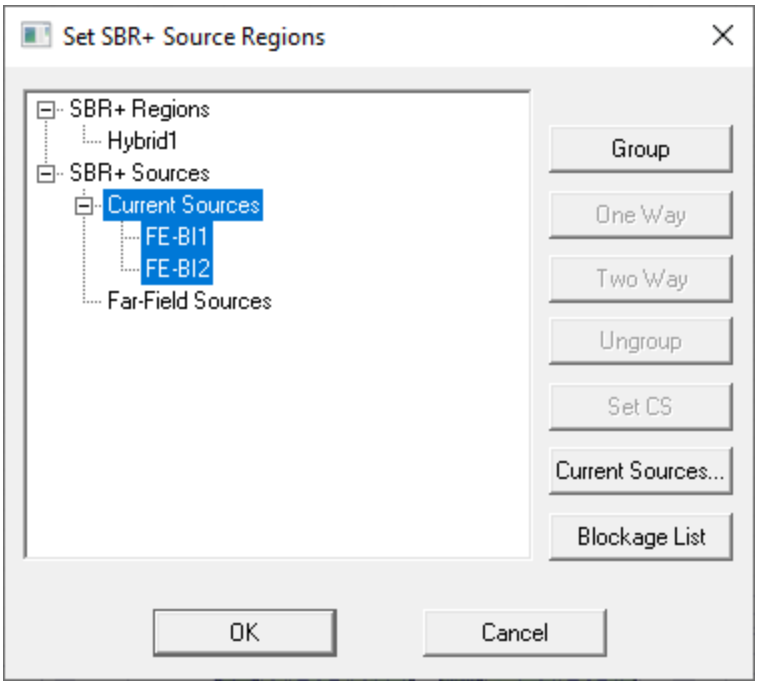


Selecting one or more sources enables the **Group** button provides a means to designate two-way couplings within a group. The regions so grouped are assigned Two Way coupling. The **One Way** button is enabled so that you can change their assignment as One Way couplings. The **Ungroup** button lets you remove coupling groupings.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



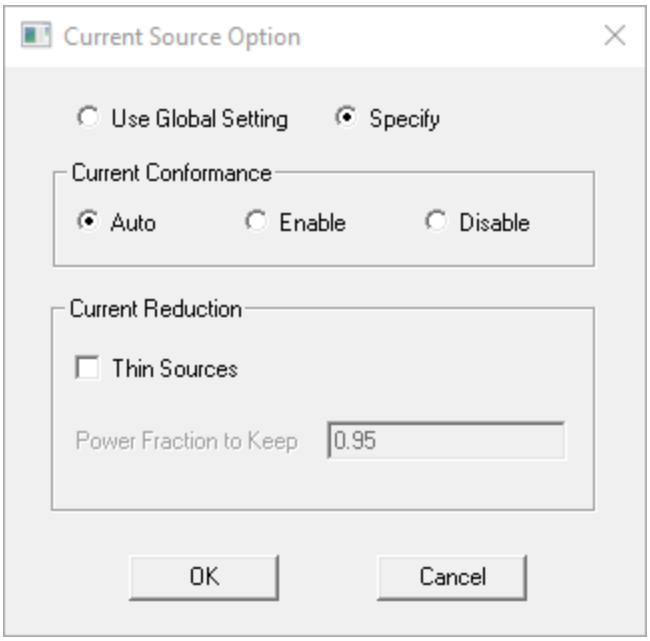
If the design includes Current Sources or Far Fields, these are listed.



The **Current Sources...** button allows you to specify customized power fraction. Use Global is the default. Enabling Thin Sources lets you specify a Power fraction should be greater than 0.1

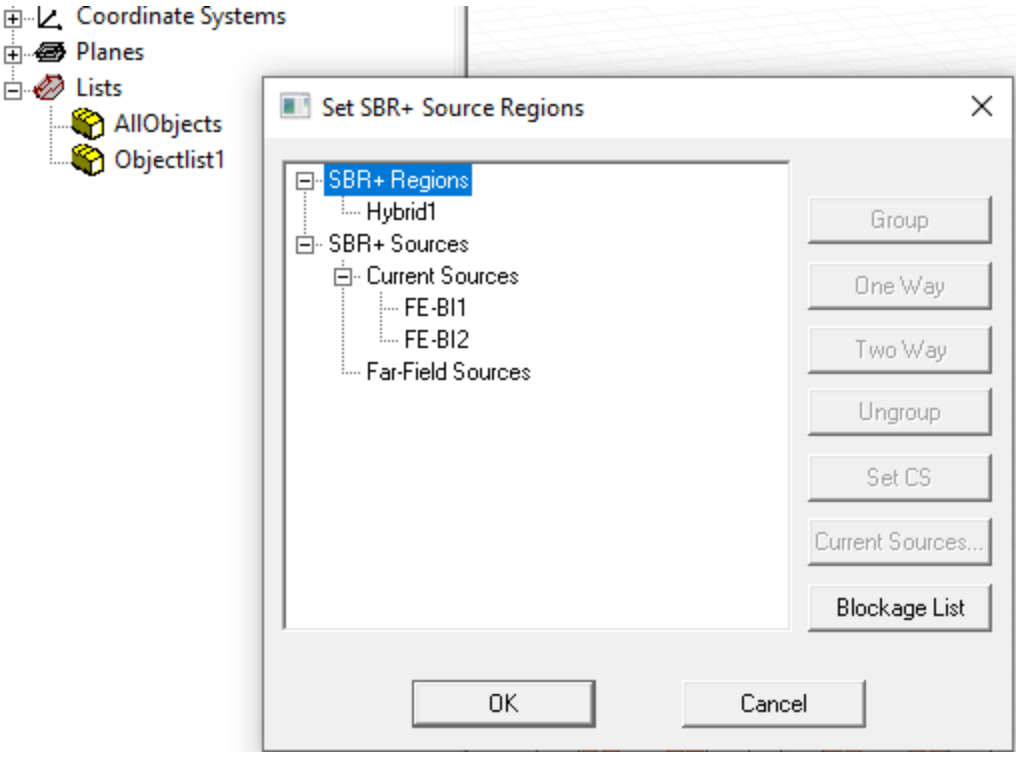
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and less than 1. Solutions are invalidated if you change current source conformance, use Thin Sources, Power Fraction, or change the Global option.



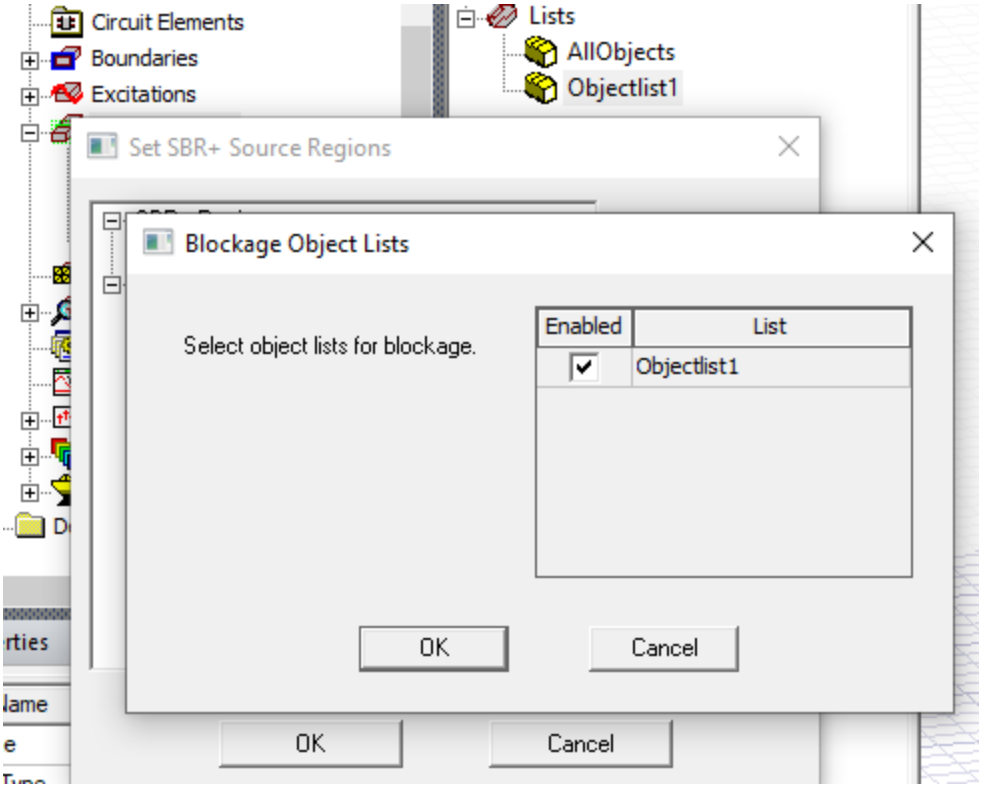
The Blockage List button opens a dialog that lets you view blockage lists. Blockages are allowed when an SBR+ region exist in driven modal and terminal designs. You can easily define an object list for simulation, and automatically assign blockage structures for each antenna.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Click the **Blockage List** button to open a table of available Objectlists. Such lists must contain at least one conducting object and have no SBR+ Regions. Enable those lists to treat the objects as blockages.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

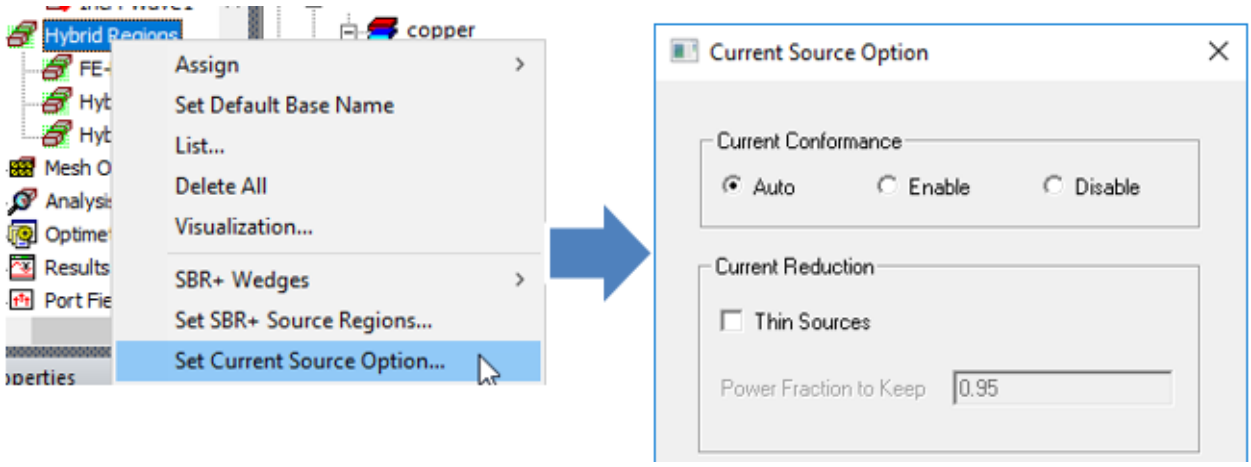


Set Current Source Option

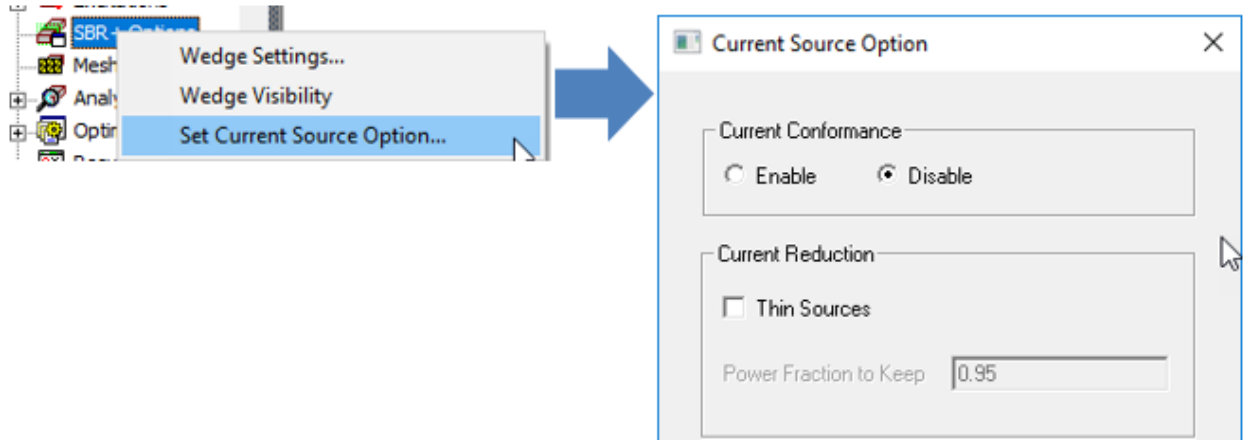
This feature provides a global current source reduction setting, which allows a customized power fraction. You can also override the global setting for each current source in an HFSS Driven design, and for each field linked antenna in an SBR+ design. Use Global is the default. Enabling Thin Sources lets you specify a Power fraction. It should be greater than 0.1 and less than 1. Solutions are invalidated if you change current source conformance, use Thin Sources, Power Fraction, or change the Global option.

For Modal and Terminal Solutions, you can right click on the Hybrid Regions icon in the Project tree and access the **Set Current Source Option...** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

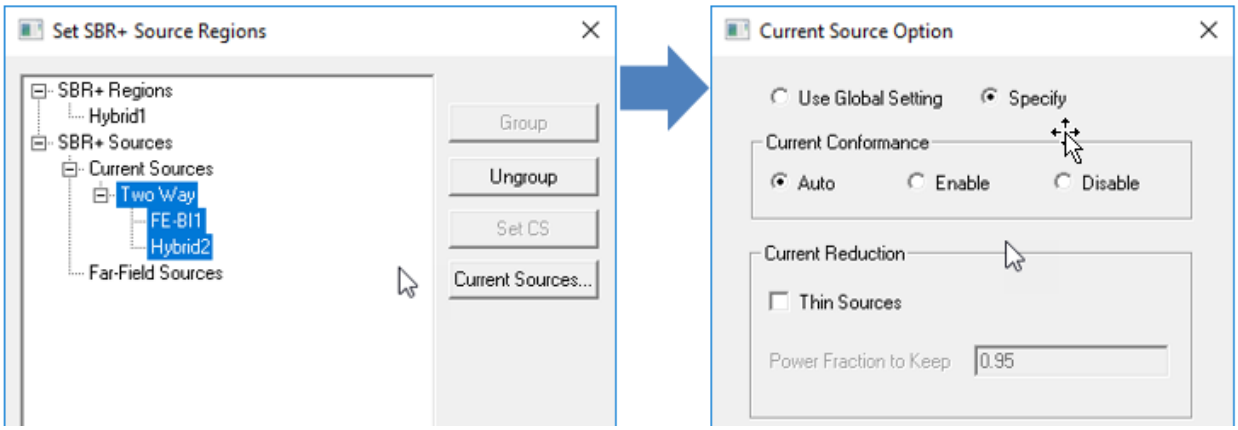


For SBR+ Designs, you can also access **Set Current Source Option** through the SBR+ Options.

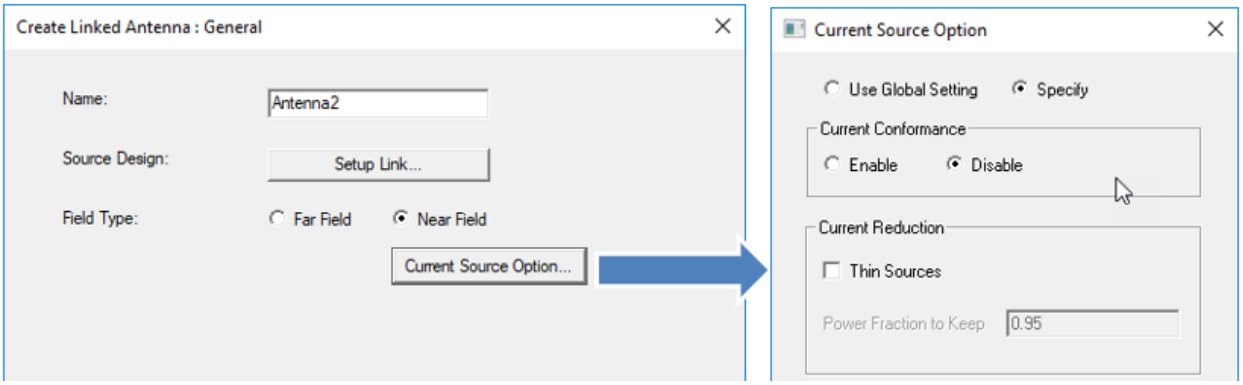


You can set Current Sources when you Set SBR+ Source Regions, but selecting the **Current Sources...** button in the dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also set specific sources when you create a create a linked a Near Field antenna from a driven Modal design to SBR+.

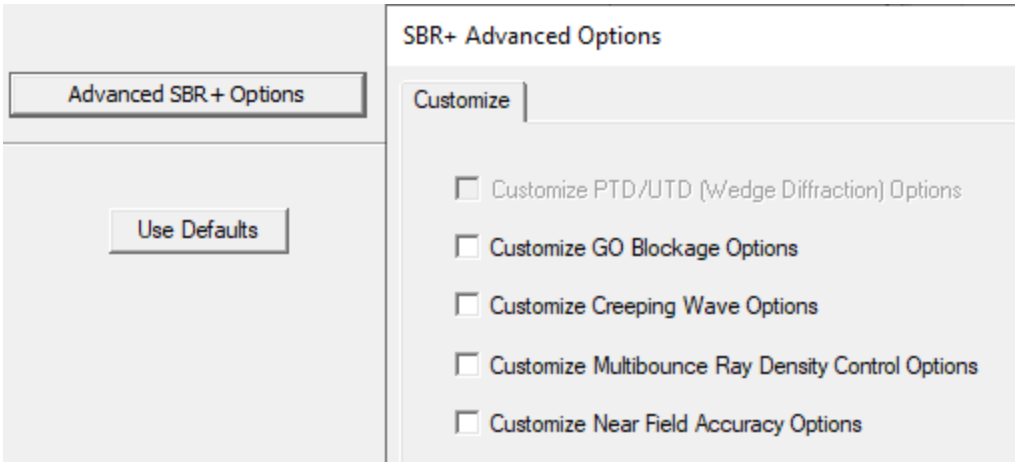


NF Accuracy Settings SBR+ Advanced Option

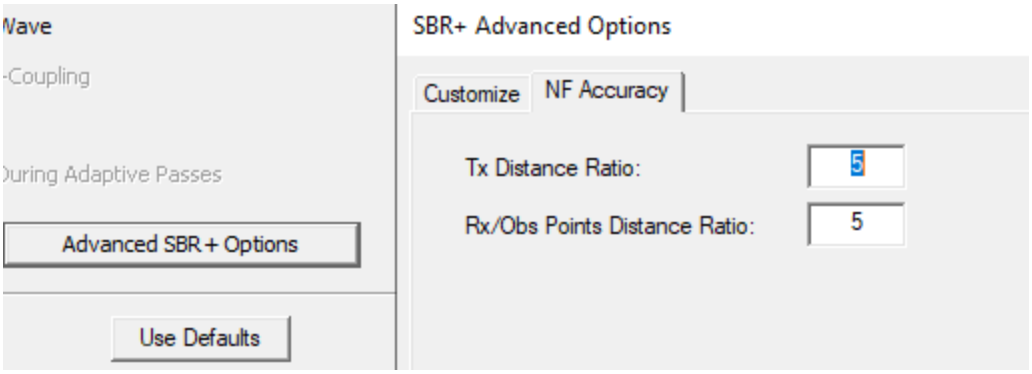
The NF Accuracy settings (NF Accuracy Controls for SBR+) provide access to methodology settings relating to cases where Tx antennas, Rx antennas, and near-field observation points are in proximity to the scattering geometry (i.e., the platform CAD model). When this feature is switched off, HFSS SBR+ uses default near-field accuracy settings that should be adequate for most situations, including close-proximity Tx/observer conditions that this feature is designed to tune.

After you enable the feature, the setting is available through the Solutions Setup for SBR+ solution types, **Options** tab, and HFSS with Hybrid and Arrays Solution Types, Solutions Setup, **Hybrid** tab, if the design contains SBR+ regions. Selecting the **SBR+ Advanced Options** button to display SBR+ Advanced Options.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Select Customize Near Field Accuracy Options to enable the **NF Accuracy** tab.



Tx Distance Ratio:

Fields change rapidly near the Tx antenna. When the Tx antenna is placed near a surface, HFSS SBR+ adaptively increases the ray density above the normal level, but only for the nearby surfaces. This ensures adequate sampling of the Tx radiation field by the rays.

For close-proximity surfaces, HFSS SBR+ further subdivides these sub-surfaces to maintain a minimum ratio between distance-from-Tx-to-surface and maximum-linear-dimension-of-surface. By default, this ratio threshold is set to 5. Normally, this setting is adequate. To increase the over-sampling, increase this ratio setting above 5. To switch off the over-sampling feature, set this ratio very low, say 0.01. Even when set very low, HFSS SBR+ will still shoot rays at a minimum density according to the Ray Density parameter in the Methodology node.

Rx/Obs-Pts Dist Ratio

This setting has the same geometric interpretation as Tx Distance Ratio, but instead controls the over-sampling of projected ray tube footprints for close-proximity observation points and

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

receivers in the scattered field radiation integral. To switch off the over-sampling feature, set Rx/Obs-Pts Dist Ratio to zero. By default, the ratio threshold is set to 5, which is normally adequate, regardless of how close the observation point is to the surface.

(GPU plug-in only): For SBR ray footprints sufficiently near the field observation point or Rx antenna for over-sampling to occur, the scattering computation will not be GPU-accelerated. Instead, it will be performed on the CPU.

This setting has no impact on the computation of far-field patterns.

Select Tx/Rx Antennas for SBR+ Solutions

The SBR+ solution type allows you to create parametric antennas for a streamlined workflow in Ansys Electronics Desktop HFSS, which helps support ADAS and other similar application areas with large scene models. To allow for more efficient simulations, SBR+ Tx/Rx Antenna Selection lets you specify how each antenna is to be used in the simulation, as either Transmit, Receive, Transmit and Receive, or Off (Tx, Rx, Tx/Rx, or Off). This can dramatically reduce the number of antenna-to-antenna coupling pairs that must be simulated for a given scenario using SBR+ so that full NxN coupling analysis is not required. Even though SBR+ can handle these larger scene models more efficiently than some other approaches, if you select the specific Tx-Rx roles for each antenna you can obtain dramatically shorter runtimes because each transmitting antenna requires a separate ray trace. Near field Analysis for SBR+ Solutions is available if you specify Tx or Rx/Tx roles.

> **Note:**
>
> When antenna are assigned NxN full coupling among all sources as in all previous SBR+ region applications, HFSS SBR+ averages the values of cross coupling S matrix terms Smn and Snm to force them to be the same. When all antenna are assigned to TxRx, Smn and Snm might not be both available in the final solution. As a result, when the TxRx feature is assigned, such averaging is performed only when HFSS SBR+ detects that both Smn and Snm are present.

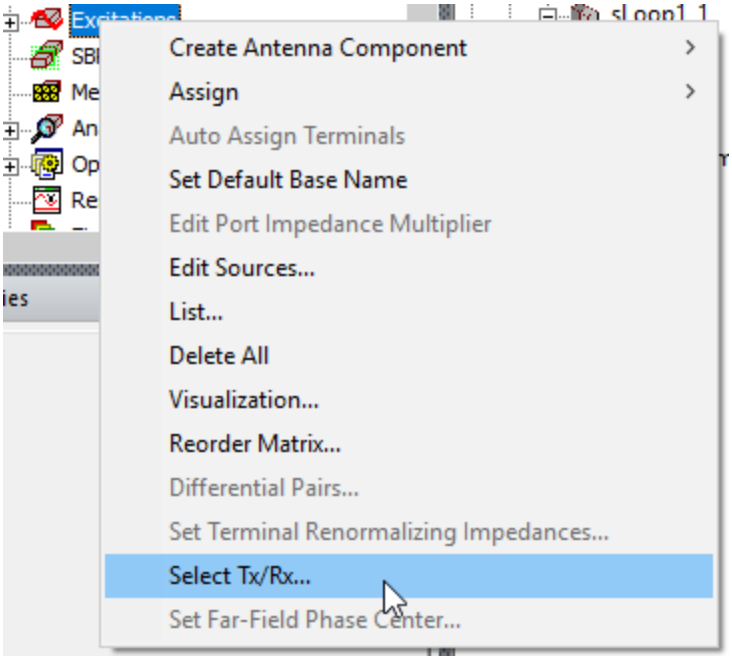
Prerequisites

- SBR+ Solution Type
- Parametric Antenna Components

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

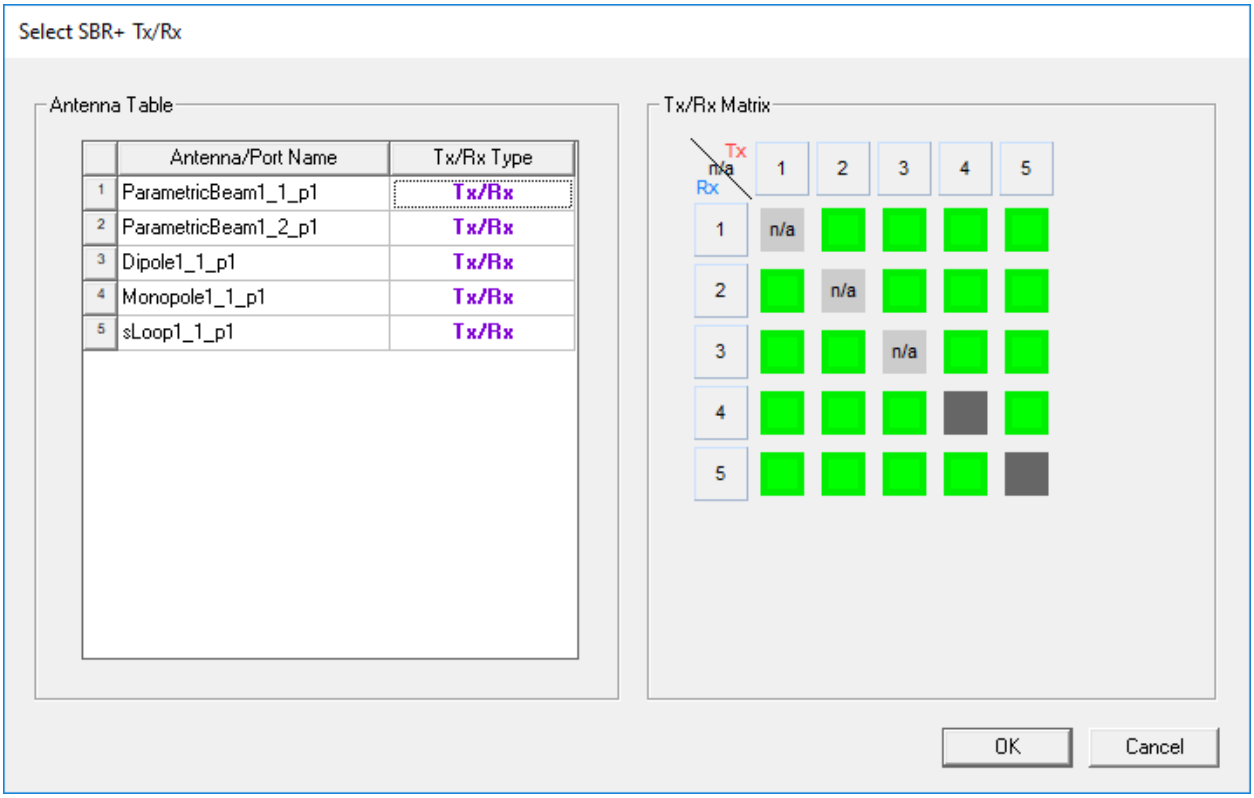
To Assign Transmit and Receive Assignments for Antennas in SBR+

1. Right-click the **Excitations** icon in the Project tree, and click, **Select Tx/Rx...** or click **HFSS>Excitations>Select Tx/Rx...**



This opens the **Select SBR+ Tx/Rx** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The left panel contains the list of all available antennas/ports available and their configured Tx/Rx Types, and the right panel contains the matrix (the S-param matrix) of active/inactive indicators for each transmit/receive (tx/rx) pair configured for the simulation. You can resize the dialog both horizontally and vertically to show more antennas in the list and more cells in the matrix. Scrollbars appear for both the table and the matrix if there are too many antennas/ports to fit in the dialog size.

Default types are set to “Tx/Rx” for all antennas/ports, with all Tx/Rx matrix cells set to active. This corresponds to a full NxN coupling configuration. If you want full NxN coupling for an SBR+ simulation there is no need to open or configure the Tx/Rx Selection settings.

Beta Feature HFSS Interactive Dialogues. If you enable this feature, you can edit the design outside the **Select Tx/Rx** dialog, and the dialog will update. The supported edits include:

Antenna Table Updates:

- New source is created: added to the last row of antenna table. Default Tx/Rx type is “Tx/Rx”.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

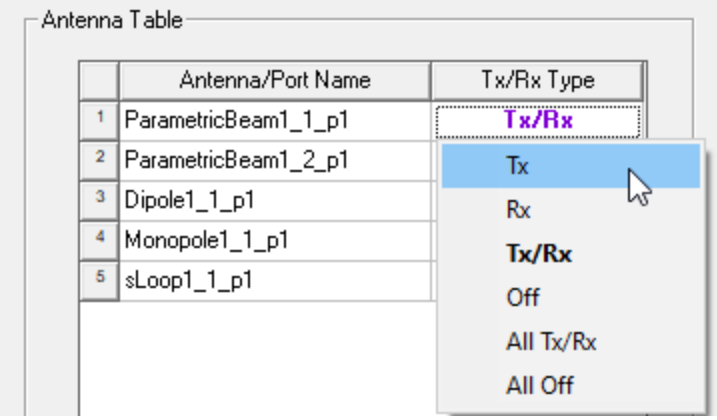
- Sources are deleted: remaining sources retain the original order on the table, with deleted sources removed.
- Source name changes: corresponding source name on the table updates to the new name.

Tx/Rx Matrix Update

- New source is created: a new row and column will be added to the matrix. Depending on the number of Tx and Rx channels in the matrix, the number of buttons added to the matrix will adjust itself to form the new matrix. Default button state is on, except the buttons that have the same source of Tx and Rx.
- Sources are deleted: corresponding Tx and Rx of those sources in the matrix are deleted.
- Source Tx/Rx type changes: row and column visibility updates based on the change of Tx/Rx matrix.

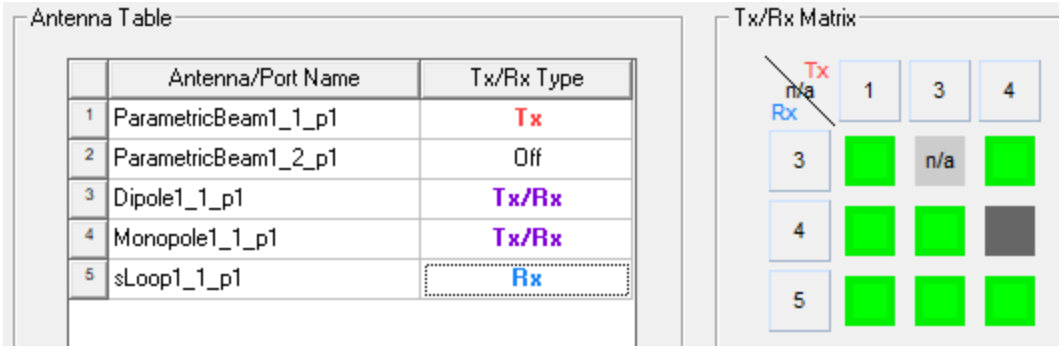
Note: The dialog supports undo and redo for creating, deleting and renaming elements. After undo/redo, added elements will recover to default Tx/Rx type and button state.

2. For the Antennas table, you can assign each antenna/port a type of **Tx**, **Rx**, **Tx/Rx**, or **Off**. This is done via a drop-down menu selection on the table cells under the **Tx/Rx Type** column. For convenience, the selection list includes **All Tx/Rx** and **All Off** selections.



The text color changes with the Tx/Rx Type selection as follows: Tx = red, Rx = blue, Tx/Rx = purple, Off = black.

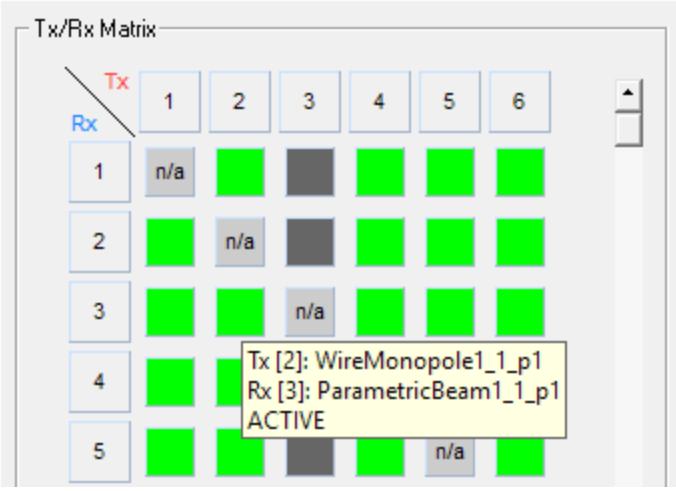
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Tx/Rx Type** also determines which rows and columns are displayed in the **Tx/Rx Matrix** panel on the right. Transmit antennas appear as columns across the top (notice the red Tx label), and receive antennas appear as rows down the left side (notice the blue Rx label), just as shown in the S-parameters matrix (available under Results>Matrix Data). If you set the type to **Tx**, an antenna/port will only show its **Tx** column in the matrix, and the **Rx** row has all its cells set to inactive and is not shown. Similarly if you set the type to **Rx**, the **Tx** column for that antenna/port in the matrix has all its cells set to inactive and hidden.

Antenna rows/columns are numbered to match those used in the table on the left. The order of both corresponds to the order of the antennas/ports in the output data, and may be different than the alphabetical order shown in the project tree under the **Excitations** folder.

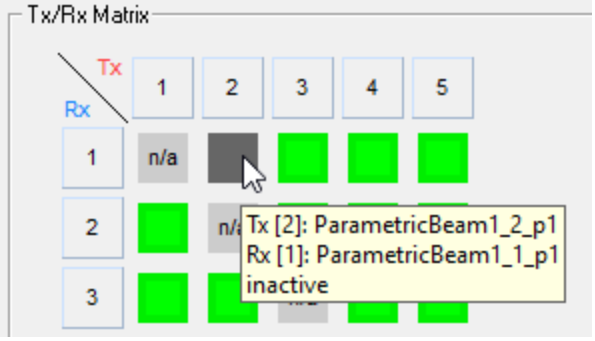
In the **Tx/Rx** matrix, tooltips are shown when the mouse is hovered over any cell in the matrix to show the corresponding antenna/port names and active/inactive status.



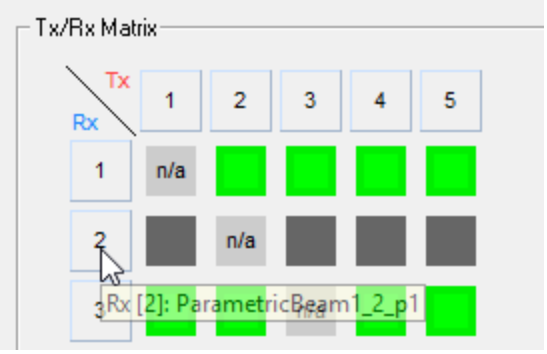
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A green indicator means this Tx/Rx combination is active and the SBR+ simulation compute results for the interactions between this antenna port pair; dark grey means it is inactive for the simulation. Any self-coupling cells are marked with "n/a" as not applicable for the SBR+ solution type and may not be modified, which includes the diagonal matrix cells as well as any self-coupling cells for multi-port antennas.

3. To toggle a Tx/Rx combination as active/inactive simply click on the corresponding cell in the matrix. The color will toggle between green (ACTIVE) and dark grey (inactive).



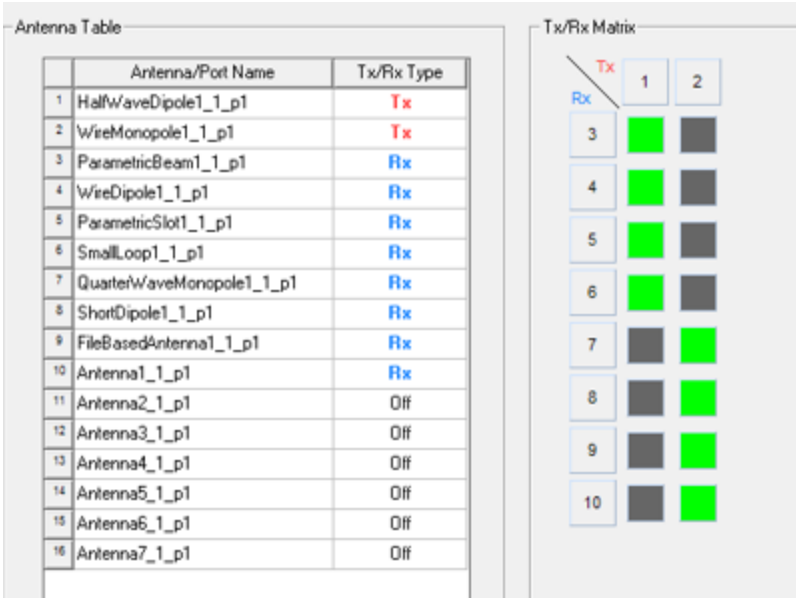
4. To toggle an entire Tx column or Rx row, click one of the numbered header cells. Clicking a Tx column has the effect of turning this Tx either on or off to all receivers, and likewise, clicking an Rx row has the effect of turning this Rx on or off to all transmitters.



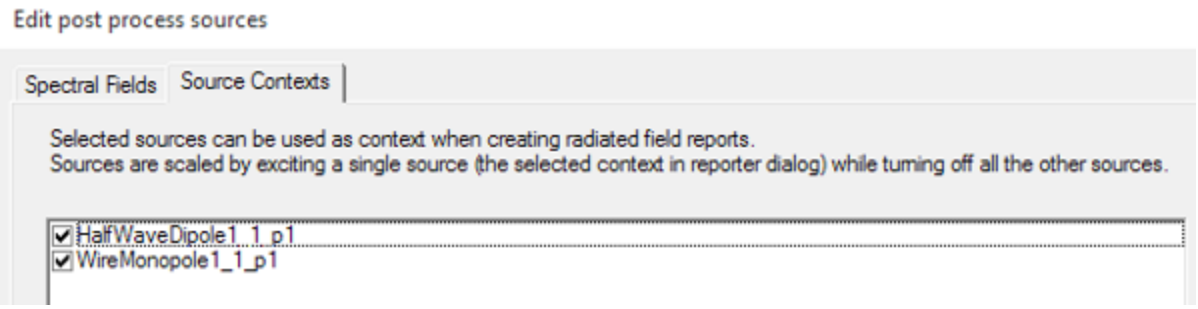
> **Note:**
>
> Turning an entire row or column off (making it inactive) achieves a similar end to specifying an antenna as either **Tx** only or **Rx** only.

The **Tx/Rx** roles you select in the SBR+ **Tx/Rx Selection** dialog can impact the post-processing sources available for the simulation. For example, suppose you configure an antenna/port to be **Rx** or **Off**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

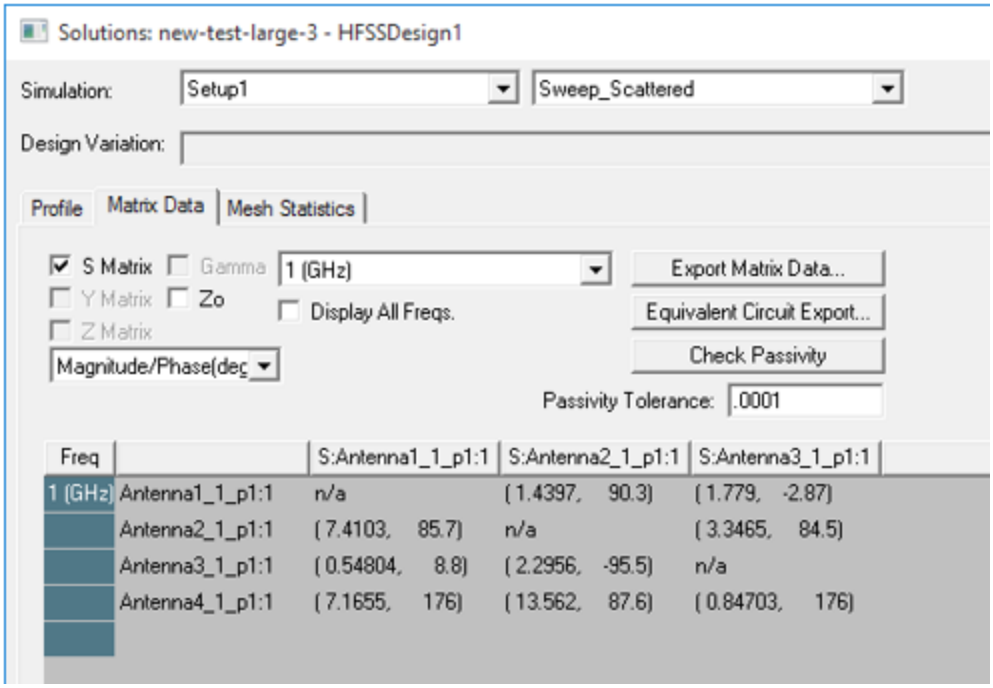


Then that item will not be available as a post-processing source context under Edit Sources. Only **Tx** or **Tx/Rx** antennas/ports will appear on the **Source Contexts** tab, since those are the only antennas/ports configured to produce radiating fields (ie, transmitting).

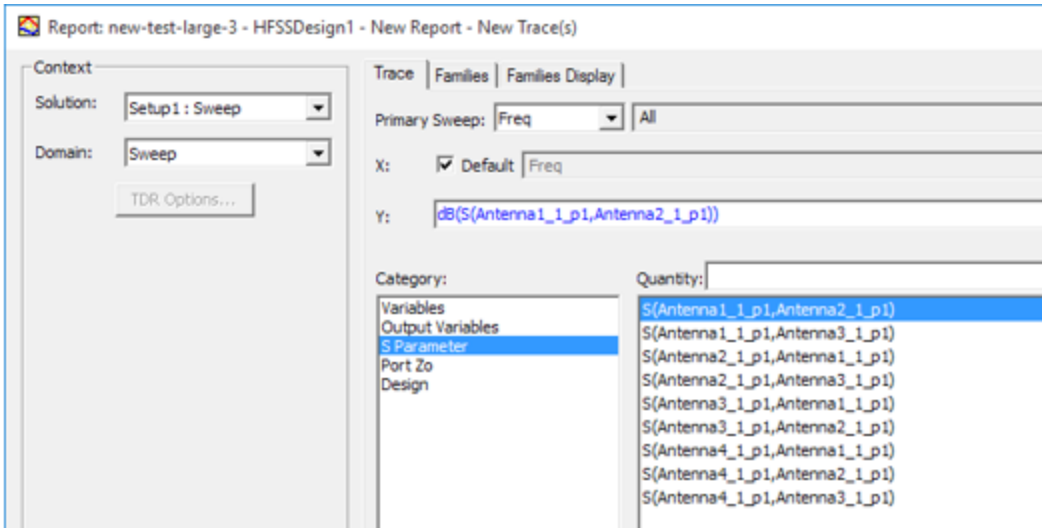


5. Upon completion of the simulation, the results in the S-parameter matrix data display will correspond directly to the Tx/Rx Matrix configuration, with simulation results only computed for ACTIVE (green) cells. All other inactive and "n/a" cells will show "n/a" results for those same entries in the S-parameter matrix since no SBR+ simulation data was computed for those coupling pairs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



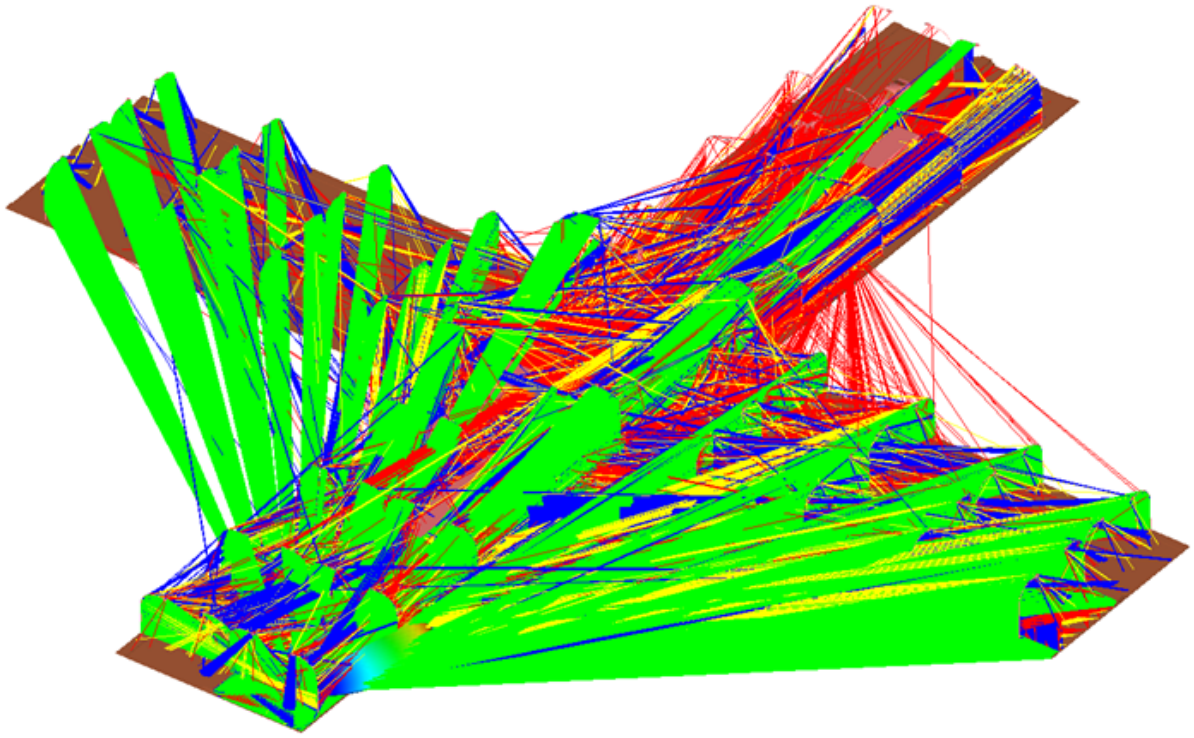
6. In the **Create Report** dialog box, you see only the active Tx/Rx coupling pairs listed as the available report quantities, not the full list of NxN coupling matrix pairs (unless configured as full NxN coupling).



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Visual Ray Trace (VRT) for SBR+ Simulation

Visual Ray Trace (VRT) is a powerful feature available in HFSS for visualizing the ray geometry and interactions for an SBR+ simulation. Since SBR+ relies on “painting” physical optics (PO) currents on the geometry, VRT can provide valuable insight for analyzing these PO contributions from the direct illumination first-bounce rays and the multi-bounce rays we propagate using geometric optics (GO). The “hit point” locations for each ray bounce may be visualized, the ray paths between each bounce may be visualized, and the “footprints” that are used to integrate the PO currents at each bounce may be visualized. The VRT feature includes a number of other rendering controls and options to filter whole ray tracks or individual ray bounces, visualize ray scattering directions, and color ray point/lines/footprints according to different criteria. The VRT feature also lets you use "SBR+ with PTD and UTD" on page 13-269 to account for additional phenomenology. You can now create SBR VRT plots with blockages, as described in "Create VRT (Visual Ray Trace) Plot with Geometry Blockage" on page 13-256



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Prerequisites

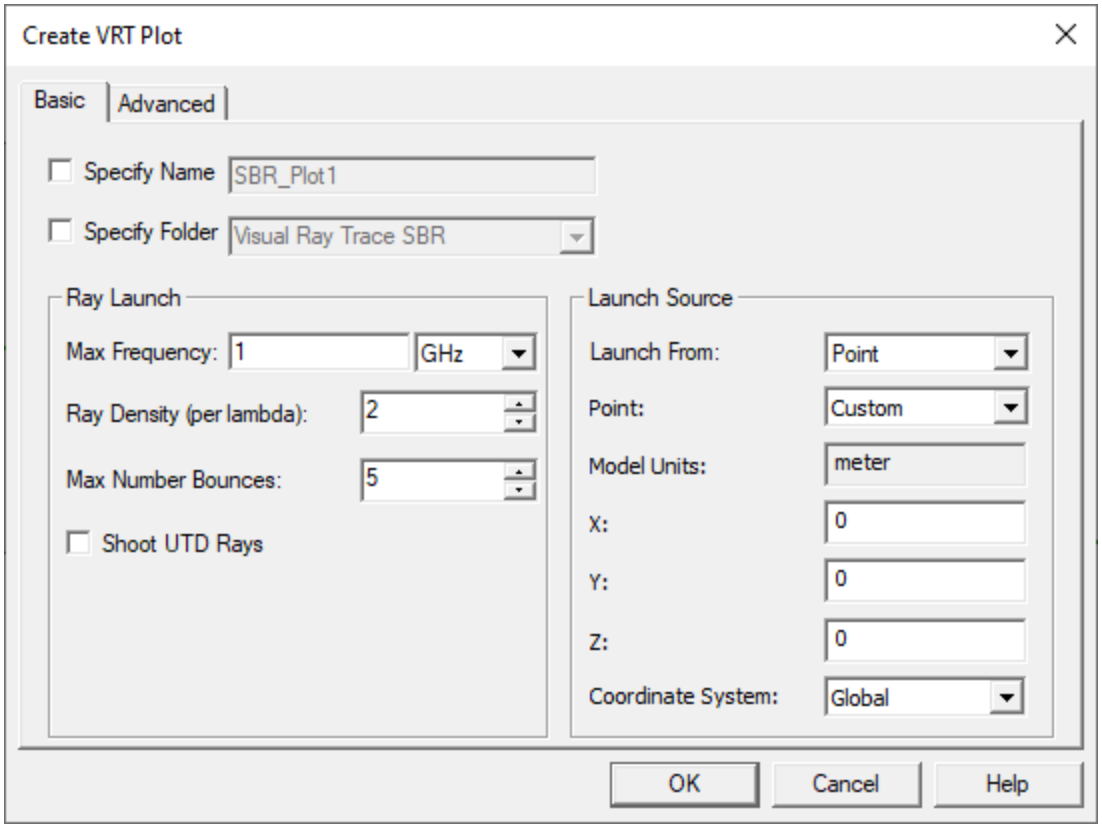
To use the VRT feature, you must have an HFSS design with either the SBR+ solution type or with the HFSS with Hybrid and Arrays solution type and one or more SBR+ hybrid regions. Various boundary conditions (BCs) and material properties may be assigned to the SBR+ geometry, as detailed in Geometry, Boundaries and Dielectric Region Support in SBR+. Use of any unsupported boundaries or material configurations will produce an error if attempted with VRT. Note that for all Layered Impedance boundaries assigned to an SBR+ region geometry the surface roughness must be set to “0”.

Using Visual Ray Trace (VRT) in Ansys Electronics Desktop

1. Once you have an appropriate HFSS design, there are three different ways in the user interface to create a new Visual Ray Trace Plot:

- Right-click the **Field Overlays** icon in the **Project Tree** and select **Plot VRT...** from the short cut menu.
- Right-click the 3D Modeler window and select **Plot VRT...** from the pop-up menu.
- Click **HFSS>Fields>Plot VRT...**

After you use one of these approaches, the **Create VRT Plot** dialog displays:

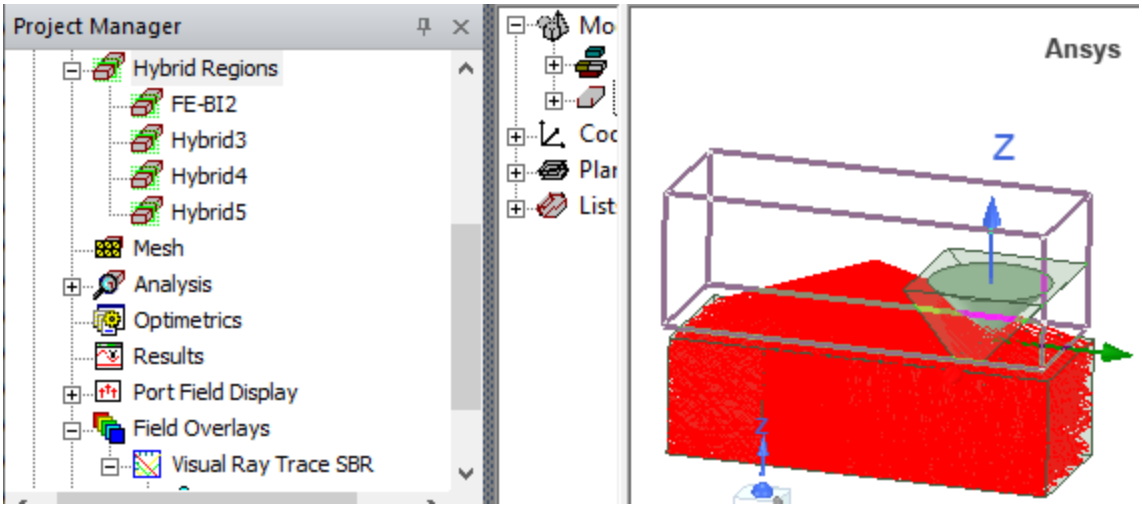


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To **Specify Name** check the box to enable the text field. To **Specify Folder**, check the box to enable the selection menu.

2. Set the Ray Launch Parameters

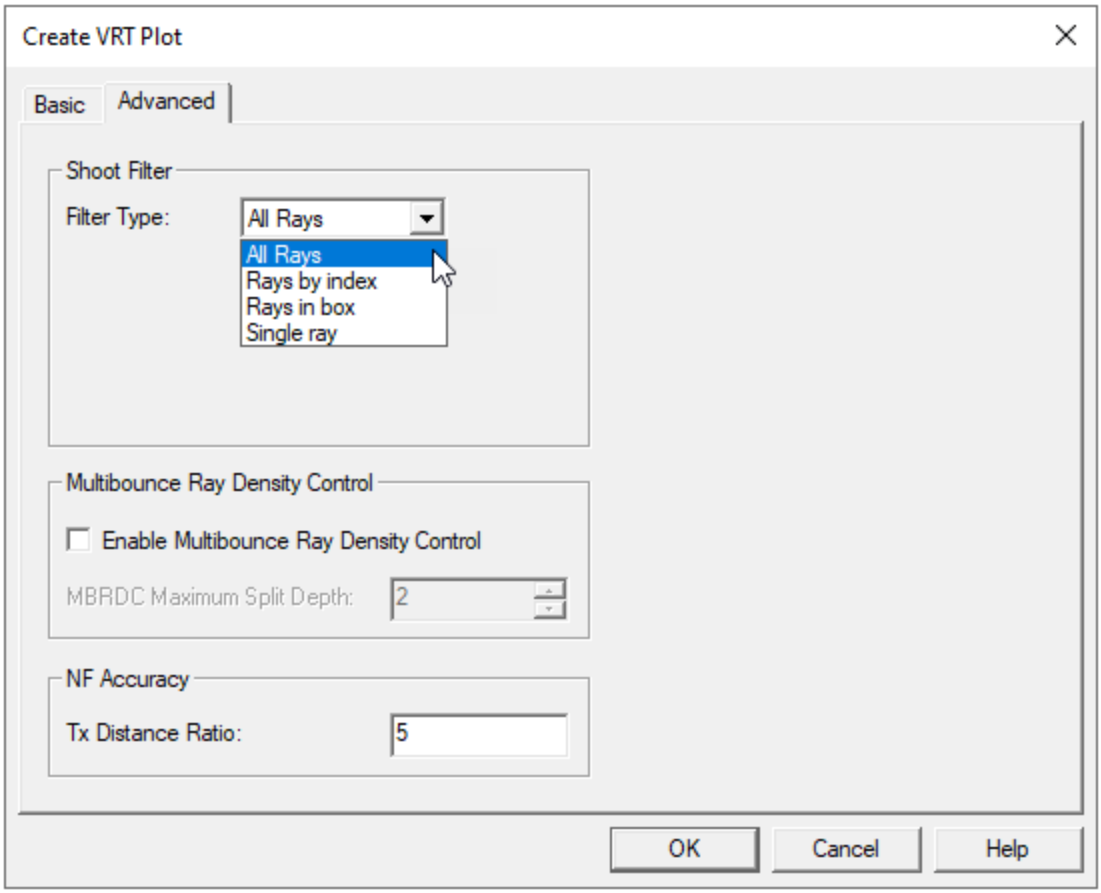
- VRT plots do not have to be configured the same as your HFSS design/analysis setup. You can use VRT to experiment and plot rays with different types/locations of ray sources independent of your full HFSS simulation run.
- **Max Frequency** and **Ray Density** parameters control the density of rays that will be launched from the source for the Visual Ray Trace.
- **Launch From** parameters configure the source ray launch setup for the VRT rays. You can use a custom specified XYZ coordinate, an existing Point object, a Plane-Wave incident direction specified in Theta/Phi, or from an Antenna 3D Component. More detailed information/instructions about using Antennas with VRT can be found under Parametric Antennas and Parametric Array Antennas for VRT Plots. By default, the **Launch From** selection is Point and launched from the global coordinate system origin. **Note**: In the case of a FE-BI Hybrid Region, by design the VRT rays launch from the center of the FE-BI, rather than a port location in the FE-BI portion of the design. For example, in the following image, the highlighted box is assigned as a FE-BI Hybrid Region, and the center of the box becomes the VRT launch point rather than the antenna within the Hybrid region.



- **Shoot UTD Rays** controls whether the plot generates additional UTD rays that represent diffraction within the Keller cone off the edge to generate rays in the shadow region. If you modify the SBR+ initial mesh and/or wedge settings, the VRT plot with UTDs rays may need to be invalidated since it depends on both. See the discussion of SBR+ Wedge settings.

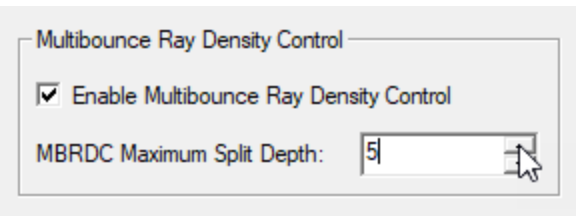
3. Optionally, select the **Advanced** tab. Here you can specify a Filter for “Shoot Filter” to limit the number of rays generated. You can select from these to diagnose model issues or to explore sub-regions of a very large or dense mesh:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- All Rays (normal, default behavior)
- Rays by index: ray indices may be used to limit the rays to a specified range/interval by Start/Stop, or can skip rays by changing the Step value from 1
- Rays in box: a Box (model or non-model) can be created or selected from the model in the area near the VRT launch point. The Box filters generation of rays whose first bounce is in the area of the SBR+ region geometry defined by the Box extents. A valid Box can be defined using either a Global/Relative coordinate system, but it cannot have additional geometry operations (Rotate, Move).
- Single Ray: specify a theta/phi angle relative to the launch point for a single ray to be generated

4. The **Advanced** tab also allows you to **Enable Multibounce Ray Density Control**.



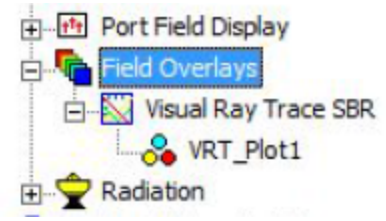
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use the checkbox to **Enable Multibounce Ray Density Control** and enter the maximum number of subdivisions to be used in the plot. Enabling the feature may increase the solve time. The **MBRDC Maximum Split Depth** field value increases the number of spawned ray tracks on each ray bounce. The MBRDC algorithm is designed to achieve the ray density per wavelength criterion at any bounce depth. If a footprint at depth *N* in a ray track does not satisfy the ray-density criterion, the MBRDC algorithm attempts a refined ray shoot from an earlier bounce, just for the ray in question. The size of the triggering footprint informs the level of refinement. If the refined shoot is not successful in achieving the desired footprint size, or new footprints are too large, the MBRDC algorithm attempts further, recursive refinements. The **MBRDC Maximum Split Depth** setting limits how aggressively the refinement algorithm is applied by specifying the maximum number of split recursions. For this reason, linear increases in **MDRC Maximum Split Depth** can yield a geometric progression in the total number of rays shot and associated solution time.

When both MBRDC and UTD are enabled, first-bounce UTD rays and bright points on wedges are recomputed during the MBRDC splitting process, but only the original raytrack is rendered up to the 1st bounce. Similarly, length-based ray filters are applied to the original UTD initial ray track up to 1st bounce rather than using the information in the additional MBRDC tracks. This is a known limitation.

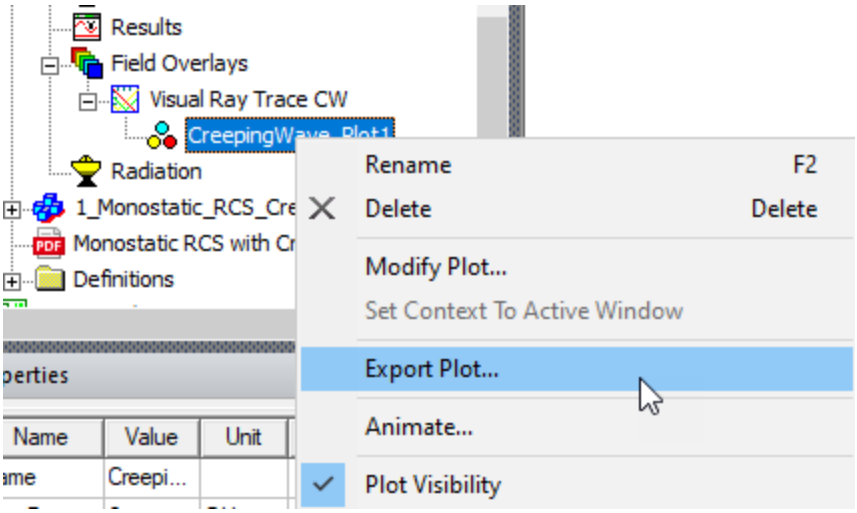
5. The **Advanced** tab allows you to specify a Tx Distance Ratio to oversample footprints near the ray source. Fields change rapidly near the Tx antenna. When you place a Tx antenna near a surface, HFSS SBR+ adaptively increases the ray density above the normal level, but only for the nearby surfaces. This ensures adequate sampling of the Tx radiation field by the rays. For close-proximity surfaces, HFSS SBR+ further subdivides these sub-surfaces to maintain a minimum ratio between distance-from-Tx-to-surface and maximum-linear-dimension-of-surface. By default, this ratio threshold is set to 5. Normally, this setting is adequate. To increase the over-sampling, increase this ratio setting above 5. To switch off the over-sampling feature, set this ratio very low, say 0.01. Even when set very low, HFSS SBR+ will still shoot rays at a minimum density according to the Ray Density parameter
6. Click **Done** to dismiss the dialog to run the VRT generation to produce SBR+ ray data. Only the geometries contained in the SBR+ regions are used for the VRT ray generation.

A new “Visual Ray Trace SBR” plot folder is created under Field Overlays, which contains one or more “VRT_Plot” items.

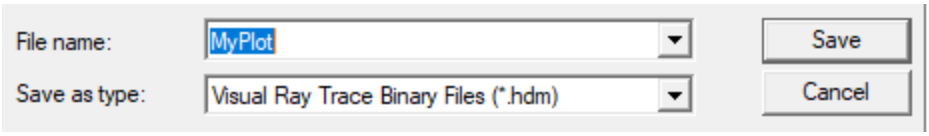


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can right-click on the VRT plot to display the **Export Plot...** command.



This allows you to export Visual Ray Trace Binary Files using a .hdm suffix.



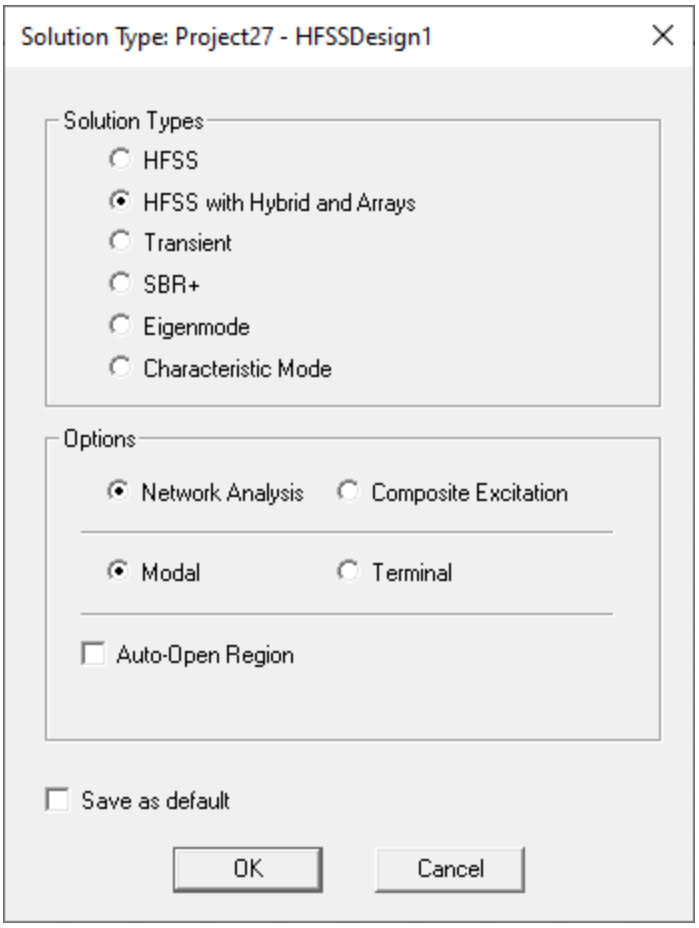
If you run **Plot VRT...** again, that creates an additional VRT plot item under the “Visual Ray Trace SBR” folder, or a different folder name if you specify one in the **Create VRT Plot** dialog.

Create VRT (Visual Ray Trace) Plot with Geometry Blockage

Ppreviously Visual Ray Trace (VRT) plots did not support use of "Model Blockage” geometry - that geometry is not considered as part of the VRT geometry since it is not part of the SBR+ region. We added VRT support for this antenna blockage geometry in the same way that it is now included for the SBR+ simulation. Any geometry objects in the SBR+ region identified as “Model Blockage” will be ignored on the first bounce and included on any multi-bounce for VRT when the associated source antenna is used as the VRT source.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

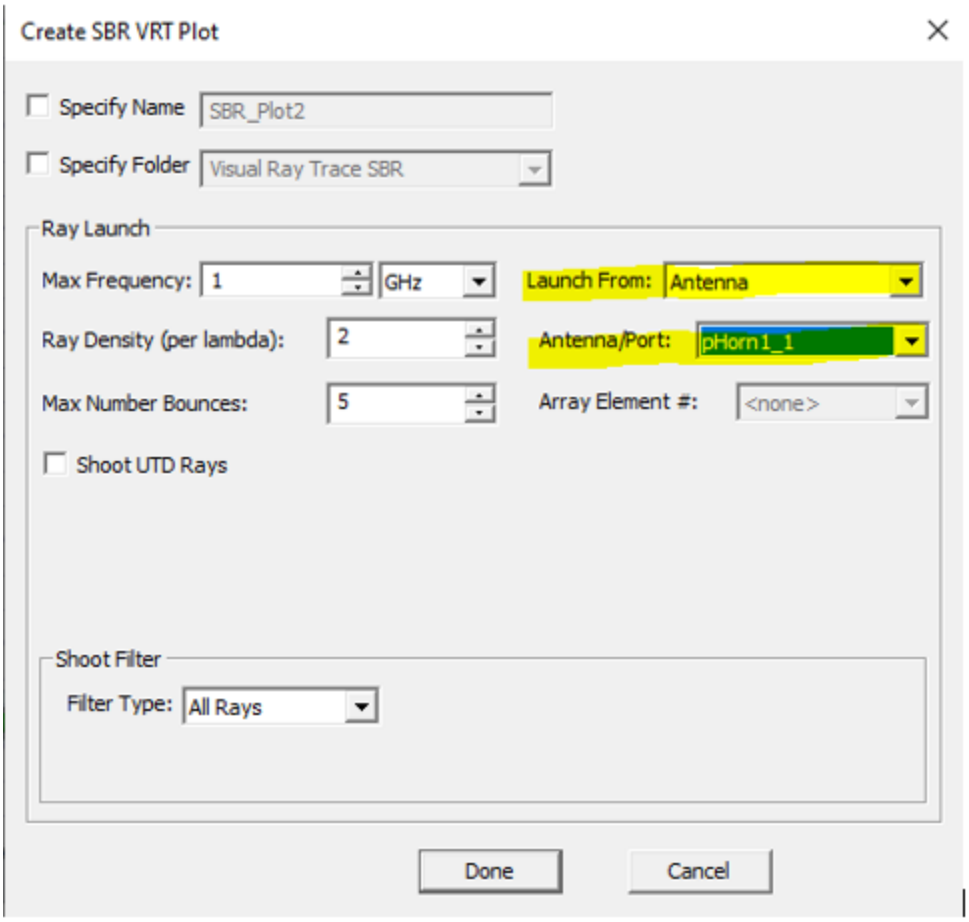
1. Create/open an design with HFSS with Hybrid and Arrays solution type.



2. Configure an HFSS design with SBR+ antenna blockage for either SBR+ or HFSS with Hybrid and Arrays solution type.
3. To create a new VRT plot select the menu option **Field Overlays > Plot VRT> SBR** to open the **Create SBR VRT Plot** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Change the Launch From: setting to “Antenna” and select the desired Antenna/Port from the available hybrid Antenna/Port source/excitation regions.

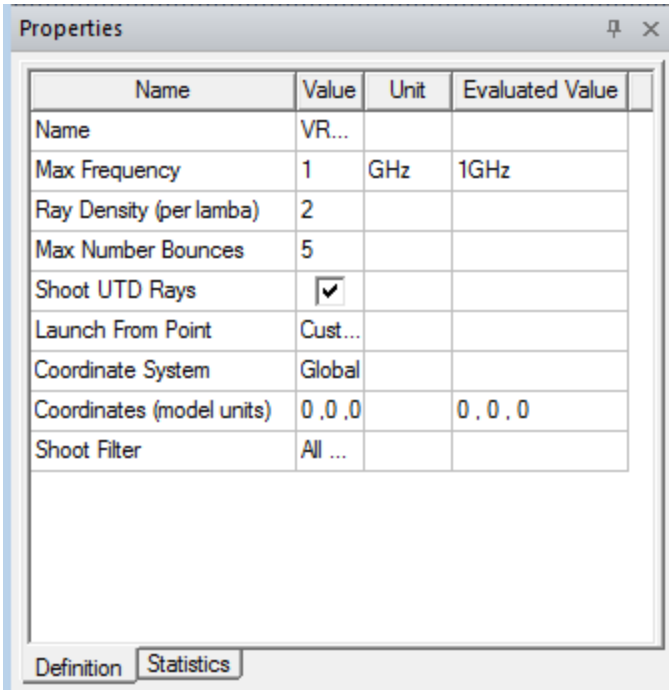


5. Click **Done** to create the plot with SBR rays, which will include the configured behavior of any SBR+ antenna blockage geometry.

VRT Plot Properties

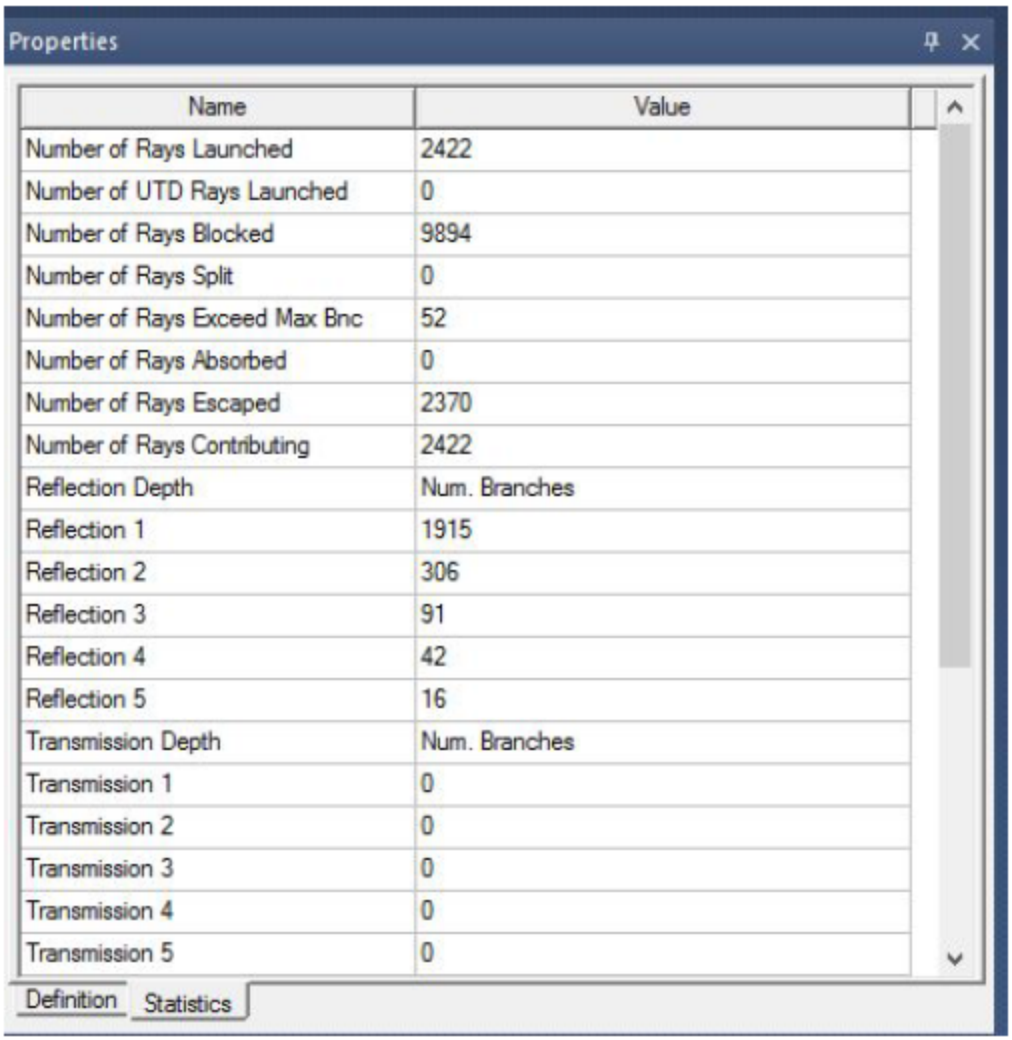
The Properties of each VRT_Plot*n* item are the same as those created in the **Create VRT Plot** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Each Visual Ray Trace SBR plot has an additional read-only tabbed property page that displays Ray Statistics output from the ray generation process.

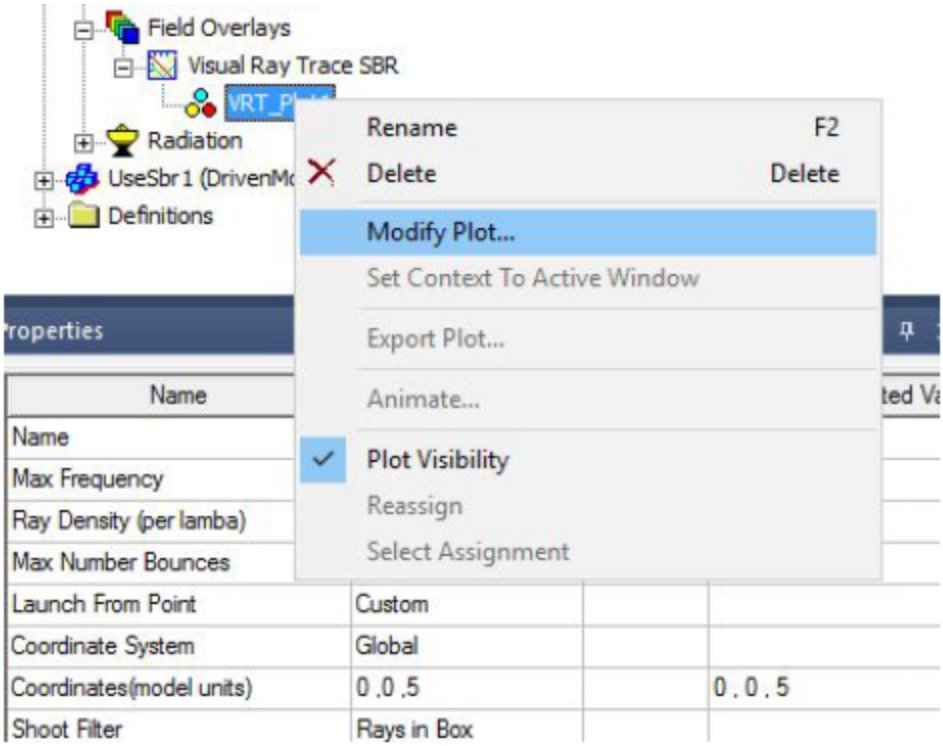
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Modifying VRT Plots

You can modify VRT Plot node properties directly in the **Properties** window to cause immediate updates to the plot data. You can also right-click the plot item and select **Modify Plot…** to again bring up the **Create VRT Plot** dialog to make multiple changes and then commit them to the same, or a differently named plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



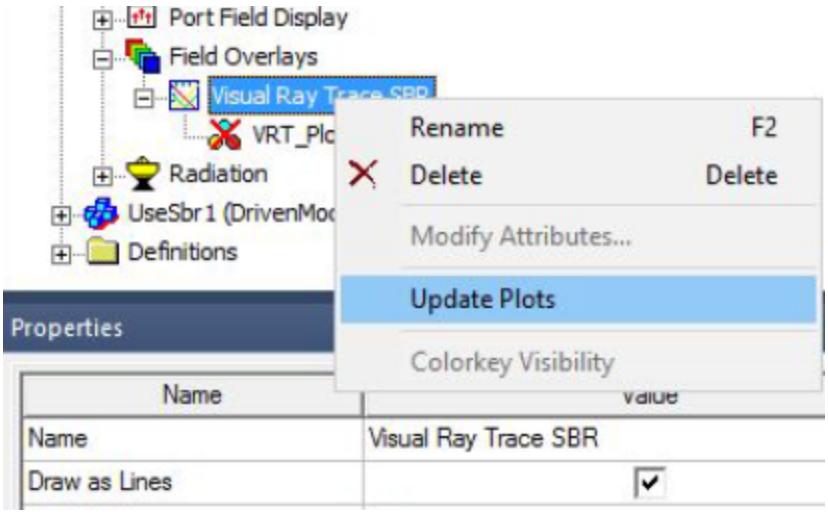
Design Edits that Invalidate the Mesh Invalidate VRT Plots

Any design edits that invalidate the mesh (geometry delete, modify, or change in boundary condition assignment) also invalidates VRT plots. HFSS does not automatically regenerate the VRT plot after a design edit. You can update plots update by right-clicking on the **Visual Ray Trace SBR** folder and selecting **Update Plots** or by double-clicking on the VRT plot item in the Project Manager, which will either be invalidated with a red ‘X’ or greyed out if it needs manual update.

Modification in SBR+ Wedge Setting parameters invalidates the VRT plot when “Shoot UTD Rays” is turned on.

In addition for UTD rays, a modification to the geometry and/or boundary conditions should automatically cause new SBR+ wedge to be generated whenever the VRT plot is manually updated.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Visual Ray Trace SBR Folder Properties

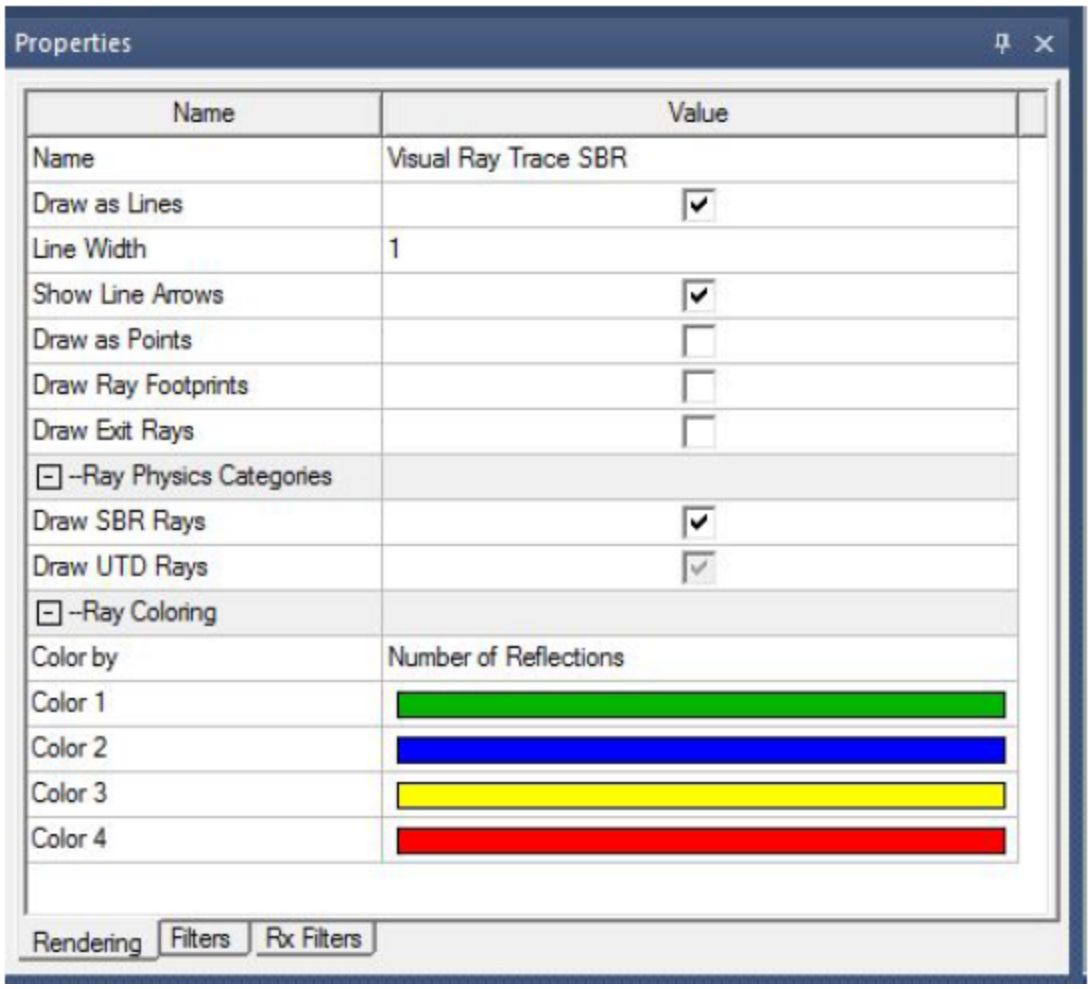
All the render and filter parameters appear as Properties of the parent **Visual Ray Trace SBR** folder. Each “Visual Ray Trace SBR” plot folder provides a grouping of render/filter attributes. Much like setting specific plot attributes like 2D plot extents, scaling, colormaps, etc., the render and filter parameters for all VRT plots are performed at this folder level. The different categories of render and filter operations are kept as separate tabs for: Rendering, Filters, and Rx Filters.

All render and filter property changes are immediate and will automatically refresh the 3D modeler window

Rendering Properties Tab

The **Rendering** tab properties control which items are drawn for each ray track:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Draw as Lines: show the “rays” connecting the launch point, each intersection point, and the exit point.
- Draw as Points: show the point for each ray bounce where we intersect the geometry.
- Draw Ray Footprints: show the ray tube projection for each ray bounce.
- Draw Exit Rays: show the final ray of a ray track if/when it exits the geometry and is scattered away (note: ray tracks that reach “Max Bounce” may not have an exit ray).
- Draw SBR Rays: toggles SBR rays on and off.
- Dray UTD Rays: toggles UTD rays on and off.

Ray Coloring controls:

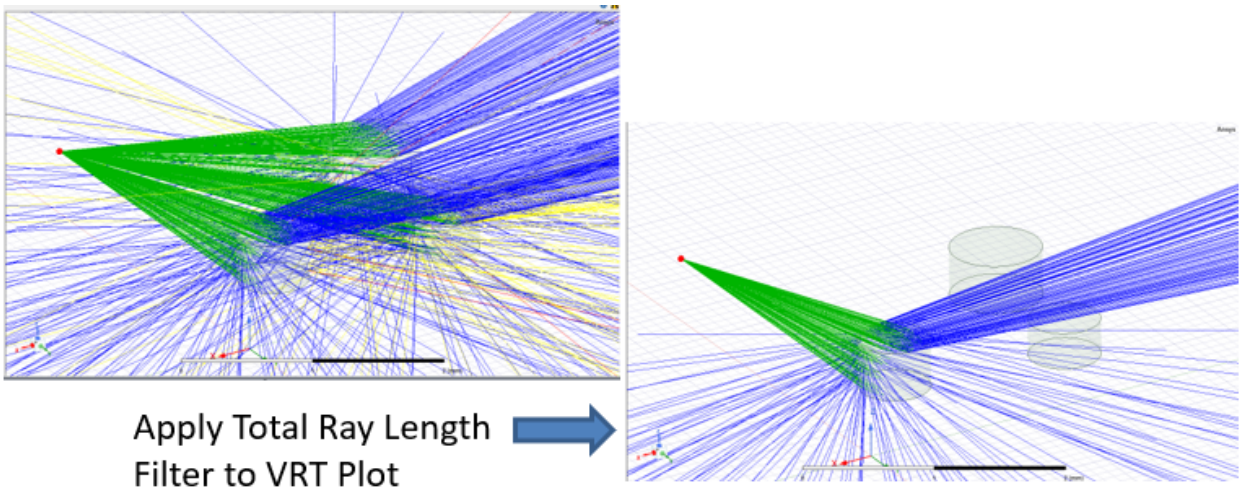
- Color by: Number of Reflections: the full ray track branch (from creation to exit) is colored according to the number of reflection bounces occurring for that ray track segment.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Color by: Number of Transmissions: the full ray track branch (from creation to exit) is colored according to the number of transmission bounces occurring for that ray track segment.
- Color by: Bounce Number: each ray bounce of the ray track branch is colored a different color, according to the bounce number of that ray

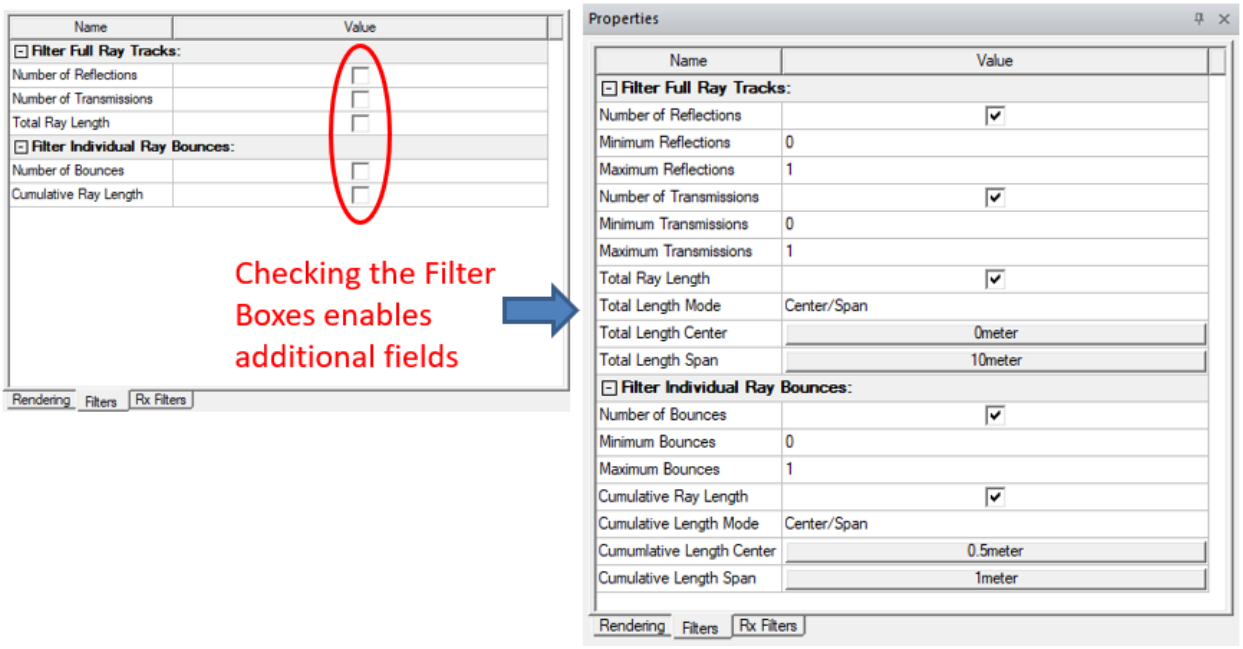
Filters Properties Tab

The **Filters** properties tab controls which ray tracks and/or bounces are shown in the 3D modeler window. For example, you could generate a VRT plot and apply a filter.

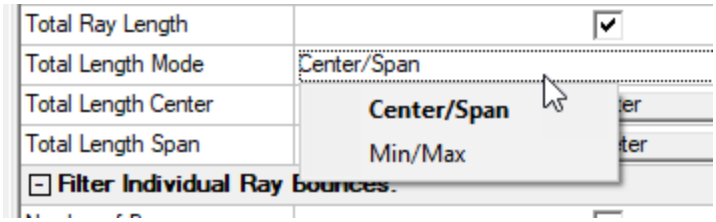


Intially, the checkboxes for Full Ray Tracks and Filter Individual Ray Bounces are unchecked. Your selections for each type of control enables additional fields for specifying exactly how and what to filter for Full Ray Tracks and/or Individual Ray Bounces.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

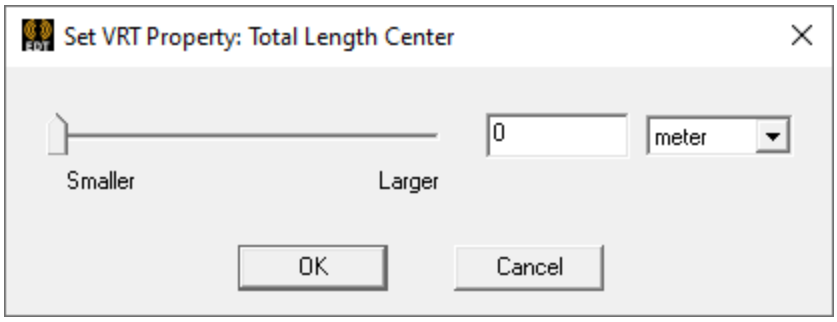


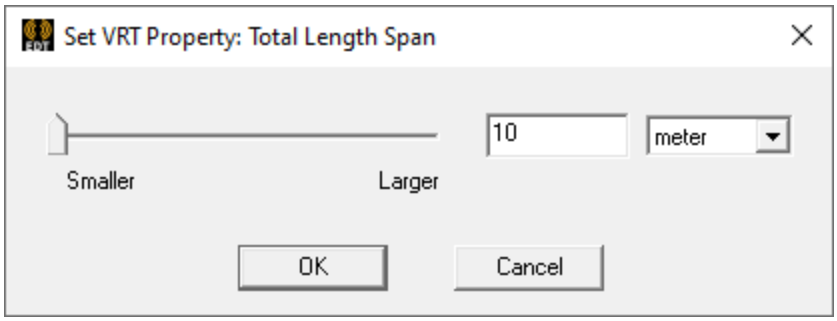
- Filter Full Ray Tracks:
  - **Number of Reflections**: use the Minimum and Maximum Reflections values to show only those ray tracks within the specified range of reflected bounces (filters out whole ray tracks).
  - **Number of Transmissions**: use the Minimum and Maximum Transmissions values to show only those ray tracks within the specified range of transmitted bounces (filters out whole ray tracks).
  - **Total Ray Length**: use Total Length Mode of either Center/Span or Min/Max. You can assign length values as a variable. Using a variable permits you to creating an animation by sweeping the variable.



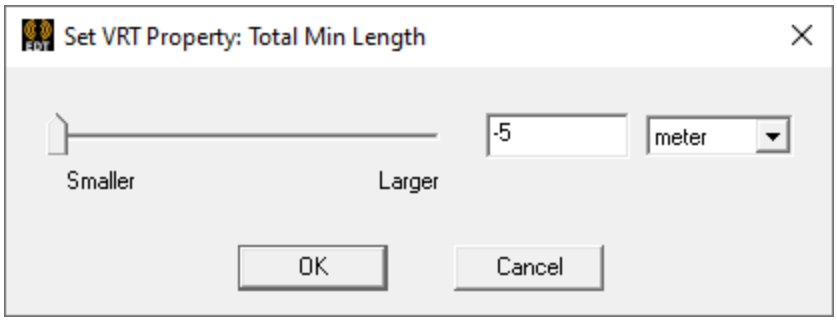
Selecting Center/Span enables buttons for **Total Length Center** and **Total Length Span**. Clicking one of these buttons opens Slider window that allows you to modify the filter values and see in real-time the changing rays in the model view window.

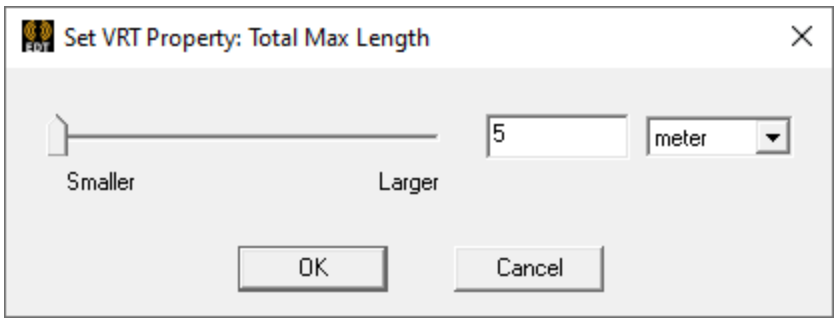
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Selecting Min/Max enables buttons for Total Min Length and Total Max Length. Clicking a button opens a Slider window for real time control of the modeler window.

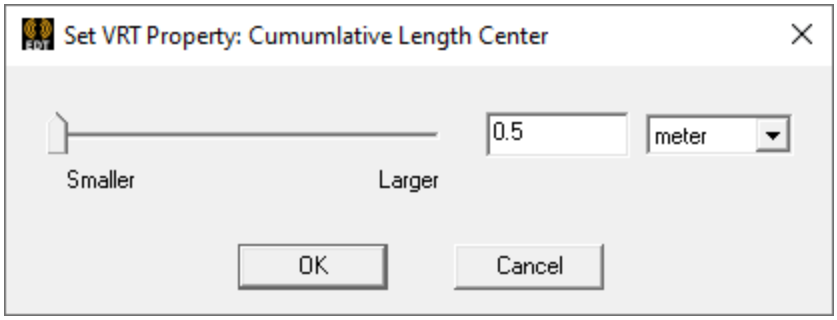


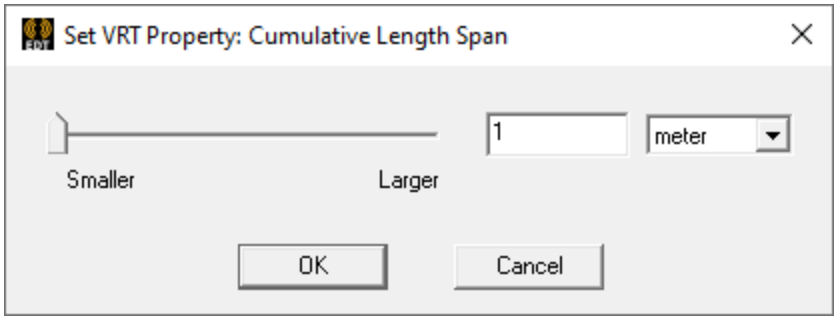


- Filter Individual Ray Bounces:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

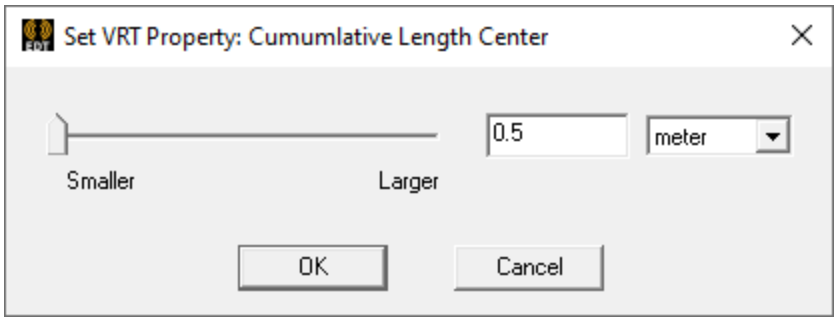
- **Number of Bounces**: use the Minimum and Maximum values to show specific bounce numbers from all ray tracks (filters out individual bounces).
- **Cumulative Ray Length**: use with the Center/Span or Min/Max selections. You can assign length variables as the value. Using a variable allows you to animate the plot by sweeping that variable. Selecting Center/Span enables buttons for **Cumulative Length Center** and **Cumulative Length Span**. Clicking one of these buttons opens Slider window that allows you to modify the filter values and see in real-time the changing rays in the model view window.

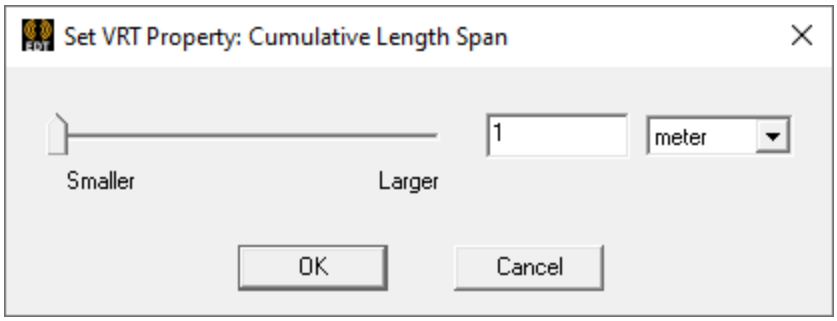




Selecting Min/Max enables buttons for Cumulative Length Center and Cumulative Length Span. Clicking a button opens a Slider window for real time control of the modeler window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



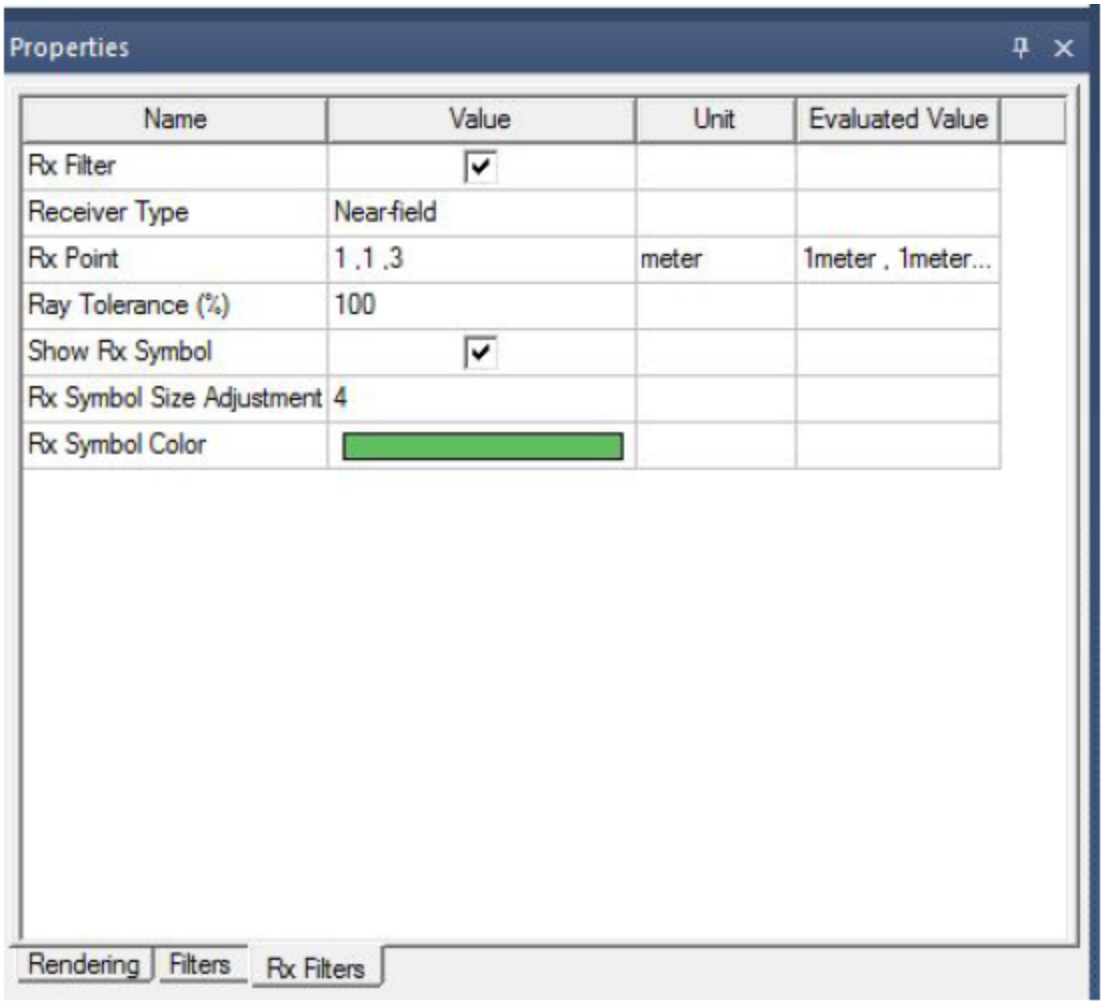


Note these Filters can be used in combination to show only the specified ray bounces from a filtered set of ray tracks (e.g., show only the 2nd and higher bounces for ray tracks that have no transmitted rays).

Rx Filter Properties Tab

Use an “Rx Filter” to show only ray tracks that satisfy a specified scatter direction (far-field) or spatial filter (near-field).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Receiver Type: Near-field: use a spatial probe type of filter to only show ray tracks that go near a specified point/location, within some tolerance
- Receiver Type: Far-field: use a far-field directional filter to only show ray tracks whose exit ray direction (scattering direction) is the specified theta/phi angle, within some tolerance

SBR+ with PTD and UTD

For SBR+ simulations, the Physical Theory of Diffraction (PTD) and Uniform Theory of Diffraction (UTD) features can account for additional phenomenology not well predicted by plain SBR due to truncation of Physical Optics (PO) currents at sharp angular discontinuities ("wedges") on metallic surfaces. PTD is a numeric correction to the results, while UTD is a method to generate additional rays that represent diffraction within the Keller cone off the edge to generate rays in the shadow region. To use these features for the SBR+ solver and Visual Ray Trace (VRT) in the desktop UI, we provide you with a way to specify the location of these "wedges" on the 3D geometry model, so that their contributions for PTD and UTD may be

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

determined when these features are enabled. Typically, there may be thousands of candidate edges on a triangular mesh model, and many hundreds or thousands of “wedges” to include for the simulation. You often need some additional guidance in determining which wedges to include in the simulation, and you may want to specify areas of the model include or exclude wedges in the analysis. You can provide an angular criteria can for the minimum exterior wedge angle to help filter the desired set of wedges, and you can choose to include or exclude wedges from “knife edges”, or open (non-shared) edges from a geometry such as a very thin flat plate. You can view the set of wedges to be used for the SBR+ or VRT simulation in the 3D modeler window, and subsequently regenerate the set of wedges if necessary with different criteria.

Scenarios for using VRT toVisualize UTD Rays include the following:

- Setup an SBR+ simulation with PTD/UTD and check the UTD rays before solving. Set the launching point for a VRT plot to a location where an antenna will be used. Open the statistics tab, and check the Number of UTD rays launched. If the value is greater than 0, then the simulation will benefit from the SBR+ PTD/UTD feature.
- Explore an antenna placement for possible UTD rays. To do this, create variables that control where the antenna will be placed. Apply that variable to a relative coordinate system and/or a point. You can then create a VRT plot where the launching point uses the relative coordinate system or point you have defined. Enable the Shoot UTD rays feature and make those rays visible. After drawing the plot, animate it, and set the swept variable as one that controls the coordinate system or point placement. You can then observe the UTD rays drawn and tune the antenna placement based on the existence of UTD rays.

Parametric Antennas and Parametric Array Antennas for VRT Plots

This chapter provides a summary workflow, and a detailed workflow for using Parametric Antennas and Arrays to create Visual Ray Traced plots with SBR rays and creeping wave (CW) rays. Any Antenna to be used with VRT must be created as a 3D Component in HFSS for it to be available as a ray launch source. Note that VRT plots for creeping wave (CW) rays are only supported for Antennas that can be represented (either synthetically or parametrically) by current sources. This means that far-field antennas created either via external file or via linked design cannot be used with CW plots, but they may still be used for SBR plots.

Summary Workflow

1. Open/create a project with an HFSS design having SBR+ solution type, and containing either a parametric antenna or parametric array antenna component.
2. Add a VRT plot for SBR by selecting **Plot VRT > SBR** from the right-click menu for the **Field Overlays** icon in the **Project Manager** window.

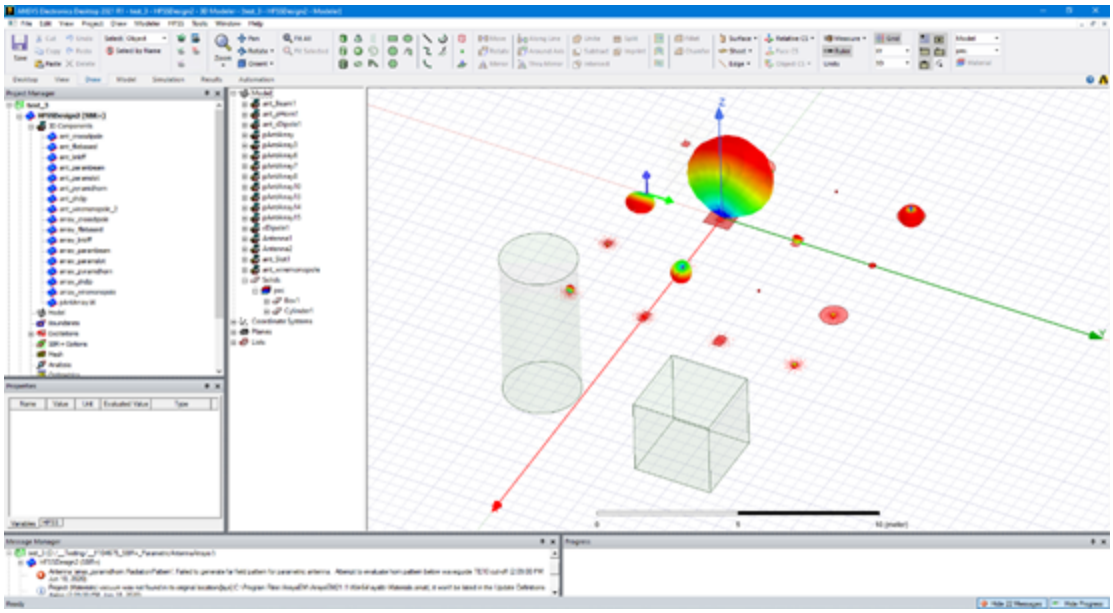
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **Create SBR VRT Plot** dialog
    a. From the “Launch From:” combo box list, select “Antenna”
    b. From the “Antenna/Port:” combo box list of available antennas, select the desired antenna source to use for launching SBR rays
    c. If the selected Antenna is an array, select the desired array element number from where rays will be launched
4. Observe the resulting set of SBR rays generated in the 3D model window

Detailed Workflow

1. Open/create a project with an HFSS design having SBR+ solution type, and containing either a parametric antenna or parametric array antenna component. Remember the design must contain some model geometry so the Visual Ray Trace (VRT) plot will produce rays.

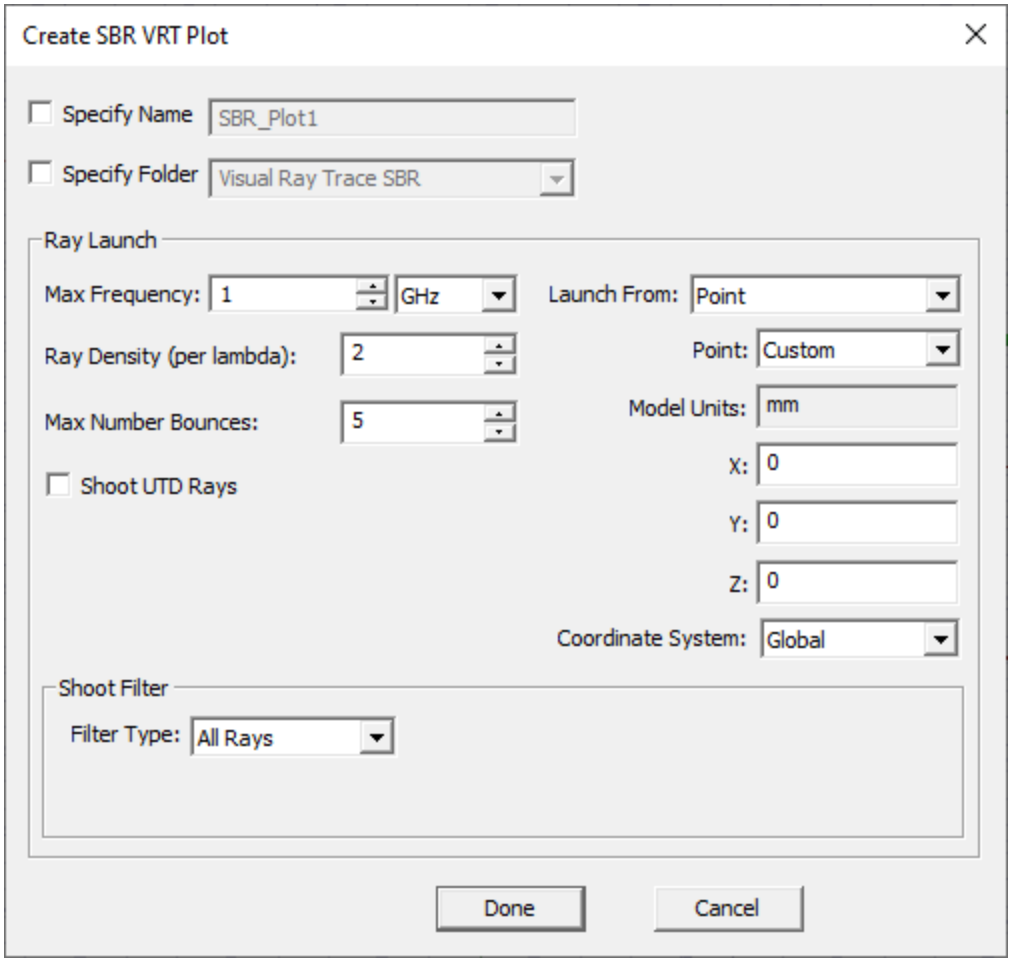
The design(s) should also contain some SBR+ geometry so the Visual Ray Trace (VRT) plot will produce rays.



2. Add a VRT plot for SBR by right clicking **Field Overlays** in the **Project Manager** window and selecting **Plot VRT > SBR**.

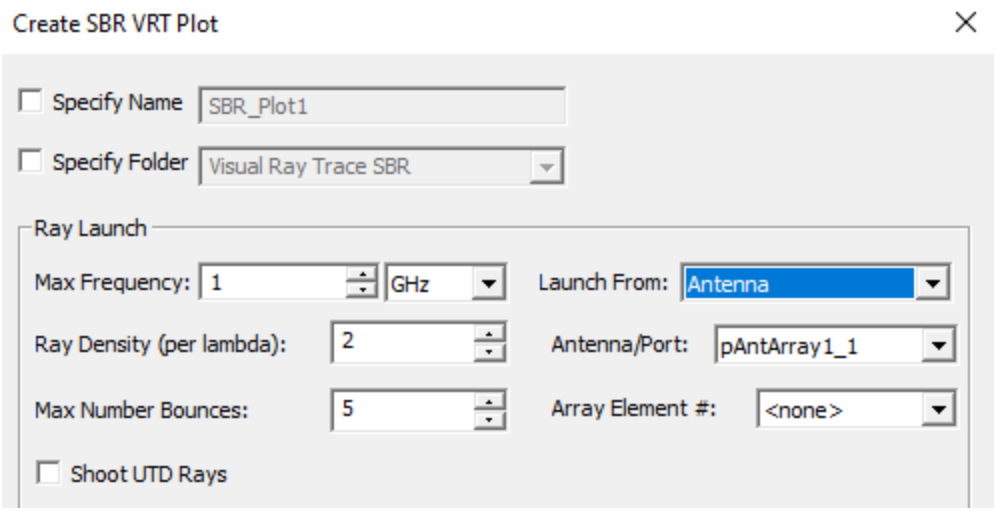
This opens the **Create SBR VRT Plot** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. In the **Create SBR VRT Plot** dialog:
    a. From the Launch From: combo box list, select Antenna.

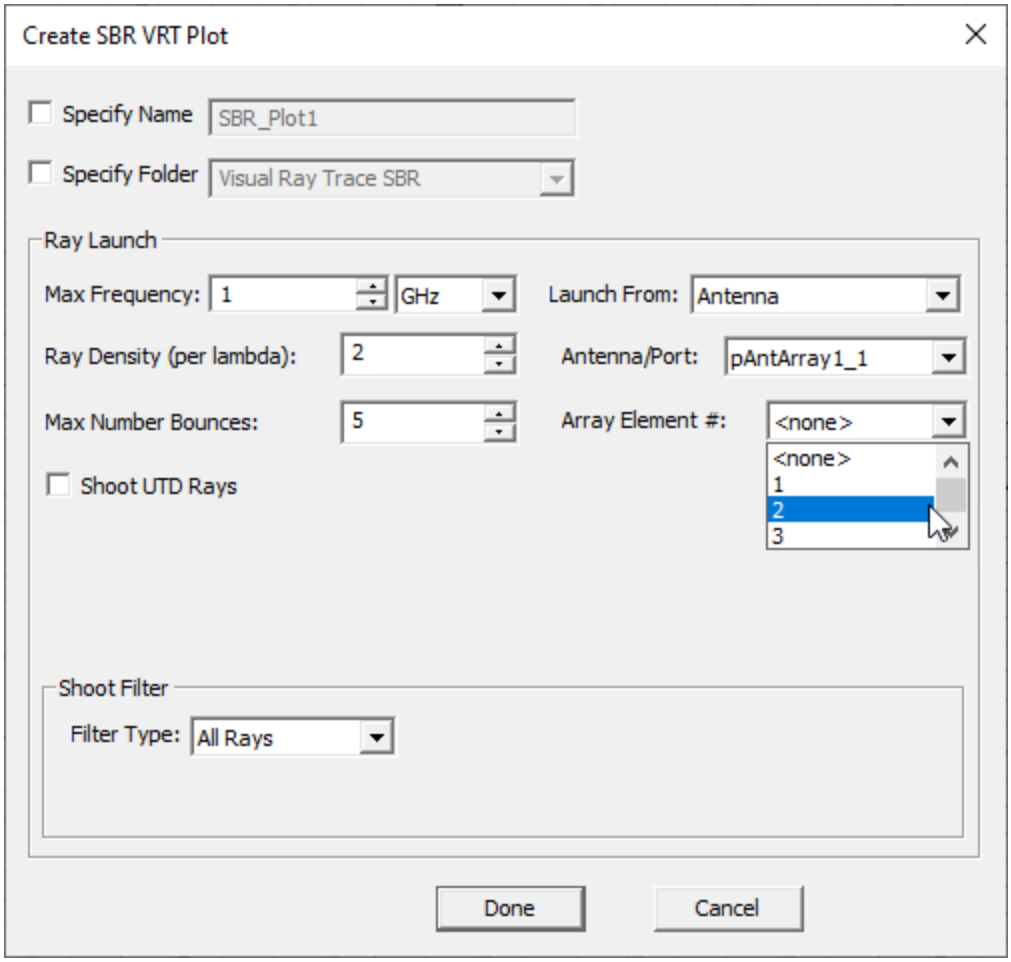
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This enables the Antenna/Port: combo box list of available antennas.

b. Select the desired antenna source to use for launching SBR rays.
c. If the selected Antenna is an array, select the desired array element number from where rays will be launched.

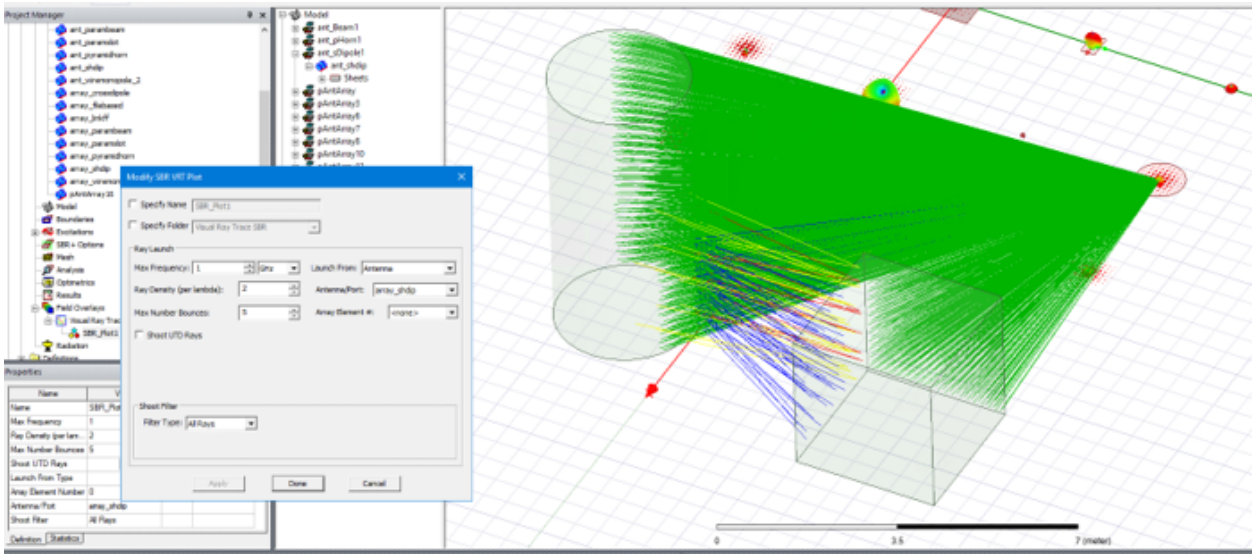
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you select <none> for the Array Element #, then the rays are launched from the placement location of the antenna (the origin of the antenna local coord sys) instead of from a specific array element location

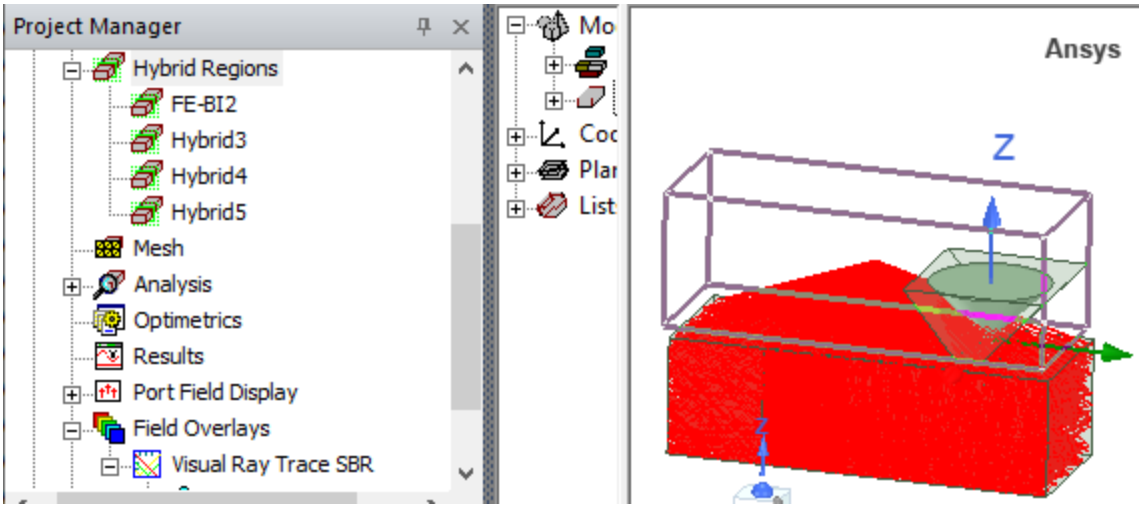
4. Observe the resulting set of SBR rays generated in the 3D model window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The steps for creating a VRT plot for CW rays with a parametric array antenna are nearly identical to the steps for a VRT plot showing SBR rays. See the page on Creeping Wave for Antenna Placement on Curved Surfaces in SBR+ Simulation for more details on the setup of a VRT plot with CW rays. As with an SBR plot, the CW plot may be configured to launch rays from a specific element number of the array antenna. Note that for CW plots with parametric array (and for CW analysis setup with parametric arrays), surface conformance of the array elements is typically required to produce suitable CW results.

Note: In the case of a FE-BI Hybrid Region, by design the VRT rays launch from the center of the FE-BI, rather than a port location in the FE-BI portion of the design. For example, in the following image, the highlighted box is assigned as a FE-BI Hybrid Region, and the center of the box becomes the VRT launch point rather than the antenna within the Hybrid region.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creeping Wave VRT Plots

This section describes how to use Visual Ray Trace (VRT) with Creeping Wave for "Using Visual Ray Trace (VRT) with Creeping Wave for Antenna Placement" on page 13-293 and for "Using Visual Ray Trace (VRT) with Creeping Wave for RCS" below.

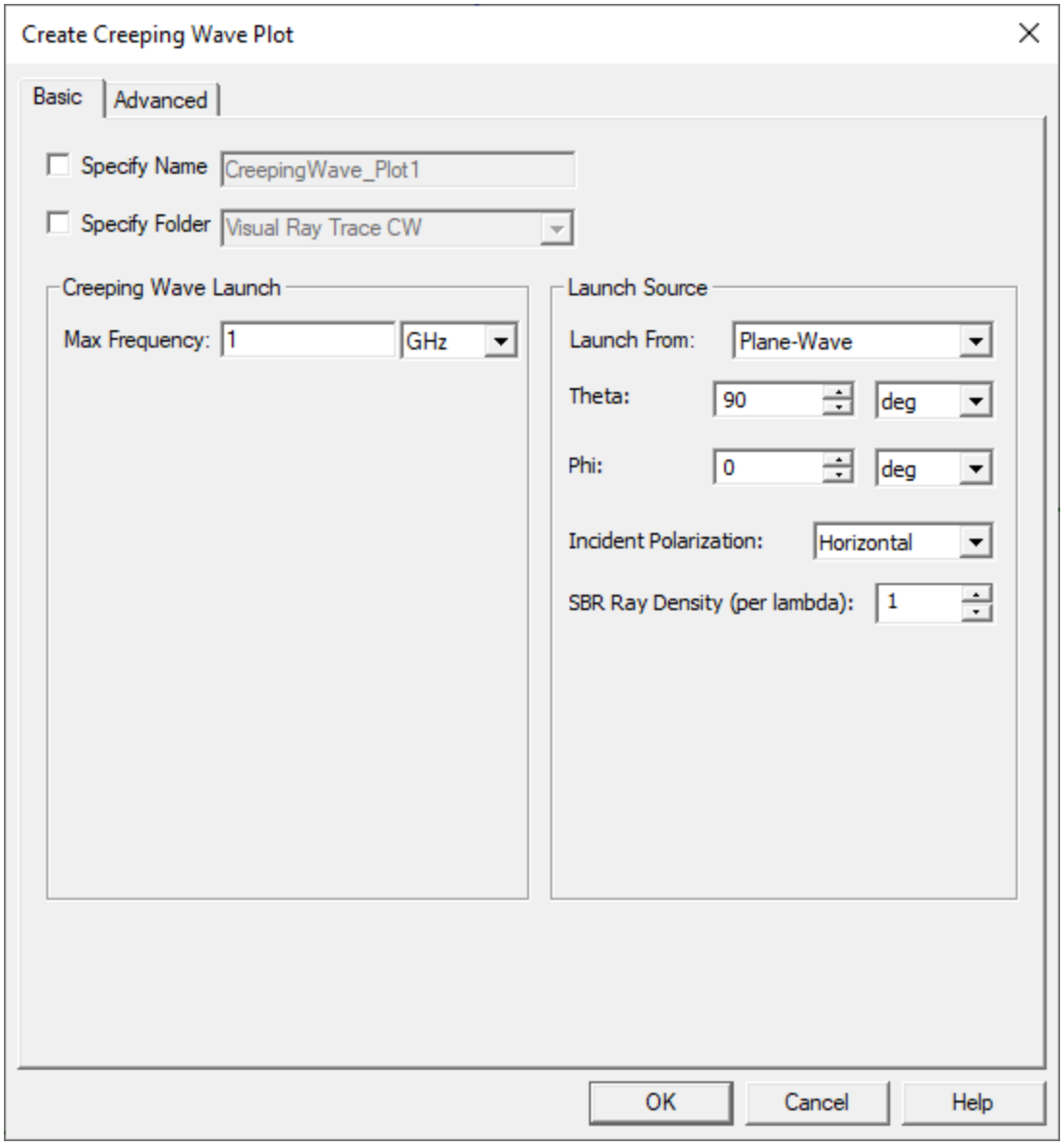
Using Visual Ray Trace (VRT) with Creeping Wave for RCS

1. Once you have an appropriate HFSS design, there are three different ways in the user interface to create a new Visual Ray Trace Plot with Creeping Rays:

- Right-click the **Field Overlays** icon in the **Project Tree** and select **Plot VRT...>Creeping Wave** from the short cut menu.
- Right-click the 3D Modeler window and select **Plot VRT...>Creeping Wave** from the pop-up menu.
- Click **HFSS>Fields>Plot VRT...>Creeping Wave**

After you use one of these approaches, the **Create Creeping Wave VRT Plot** dialog displays:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



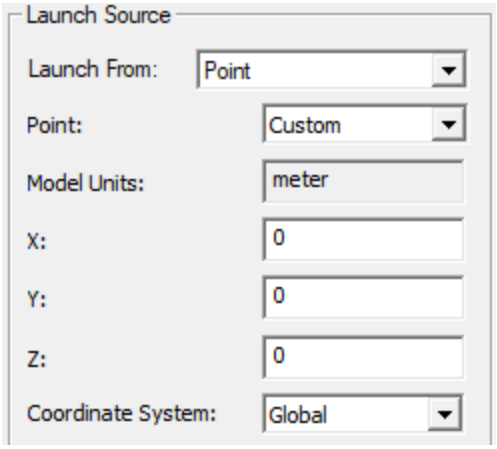
To **Specify Name** check the box to enable the text field. To **Specify Folder**, check the box to enable the selection menu.

2. Set the **Creeping Wave Ray Launch** parameters

- **Max Frequency** should be set to the maximum frequency desired for the simulation. For simulations spanning multiple frequencies, the creeping wave settings should be tested at the highest simulation frequency.
- **Launch from** parameters includes a menu for selecting the source as Plane-Wave or Point. For Plane-Wave, you specify the plane wave's incident direction by specifying **Theta** and **Phi** in spherical coordinates. If you select Point, the fields change to X, Y, and

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Z for a Global CS.

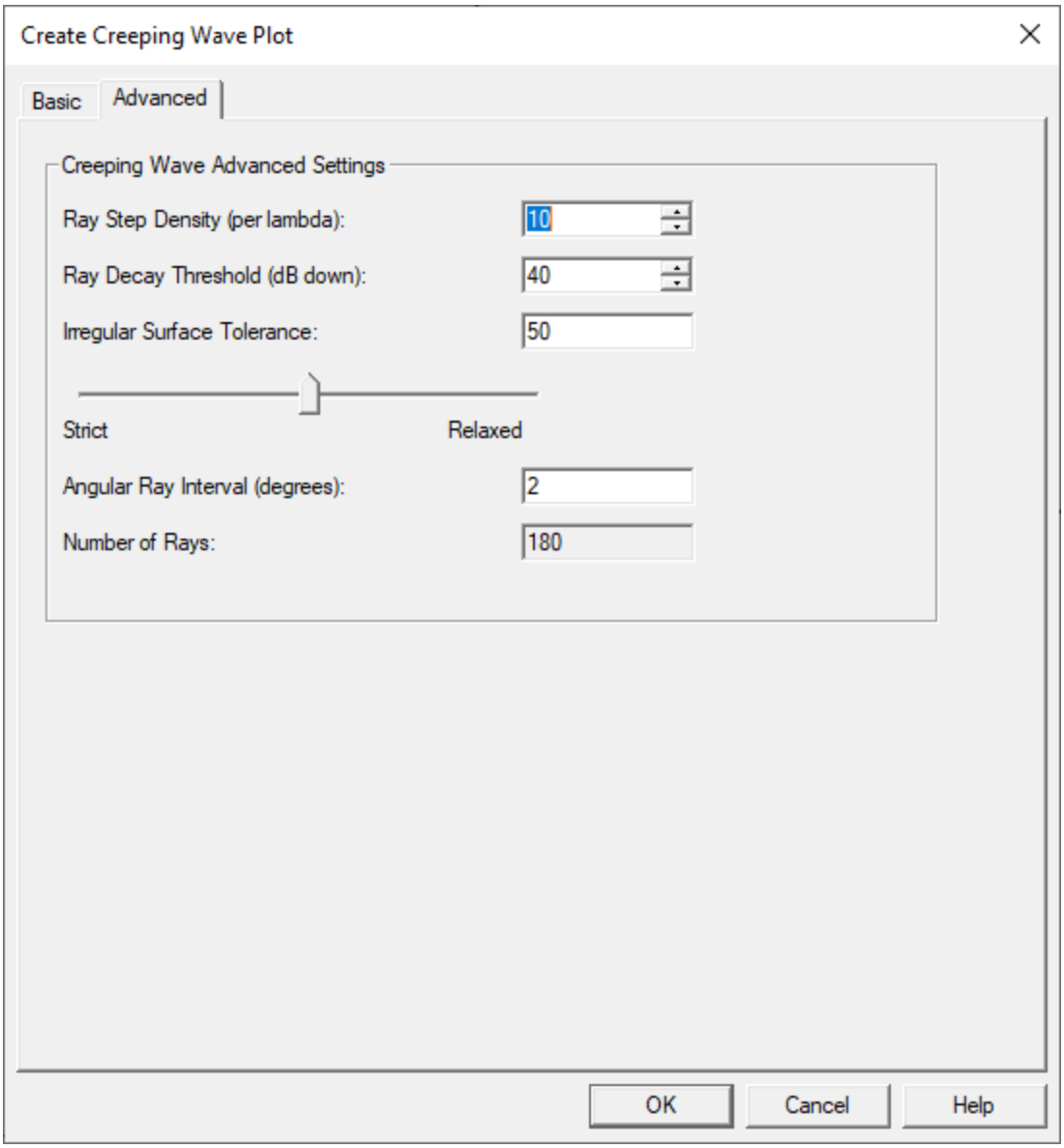


- **Incident Polarization** specifies the polarization of the incident plane wave as horizontal (Phi directed) or vertical (Theta directed). In conjunction with the **Ray Decay Threshold** setting, the choice of incident polarization differentially influences the propagation distance of individual creeping rays since their rate of decay depends on the polarization of their associated fields. In general, creeping rays launched from the shadow boundary where the incident electrical field is perpendicular to the surface decay more slowly than for those where the field is parallel to the surface.
- **SBR Ray Density (per lambda)** controls SBR ray density per lambda

Caution: The settings in this diagram are used by the VRT to test if the creeping waves are propagating as expected. **These parameters are not automatically transferred to the Creeping Wave Settings for simulation. You need to set the simulation Creeping Wave Settings manually.**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Optionally select the **Advanced** tab.



Ray Step Density (per lambda) controls lateral sampling along the direction of propagation along the surface.

Ray Decay Threshold controls how far the creeping rays propagate by terminating each ray once its power density decay (defined as dB down from its level at its starting point at the shadow boundary) exceeds this threshold.

Irregular Surface Tolerance is a single control for a collection of tolerances that determine creeping rays' sensitivities to local surface irregularities that theoretically should block their further propagation. Moving the slider toward **Relaxed** allows

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

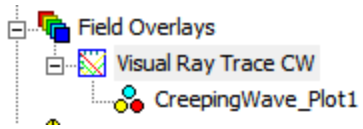
creeping rays to propagate through small surface irregularities that may arise through defects in the input geometry or due to limitations of curvature extraction accuracy. However, setting this tolerance too relaxed may allow creeping waves to pass through concave surfaces, saddle surfaces, or wedges, which are not supported by creeping wave physics. The intention is that creeping rays only propagate over the convex portions of surfaces, and some user judgment is required in correctly setting the Irregular Surface Tolerance for each situation.

Angular Ray Interval (degrees) controls the density of rays that are generated along the nearest surface, starting under the current source, and spanning the 360 degrees around the current source.

Number of Rays indicates how many CW rays will be launched. This is a read-only field that is internally computed by dividing 360 degrees by the **Angular Ray Interval**

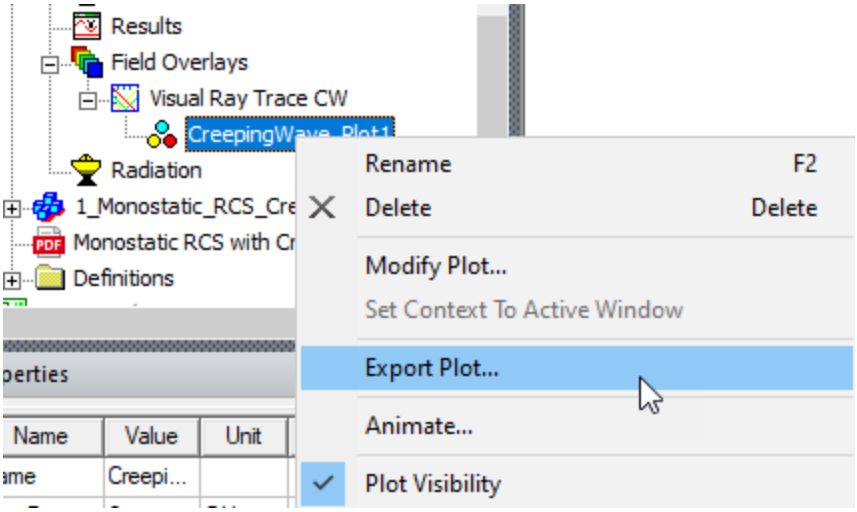
3. Click **OK** to dismiss the dialog to runs the VRT generation to produce SBR+ Creeping ray data. Only the geometries contained in the SBR+ regions are used for the VRT ray generation.
4. The “Shoot Filter” selections are disabled in the dialog box, and moved to the **Visual Ray Trace CW** folder properties as described below.

A new “Visual Ray Trace CW” plot folder is created under Field Overlays, which contains one or more “CreepingWave_Plot” items.

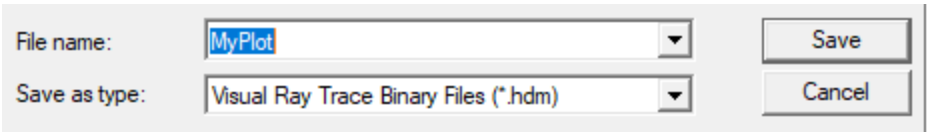


You can right-click on the VRT plot to display the **Export Plot...** command.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This allows you to export Visual Ray Trace Binary Files using a .hdm suffix.

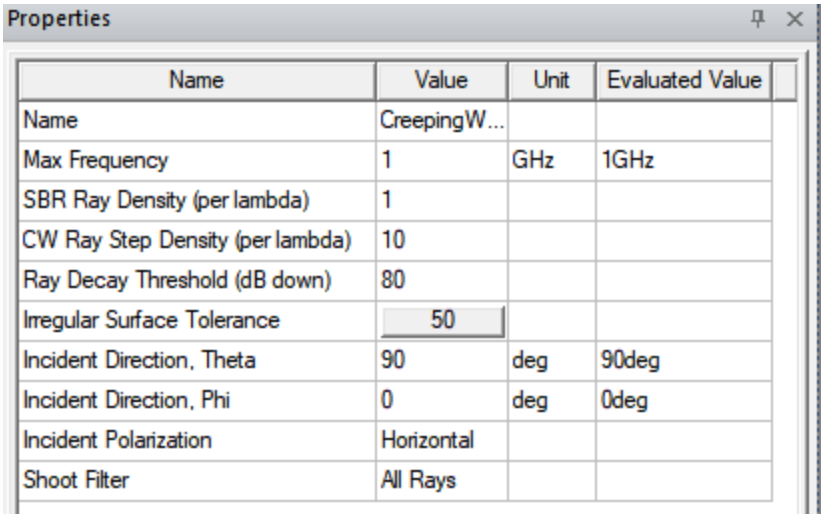


If you run **Plot VRT...>Creeping Wave** again, that creates an additional CreepingWave_Plot item under the **Visual Ray Trace CW** folder, or a different folder name if you specify one in the **Create Creeping Wave VRT Plot** dialog.

Creeping Wave Plot Properties

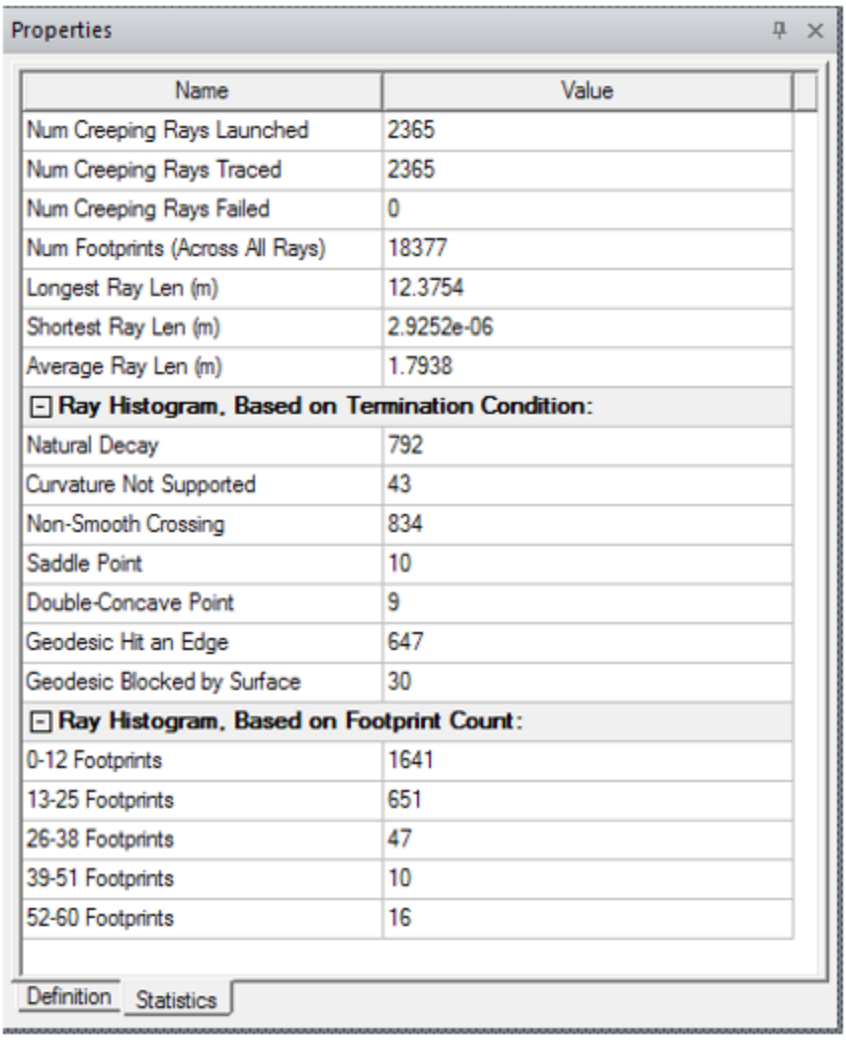
The Properties of each CreepingWave_Plot*n* item are the same as those created in the **Create Creeping Wave VRT Plot** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Each Creeping Wave plot has an additional read-only tabbed property page that displays Ray Statistics output from the ray generation process.

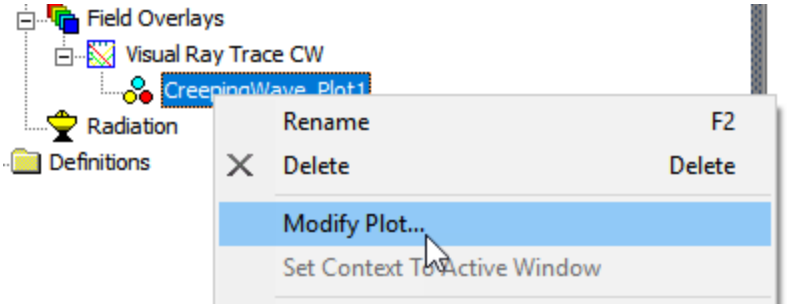
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



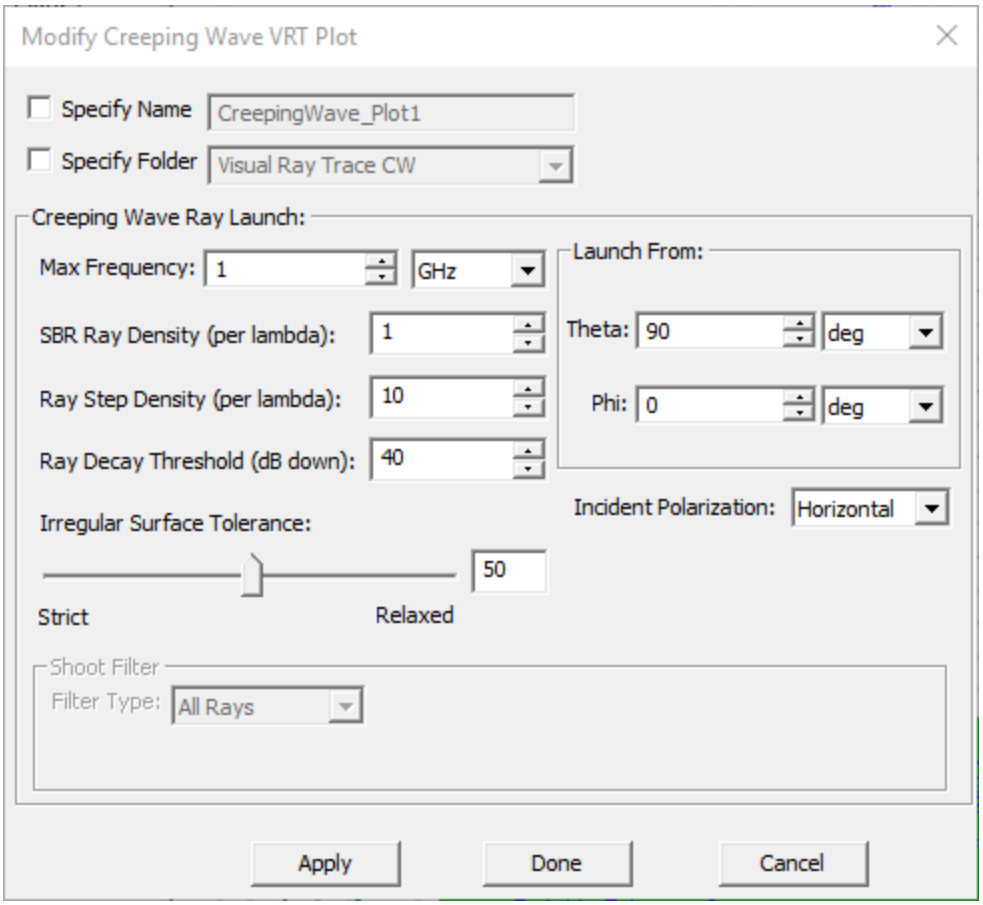
Modifying Creeping Wave Plots

You can modify Creeping Wave properties directly in the **Properties** window to cause immediate updates to the plot data. You can also right-click the plot item and select **Modify Plot…** to again bring up the **Create VRT Plot** dialog to make multiple changes and then commit them to the same, or a differently named plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This opens a **Modify Creeping Wave VRT Plot** dialog.



Design Edits that Invalidate the Mesh Invalidate Creeping Ray VRT Plots

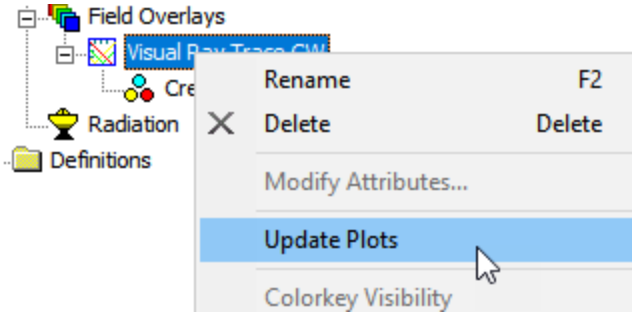
Any design edits that invalidate the mesh (geometry delete, modify, or change in boundary condition assignment) also invalidates VRT plots. HFSS does not automatically regenerate the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VRT plot after a design edit. You can update plots update by right-clicking on the **Visual Ray Trace SBR** folder and selecting **Update Plots**.

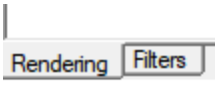
Modification in SBR+ Wedge Setting parameters invalidates the VRT plot when “Shoot UTD Rays” is turned on.

In addition for UTD rays, a modification to the geometry and/or boundary conditions should automatically cause new SBR+ wedge to be generated whenever the VRT plot is manually updated.



Visual Ray Trace CW Folder Properties

All the render and filter parameters appear as Properties of the parent **Visual Ray Trace CW** folder. Each **Visual Ray Trace CW** plot folder provides a grouping of render/filter attributes. Much like setting specific plot attributes like 2D plot extents, scaling, colormaps, etc., the render and filter parameters for all VRT plots are performed at this folder level. The different categories of render and filter operations are kept as separate tabs for: Rendering and Filters.

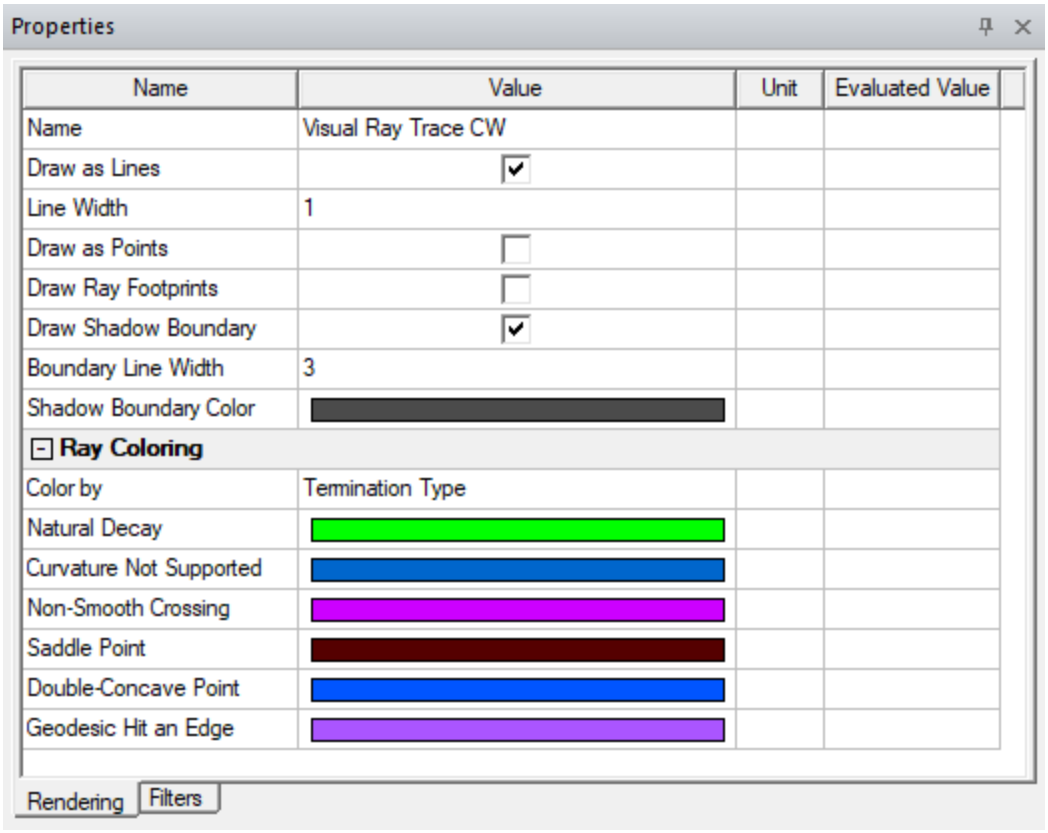


All render and filter property changes are immediate and will automatically refresh the 3D modeler window

Rendering Properties Tab

The **Rendering** tab properties control which items are drawn for each ray track, as well as width and shadow boundary controls:

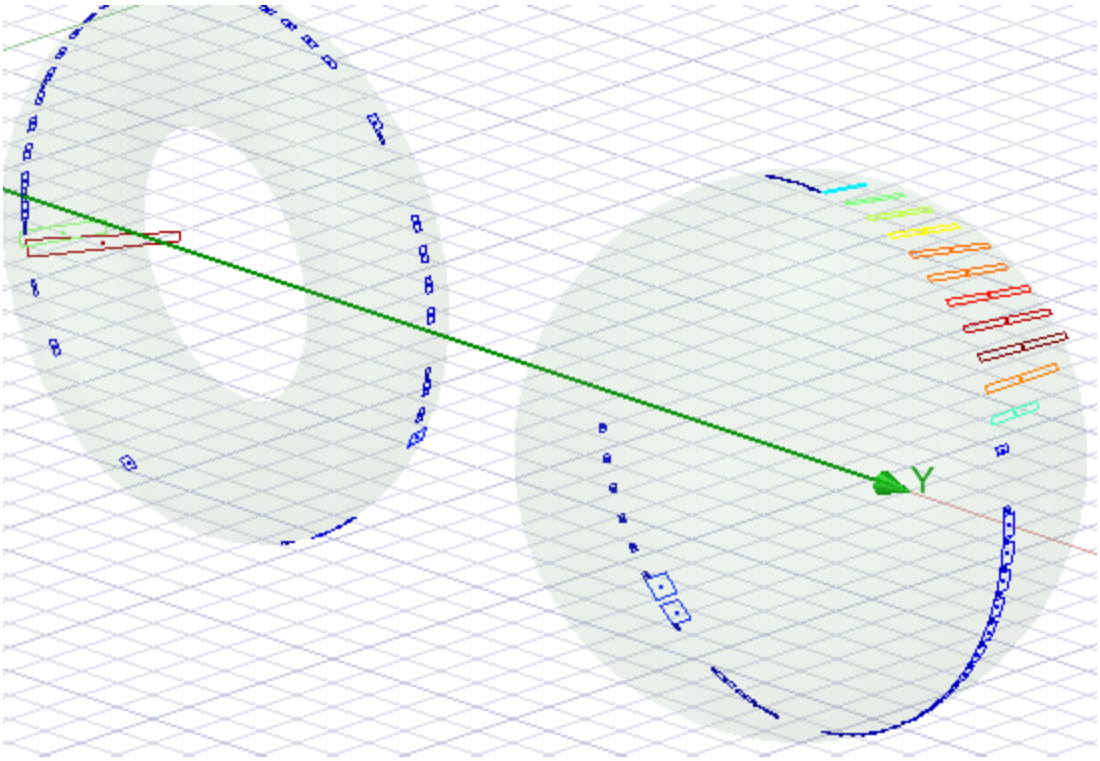
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



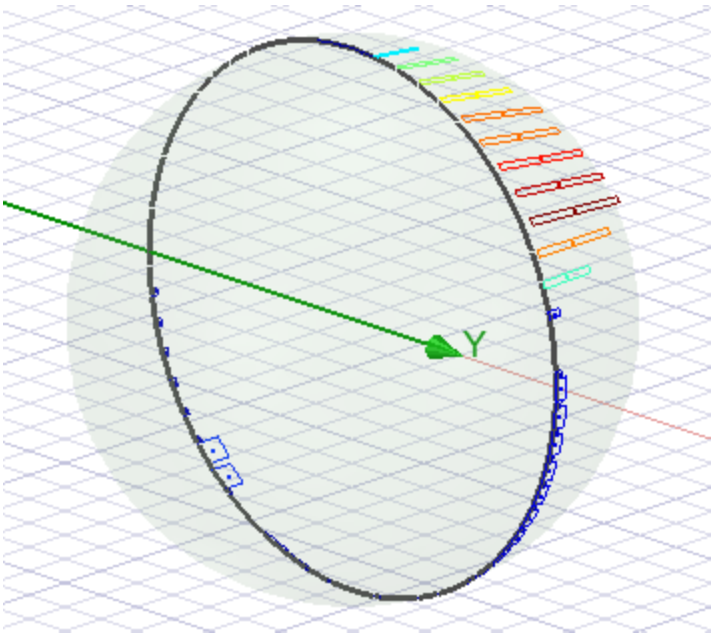
- Draw as Lines: show the “rays” connecting the launch point, each intersection point, and the exit point.
- Draw as Points: show the point for each ray bounce where we intersect the geometry.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Draw Ray Footprints: show the ray tube projection for each ray bounce.



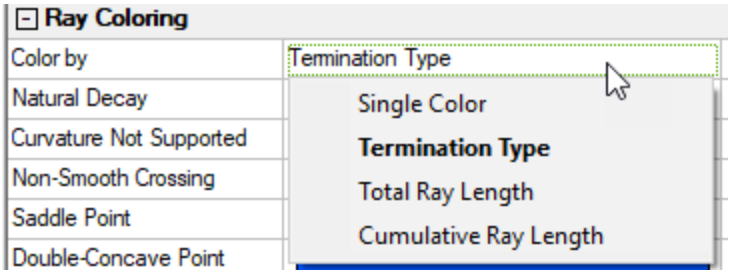
- Draw Shadow Boundary: shows the edge that defines the shadow region.



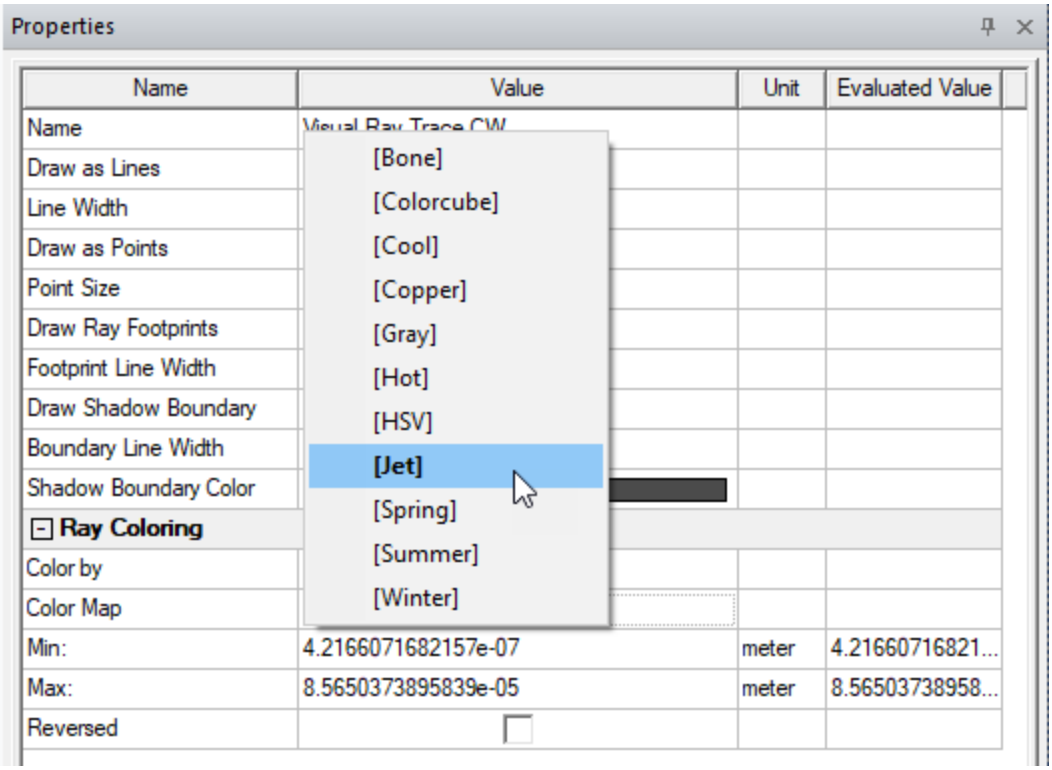
Ray Coloring controls:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Color by: Can be Single Color, Termination Type, Total Ray Length, or Cumulative Ray Length.



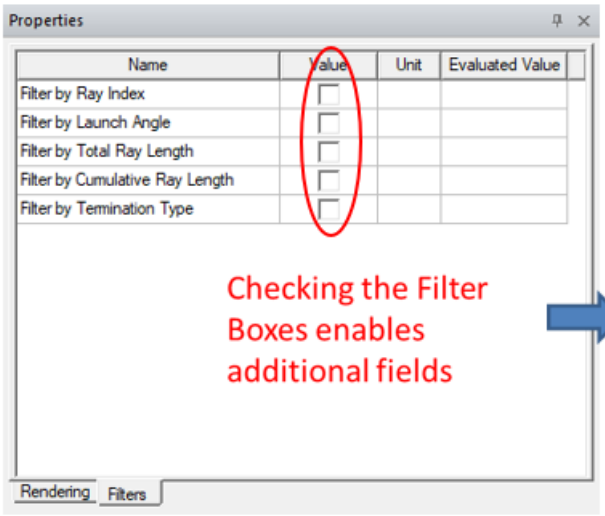
- If you select Termination Type, the terminations listed depend on those generated by the plot.
- If you select Single Color, the properties are Line Color, Point Color, and Footprint Color.
- If you select Total Ray Length or Cumulative Ray Length, the properties are Color Map (which offers many choices, shown below), Min, Max, and Reversed.

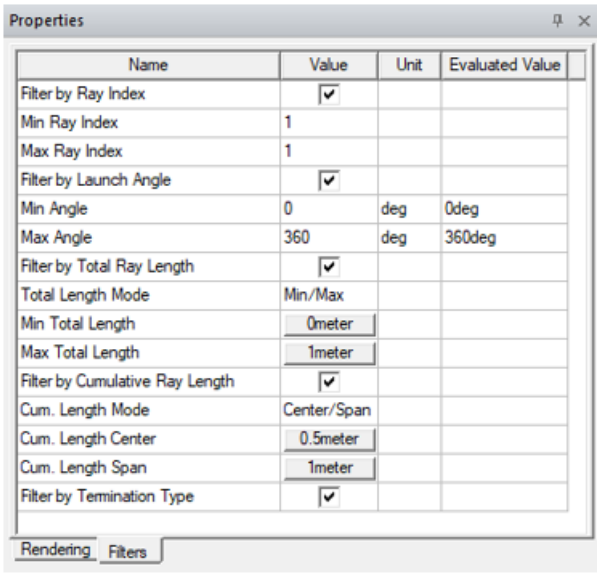


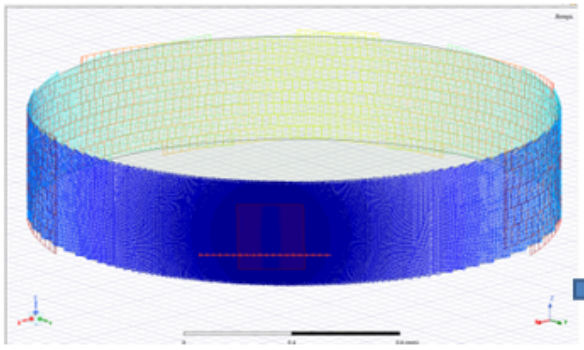
Filter Properties Tab

The Filter properties tab controls which ray tracks and/or bounces are shown in the 3D modeler window. Filter by Launch Angle is unique to CW for antennas (vs. CW for RCS).

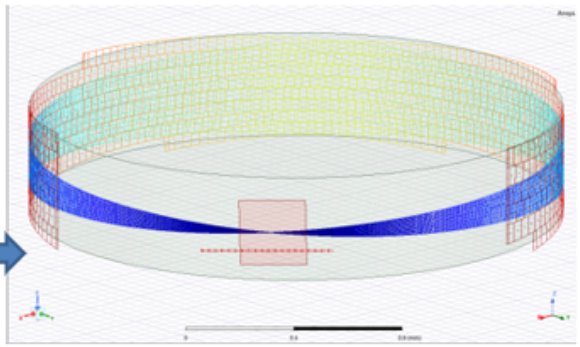
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



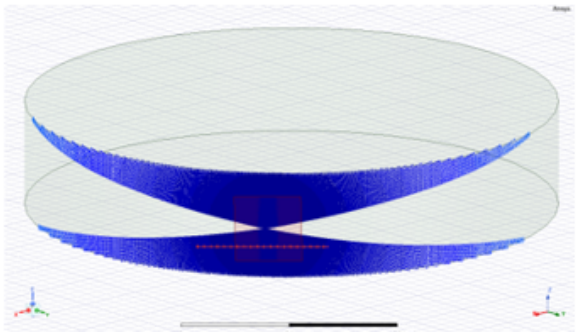


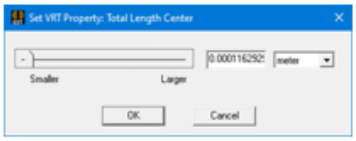


Apply a Total Ray Length Filter



Modify the Properties of the Filter with sliders and Analyze the Results

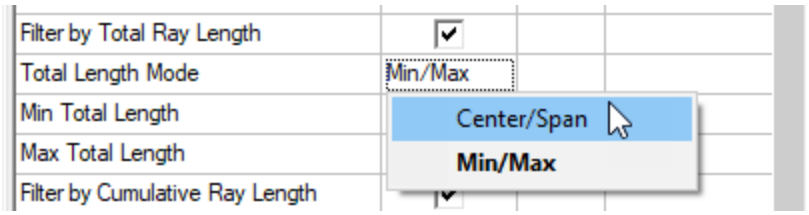




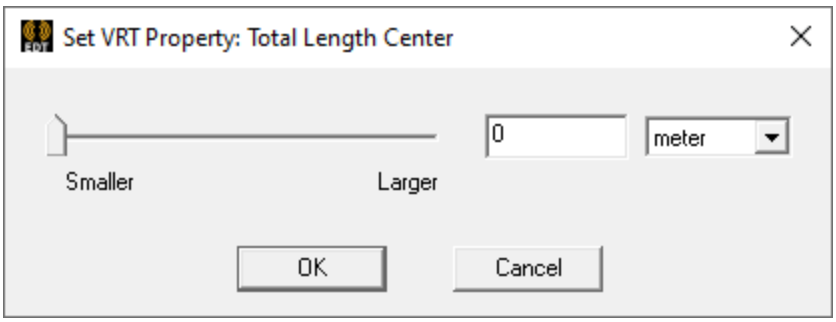
Selecting any of Ray Index, Launch Angle, Total Ray Length, opens Min and Max parameters for that filter.

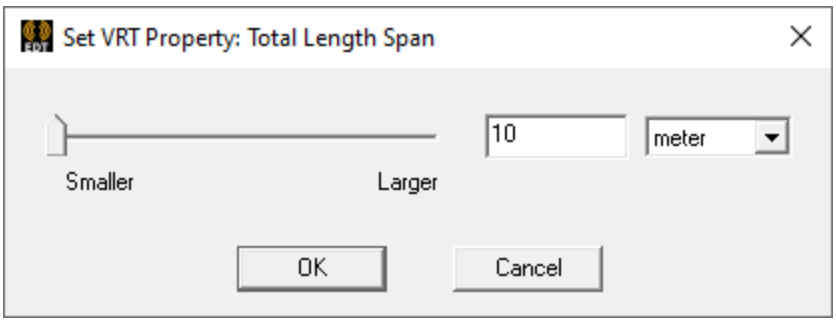
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Filter By Total Ray Length: use Total Length Mode of either Center/Span or Min/Max. You can assign length values as a variable. Using a variable permits you to creating an animation by sweeping the variable.



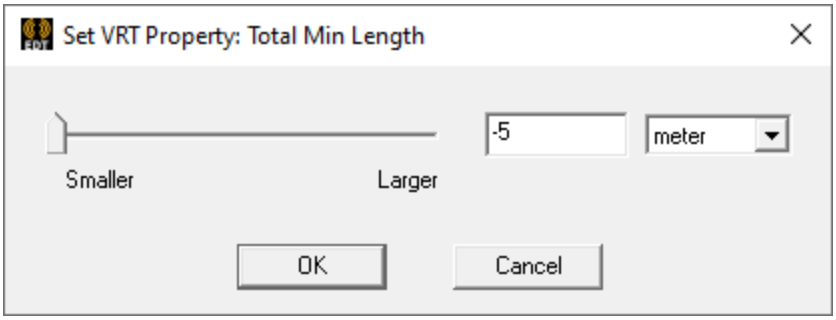
Selecting Center/Span enables buttons for **Total Length Center** and **Total Length Span**. Clicking one of these buttons opens Slider window that allows you to modify the filter values and see in real-time the changing rays in the model view window.

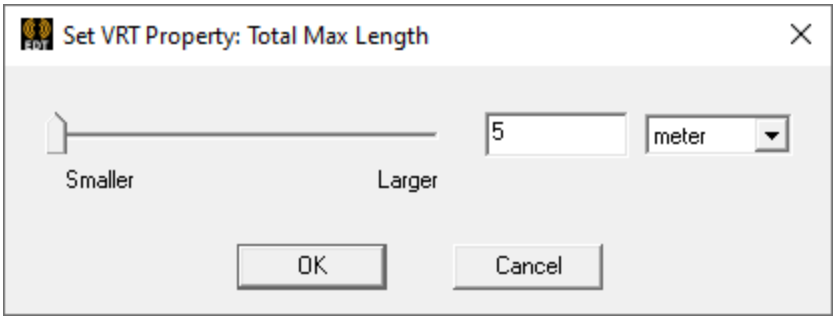




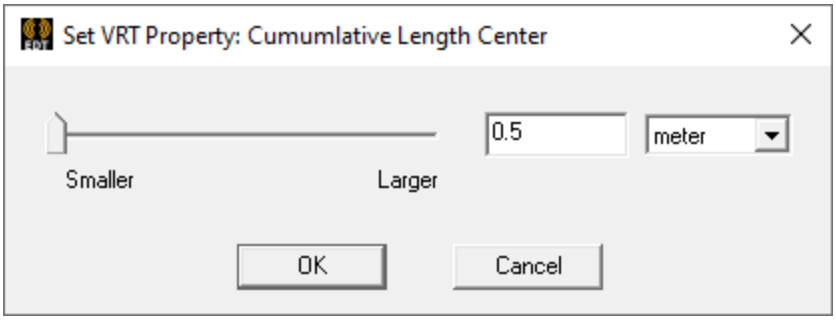
Selecting Min/Max enables buttons for Total Min Length and Total Max Length. Clicking a button opens a Slider window for real time control of the modeler window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

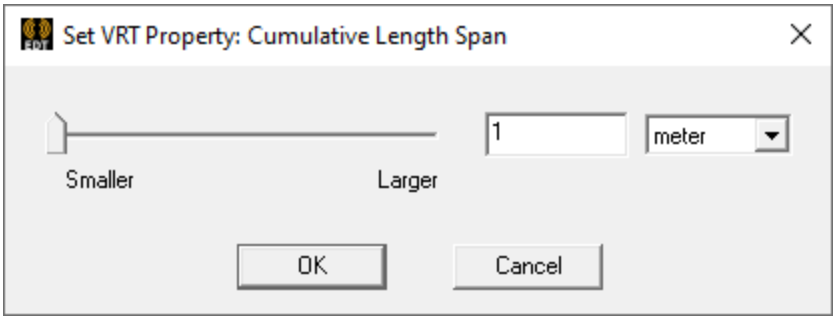




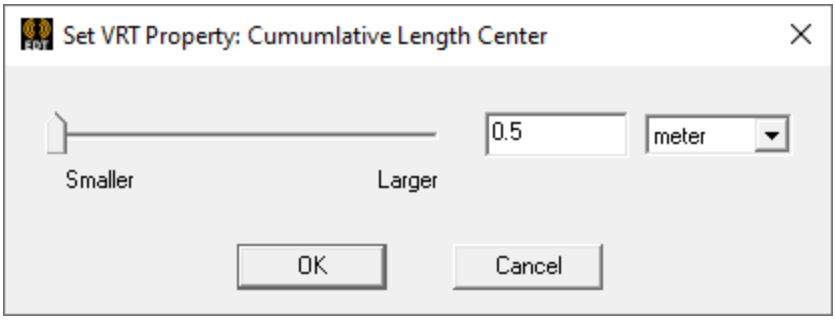
Filter By Cumulative Ray Length: use with the Center/Span or Min/Max selections. You can assign length variables as the value. Using a variable allows you to animate the plot by sweeping that variable. Selecting Center/Span enables buttons for **Cumulative Length Center** and **Cumulative Length Span**. Clicking one of these buttons opens Slider window that allows you to modify the filter values and see in real-time the changing rays in the model view window.

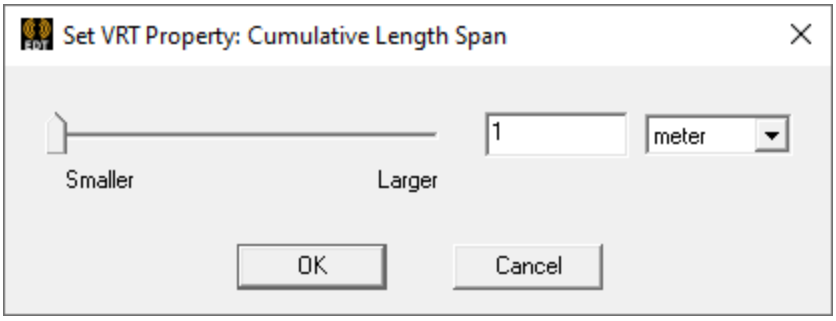


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



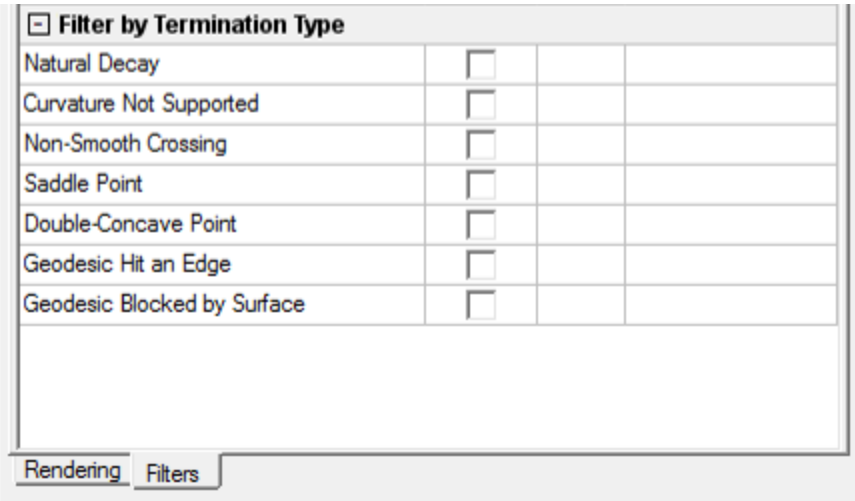
Selecting Min/Max enables buttons for Cumulative Length Center and Cumulative Length Span. Clicking a button opens a Slider window for real time control of the modeler window.





Selecting **Filter by Termination Type** lists the terminations available in the model.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



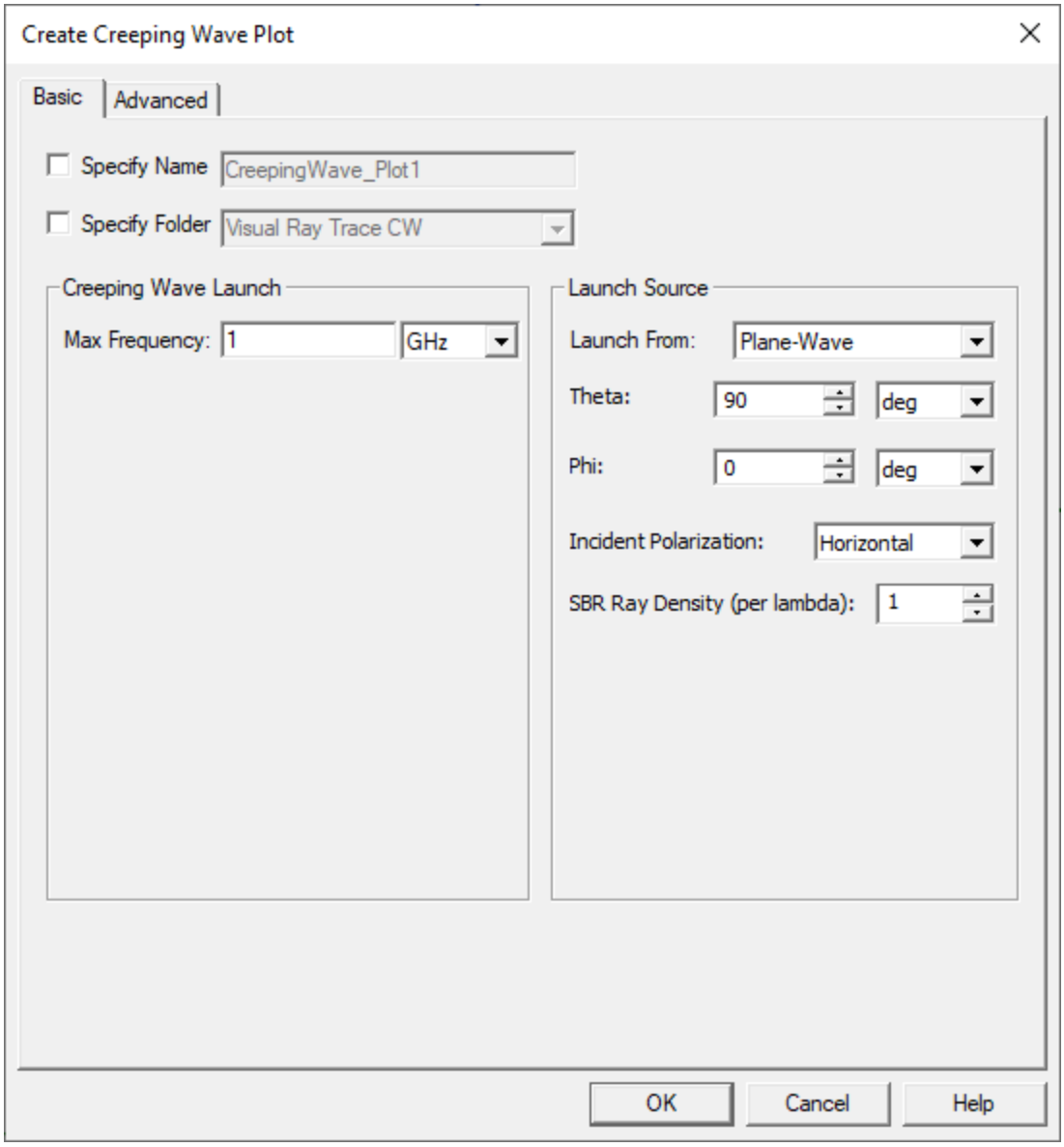
Using Visual Ray Trace (VRT) with Creeping Wave for Antenna Placement

1. Once you have an appropriate HFSS design, there are three different ways in the user interface to create a new Visual Ray Trace Plot with Creeping Rays:

- Right-click the **Field Overlays** icon in the **Project Tree** and select **Plot VRT...>Creeping Wave** from the short cut menu.
- Right-click the 3D Modeler window and select **Plot VRT...>Creeping Wave** from the pop-up menu.
- Click **HFSS>Fields>Plot VRT...>Creeping Wave**

After you use one of these approaches, the **Create Creeping Wave VRT Plot** dialog displays.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



From the **Launch From**: drop down menu, select the Current-Source antenna source and then select the desired **Antenna/Port** for launching creeping waves. This will change some dialog fields, from the Plane Wave configuration, as follows.

To **Specify Name** check the box to enable the text field. To **Specify Folder**, check the box to enable the selection menu.

2. Set the **Creeping Wave Ray Launch** parameters

- **Max Frequency** should be set to the maximum frequency desired for the simulation. For simulations spanning multiple frequencies, the creeping wave settings should be tested at the highest simulation frequency.

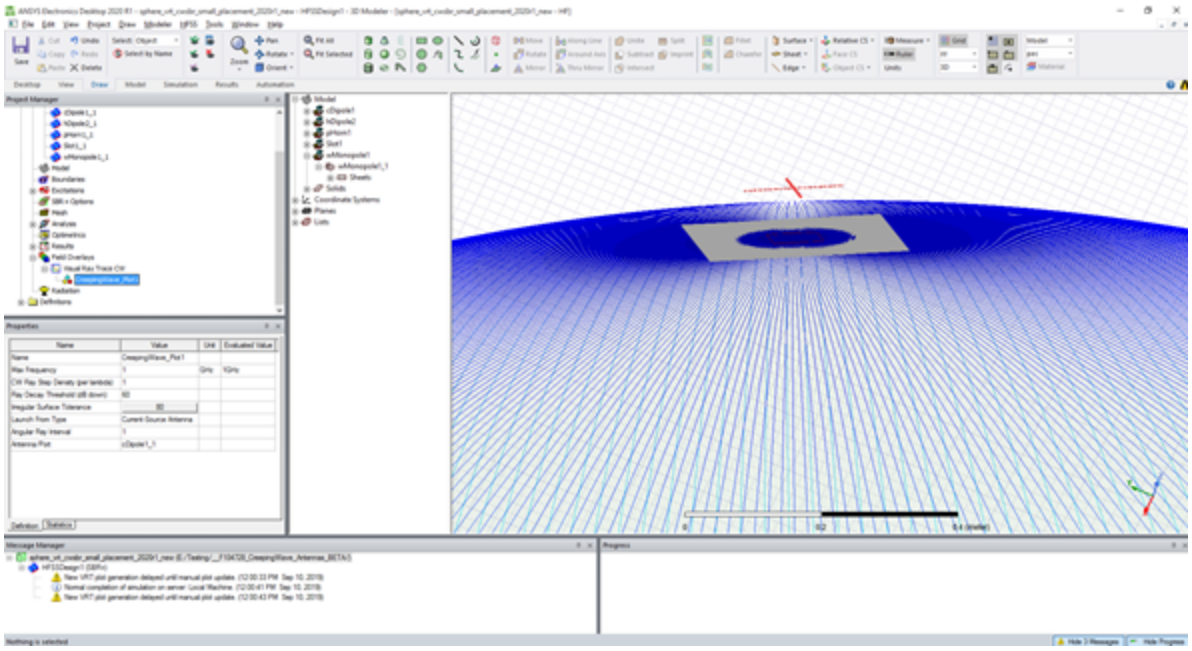
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Ray Step Density (per lambda)** controls longitudinal sample along the direction of propagation along the surface.
- **Ray Decay Threshold** controls how far the creeping rays propagate by terminating each ray once its power density decay (defined as dB down from its level at its starting point at the shadow boundary) exceeds this threshold.
- **Irregular Surface Tolerance** is a single control for a collection of tolerances that determine creeping rays' sensitivities to local surface irregularities that theoretically should block their further propagation. Moving the slider toward **Relaxed** allows creeping rays to propagate through small surface irregularities that may arise through defects in the input geometry or due to limitations of curvature extraction accuracy. However, setting this tolerance too relaxed may allow creeping waves to pass through concave surfaces, saddle surfaces, or wedges, which are not supported by creeping wave physics. The intention is that creeping rays only propagate over the convex portions of surfaces, and some user judgment is required in correctly setting the Irregular Surface Tolerance for each situation.
- **Launch from** parameters includes a menu for selecting the source for Antenna placement Creeping Wave, the near field parametric antenna or Near Field link you specified.
- **Antenna /Port** lists available antennas/ports that can be represented by current sources.
- **Angular Ray Interval (degrees)** controls the density of rays that are generated along the nearest surface, starting under the current source, and spanning the 360 degrees around the current source
- **Number of Rays** indicates how many CW rays will be launched. This is a read-only field that is internally computed by dividing 360 degrees by the **Angular Ray Interval**.

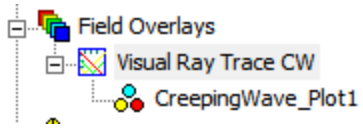
> **Caution**: The settings in this diagram are used by the VRT to test if the creeping waves are propagating as expected. **These parameters are not automatically transferred to the Creeping Wave Settings for simulation. You need to set the simulation Creeping Wave Settings manually.**

3. Click **Done** to dismiss the dialog and run the VRT generation to produce SBR+ Creeping ray data. Only the geometries contained in the SBR+ regions are used for the VRT ray generation.
4. The "Shoot Filter" selections are disabled in the dialog box, and moved to the **Visual Ray Trace CW** folder properties as described below.
5. After the plot is generated, you can use the plot properties to update or change the appearance of the plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A new “Visual Ray Trace CW” plot folder is created under Field Overlays, which contains one or more “CreepingWave_Plot” items.

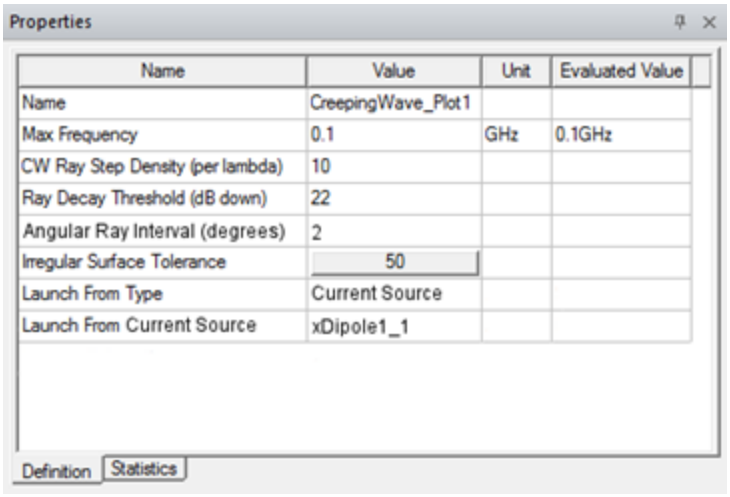


If you run **Plot VRT...>Creeping Wave** again, that creates an additional CreepingWave_Plot item under the **Visual Ray Trace CW** folder, or a different folder name if you specify one in the **Create Creeping Wave VRT Plot** dialog.

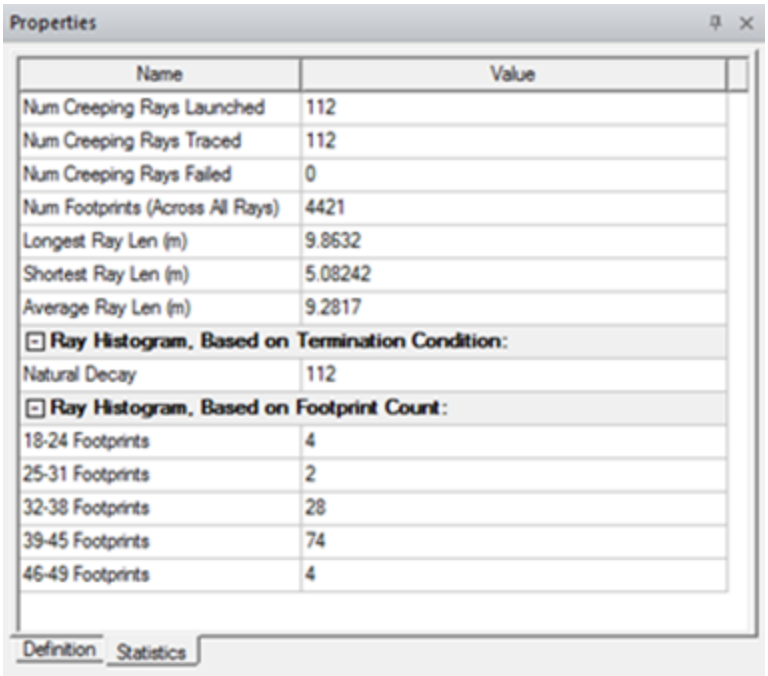
Creeping Wave Plot Properties

The Properties of each CreepingWave_Plot*n* item are the same as those created in the **Create Creeping Wave VRT Plot** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Each Creeping Wave plot has an additional read-only tabbed property page that displays Ray Statistics output from the ray generation process.

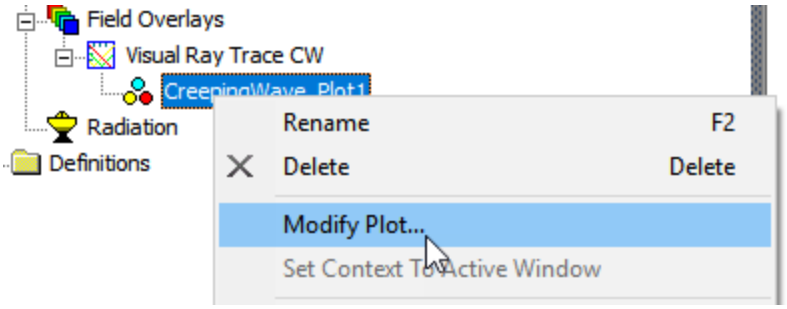


Modifying Creeping Wave Plots

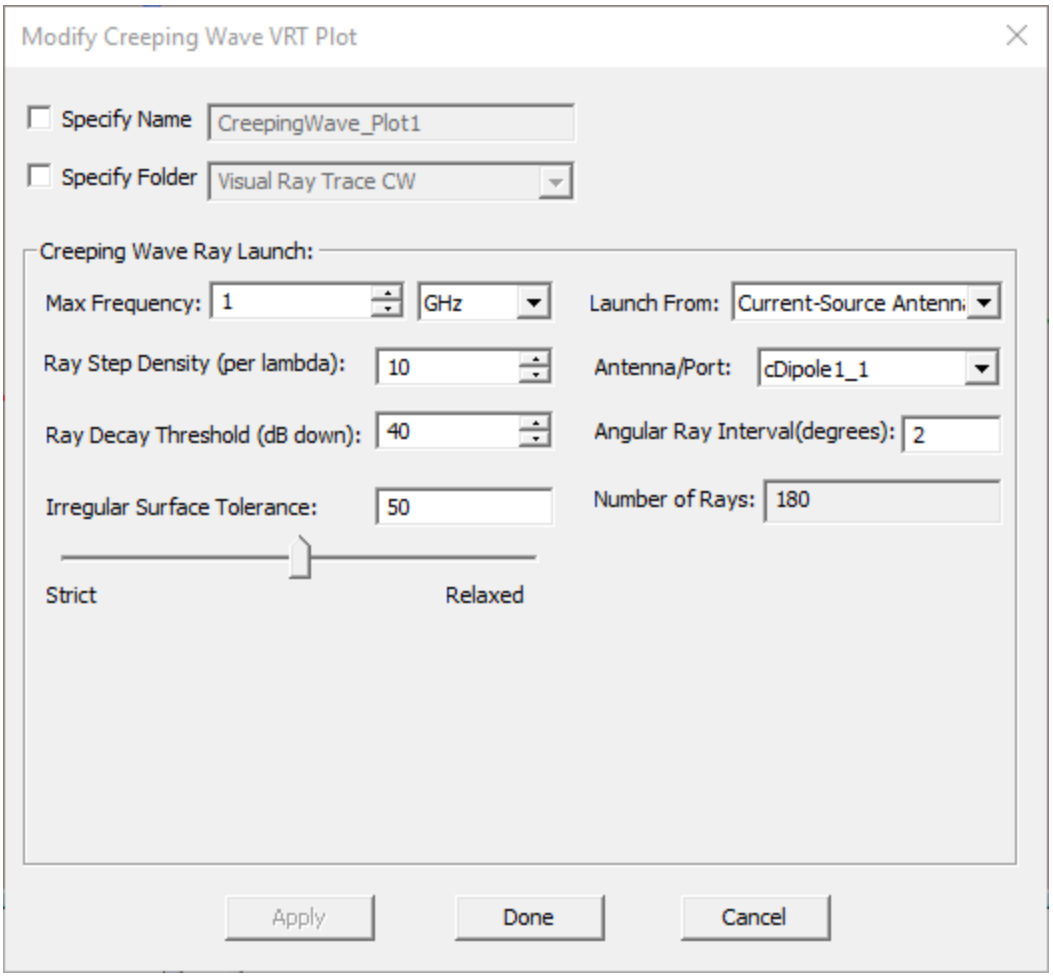
You can modify Creeping Wave properties directly in the **Properties** window to cause immediate updates to the plot data. You can also right-click the plot item and select **Modify**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot… to again bring up the **Create VRT Plot** dialog to make multiple changes and then commit them to the same, or a differently named plot.



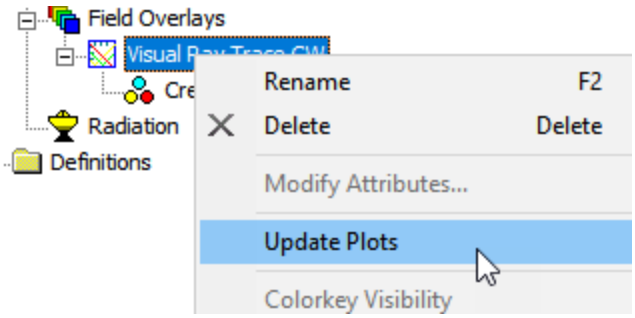
This opens a **Modify Creeping Wave VRT Plot** dialog.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

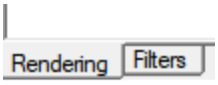
Design Edits that Invalidate the Mesh Invalidate Creeping Ray VRT Plots

Any design edits that invalidate the mesh (geometry delete, modify, or change in boundary condition assignment) also invalidates VRT plots. HFSS does not automatically regenerate the VRT plot after a design edit. You can update plots update by right-clicking on the **Visual Ray Trace SBR** folder and selecting **Update Plots**.



Visual Ray Trace CW Folder Properties

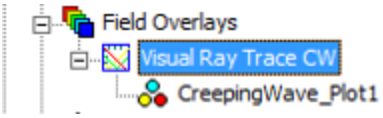
All the render and filter parameters appear as Properties of the parent **Visual Ray Trace CW** folder. Each **Visual Ray Trace CW** plot folder provides a grouping of render/filter attributes. Much like setting specific plot attributes like 2D plot extents, scaling, colormaps, etc., the render and filter parameters for all VRT plots are performed at this folder level. The different categories of render and filter operations are kept as separate tabs for: Rendering and Filters.



All render and filter property changes are immediate and will automatically refresh the 3D modeler window

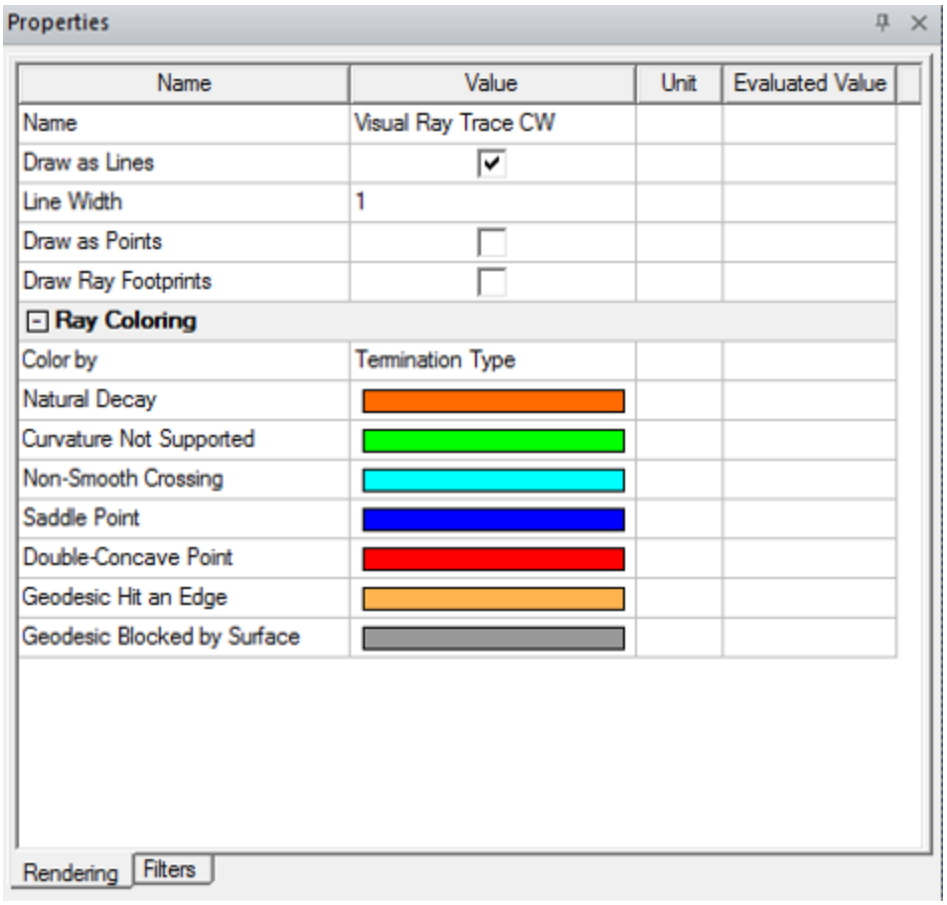
Rendering Properties Tab

To change the render and filter settings applied to the VRT plot, the plot folder containing the plot item may be selected to access the Plot Folder properties for all plots contained in the folder.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

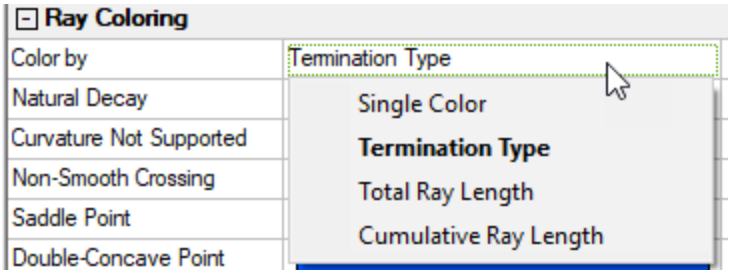
The **Rendering** tab properties control which items are drawn for each ray track, as well as width and shadow boundary controls:



Render properties include being able to select display of lines, points, and footprints, and with Color-by options for Single Color, Termination Type, Total Ray Length, and Cumulative Ray Length.

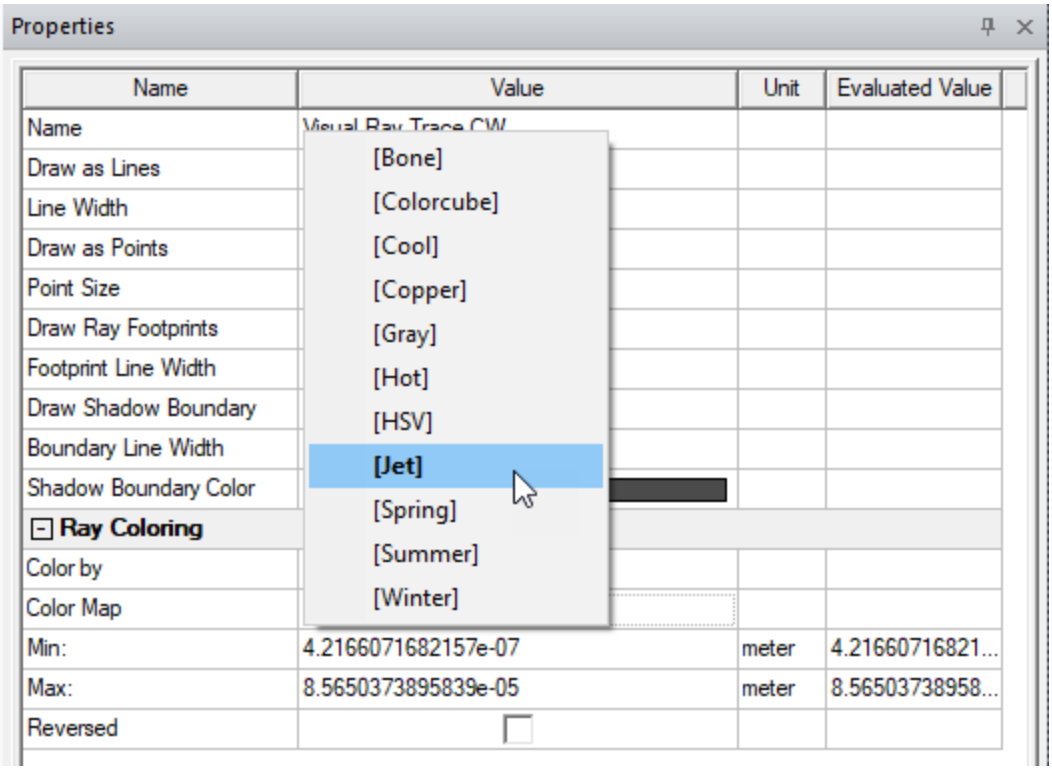
Ray Coloring controls:

- Color by: Can be Single Color, Termination Type, Total Ray Length, or Cumulative Ray Length.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

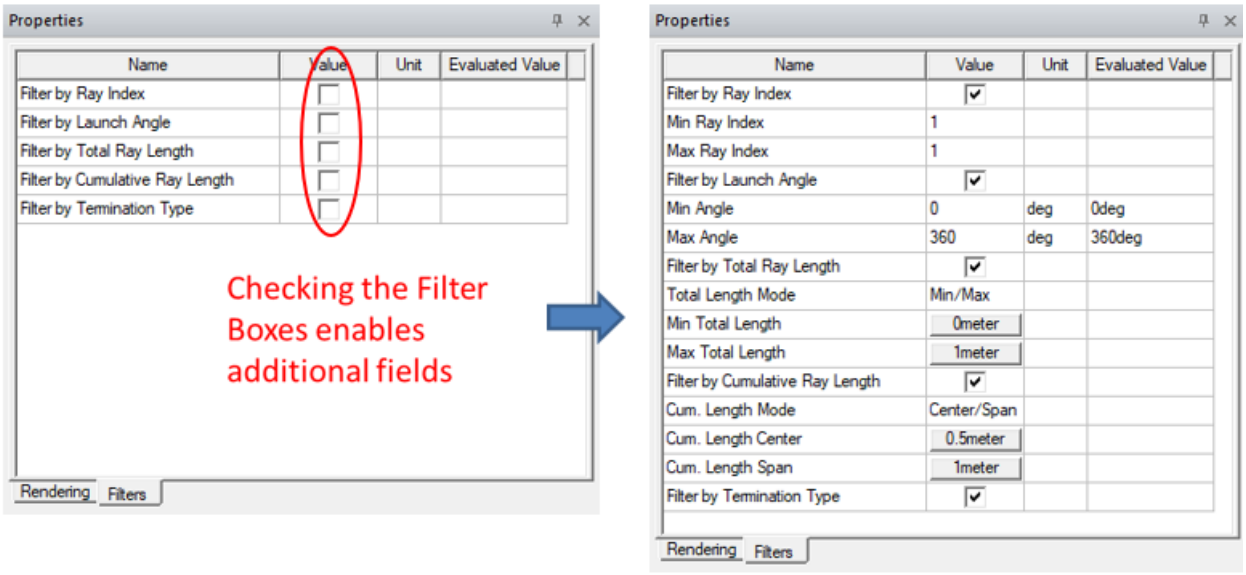
- If you select Termination Type, the terminations listed depend on those generated by the plot.
- If you select Single Color, the properties are Line Color, Point Color, and Footprint Color.
- If you select Total Ray Length or Cumulative Ray Length, the properties are Color Map (which offers many choices, shown below), Min, Max, and Reversed.



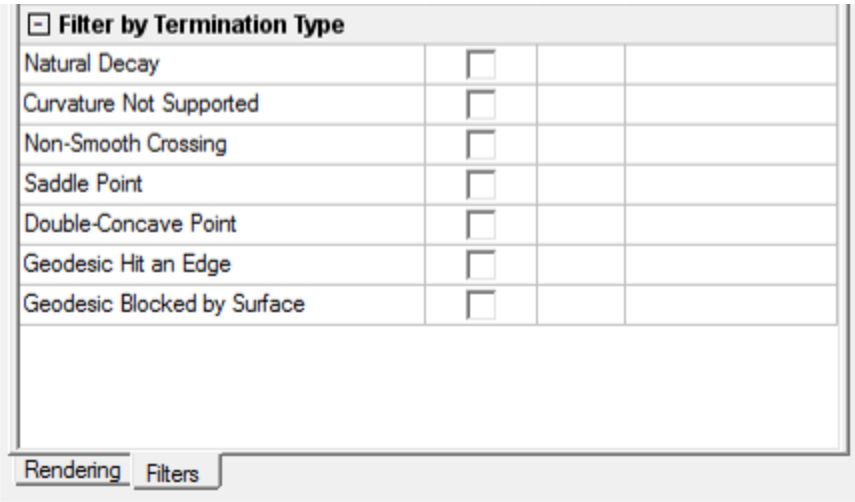
Filter Properties Tab

The Filter properties tab controls which ray tracks and/or bounces are shown in the 3D modeler window. Filter by Launch Angle is unique to CW for antennas (vs. CW for RCS).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Selecting any of Launch Angle, Total Ray Length, Cumulative Ray Length opens Min and Max parameters for that filter. Selecting Termination Type lists the terminations available in the model.



Creeping Wave for Antenna Placement on Curved Surfaces in SBR+ Simulation

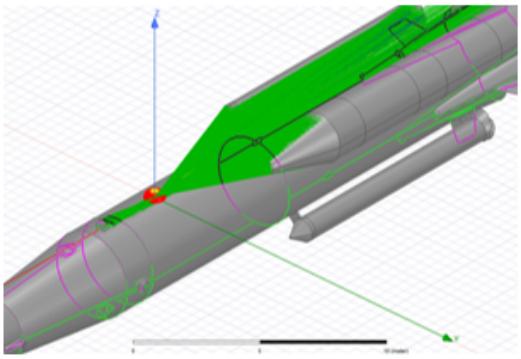
An antenna produces a creeping wave when in close proximity to curved objects with smooth surfaces. The physics of the near-field antenna CW can be modeled with SBR+ via creeping rays that travel along the geodesics of the surface, and can provide more accurate results when

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

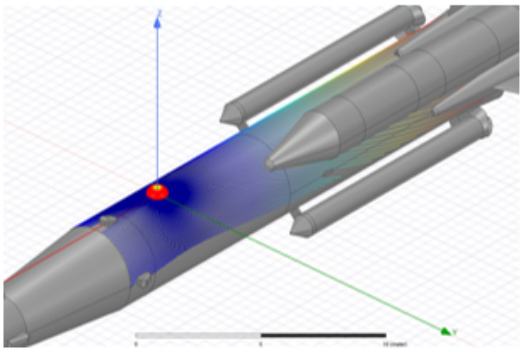
used with SBR+ advanced physics than a purely physical optics (PO) or SBR based approach. This SBR+ functionality is available in AEDT for both HFSS-SBR+ simulation configuration and for visual ray tracing (VRT) of these near-field CW antenna rays.

Key limitation: CW physics formulations for transmitting antennas are limited to current based antennas. Further, the antenna must be placed in close proximity to a metallic surface of the SBR+ region. This capability is therefore limited to near-field HFSS sources such as FEBI regions, IE regions, and parametric current sources with a built-in ground plane. Near-field link antennas in the SBR+ solution type must also have a finite or infinite ground plane. The interface enforces this limitation by not allowing attempts to deploy CW with far-field source representations (e.g., far-field links, far-field parametric antennas).

The VRT is an essential tool for tuning CW parameters and diagnosing problems. The figure shows a VRT. The CW rays travel along the surface starting under the antenna at its closest point to the platform. Users are **strongly encouraged** to perform a VRT before running the simulation to help ensure the rays are propagating as expected. A VRT example is show below, first with SBR rays alone and then with CW rays alone.



VRT: SBR Rays



VRT: Creeping Rays

When you use the Creeping Wave feature, before you simulate the design, you must set parameters in the Hybrid Regions and in the Solution Setup.

Prerequisites

Create an HFSS design with SBR+ regions, either driven model or terminal solution type or SBR+ solution type.

Creeping wave simulations require FEBI regions, IE regions, and parametric current sources with a built-in ground plane. Near-field link antennas in the SBR+ solution type must also have a finite or infinite ground plane. The interface enforces this limitation by not allowing attempts to

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

deploy CW with far-field source representations (e.g., far-field links, far-field parametric antennas).

Process Flow for Setting up and Simulating an SBR+ Design for Creeping Waves for Antenna Placement

1. Create an HFSS design with SBR+ region (either driven modal solution type or SBR+ solution type).
2. Add scene geometry components that have curvature (spheres, cylinders, curved surfaces, etc.).

These can be either light weight geometries, or conventional Modeler geometries.

Creeping wave deployment depends on surface curvature information. For curved-surface geometry that is imported or that is synthesized from HFSS primitives, make sure curvilinear meshing is enabled. Also, the highest mesh resolution is recommended for best results.

For imported polygonal geometry (e.g., triangle meshes), surface curvature will be extracted by the solver. In general, do not disable curvature extraction on surfaces where you wish creeping rays to travel. There is one exception to this guidance: if a surface in an imported polygonal geometry is known by you to actually represent a flat surface in the intended model, it is better to disable curvature extraction on that face. This will force the face to be explicitly flat during SBR+ simulation, and CW rays can still launch on and propagate over perfectly flat surfaces. Otherwise, curvature extraction will estimate the curvature on that surface based on the bending of adjoining faces, and this can lead it to assign a non-zero curvature that is not intended.

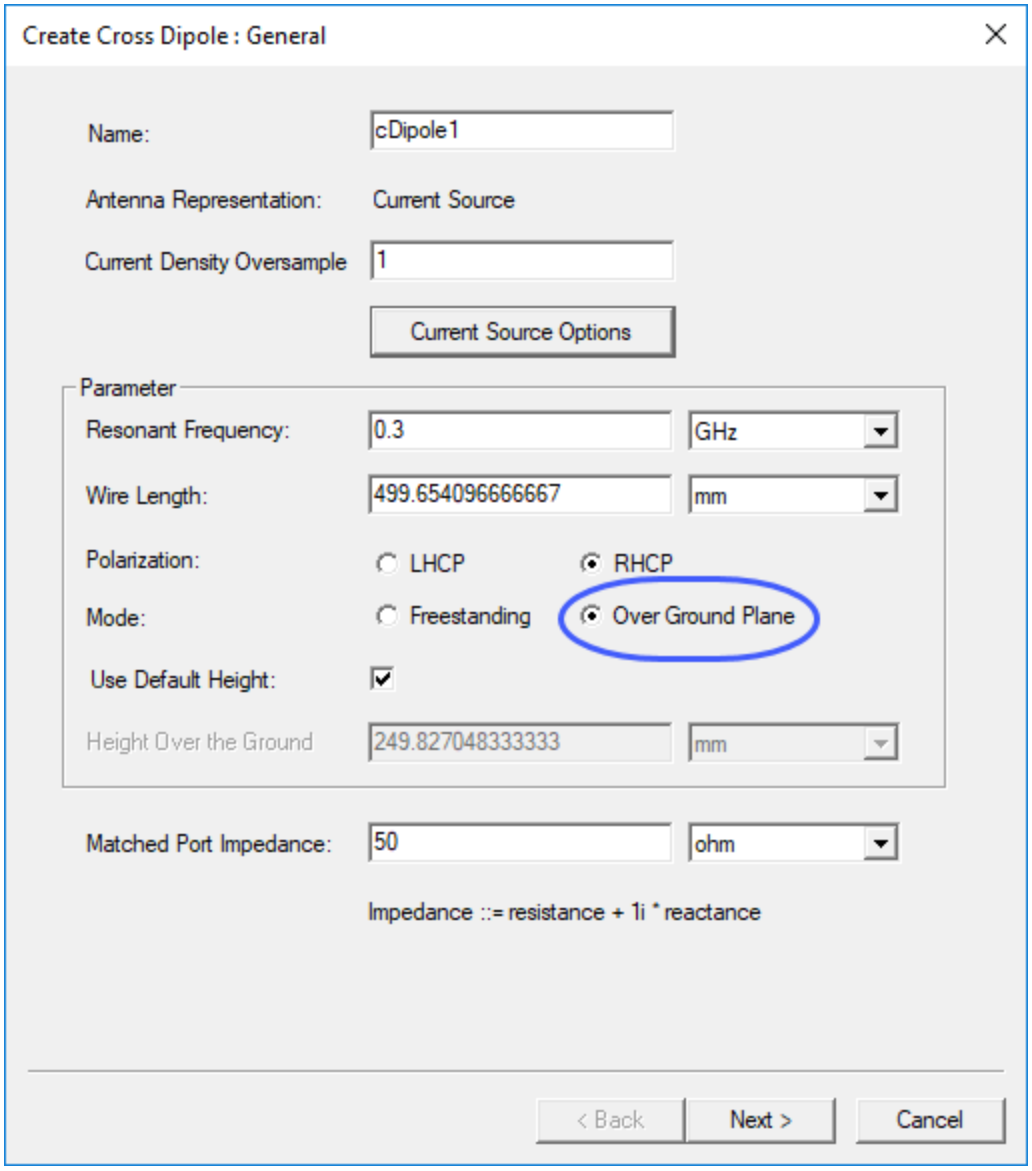
You can specify how SBR+ curvature extraction is to be handled per geometry component/part. By default, curvature extraction is applied on all SBR+ faces in the design unless overridden by the application of curvilinear meshing. You can disable curvature extraction on selected faceted faces or objects by using this mesh operation.

When object curvature information is available in the original geometry, the Apply Curvilinear Elements option may be used. This option avoids the curvature extraction approximation by passing curvature information directly to the SBR+ solver, which is more accurate.

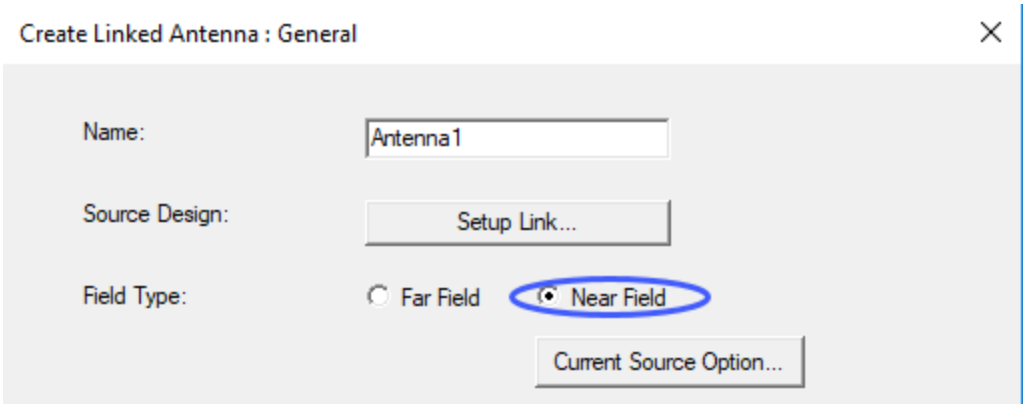
3. Add an antenna with a current-source (near-field) representation, placing the antenna on a metallic surface of the SBR+ region

For SBR+ solution type add a 3D Component for a parametric current-source antenna with a built-in ground plane, or for a near-field linked antenna with a ground plane.

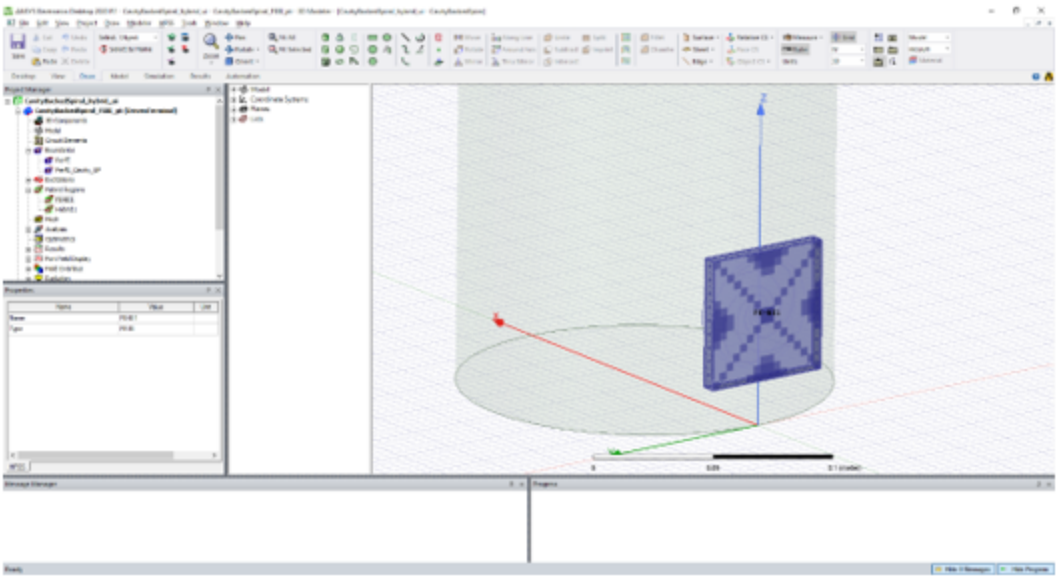
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For a hybrid HFSS-SBR+ simulation (Driven Modal or Driven Terminal) the only supported way to generate and use current sources for VRT is by synthesizing a representative current source for a FE-BI hybrid boundary. The current-source synthesis is done automatically and behind-the-scenes (hidden from user) based on the geometry of the FEBI surface.

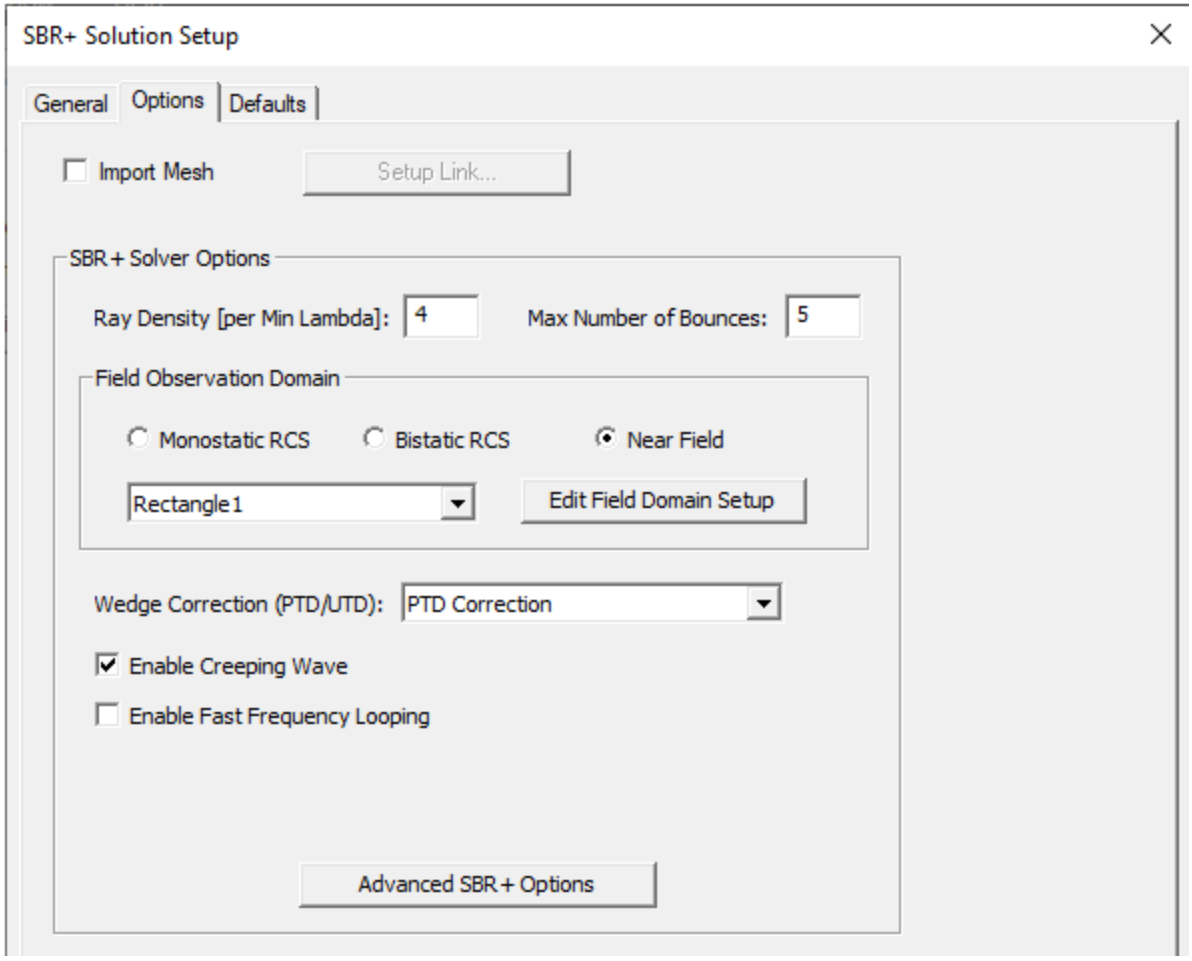


FE-BI hybrid region current-source antenna

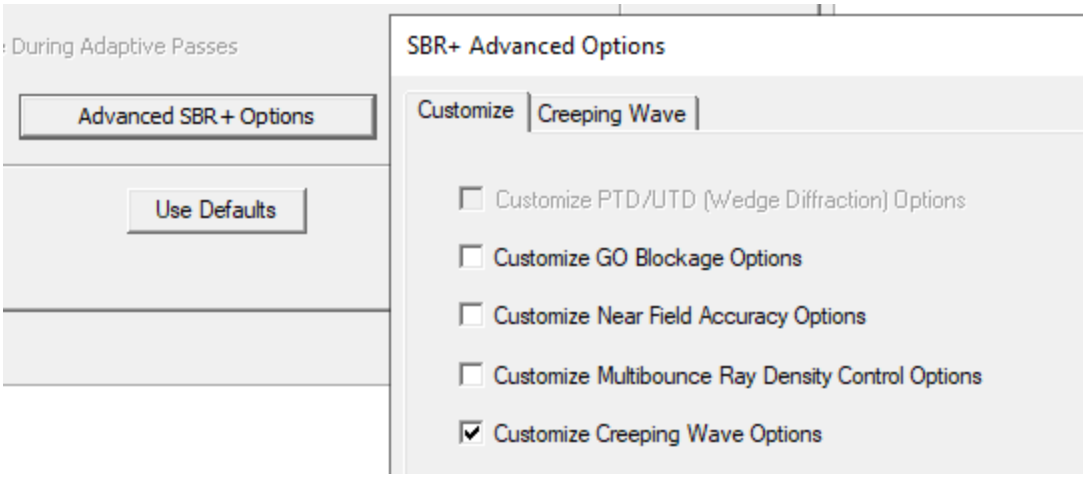
4. Add a Solution setup and in the **SBR+ Solution Setup**, **Options** tab or or **HFSS with Hybrid and Arrays** driven modal solution setup **Hybrid** tab select **Enable Creeping Wave**. (Note that this option is only available in the solution setup dialog if a near-field link to an HFSS source such as a FEBI or IE region excitation/boundary is present in the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

design.)

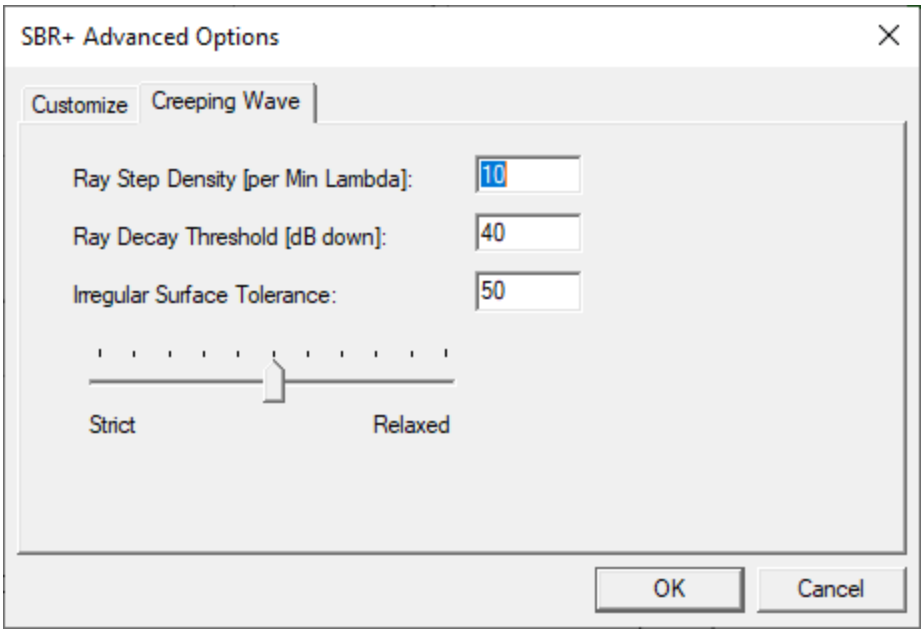


Select **Advanced SBR+ Options** and in the **SBR+ Advanced Options** dialog check Customize Creeping Wave Options to enable the **Creeping Wave** tab..



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select the **Creeping Wave** tab to set parameters.



Ray Step Density controls the longitudinal sampling along the direction of propagation.

Ray Decay Threshold controls how far the creeping rays propagate by terminating each ray once its power density decay (defined as dB down from its level at its starting point at the shadow boundary) exceeds this threshold.

Angular Ray Interval controls the density of rays that are generated along the nearest surface, starting under the current source, and spanning the 360 degrees around the current source (note the Number of Rays is a read-only field that is internally computed by dividing 360 degrees by the Angular Ray Interval).

Irregular Surface Tolerance is a single control for a collection of tolerances that determine creeping rays' sensitivities to local surface irregularities that theoretically should block their further propagation. Moving the slider toward **Relaxed** allows creeping rays to propagate through small surface irregularities that may arise through defects in the input geometry or due to limitations of curvature extraction accuracy. However, setting this tolerance too relaxed may allow creeping waves to pass through concave surfaces, saddle surfaces, or wedges, which are not supported by creeping wave physics. Therefore, you are strongly encouraged to perform a creeping-wave visual ray trace (VRT) to determine an acceptable setting for this tolerance, which will differ from model to model. The intention is that creeping rays only propagate over the convex portions of surfaces, and some user judgment is required in correctly setting the **Irregular Surface Tolerance** for each situation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Run simulation.

You should see the progress bar reporting both SBR and Creeping Wave ray shoots during that part of the run. The solution profile will indicate how many antennas successfully deployed CW rays. CW rays may fail deploy for some antennas because they are too far away from the platform or some other problem with the placement surface underneath. The Messages panel will provide more detailed information on which antennas CW successfully deployed, on which it failed, and why.

6. Generate VRT Creeping Wave plots.

This step is optional, and as noted earlier, it is usually more important to perform creeping wave VRT before simulation to verify that CW rays will propagate where intended and make adjustments to the creeping wave settings to ensure this.

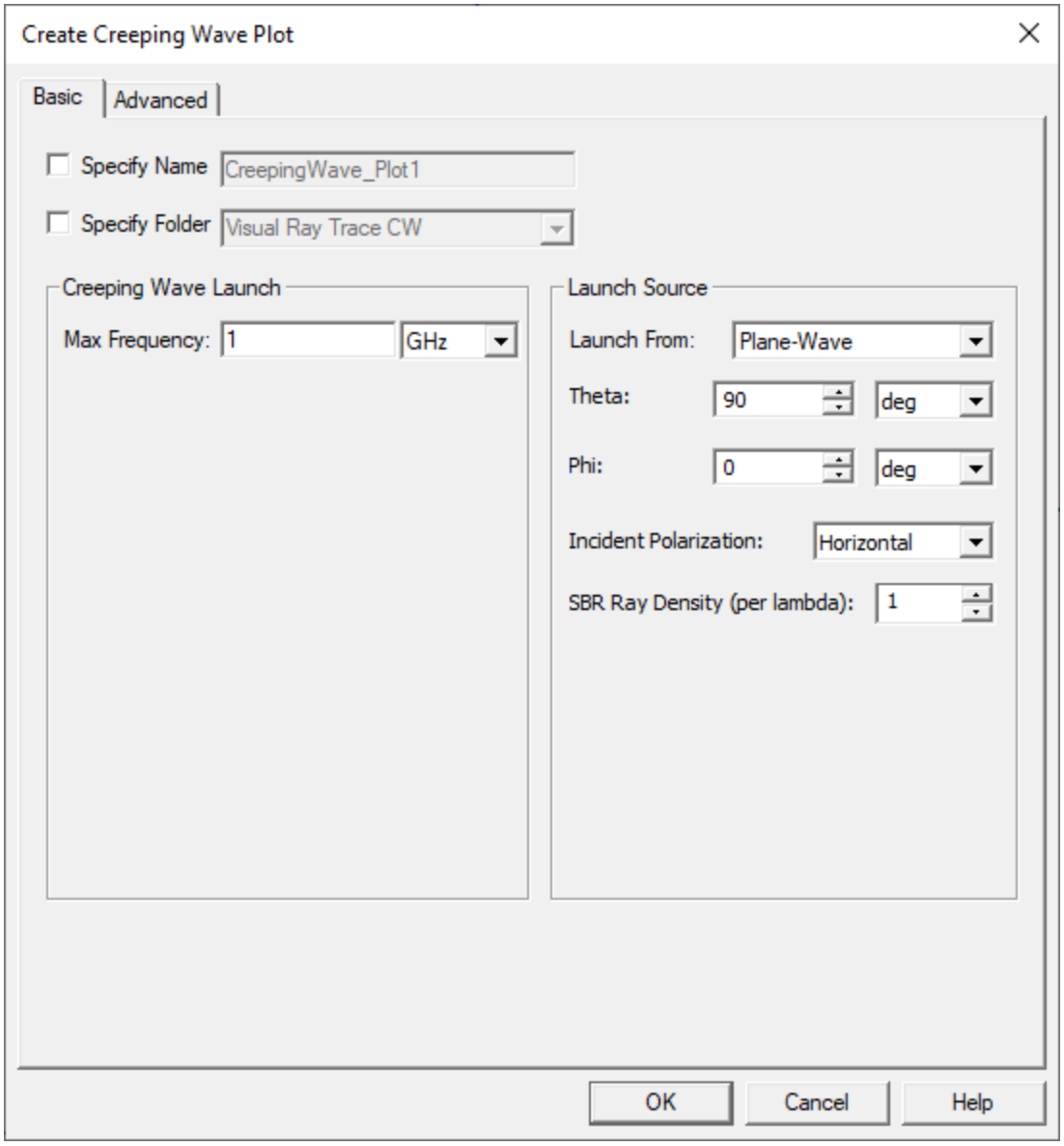
Using Visual Ray Trace (VRT) with Creeping Wave for Antenna Placement

1. Once you have an appropriate HFSS design, there are three different ways in the user interface to create a new Visual Ray Trace Plot with Creeping Rays:

- Right-click the **Field Overlays** icon in the **Project Tree** and select **Plot VRT...>Creeping Wave** from the short cut menu.
- Right-click the 3D Modeler window and select **Plot VRT...>Creeping Wave** from the pop-up menu.
- Click **HFSS>Fields>Plot VRT...>Creeping Wave**

After you use one of these approaches, the **Create Creeping Wave VRT Plot** dialog displays. From the **Launch From**: drop down menu, select the Current-Source antenna source and then select the desired **Antenna/Port** for launching creeping waves. This will change some dialog fields, from the Plane Wave configuration, as follows.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To **Specify Name** check the box to enable the text field. To **Specify Folder**, check the box to enable the selection menu.

2. Set the **Creeping Wave Ray Launch** parameters

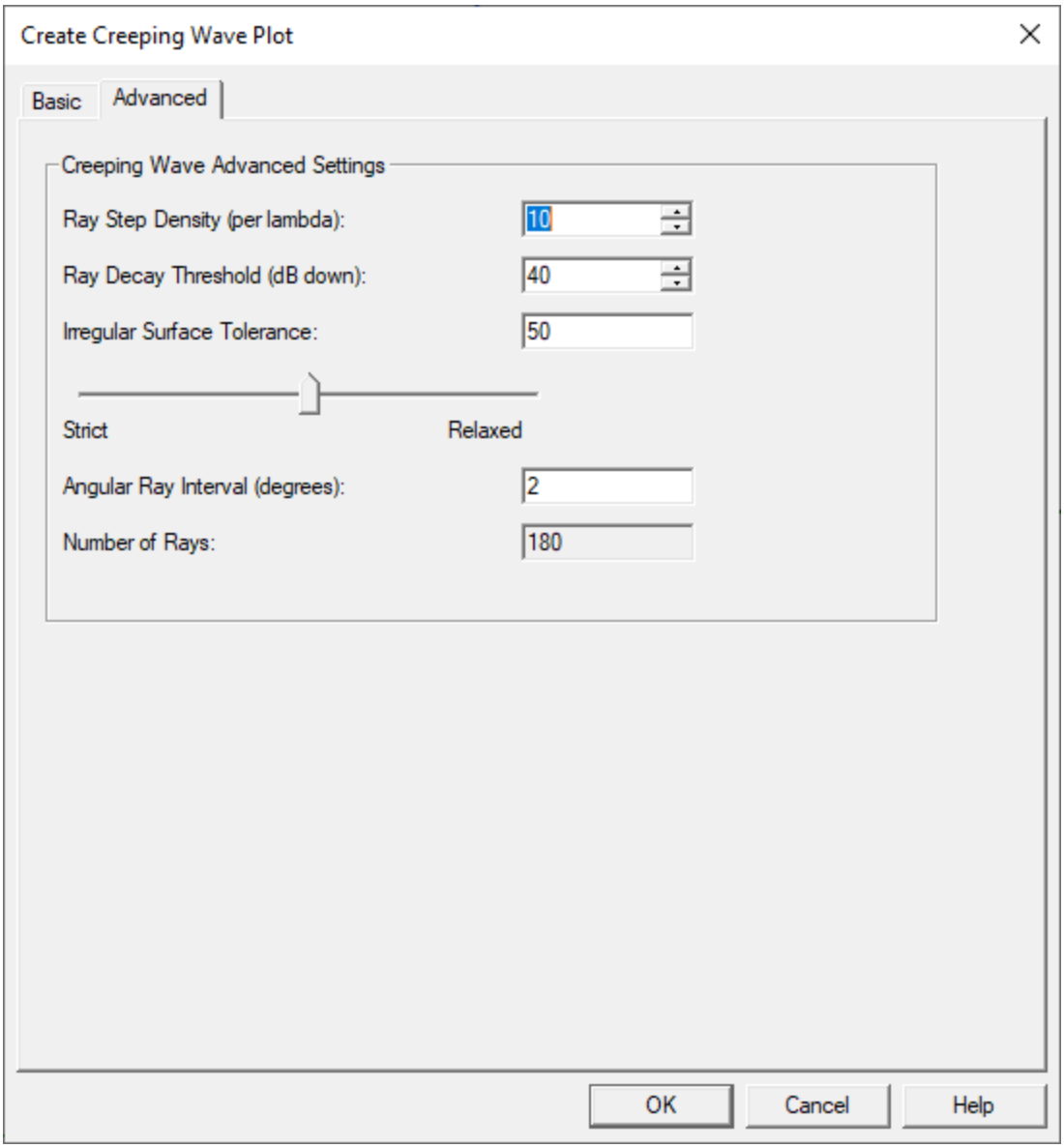
- **Launch from** parameters includes a menu for selecting the source for Antenna placement Creeping Wave, the near field parametric antenna or Near Field link you specified.
- **Max Frequency** should be set to the maximum frequency desired for the simulation. For simulations spanning multiple frequencies, the creeping wave settings should be tested at the highest simulation frequency.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Antenna /Port** lists available antennas/ports that can be represented by current sources.
- **Angular Ray Interval (degrees)** controls the density of rays that are generated along the nearest surface, starting under the current source, and spanning the 360 degrees around the current source
- **Number of Rays** indicates how many CW rays will be launched. This is a read-only field that is internally computed by dividing 360 degrees by the **Angular Ray Interval**.

Caution: The settings in this diagram are used by the VRT to test if the creeping waves are propagating as expected. **These parameters are not automatically transferred to the Creeping Wave Settings for simulation. You need to set the simulation Creeping Wave Settings manually.**

3. Option select the **Advanced** tab.

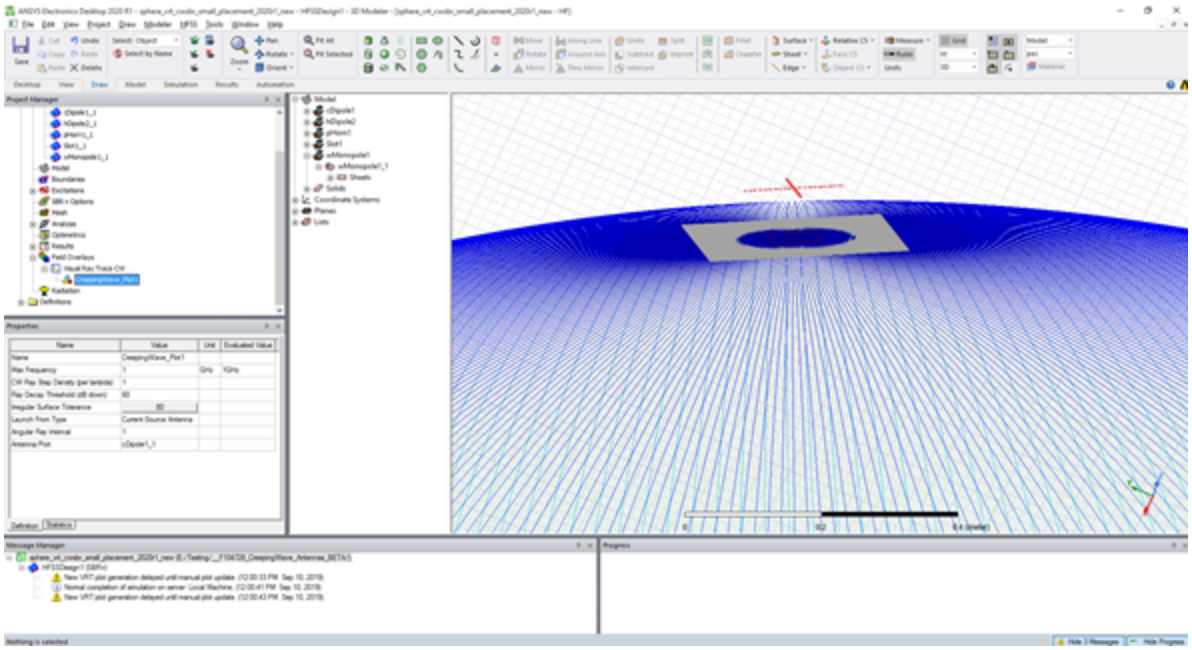


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

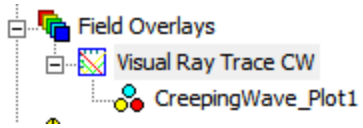
- **Ray Step Density (per lambda)** controls longitudinal sample along the direction of propagation along the surface.
- **Ray Decay Threshold** controls how far the creeping rays propagate by terminating each ray once its power density decay (defined as dB down from its level at its starting point at the shadow boundary) exceeds this threshold.
- **Irregular Surface Tolerance** is a single control for a collection of tolerances that determine creeping rays' sensitivities to local surface irregularities that theoretically should block their further propagation. Moving the slider toward **Relaxed** allows creeping rays to propagate through small surface irregularities that may arise through defects in the input geometry or due to limitations of curvature extraction accuracy. However, setting this tolerance too relaxed may allow creeping waves to pass through concave surfaces, saddle surfaces, or wedges, which are not supported by creeping wave physics. The intention is that creeping rays only propagate over the convex portions of surfaces, and some user judgment is required in correctly setting the Irregular Surface Tolerance for each situation.
- **Angular Ray Interval (degrees)** controls the density of rays that are generated along the nearest surface, starting under the current source, and spanning the 360 degrees around the current source.
- **Number of Rays** indicates how many CW rays will be launched. This is a read-only field that is internally computed by dividing 360 degrees by the **Angular Ray Interval**

3. Click **OK** to dismiss the dialog and run the VRT generation to produce SBR+ Creeping ray data. Only the geometries contained in the SBR+ regions are used for the VRT ray generation.
4. The "Shoot Filter" selections are disabled in the dialog box, and moved to the **Visual Ray Trace CW** folder properties as described below.
5. After the plot is generated, you can use the plot properties to update or change the appearance of the plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A new “Visual Ray Trace CW” plot folder is created under Field Overlays, which contains one or more “CreepingWave_Plot” items.

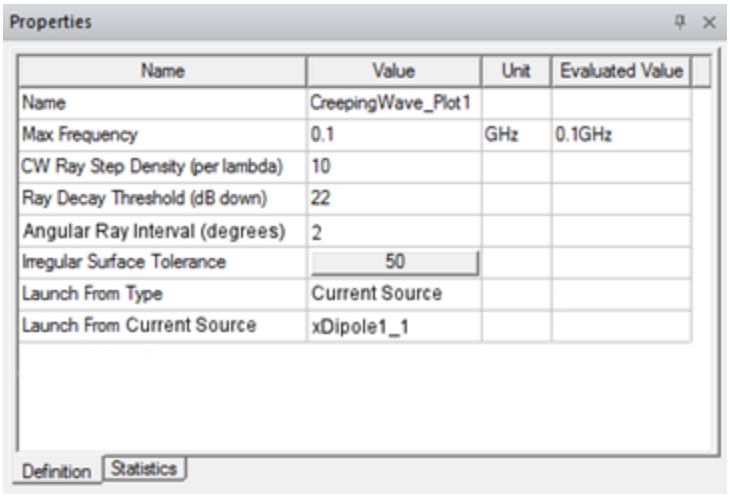


If you run **Plot VRT...>Creeping Wave** again, that creates an additional CreepingWave_Plot item under the **Visual Ray Trace CW** folder, or a different folder name if you specify one in the **Create Creeping Wave VRT Plot** dialog.

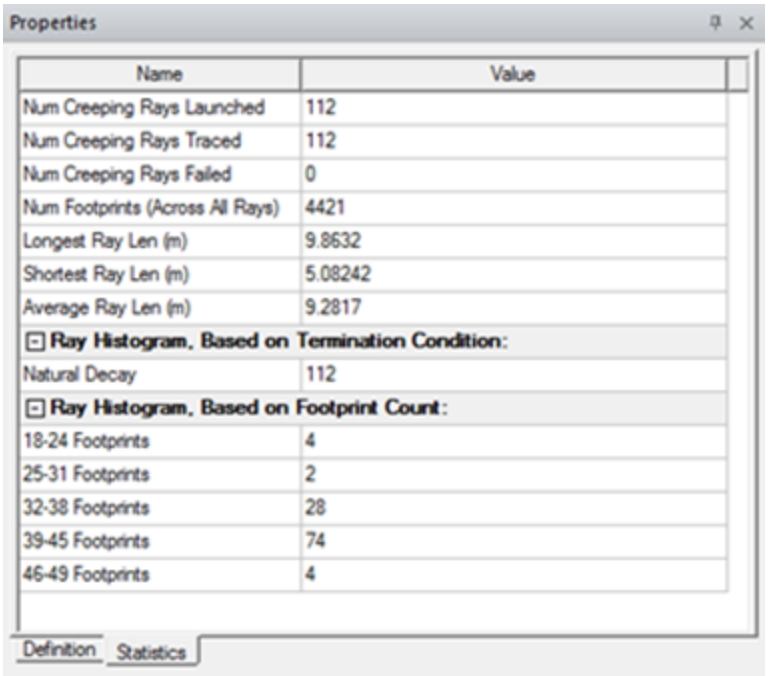
Creeping Wave Plot Properties

The Properties of each CreepingWave_Plot*n* item are the same as those created in the **Create Creeping Wave VRT Plot** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Each Creeping Wave plot has an additional read-only tabbed property page that displays Ray Statistics output from the ray generation process.

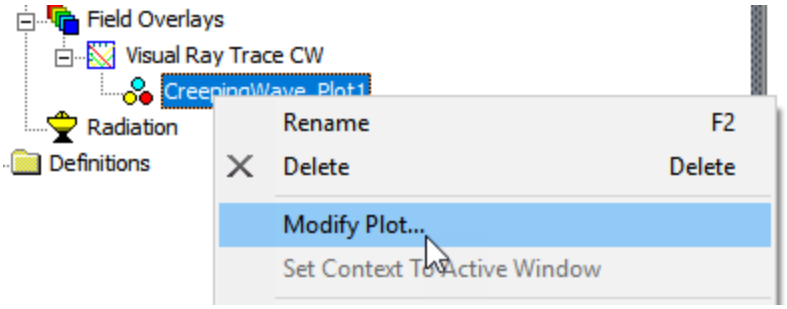


Modifying Creeping Wave Plots

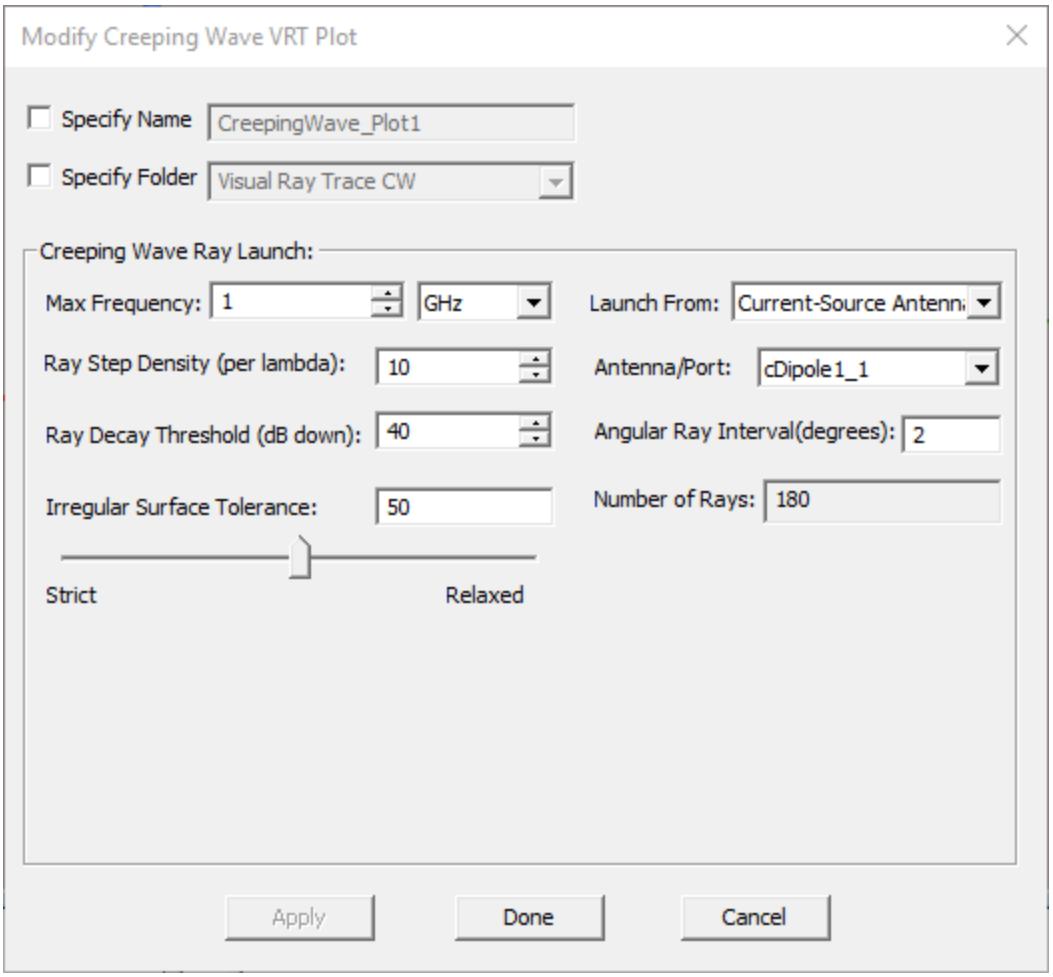
You can modify Creeping Wave properties directly in the **Properties** window to cause immediate updates to the plot data. You can also right-click the plot item and select **Modify**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot… to again bring up the **Create VRT Plot** dialog to make multiple changes and then commit them to the same, or a differently named plot.



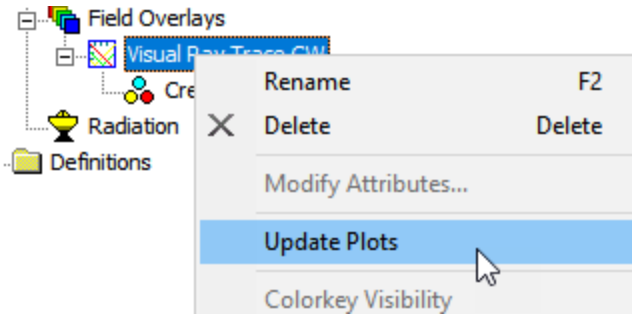
This opens a **Modify Creeping Wave VRT Plot** dialog.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

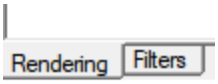
Design Edits that Invalidate the Mesh Invalidate Creeping Ray VRT Plots

Any design edits that invalidate the mesh (geometry delete, modify, or change in boundary condition assignment) also invalidates VRT plots. HFSS does not automatically regenerate the VRT plot after a design edit. You can update plots update by right-clicking on the **Visual Ray Trace SBR** folder and selecting **Update Plots**.



Visual Ray Trace CW Folder Properties

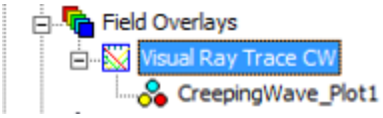
All the render and filter parameters appear as Properties of the parent **Visual Ray Trace CW** folder. Each **Visual Ray Trace CW** plot folder provides a grouping of render/filter attributes. Much like setting specific plot attributes like 2D plot extents, scaling, colormaps, etc., the render and filter parameters for all VRT plots are performed at this folder level. The different categories of render and filter operations are kept as separate tabs for: Rendering and Filters.



All render and filter property changes are immediate and will automatically refresh the 3D modeler window

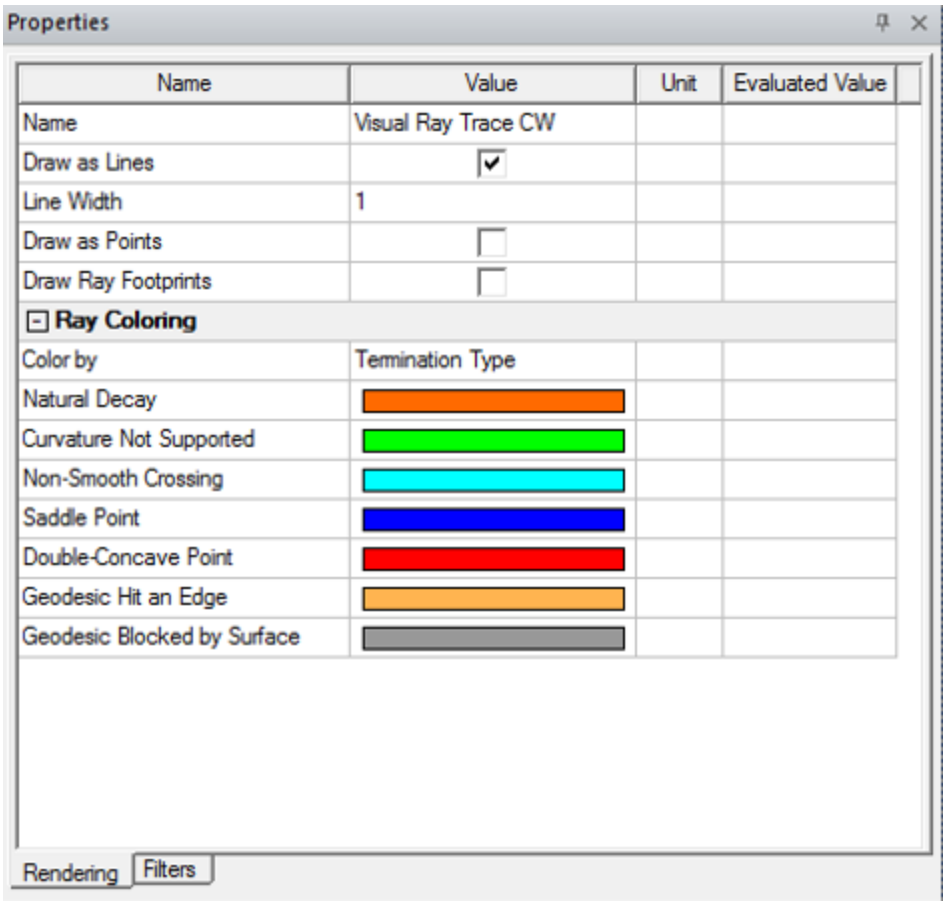
Rendering Properties Tab

To change the render and filter settings applied to the VRT plot, the plot folder containing the plot item may be selected to access the Plot Folder properties for all plots contained in the folder.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

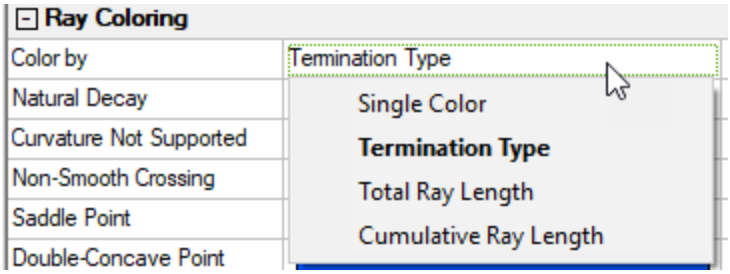
The **Rendering** tab properties control which items are drawn for each ray track, as well as width and shadow boundary controls:



Render properties include being able to select display of lines, points, and footprints, and with Color-by options for Single Color, Termination Type, Total Ray Length, and Cumulative Ray Length.

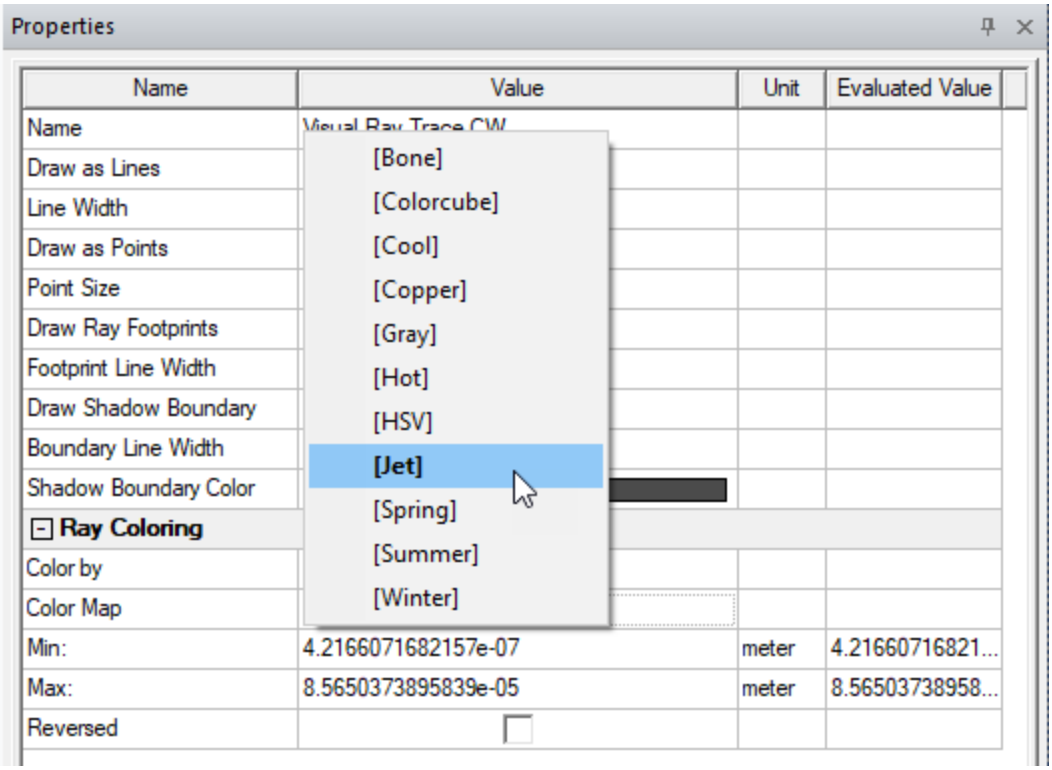
Ray Coloring controls:

- Color by: Can be Single Color, Termination Type, Total Ray Length, or Cumulative Ray Length.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

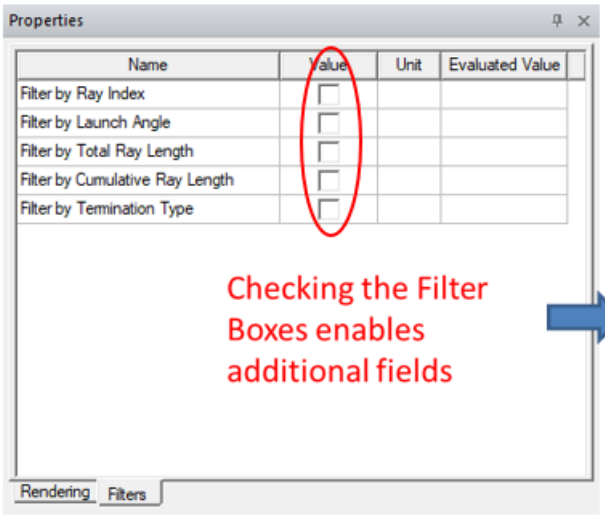
- If you select Termination Type, the terminations listed depend on those generated by the plot.
- If you select Single Color, the properties are Line Color, Point Color, and Footprint Color.
- If you select Total Ray Length or Cumulative Ray Length, the properties are Color Map (which offers many choices, shown below), Min, Max, and Reversed.

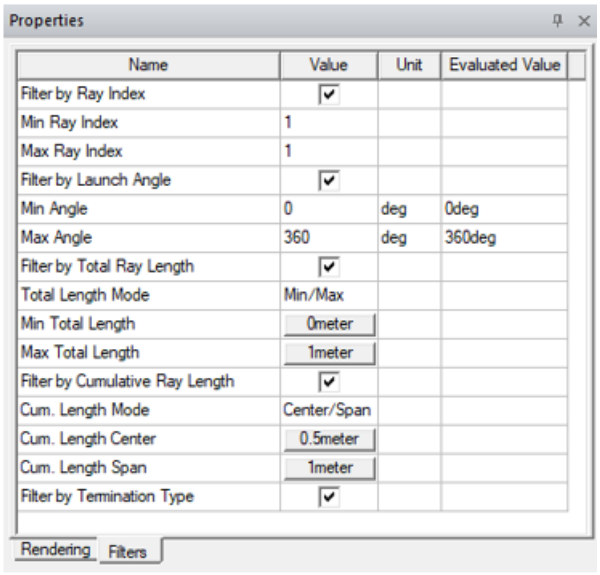


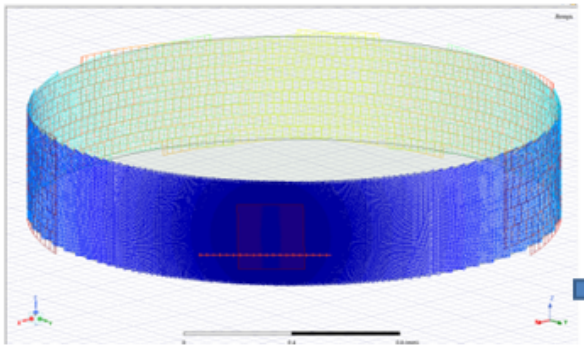
Filter Properties Tab

The Filter properties tab controls which ray tracks and/or bounces are shown in the 3D modeler window. Filter by Launch Angle is unique to CW for antennas (vs. CW for RCS).

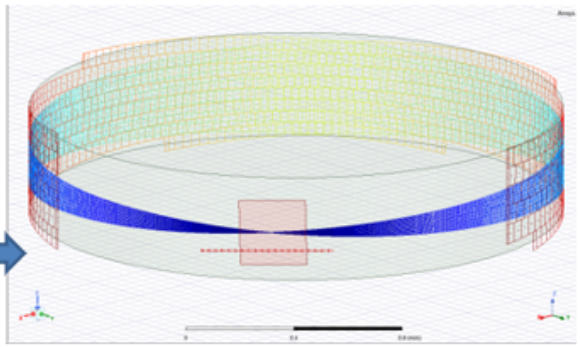
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



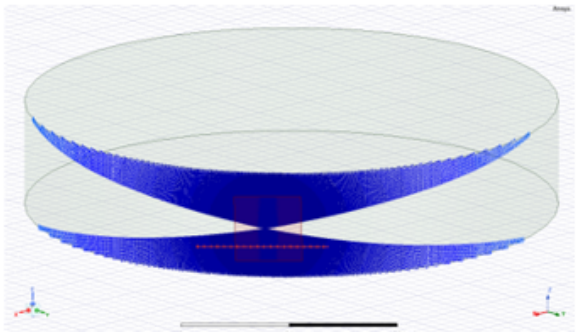


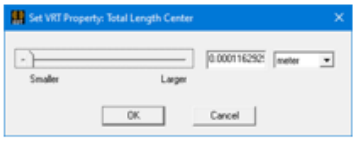


Apply a Total Ray Length Filter



Modify the Properties of the Filter with sliders and Analyze the Results

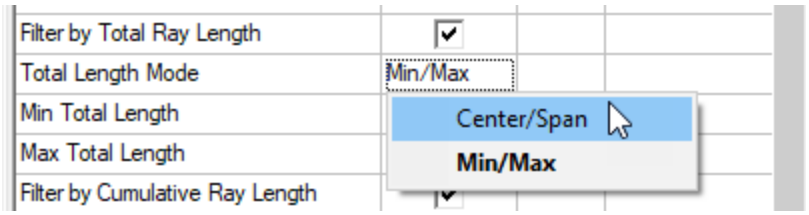




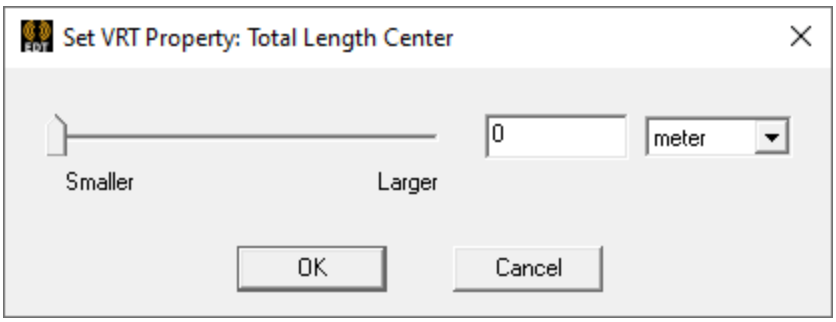
Selecting any of Ray Index, Launch Angle, Total Ray Length, opens Min and Max parameters for that filter.

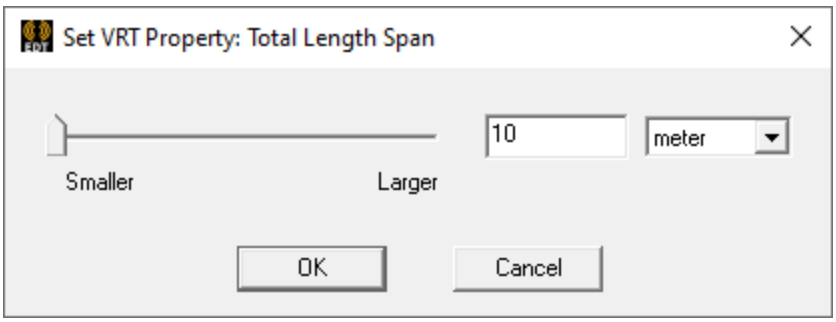
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Filter By Total Ray Length: use Total Length Mode of either Center/Span or Min/Max. You can assign length values as a variable. Using a variable permits you to creating an animation by sweeping the variable.



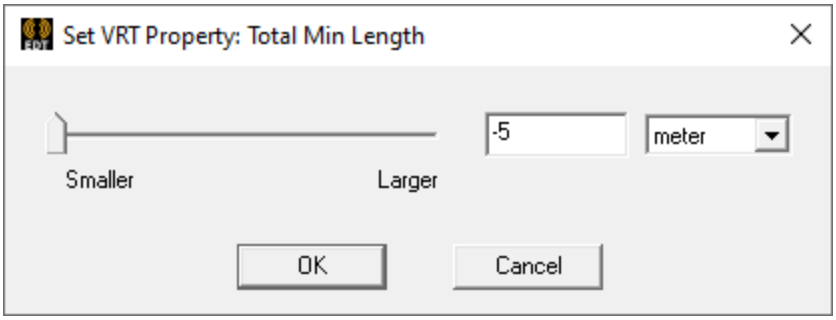
Selecting Center/Span enables buttons for **Total Length Center** and **Total Length Span**. Clicking one of these buttons opens Slider window that allows you to modify the filter values and see in real-time the changing rays in the model view window.

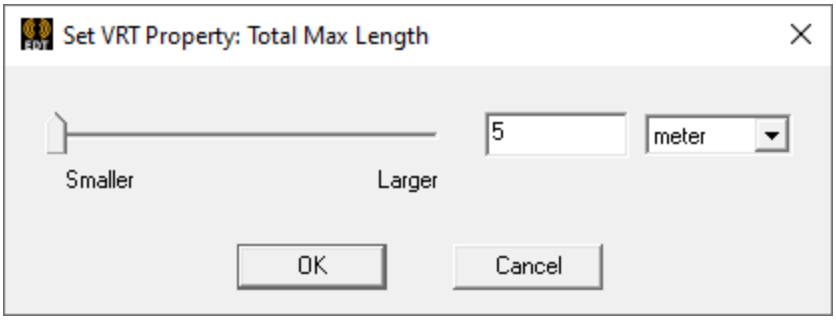




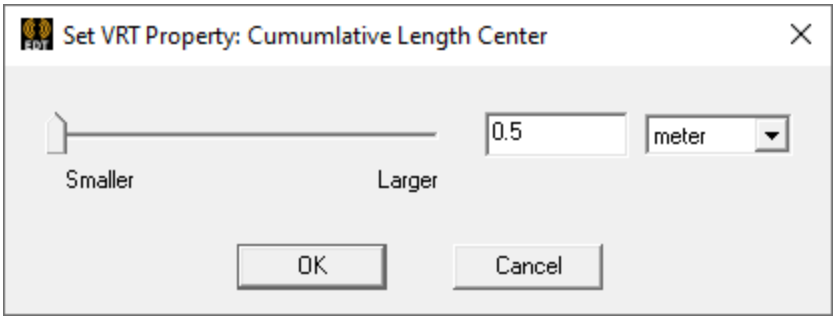
Selecting Min/Max enables buttons for Total Min Length and Total Max Length. Clicking a button opens a Slider window for real time control of the modeler window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

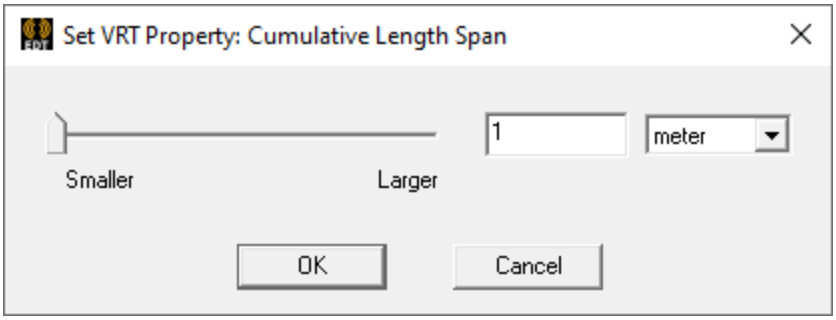




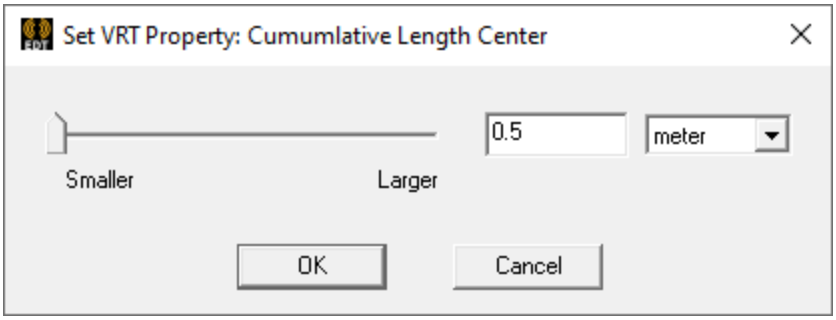
Filter By Cumulative Ray Length: use with the Center/Span or Min/Max selections. You can assign length variables as the value. Using a variable allows you to animate the plot by sweeping that variable. Selecting Center/Span enables buttons for **Cumulative Length Center** and **Cumulative Length Span**. Clicking one of these buttons opens Slider window that allows you to modify the filter values and see in real-time the changing rays in the model view window.

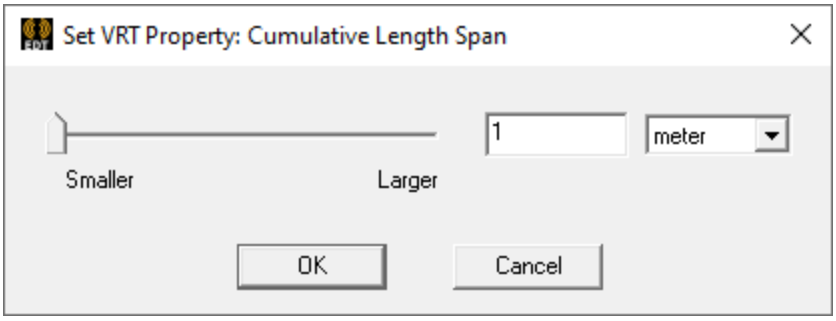


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



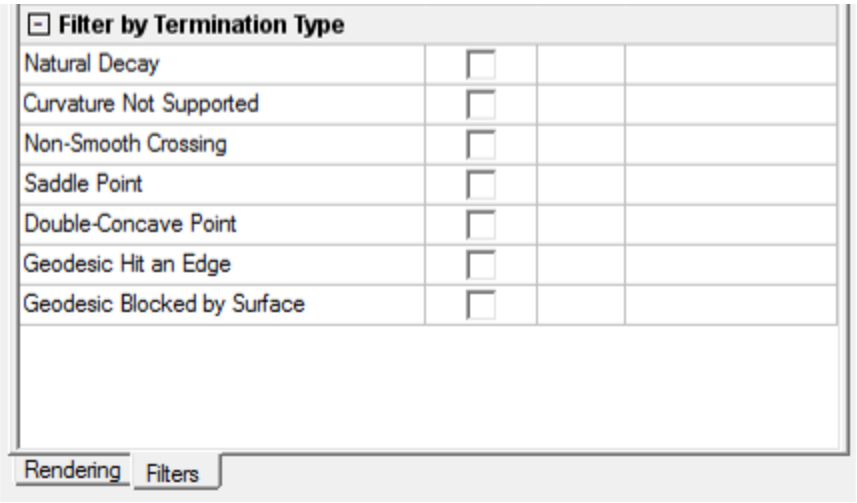
Selecting Min/Max enables buttons for Cumulative Length Center and Cumulative Length Span. Clicking a button opens a Slider window for real time control of the modeler window.





Selecting **Filter by Termination Type** lists the terminations available in the model.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Creeping Wave for RCS Analysis of Curved Surfaces in SBR+ Simulation

For advanced radar cross section (RCS) analysis of curved surfaces, creeping wave (CW) rays can provide more accurate results when used with SBR+ advanced physics than a purely PO or SBR based approach. This new SBR+ functionality is a feature in Ansys Electronics Desktop. Creeping waves can only be deployed on convex regions of curved PEC surfaces.

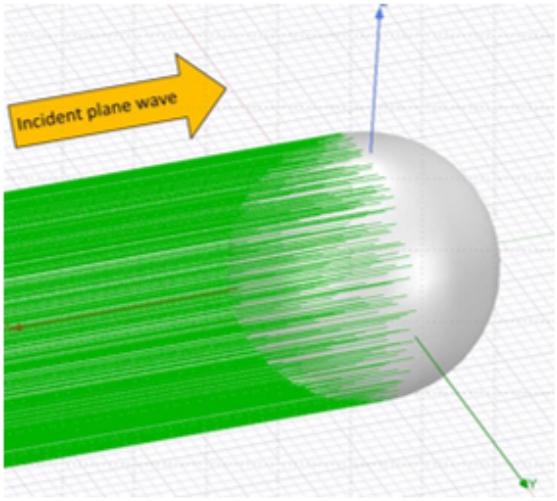
For simulations involving transmitting antennas, CW analysis can only be performed from either 1) a near-field link to an HFSS source such as a FEBI or IE region, or 2) one of the current source based parametric antennas, which are available in the SBR+ solution type. In either case, the antenna must also have an infinite PEC ground plane, and part of the antenna must be located close (distance less than λ/2) to the nearest PEC or PEC-like surface.

The latter requirement is why only near-field sources are supported. With transmitting antennas, creeping waves launch from the closest point on the surface below each current source.

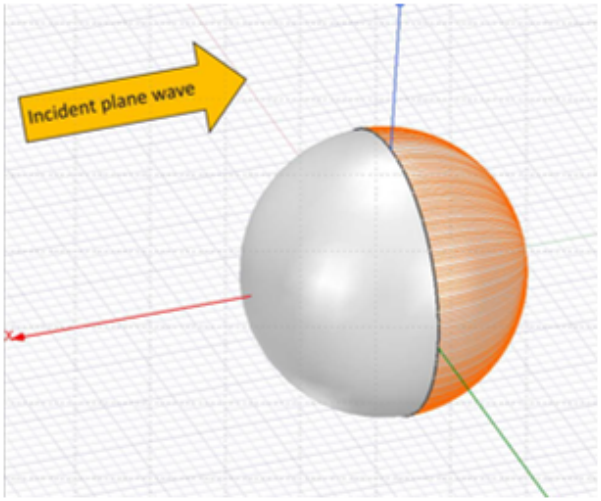
This feature includes a Visual Ray Trace (VRT) capability, which allows you to visualize creeping rays before running the simulation. **The VRT is an essential tool for tuning CW parameters and diagnosing problems**. The figure shows a VRT performed in the AEDT GUI. The CW rays travel along the surface starting from the shadow boundary, which divides the regions that are directly visible and shadowed with respect to the direction of incidence of the plane wave. Users are **strongly encouraged** to perform a VRT before running the simulation to help ensure the rays are propagating as expected

A VRT example is show below, first with SBR rays alone and then with plane wave CW rays alone.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

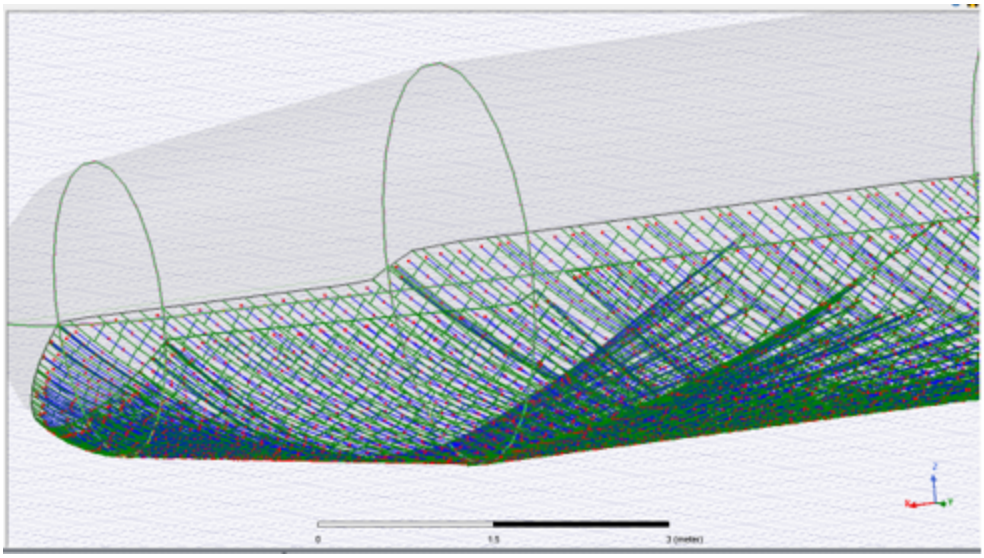


VRT with SBR alone



VRT with CW alone

When you use the Creeping Wave feature, before you simulate the design, you must set parameters in the Hybrid Regions and in the Solution Setup. After running a simulation, you can create RCS reports and create field plots of creeping rays.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Prerequisites

Create an HFSS design with SBR+ regions, either driven model or terminal solution type or SBR+ solution type.

Create a plane wave source for an RCS simulation with an SBR+ Region.

Process Flow for Setting up and Simulating an SBR+ Design for Creeping Waves for RCS

1. Create an HFSS design with SBR+ region (either driven modal solution type or SBR+ solution type).
2. Add geometries.

   These can be either light weight geometries, or conventional Modeler geometries.

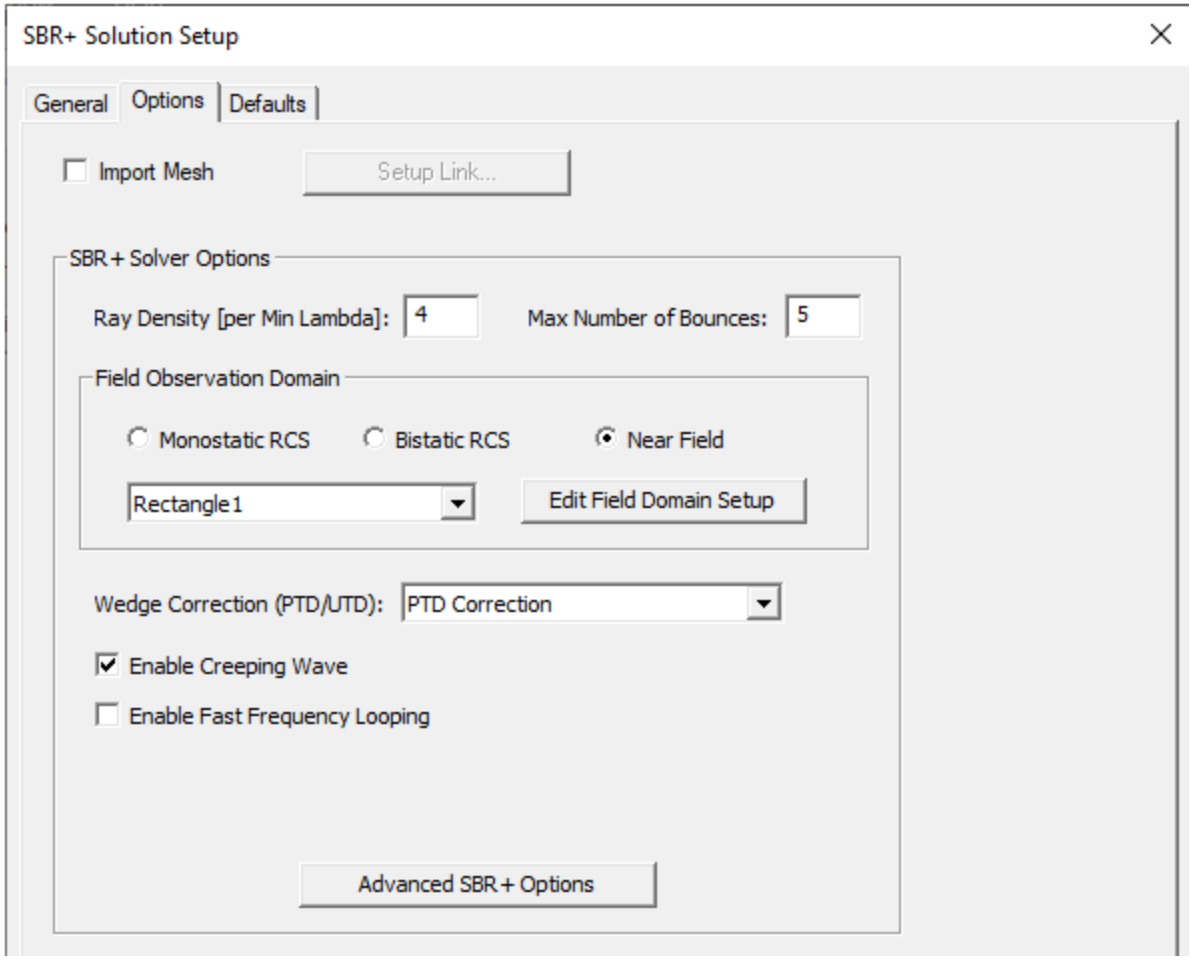
   You can specify how SBR+ curvature extraction is to be handled per geometry component/part. By default, the curvature extraction is applied on all SBR+ faces in the design. You can disable curvature extraction on selected faceted faces or objects by using this mesh operation.

   When object curvature information is available in the original geometry, the Apply Curvilinear Elements option may be used. This option avoids the curvature extraction approximation by passing curvature information directly to the SBR+ solver, which is more accurate.

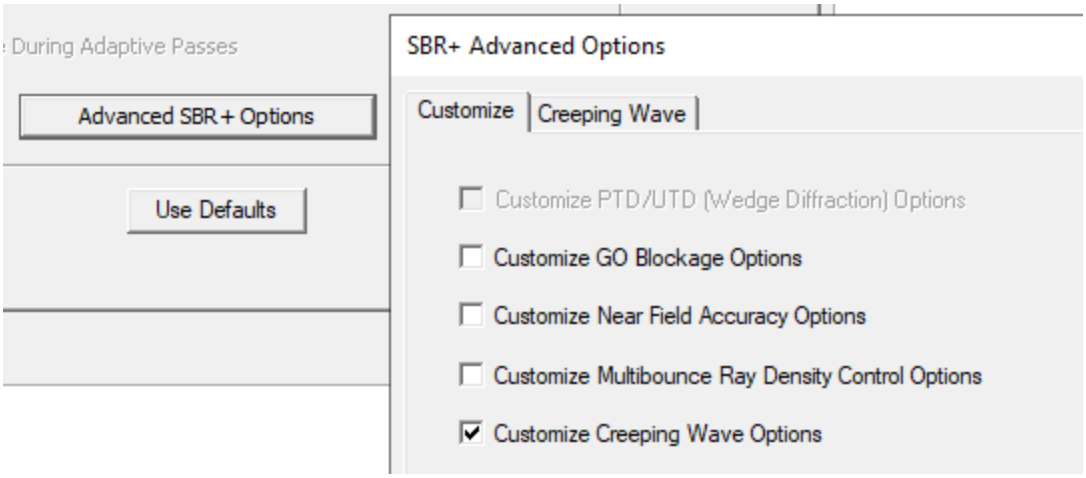
3. Add an Incident Plane Wave source.
4. Add a Solution setup and in the **SBR+ Solution Setup**, **Options** tab or or **HFSS with Hybrid and Arrays** driven modal solution setup **Hybrid** tab select **Enable Creeping Wave**. (Note that this option is only available in the solution setup dialog if a near-field link to an HFSS source such as a FEBI or IE region excitation/boundary is present in the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

design.)

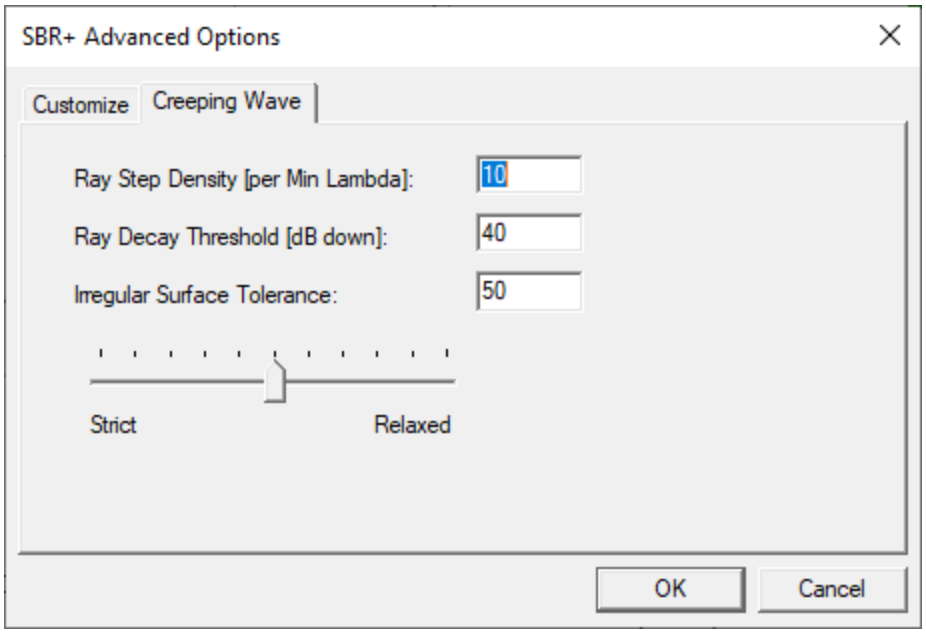


Select **Advanced SBR+ Options** and in the **SBR+ Advanced Options** dialog check **Customize Creeping Wave Options** to enable the **Creeping Wave** tab..



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select the **Creeping Wave** tab to set parameters.



Ray Step Density controls the longitudinal sampling along the direction of propagation. The lateral sampling is determined by the SBR ray density.

Ray Decay Threshold controls how far the creeping rays propagate by terminating each ray once its power density decay (defined as dB down from its level at its starting point at the shadow boundary) exceeds this threshold.

Irregular Surface Tolerance is a single control for a collection of tolerances that determine creeping rays' sensitivities to local surface irregularities that theoretically should block their further propagation. Moving the slider toward **Relaxed** allows creeping rays to propagate through small surface irregularities that may arise through defects in the input geometry or due to limitations of curvature extraction accuracy. However, setting this tolerance too relaxed may allow creeping waves to pass through concave surfaces, saddle surfaces, or wedges, which are not supported by creeping wave physics. Therefore, you are strongly encouraged to perform a creeping-wave visual ray trace (VRT) to determine an acceptable setting for this tolerance, which will differ from model to model. The intention is that creeping rays only propagate over the convex portions of surfaces, and some user judgment is required in correctly setting the **Irregular Surface Tolerance** for each situation.

5. Run simulation.

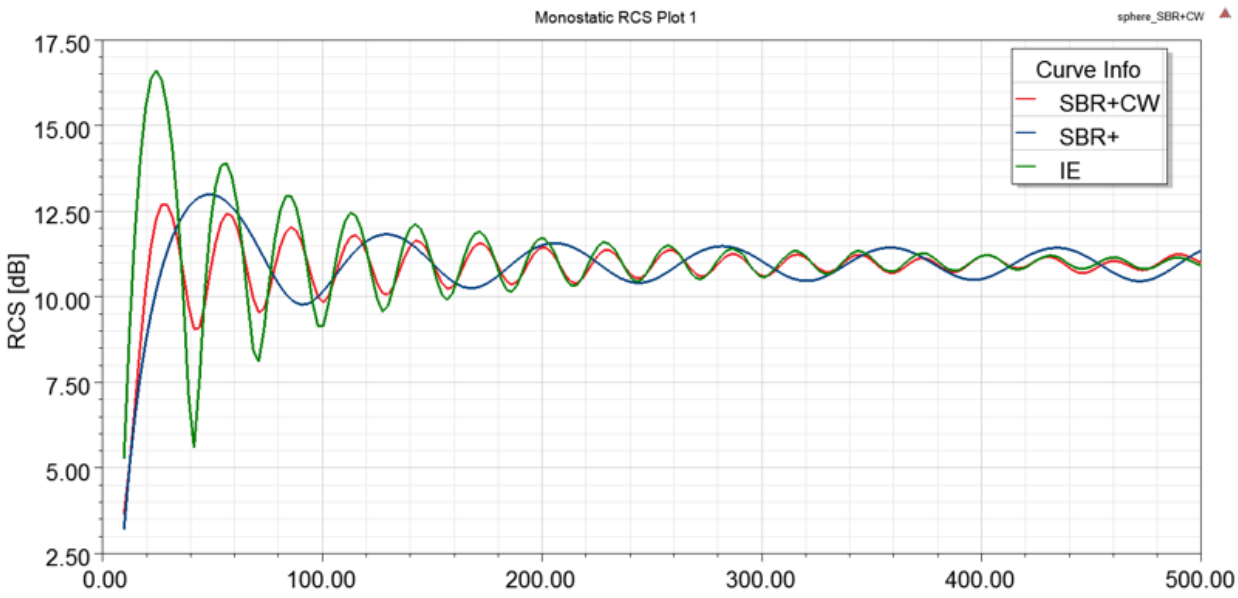
You should see the progress bar reporting both SBR and Creeping Wave ray shoots during that part of the run.

6. Generate RCS plots

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Right click on **Result** and select **Create Monostatic RCS Report > Rectangular Plot**. For the **Report** window, select **Monostatic RCS** as the Category, **MonostaticRCSTotal** as the Quantity, with **dB** as the Function, and click **New Report**.

A comparison of RCS solved by various methods for a 1-meter radius sphere is shown below. Comparing with the reference integral equation (IE) solution, and also the exact Mie series solution (not shown), the SBR+ solution with creeping wave (SBR+CW) better captures the oscillatory behavior of RCS over the simulation frequency band. For a sphere illuminated by a plane wave, the “shadow boundary” is the great circle perpendicular to the incident wave direction. The slower oscillation versus frequency predicted by SBR+ alone is caused by the wave interference of the back-scattered direct return from the leading surface of the sphere with a false diffraction from the shadow boundary where physical optics (PO) currents abruptly (and non-physically) terminate. Instead, currents should smoothly weaken when crossing from the lit half to the shadowed half of the sphere. The faster (and correct) oscillation of the IE and SBR+CW solutions is caused by the wave interference of the direct return with the delayed return of a creeping wave passing around the back of the sphere and coming out toward the front again. Since the difference in path length between these two scattering mechanisms (direct and around-the-back) is larger than the difference in that of the former (direct and false shadow boundary diffraction), the beat frequency of the interference is higher.



Exporting ROM Radar to VRXPERIENCE (Beta)

The VRXPERIENCE Sensors simulation environment includes a ROM-based automotive radar model based on the Ansys FF-RRP ROM (Far-Field Rotated Range Profile Reduced Order Model). This ROM is generated by HFSS SBR+. It consists of coherent radar cross-section

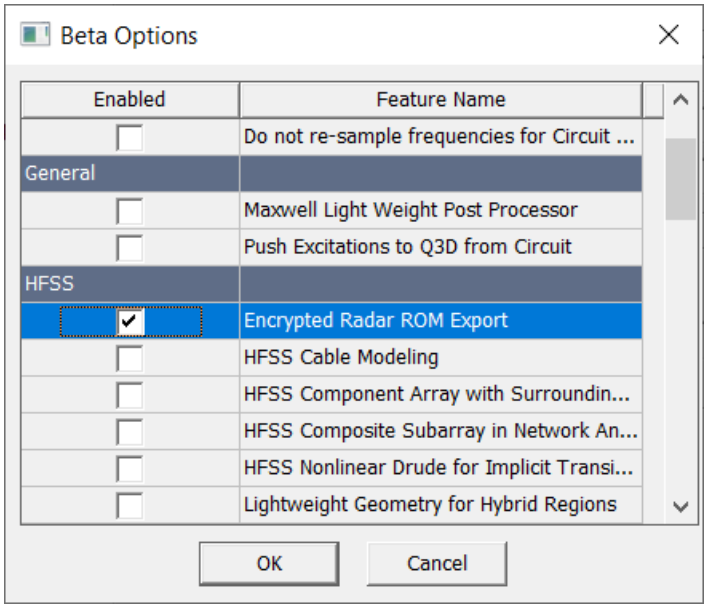
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(RCS) data of a radar target (e.g., car, truck, infrastructure) collected over a range of frequencies and aspect angles. The RCS data are captured in four polarization combinations: *VV*, *HV*, *VH*, and *HH*. The frequency sweep is typically broad enough to cover the range of frequencies permitted in a particular automotive radar band (e.g., 76 - 81 GHz). VRXPERIENCE Sensors can ingest multiple ROM files to simulate complex traffic engagements in real time. Details and instructions for using HFSS-exported FF-RRP ROM files in VRXPERIENCE Sensors can be found in its end-user documentation. This help page describes the HFSS work flow for synthesizing and exporting an FF-RRP ROM file for a single radar target.

Prerequisites

Requirements for generating a FF-RRP ROM file:

- Use **Tools>Options>General Options** to open **Options** dialog box, and under Desktop Configuration, select the **Beta Options** button, enable Encrypted Radar ROM Export. You will need to restart Ansys Electronics Desktop to use the enabled Beta Options.



- A CAD model for the radar target. Alternatively, the model can be drawn in HFSS.
- Solution type must be SBR+.
- Bounds for the intended radar frequency band. For example, based on international standards, the 77-GHz automotive radar band extends from 76 to 81 GHz. To maximize their generality of application, FF-RRP ROM files intended for simulating “77-GHz” automotive radars should use a frequency sweep from 76 to 81 GHz. The fact that a particular radar only uses a portion of the 76 – 81 GHz band should not weigh in setting the frequency sweep of an FF-RRP ROM file, as that would block its use with other radars that operate in a different portion of that band. At the same time, it is not recommended to

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

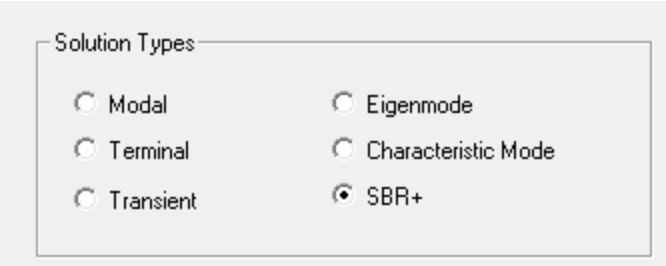
create an FF-RRP ROM file that spans the 24-GHz and 77-GHz automotive radar bands since that results in immense wasted computation and storage covering the large gap between these band where no automotive radars operate.

Synthesizing and Exporting an FF-RRP ROM File

1. Create an HFSS design and set its Solution Type to SBR+.

Right-click on the HFSS design node in the **Project** tree and select **Solution Type…** . In the dialog that appears, select **SBR+** and then **OK**.

Note: The other HFSS solution types do not support generating a ROM file.



2. Import the CAD model(s) representing the radar target.

CAD models in various formats can be imported via **Modeler> Import…**. If the target is to be assembled from multiple parts, be sure they are properly arranged relative to each other and then group them together before proceeding to the next step.

3. Position and orient the radar target.

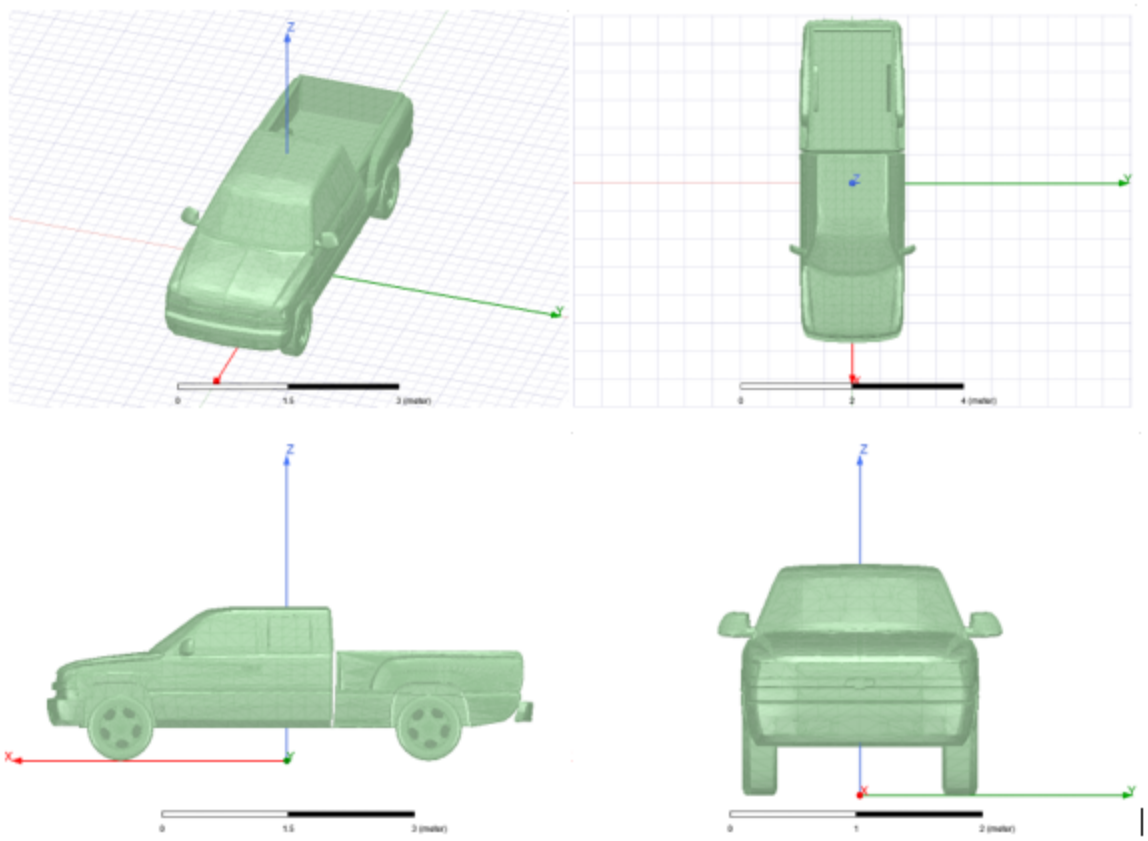
Perform Move and Rotate operations on the target geometry model so that it is positioned and oriented correctly in HFSS global coordinates. It is important that vehicles be positioned and oriented as follows in HFSS global coordinates:

- X-, Y-, and Z-axis point out the front, right side, and top of the vehicle, respectively
- Bottom of wheels touch XY-plane at Z = 0
- Vehicle bounding box extends from (-*L*/2,-*W*/2,0) to (*L*/2,*W*/2,*H*):

  L = length, *W* = width, *H* = height .

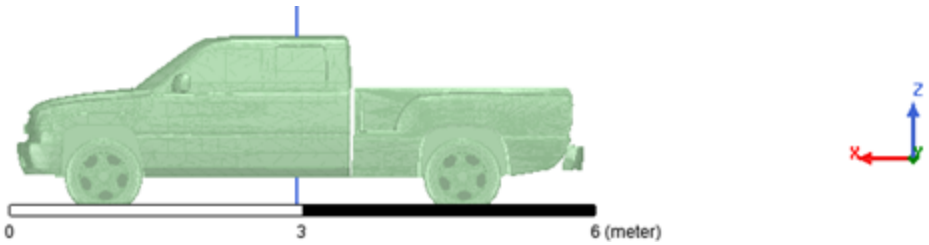
  Laterally centering the vehicle about the origin of the XY-plane minimizes the opportunity for range aliasing in range-Doppler imagery generated by the VRXPERIENCE ROM-based radar sensor model.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note: It is important that HFSS and VRXPERIENCE Sensors use an identical or nearly identical CAD model for visualization and identical local coordinate systems for the target.

In preparation for later configuring the density of the frequency sweep in the solution setup, determine the longest dimension *D* of the radar target. This can be done by aligning the target with the ruler at the bottom of the display or by using the Measure tool found in the **Draw** ribbon. It is not important to measure *D* with great precision, but in that case, one should err on the high side. In the truck example illustrated below, *D* = 6.0 m.



4. Apply boundary assignments to the surfaces.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Since this is an SBR+ simulation, material properties of the radar target should be configured via boundary assignments, **not** volumetric material assignments. In the absence of better information, the entire vehicle can be configured as Perfect E. Fidelity can be improved by investing the effort to assign more realistic boundaries, particularly for glass windows, tires, and exterior plastic/rubber. The following types of boundary assignments are recommended:

Perfect E: Metal surfaces of body.

Layered Impedance (one-sided): Tire rubber, body plastic, mirrors, interior seats. Can also be used for painted metal surfaces to improve fidelity. One-sided Layered Impedance boundaries only reflect SBR rays; they generate no transmission rays.

Layered Impedance (two-sided): glass windows. Two-sided layered impedance boundaries are penetrable to rays. For this reason, they can also be used for things like non-metallic bumpers provided the vehicle structure underneath these covering surfaces is adequately captured in the CAD model. Barring that, it is recommended to model non-metallic exterior parts using one-sided Layered Impedance boundaries. When using two-sided Layered Impedance boundaries for things like windows, they should be represented by a single thin sheet in the CAD model, not a closed volumetric object nor a pair of sheets to represent the inside and outside surfaces. The finite thickness of the glass should be captured in the assigned thicknesses of the window layer(s) in the Layered Impedance boundary configuration. Also, be careful when applying two-sided Layered Impedance boundaries to the face(s) of a closed object (as opposed to a sheet). The object volume material must be set to vacuum, and it should be understood that each SBR ray will branch into a transmission and reflection ray upon hitting the object face, whether incident from the outside or (later) incident from the inside after a transmission rays has penetrated into the object. As a result, many rays will repeatedly bounce inside the closed object volume until reaching the Maximum Number of Bounces limit. In special cases, such a configuration can make sense for some components, but if the closed object volume is not actually vacuum (or air filled), this would be the incorrect way to model it and one should instead fall back on applying a one-sided Layer Impedance boundary to all faces of the object and setting the infinite-thickness layer to the dielectric properties of the volume object.

Perfect Absorber: When the CAD model has gaps, making it leaky to rays, adding **Fresnel SBR+** Perfect Absorber surfaces to the interior of the target is a good way to trap rays that leak inside and prevent them from rattling around the interior, which would tend to harm accuracy.

5. Create a pair of incident plane-wave excitations with an angular sweep, one for vertical polarization and one for horizontal polarization.

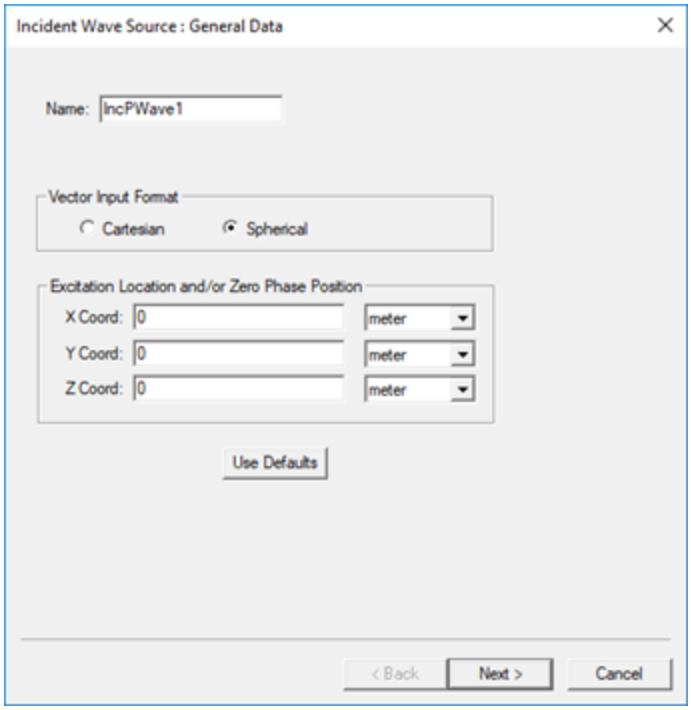
The ROM model content is coherent, monostatic RCS in four polarization combinations that provide a complete polarimetric description of the scattering: VV, HV, VH, and HH. There are two independent incident polarizations (indicated by the second letter), and for each of these there are two independent observation polarizations (indicated by the first letter).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In following steps, the polarization ordering of the two incident plane-wave excitations is interchangeable without affecting the content or data ordering in the ROM file. Also, the default excitation names of IncPWave1 and IncPWave2 are not important and can be changed if desired.

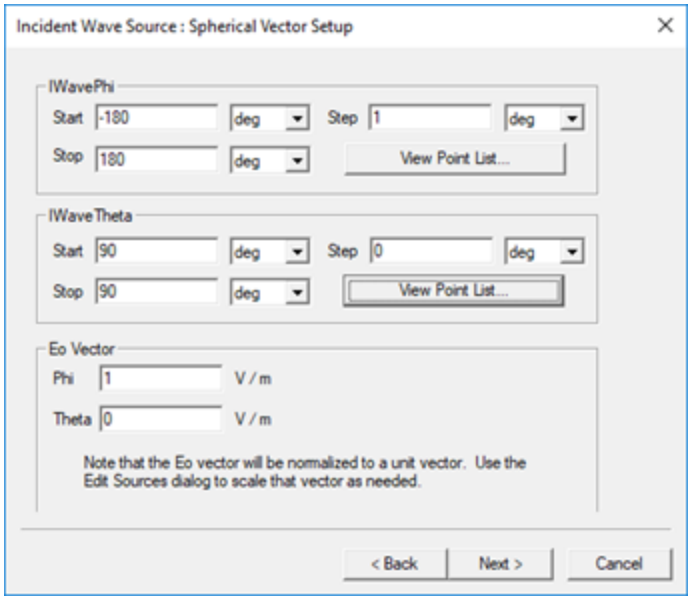
First, configure the incident plane-wave with horizontal polarization:

a. In the **Project** tree, right-click on the Excitations node and select **Assign Incident Wave>Plane Wave…**.
b. In the **Incident Wave Source: General Data** dialog that appears, set the **Vector Input Format** to Spherical. As a clarification for post processing, you may want to change the default name to something like IncPlaneWave_H for the first wave, and IncPlaneWave_V for the second wave. Then click **Next**.



c. In the updated dialog box, set IWavePhi **Start** and **Stop** to -180 deg and 180 deg for a full circular sweep around the target. The **Step** size should be set no higher than 1 deg. A lower Step size can be selected to improve resolution, but realize this will lead to a longer HFSS synthesis time and a larger ROM file. **Regardless, make sure that the Step size fits evenly into 360 deg. Otherwise, the final IWavePhi sample at +180 deg might be omitted, resulting in an incomplete ROM model and unexpected results when it is used in VRXPERIENCE**. Confirm that the final IWavePhi angle sample is +180 deg by selecting **View Point List…** and scrolling down to the end of the list.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

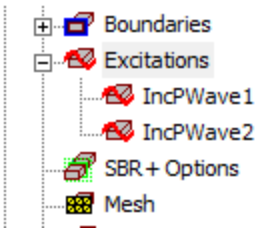


d. In the same dialog box, set IWave Theta **Start**, **Stop**, and **Step** to 90 deg, 90 deg, and 0 deg, respectively. This fixes the elevation angle at zero (0), the XY-plane.

e. In the same dialog box, configure Eo Vector as **Phi** = 1 *V*/*m*, **Theta** = 0. This sets up a horizontally polarized incident field that yields the *VH* and *HH* RCS polarization components.

f. Click **Next**. In the updated dialog box, accept the default selection (Regular/Propagating) and click **Finish**. The incident and observation angles of the monostatic RCS sweep have now been configured for *HV* and *HH* polarization combinations.

g. To set up the incident vertical polarization, repeat Steps 5a – 5f, but with the following change: in Step 5e, configure Eo Vector as **Phi** = 0, **Theta** = 1 *V*/*m*. The monostatic RCS sweep for the *VV* and *HV* polarization combinations has now also been configured. During simulation, these two incident polarizations will be simulated independently.

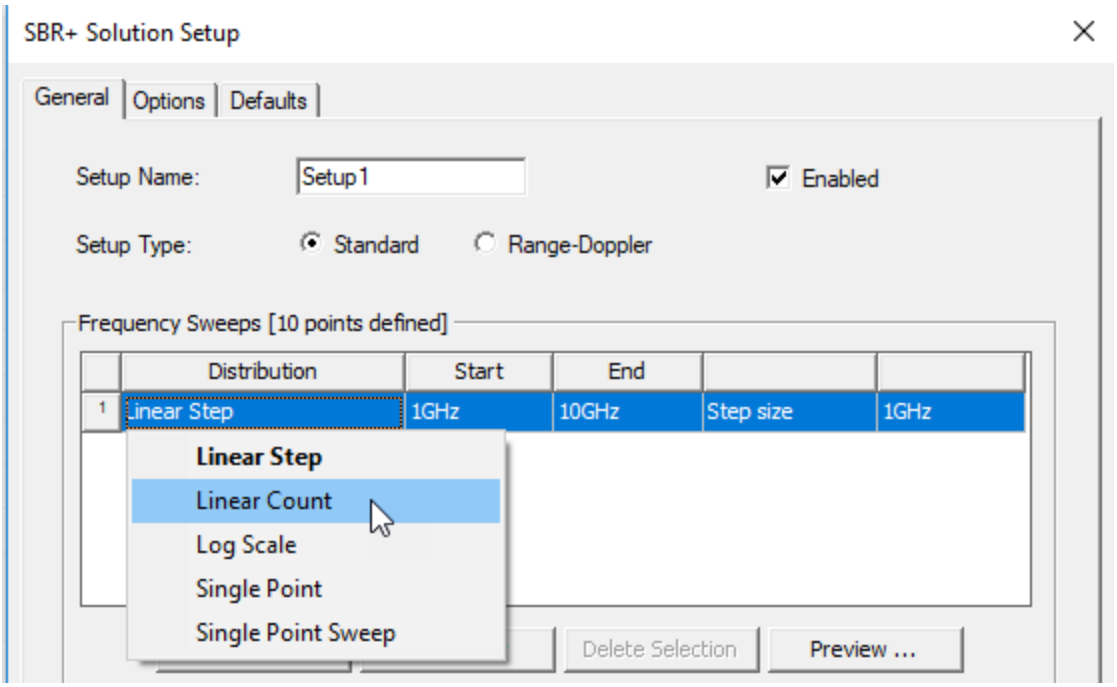


6. Create and configure the solution setup.

Create a solution setup that configures a uniform frequency sweep, sets the RCS modality to monostatic (observation angle tracks incident angle), and configures the SBR+ methodology settings.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

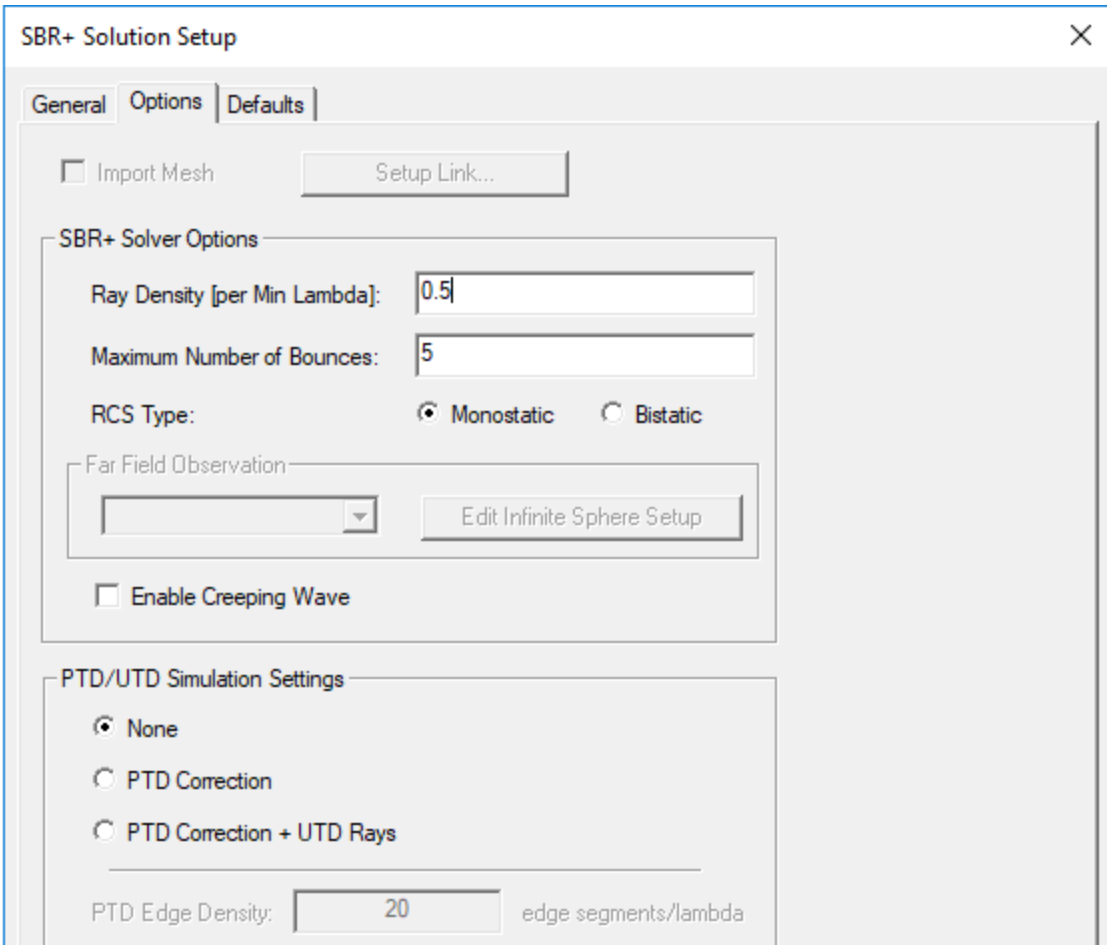
a. In the **Project** tree, right-click **Analysis** and select **Add Solution Setup…**.
b. In the **SBR+ Solution Setup** dialog that appears, leave the Setup Type as **Standard**. There will be a table with a single row whose Distribution is set to Linear Step. Left-click on **Linear Step** and select **Linear Count** instead.



c. Using the longest dimension *D* of the radar target that was established in Step 3, determine the required frequency interval as $\Delta f = c/6D$ where, c is the speed of light. Sampling at this frequency (or lower) will minimize the opportunity for range aliasing in radar imagery generated by the ROM-based radar model in VRXPERIENCE Sensors. The above formula includes a safety margin of *D* behind the vehicle (relative to the radar) for multi-bounce delayed response. In the truck example above, $D = 6$ m, so $\Delta f = 8.3333$ MHz.
d. To generate a ROM model applicable to any automotive radar operating in the 77-GHz band, set the Start and End fields to 76GHz and 81GHz (these are the maximum extents allotted to the 77-GHz automotive radar band). This corresponds to a bandwidth of 5 GHz. Divide the bandwidth by Δf and then round up to obtain an integer number of frequency intervals. Then add one (1) to obtain the number of points and enter this in the far right (unlabeled) field of the table row. In this example, 5 GHz/8.3333 MHz = 600, so enter 601 for the number of points.
e. Make sure that **Compute Far Fields** is toggled on.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

f. Select the **Options** tab.



g. **Important**: Change **Ray Density per Min Lamda** from its default (4.0) to 0.5. At this nominal 77-GHz frequency, we recommend a setting of 1.0 or less for ray density. Otherwise, far more rays are shot than are really needed, harming simulation time for generating the FF-RRP ROM model (it does not affect dynamic simulation time in VRXPERIENCE Sensors, however).

h. Set RCS Type to **Monostatic**.

i. Set the **Maximum Number of Bounces**. The default value of 5 is recommended.

j. If you elect to switch on PTD or PTD + UTD to improve accuracy, you may optionally choose to extract and filter wedges. This can be done by right-clicking **SBR+ Options** in the **Project** tree and selecting **Wedge Settings…**.

7. Run the simulation (i.e., **Analyze** the solution setup).

In general, we recommend that you use Ansys HPC to add more cores to speed the solve process.

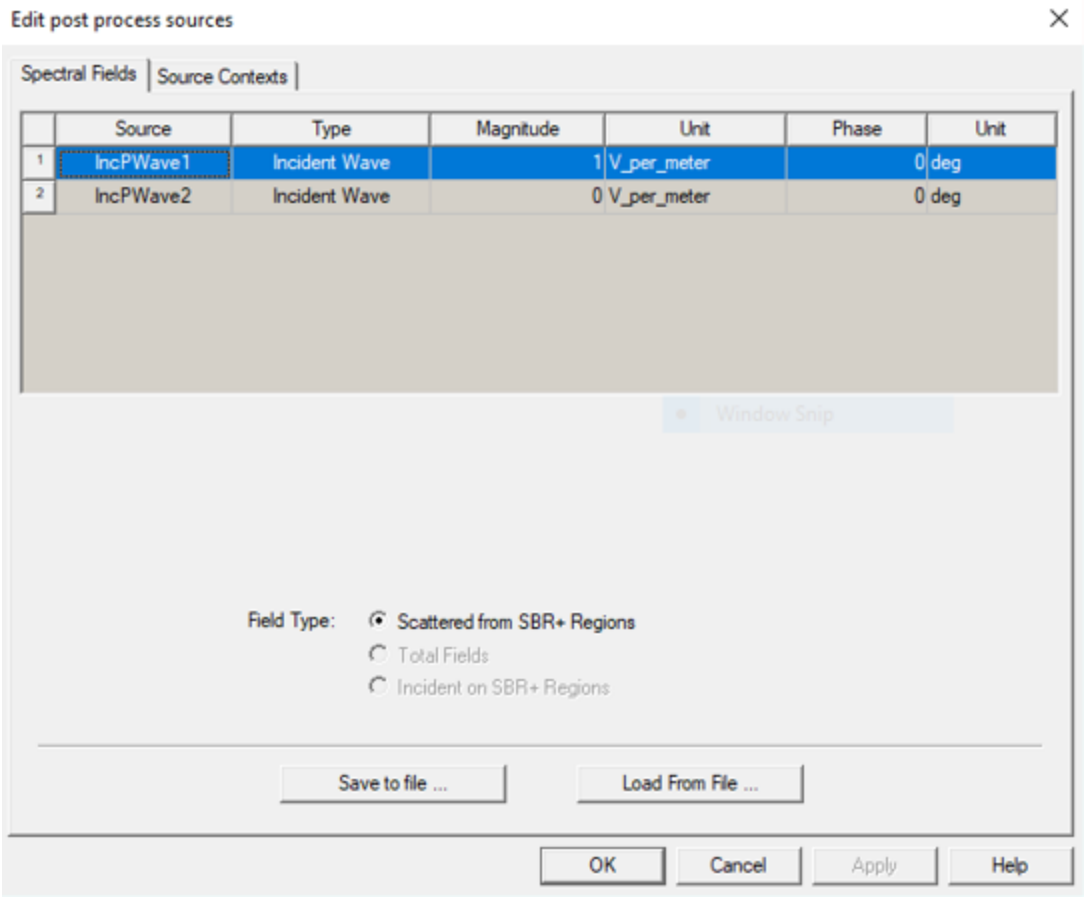
8. Plot the RCS results (optional).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Generate a plots of RCS vs. angle to verify the results are reasonable.

a. Before generating the plot, set/verify the active incident polarization for the plot. In the **Project** tree, right-click on **Excitations** and select **Edit Sources…**.

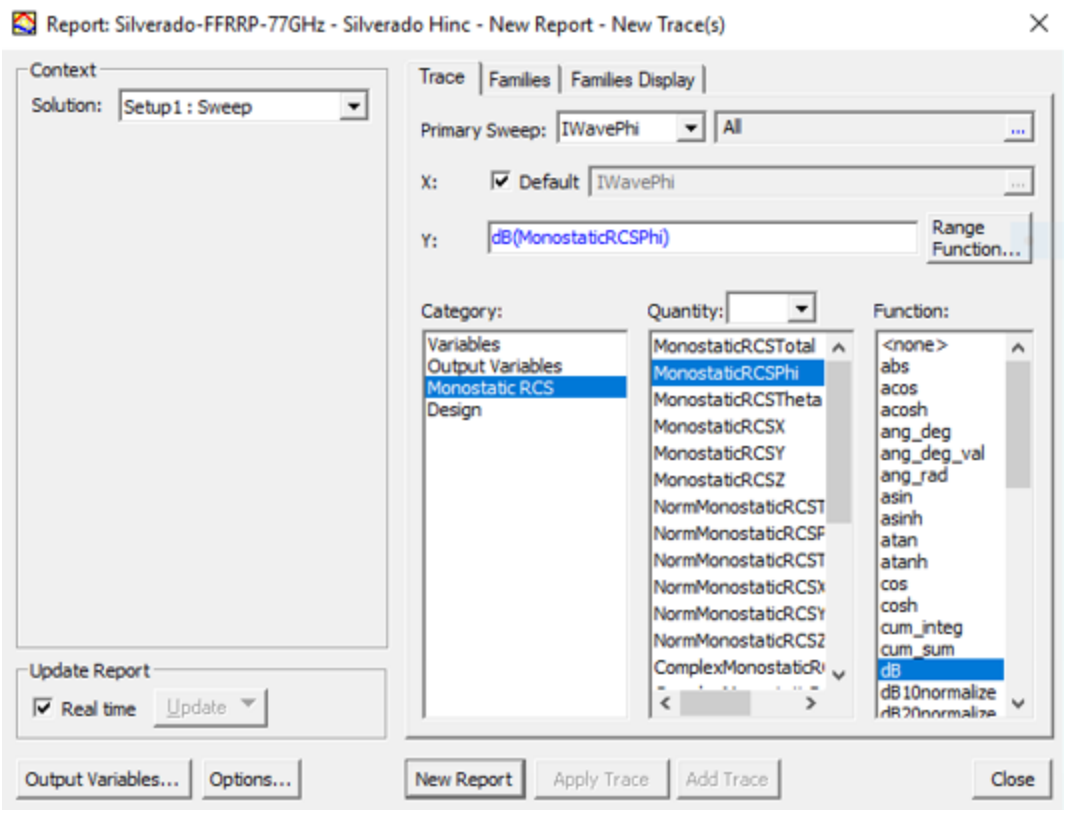
b. In the **Edit post process sources** dialog that appears, there will be a table with two rows for the two configured plane-wave excitations. If not already configured this way, set **Magnitude** of IncPWave1 to 1 V/m and that of IncPWave2 to 0 V/m, and then click **OK**. Since in this example, IncPWave1 was configured as the incident horizontal polarization, any existing or new plot while this edit-sources configuration is active will show results for incident horizontal polarization.



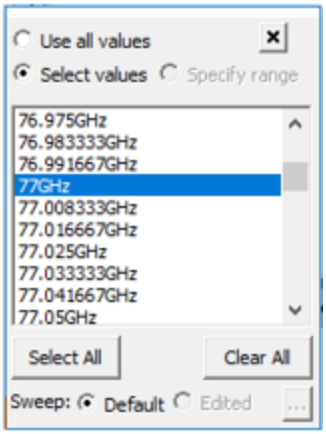
c. In the **Project** tree, right-click on **Reports** and select **Create Monostatic RCS Report >Rectangular Plot**.

d. In the **Report** dialog box, set Primary Sweep to IWavePhi, under Category select Monostatic RCS, under Quantity select MonostaticRCSPhi, and under Function select dB.

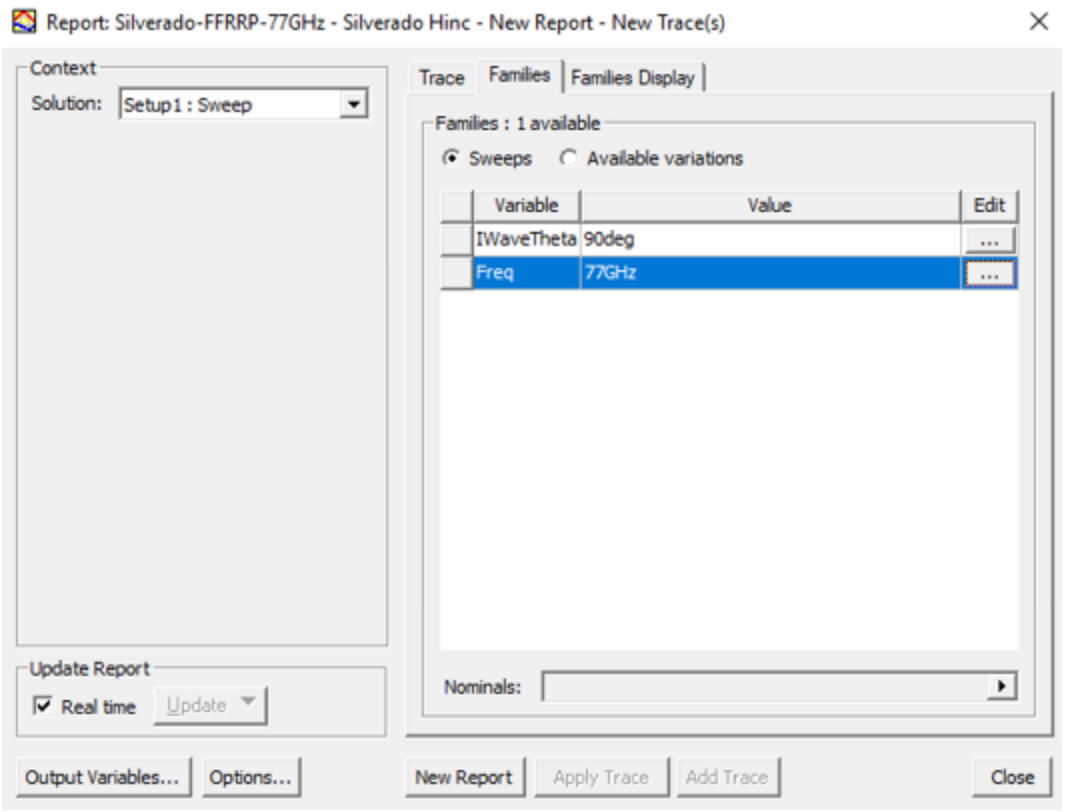
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



e. Select the **Families** tab in the same dialog box. This controls the multi-dimensional slicing to configure one or more plot traces. In the table, click the ellipses **[…]** button in the **Edit** column for the Freq row. This pops up a small dialog for selecting the discrete frequency samples to plot as separate traces. Scroll down, select 77GHz, and close the dialog by clicking the "**X**" button in its upper right corner.

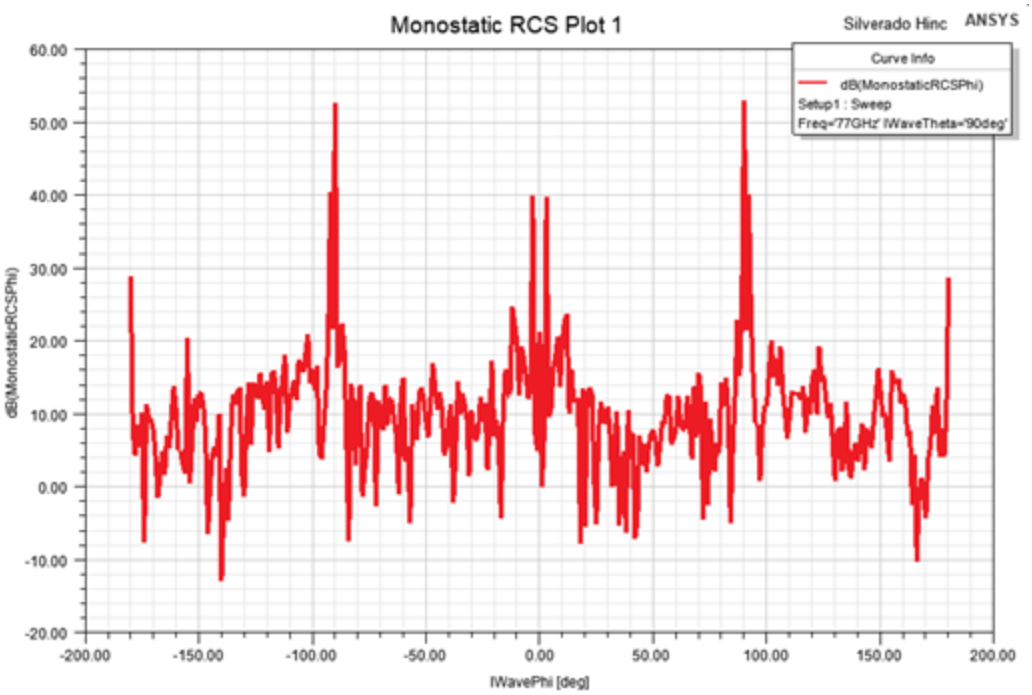


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



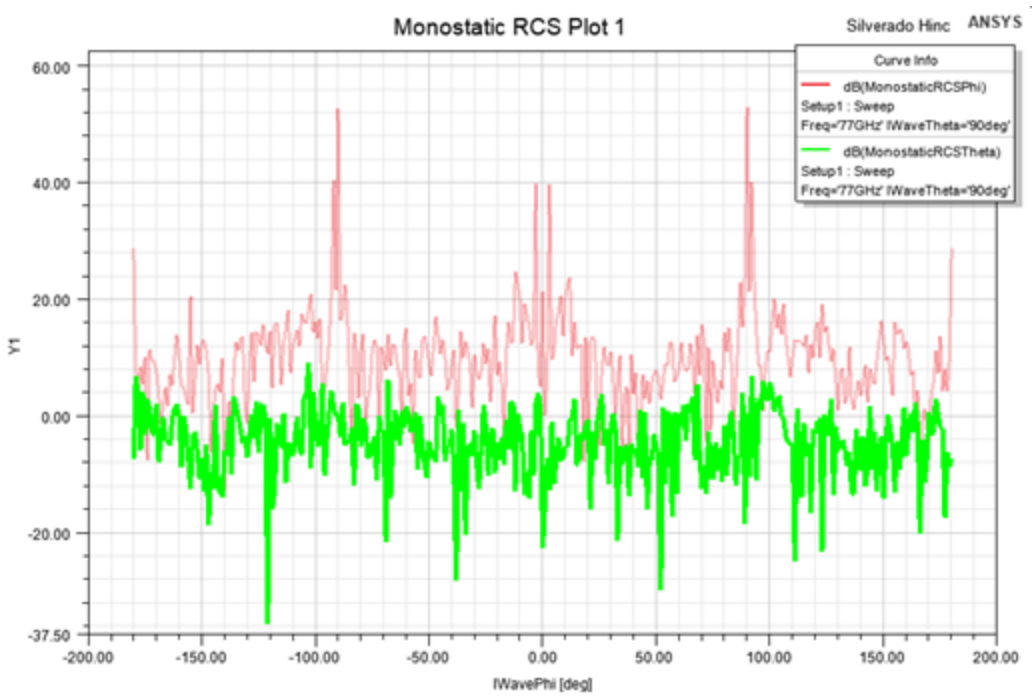
f. At the bottom of the **Report** dialog box, click **New Report**. A new plot is created showing the HH-polarization RCS vs. Phi angle at 77 GHz. It is HH polarization because of the combination of the **Edit Sources** weighting from Step 8b and the Quantity selection from Step 8d. Leave the **Report** dialog open.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



g. In the **Report** dialog box, select the **Trace** tab again and then set **Quantity** to MonostaticRCSTheta. Also make sure to set **Function** to dB again, then click **Add Trace**. A second trace is added to the plot that shows the VH-polarization RCS. It is significantly weaker than the HH-polarization, which is typical when comparing cross-polarized to co-polarized RCS.

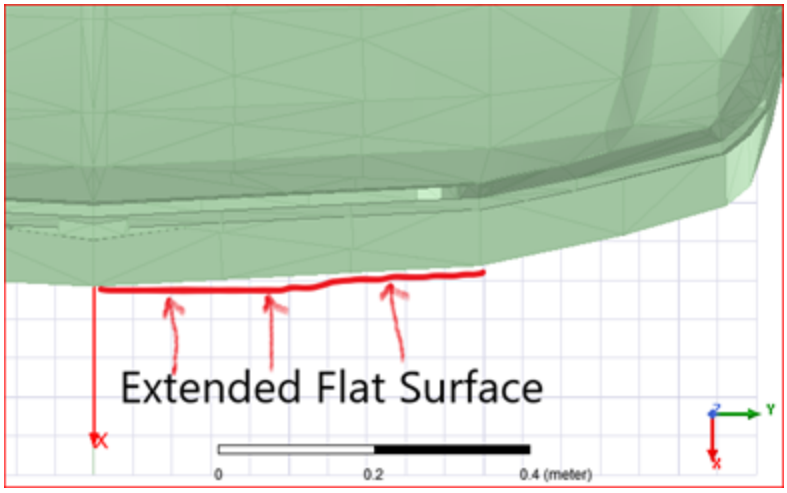
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



h. Note the sharp peaks at IWavePhi = +/- 180^{o} and +/- 90^{o} for the *HH*-polarization. These are specular flashes off the large and nearly flat surfaces of the back and sides of the truck (in this example). The side flashes are particularly pronounced, exceeding 50 dBsm (dB square-meters) because truck doors and bed walls are nearly perpendicular to the radar and have a very large surface area.

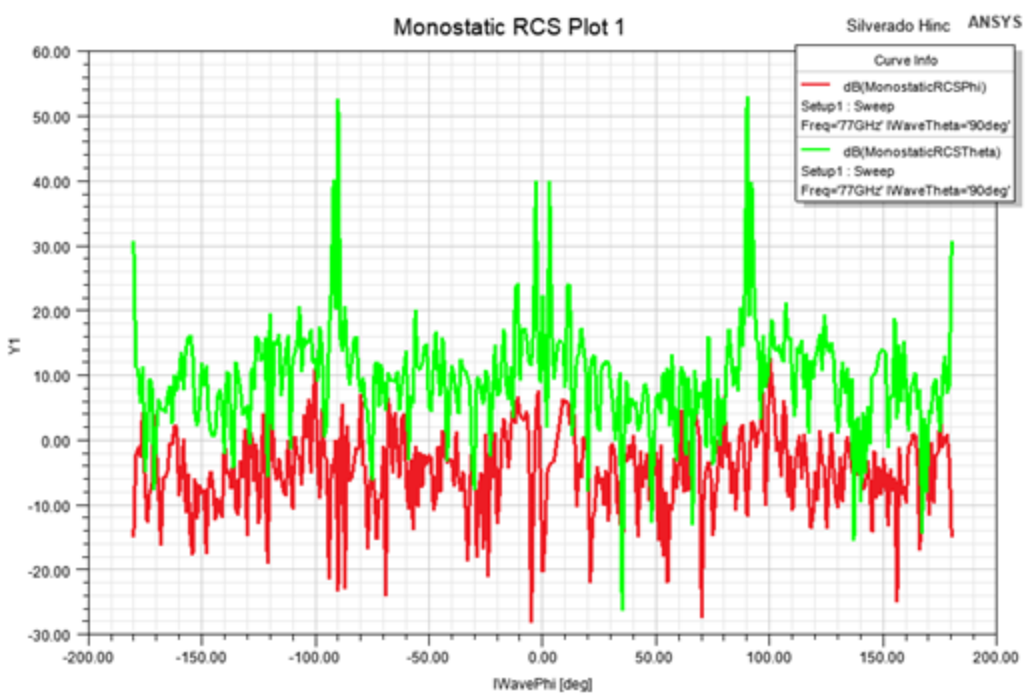
   There are also a pair of specular flashes near IWavePhi = +/- 3^{o}. These are caused by extended flat surfaces on the front bumper and are therefore likely an artifact of the coarse meshing resolution of the bumper. This leads to an important observation: at 77 GHz, the wavelength is approximately 4 mm. Meshings of radar targets that might be perfectly adequate for 10 GHz and below may be too coarse at 77 GHz because flat surfaces of the mesh increase in electrical size, leading to artificial specular flashes. This is likely the situation with the pickup truck model used in this example. The real bumper is a smoothly curved surface, so the observed coarse meshing of the bumper and the correlation of these specular flashes with the angles of the bumper surfaces indicate that it has inadequate mesh resolution for RCS prediction at 77 GHz.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



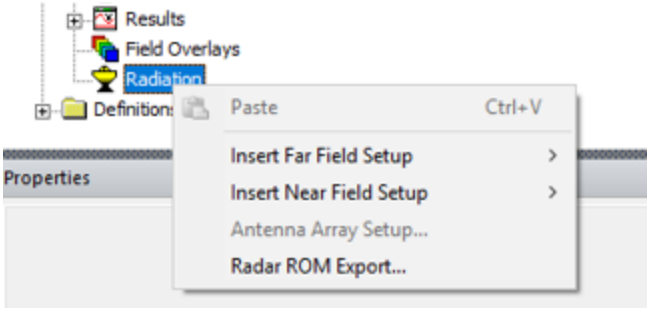
i. Open the **Edit Sources** dialog again (see Step 8a) and set the Magnitude for IncPWave1 to 0 and IncPWave2 to 1 *V*/*m*. Click **OK**. Allow some time for post-processing to update field results. This switches the incident polarization from horizontal to vertical (given how we configured the incident plane waves in Step 5). The RCS plot updates to show *HV* instead of *HH* (red trace) and *VV* instead of *VH* (green trace). Comparing with the earlier plot, the *HH* and *VV* polarizations are similar, but different in the details, whereas *VH* and *HV* show a more pronounced difference. This behavior is typical for RCS. *VV* and *HH* RCS are identical for the dominant first-bounce returns off metal surfaces, so co-polarized RCS tends to be similar when comparing *VV* and *HH*. Cross-polarized RCS is zero for metal surfaces on the first bounce and is weaker, in general, than co-polarized RCS, so it is more sensitive to second-order effects that are polarization dependent.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



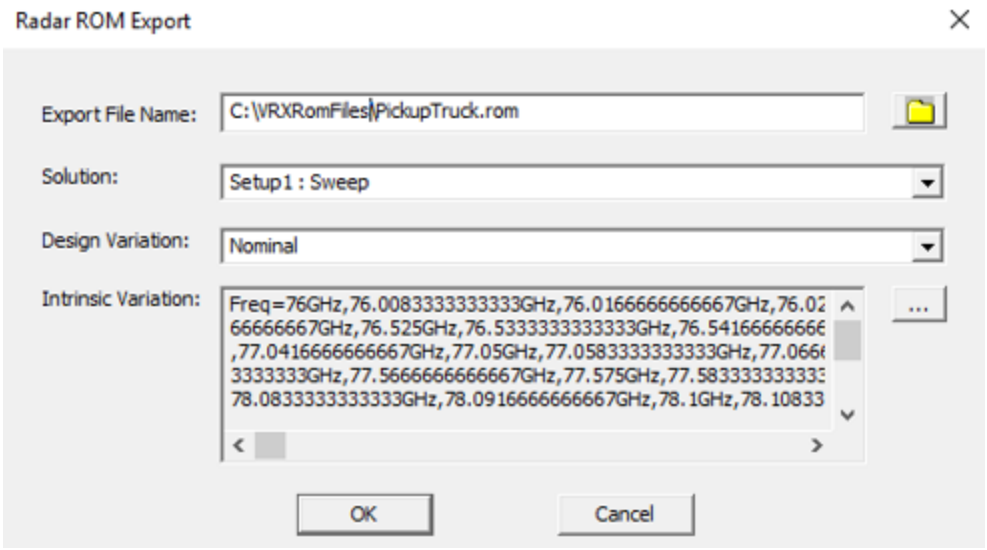
9. Export the ROM file.

In the **Project** tree, right-click on the **Radiation** icon and select **Radar ROM Export…**.



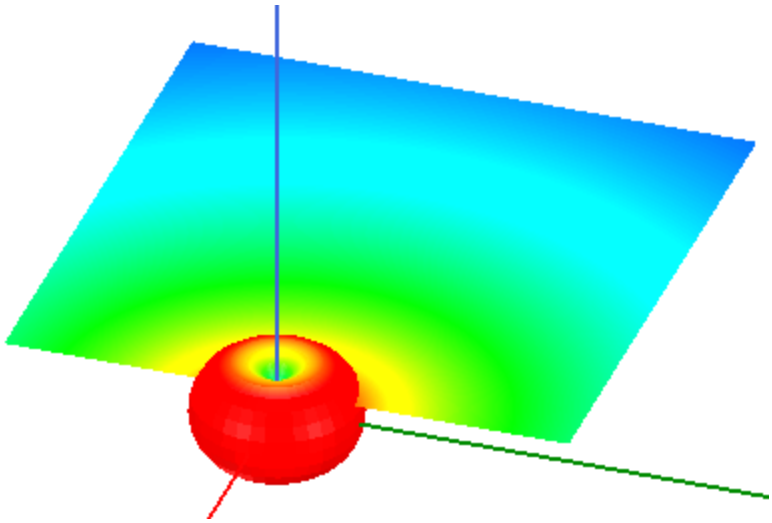
You can then specify the directory and file name for the exported ROM file. Generating the ROM file may take a few minutes while the HFSS post-processor prepares the coherent RCS result data. The exported ROM file is not human readable, but it can be directly imported and used by the ROM-based radar model in VRXPERIENCE Sensors.

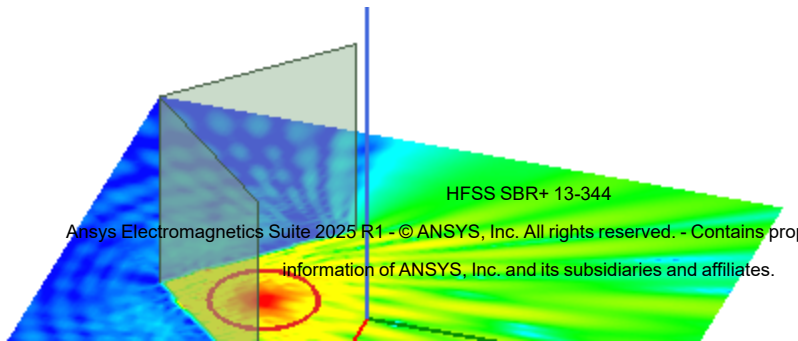
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Near Field Analysis using Antenna Ports in SBR+ Design

SBR+ designs and HFSS designs with SBR+ Hybrid regions can provide near field analysis and overlay rectangular near field plots on the modeler window. For SBR+, the near field setup must be defined before simulation, as is currently done for an SBR+ far field setup. The near field setups allow for a 3-D grid of sample points to be defined.





Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This chapter describes the process flows for Near Field Analysis in SBR+ designs, and for HFSS Terminal or Modal designs with SBR+ Hybrid Regions.

Prerequisites

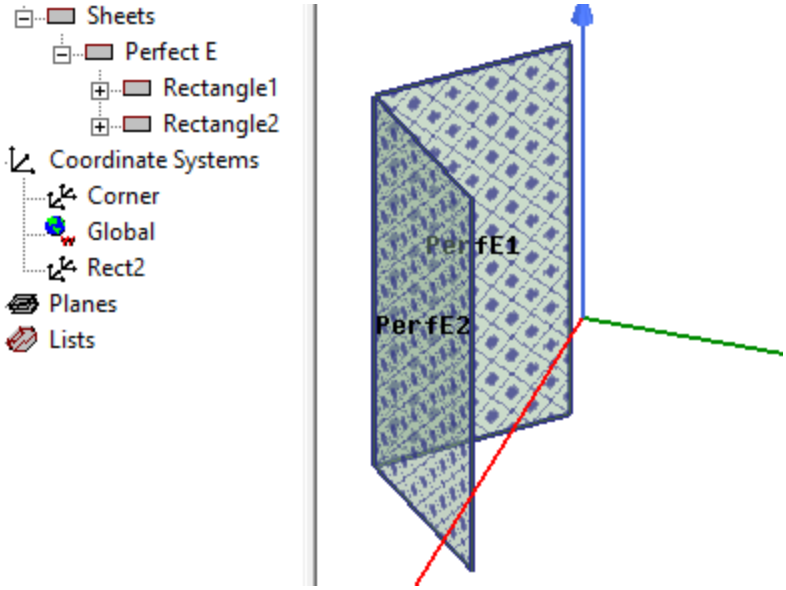
- Create an HFSS design with SBR+ regions, either driven model or terminal solution type or an SBR+ solution type.
- Near Field analysis with SBR+ supports antenna port excitations and support Incident Plane wave sources.
- You must specify Tx or Tx/Rx roles for antenna ports, or Incident Plane wave source to make Near Field analysis available.
- **Caution:** When there are any encrypted components in the design, Near Field Analysis will be aborted.
- **Caution**: Placing near-field observation points very close to an SBR+ region surface can increase SBR+ simulation time, sometimes significantly; when a near-field observation point is near an SBR ray footprint, the solver needs to finely subdivide the ray footprint to maintain accuracy in its scattered field contribution to that observation point."
- **Caution**: In the SBR+ methodology, final (total) field results are always coherently assembled upon the base of an incident field from the source radiation radiating in free space. That means the fields start strong everywhere before the SBR rays are launched. Any reduction of this initial field due to structural blockage is achieved through destructive interference with a comparably strong scattered field arising from SBR currents painted by the rays hitting the blocking structure.

  Under conditions of strong shadowing, as in the extreme example of volume enclosed by a metal surface, one expects very weak or zero fields in the shadowed region. Destructive interference is rarely perfect, and one can generally expect some finite field at every near-field observation point. Field extinction (or nulling) in an enclosed volume is only achieved to the extent of predicting currents with high fidelity upon the enclosing surface. If you observe unrealistically large fields in a deeply shadowed or enclosed region, try increasing SBR Ray Density until the results converge. You can also try deploying SBR+ physics enhancements, such as UTD Edge Diffraction Rays and, if the source is an antenna mounted on a smooth metal surface with convex curvature, Creeping Rays. However, SBR+ remains an asymptotic (high-frequency) approximation best suited for electrically large scattering geometry, so the accuracy of painted currents and therefore field extinction in shadowed spatial regions is ultimately limited by the accuracy of multi-bounce geometric optics (GO), UTD edge diffraction, and UTD creeping-wave formulations.

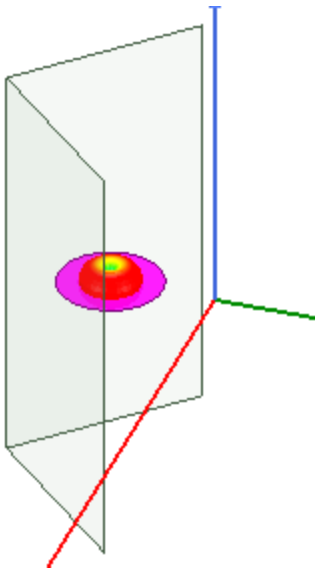
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Process Flow for Near Field Analysis in SBR+ Design

1. Create an HFSS design with SBR+ solution type.
2. Add scene geometry components. This example contains two sheets at a right angle, and assigned Perfect E boundaries.



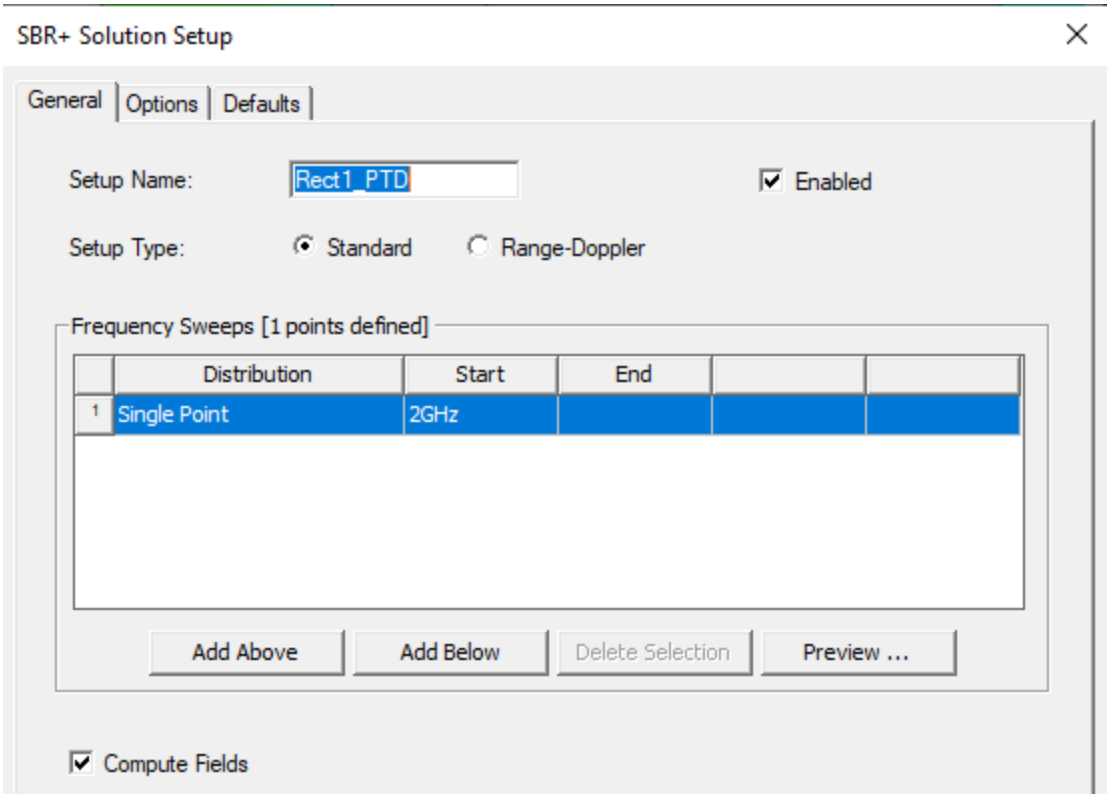
3. For SBR+ solution type create a parametric antenna source or Near Field link. This example shows a dipole antenna positioned between the Perfect E boundary sheets.



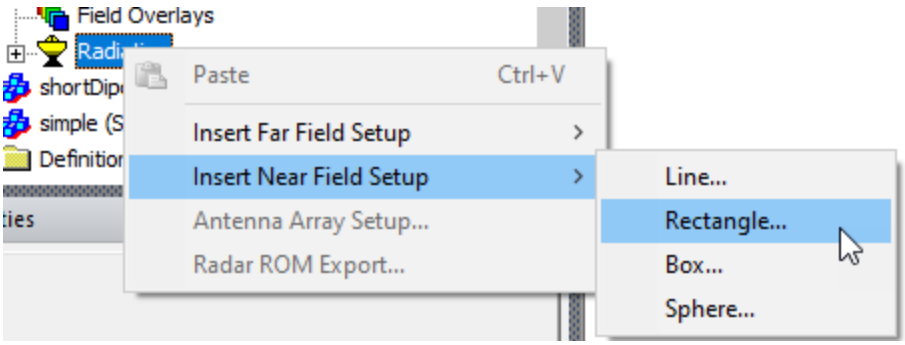
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Right-click on the **Analysis** icon in the Project tree and select **Add Solution Setup...**, or select the **Simulation** tab of the ribbon and select the **Setup** icon.

The **SBR+ Solution Setup** dialog displays. On the **General** tab check the box to **Compute Fields**.



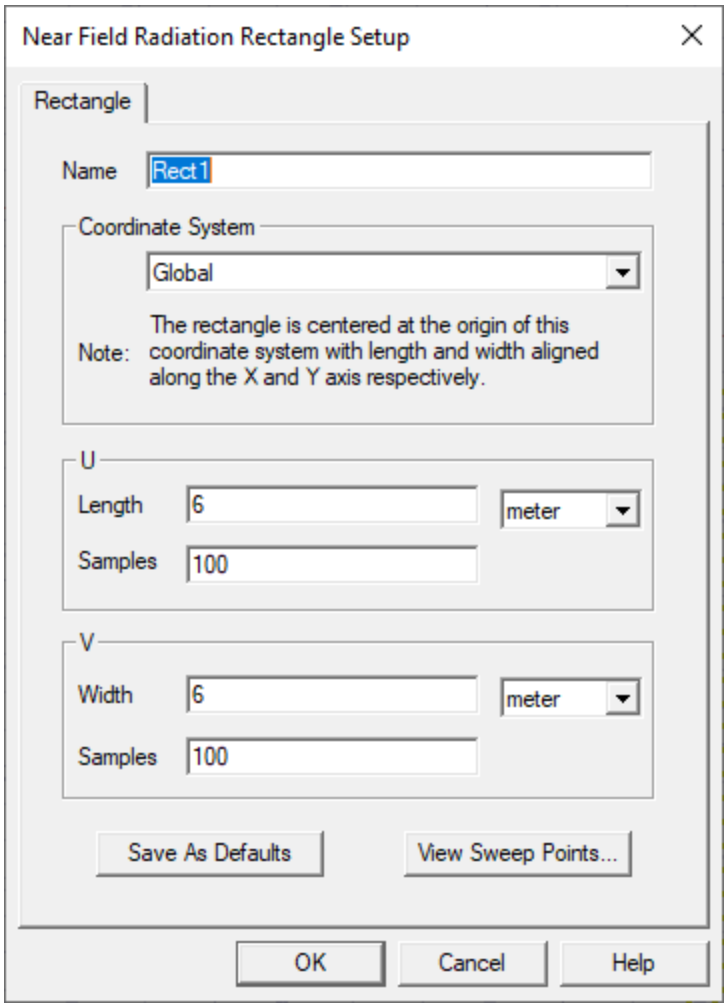
Right click on the **Radiation** icon in the **Project** tree and select **Insert Near Field Setups>** and select **Rectangle...** from the menu.



This opens a **Near Field Radiation Rectangle Setup** dialog box. You specify the name, Coordinate System, U and V length, units, and number of samples. To move or re-oriented

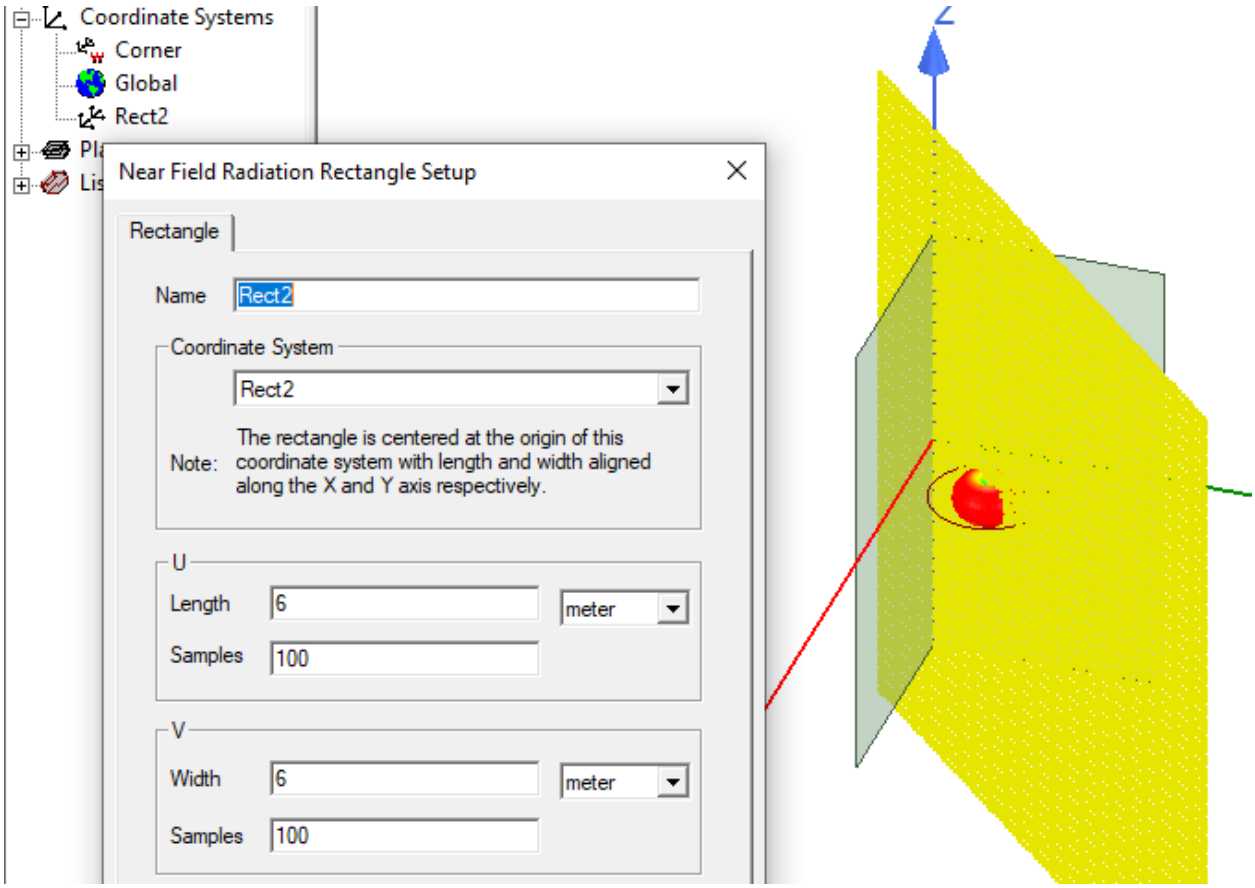
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the near-field sampling grid, set the Coordinate System to one with the desired translation and rotation, first creating a new coordinate system for this purpose if needed. This first example shows a setup based on the Global Coordinate system.

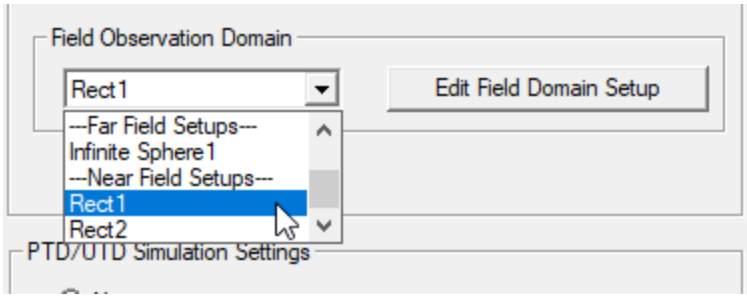


In this example, a second Near Field Radiation Rectangle setup (Rect2) uses a different coordinate system (Rect2) that is rotated by theta = -90 deg around the original (global) Y axis. The point locations will be defined relative to the _u (X) and _v (Y) axes of the Rect2 Relative CS. However, the reported polarized field components are always aligned with global coordinates (i.e., Ex means the electric field along the X axis of global coords, not the X axis of Rect 2 Relative CS).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

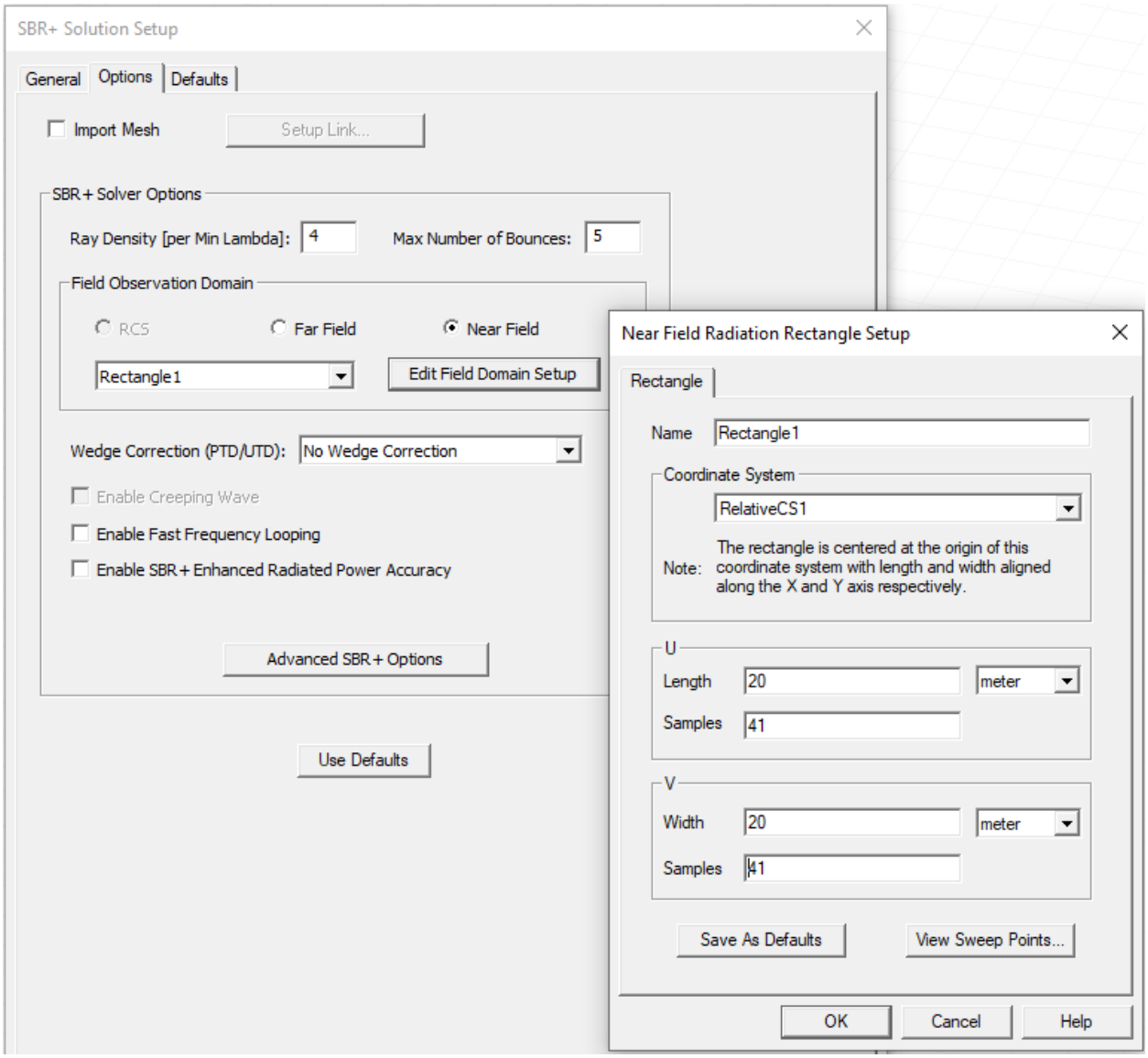


Any Near Field or Far Field setups that you create, or that exist for the design, are listed in the **Field Observation Domain** drop down menu of **Options** tab of the **SBR+ Solution Setup**.



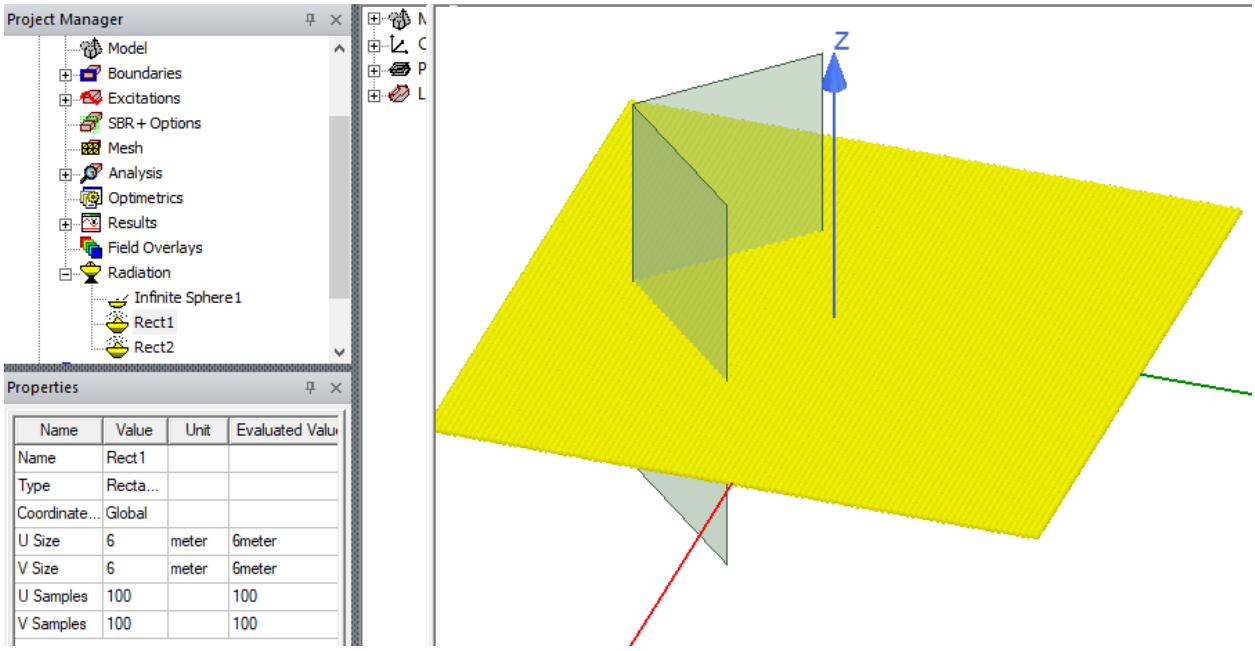
Optionally, you can use the **Options** tab create specify Ray Density, and to specify a **Field Observation Domain** by clicking on the dropdown list and selecting **Create Near Field...**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. To visualize the sample points in the Modeler window, select the Radiation setup in the Project tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

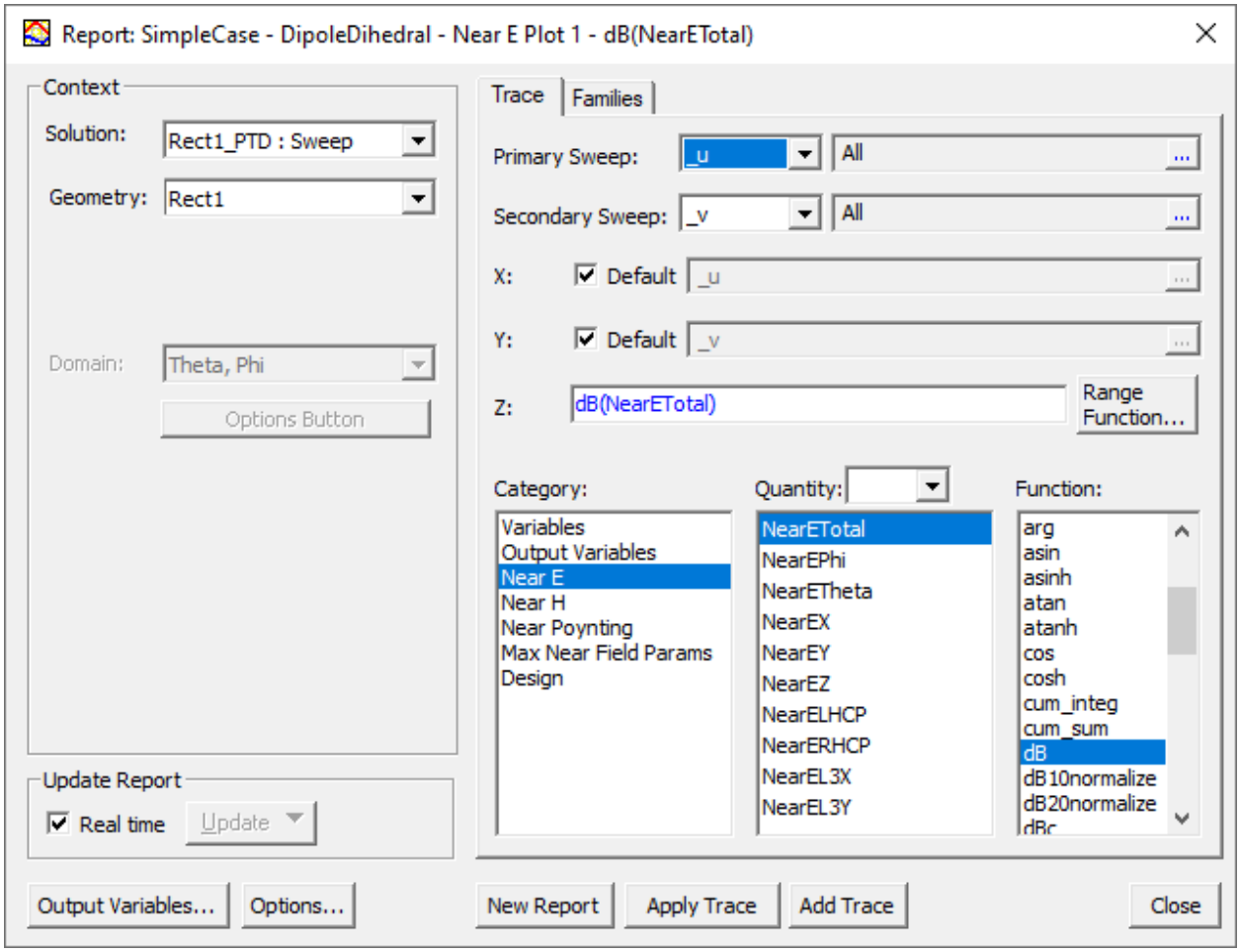


Near Field analysis is computed based on Tx or Tx/Rx role specified in the Select SBR+ Tx/Rx dialog.

6. Right click on the **Solution** setup icon the **Project** tree and click **Analyze** to run the simulation.
7. Generate a Rectangular Contour Plot of the Near Field.

Right-click on the **Results** icon in the **Project** tree and select **Results>Create Near Fields Report...>Rectangular Contour Plot**, or select the **Results** tab of the ribbon, and click on the **Near Fields Report** icon and select the 2D Contour icon. These bring up the **Reporter** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

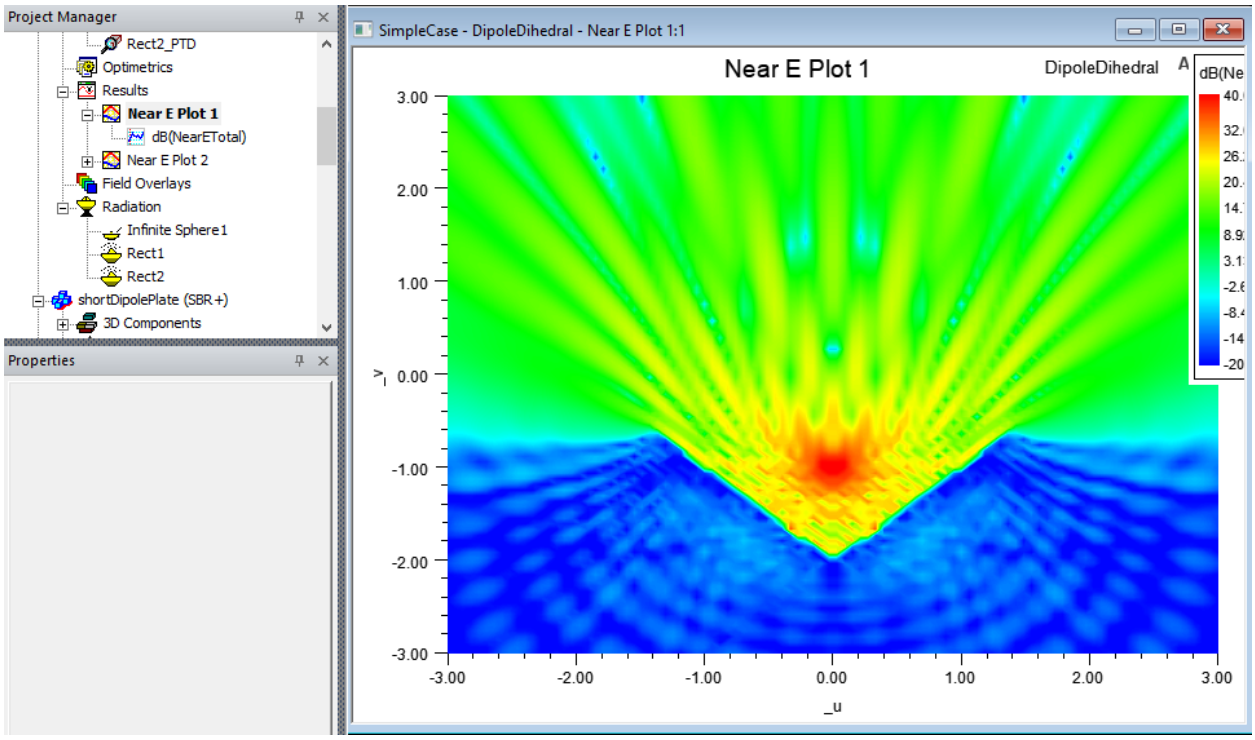


Select the Near Field radiation Field Observation Domain as the Geometry for the Context. Select Near E as the Category, NearETotal as the Quantity, and dB as the Function., and click **New Report**. This creates the Near E Rectangular Contour plot.

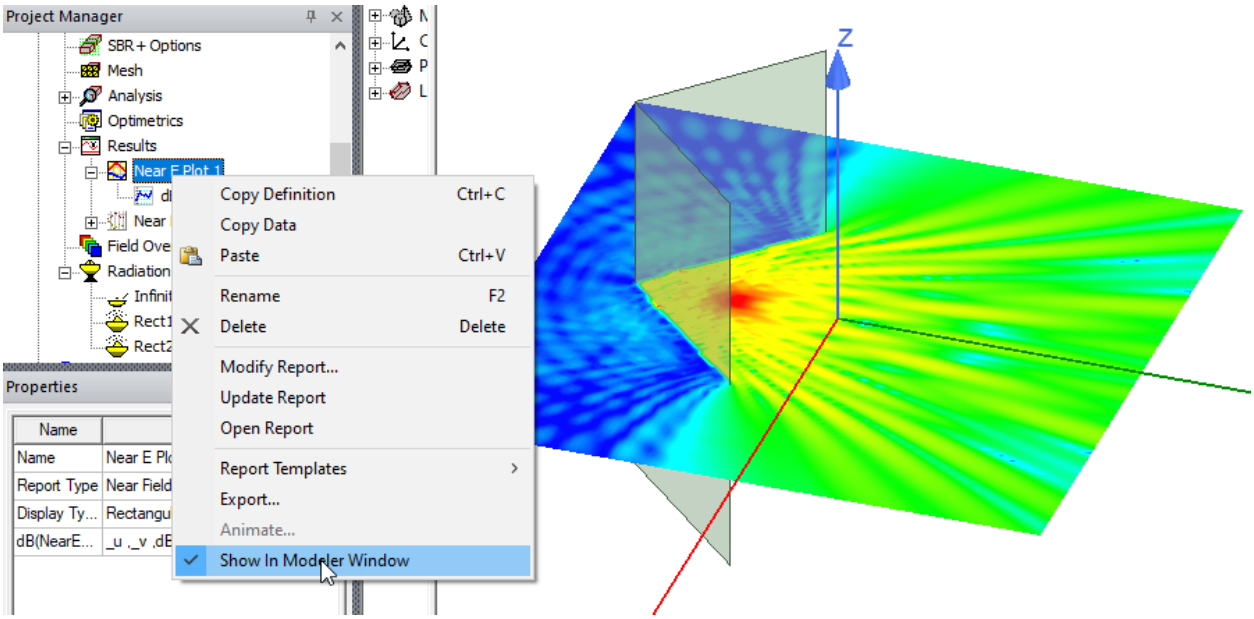
Note that these plots are associated with the local coordinate system for positions and global coordinate system for field polarizations. Point positions of Sweeps _u and _v are associated with X and Y of the local Relative CS in which the near-field sample points are defined, while the reported polarized field components are associated with the orientation of the global coordinate system. In this example, the Relative CS, named Rect2, for the Rect2 grid is rotated by theta = -90 deg around the Y axis of global coordinates. Thus, the near-field sample points are located in the XY plane of Rect2 Relative CS and also in the YZ plane of global coordinates. Point positions of the sweep _u are associated with the X axis of the local coordinate system, which is the Z axis of the global coordinate system. On the other hand, selecting NearEZ in a field report will show the electric field component oriented along the Z axis of global coordinates, not the Z axis of the Rect2 RelativeCS. It is important to

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

understand that the designated field polarizations are always oriented according to global coordinates.

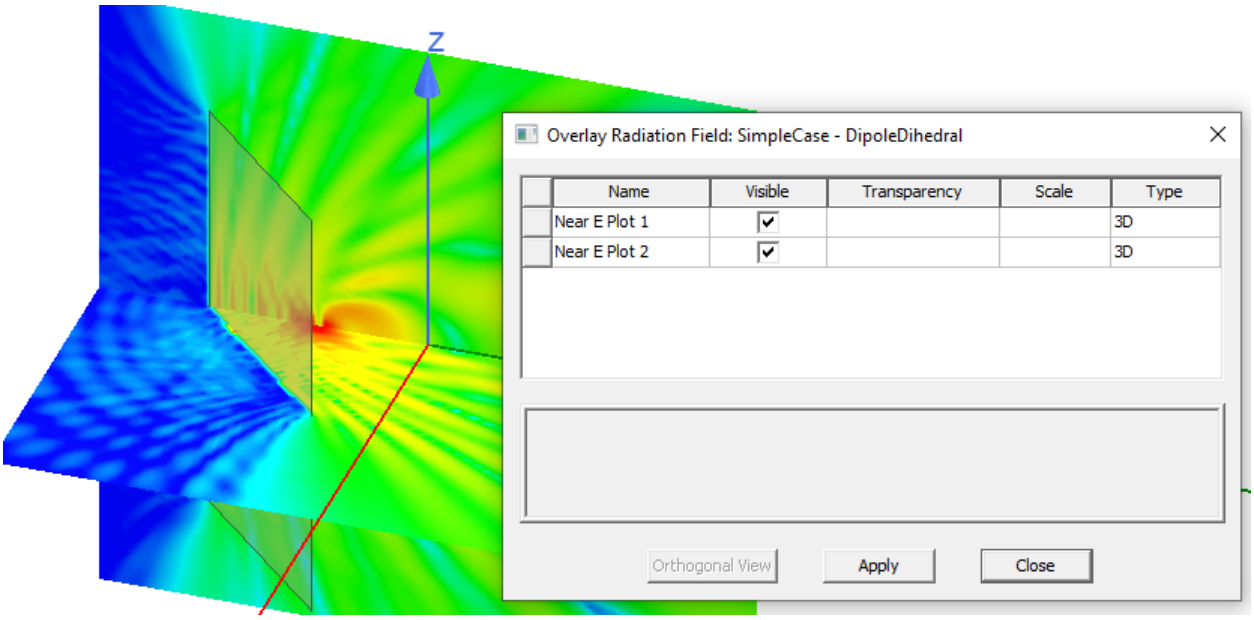


8. To overlay the near field report contour plot in **Model** window, right click on the Plot in the **Project** tree and select **Show in Modeler Window**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also right click in the Modeler window and select **Plot Fields...>Radiation Field...** to open the **Overlay Radiation Field** dialog box, and check for the plot to be visible. If more than one plot exists, you can overlay any or all of them that have current solution data.

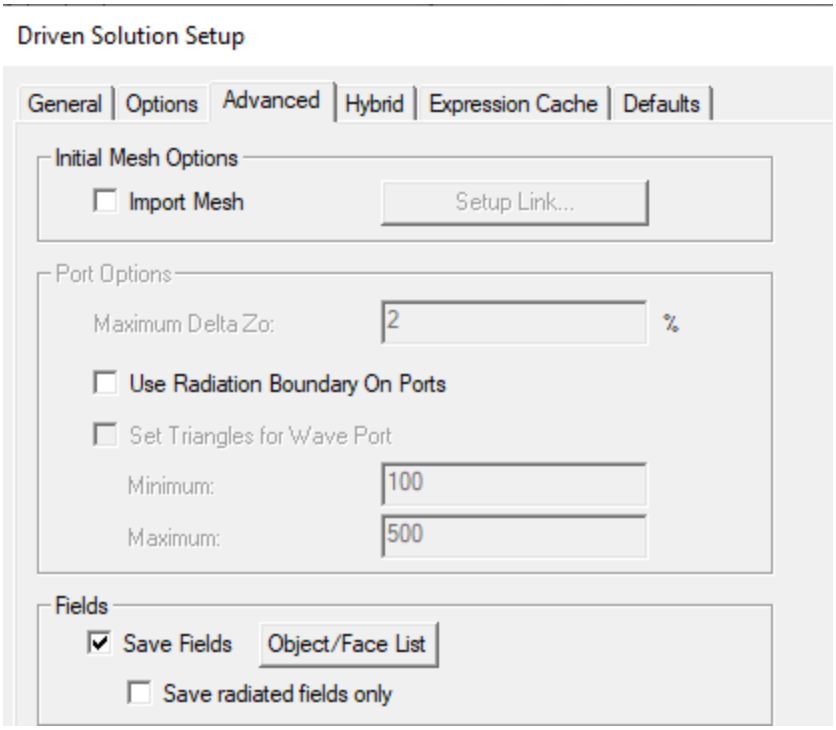


Process Flow for Near Field Analysis in HFSS Modal or Terminal Design with SBR+ Hybrid Regions

1. Create an HFSS design of either driven Modal/Terminal solution type.
2. Add scene geometry components.
3. Assign SBR+ hybrid region(s).
4. Add a source excitation.
5. For driven Modal/Terminal solution types, the antenna geometries with the excitation must be surrounded by a FE-BI, a Radiation boundary, or a Perfect E boundary.
6. Add an Analysis Setup.

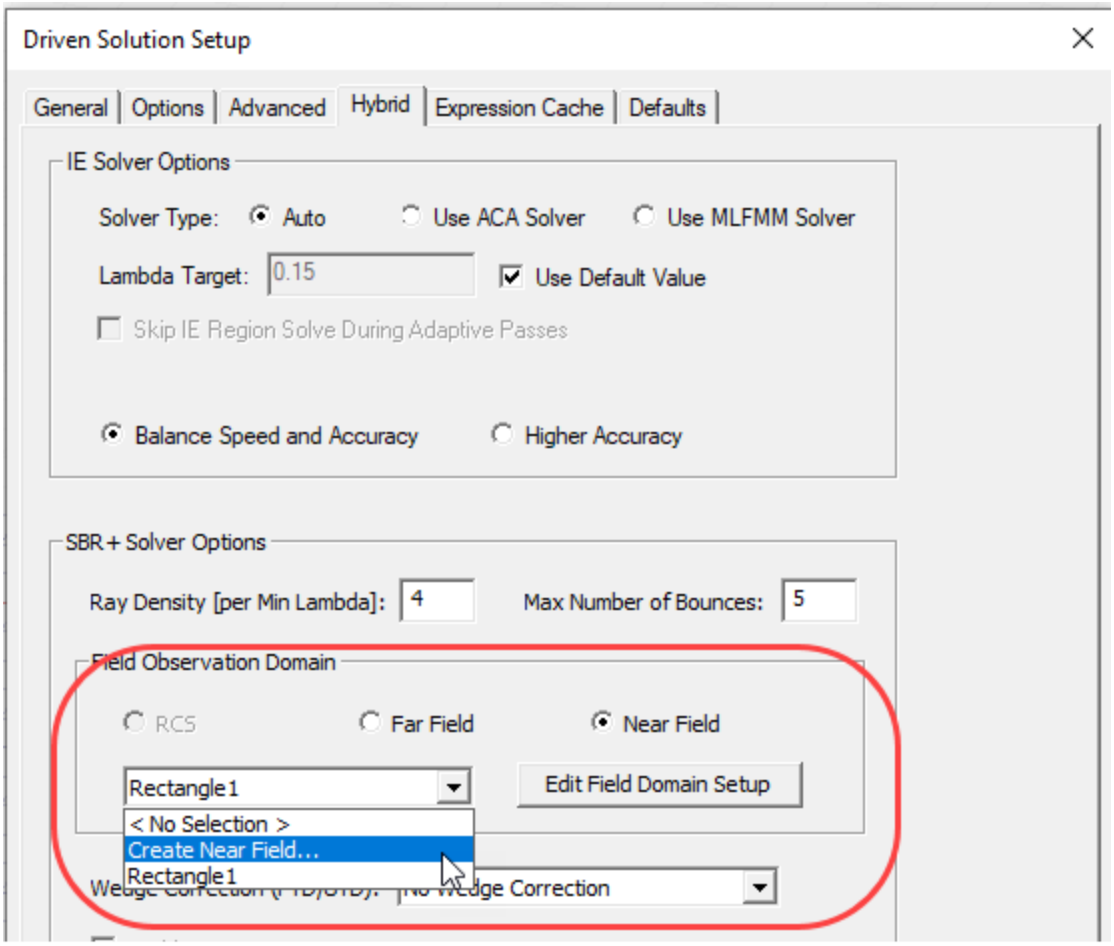
   On the **Advanced** tab check the box to Save Fields.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



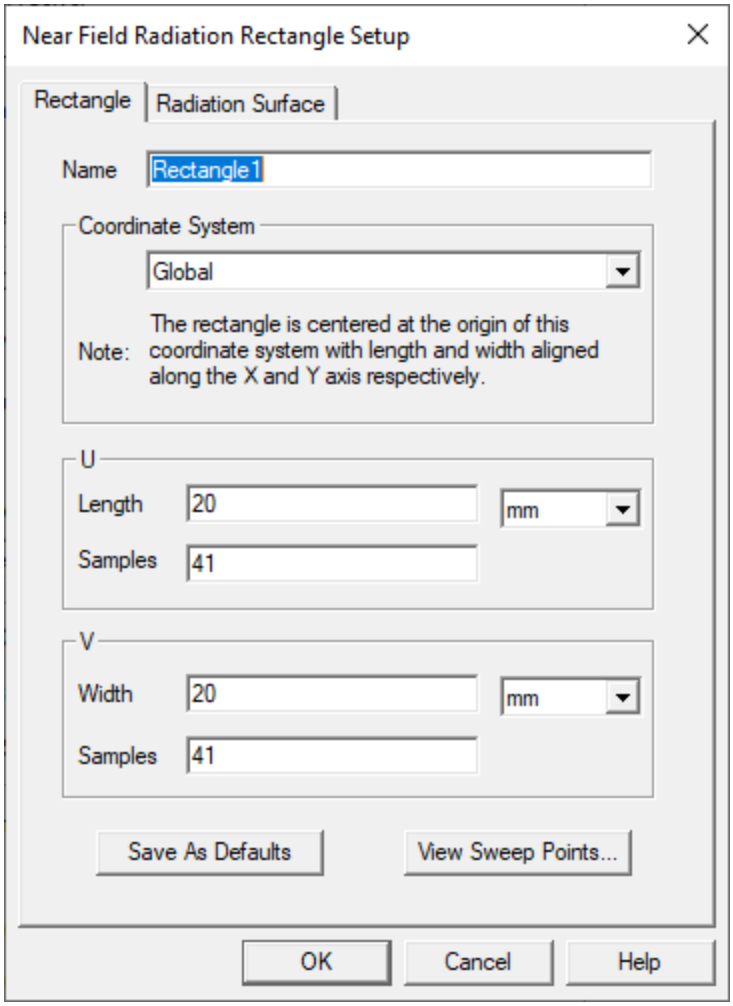
On the **Hybrid** tab, select a Field Observation Domain by clicking on the dropdown list and selecting **Create Near Field...** (or select an existing one, if already exists).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



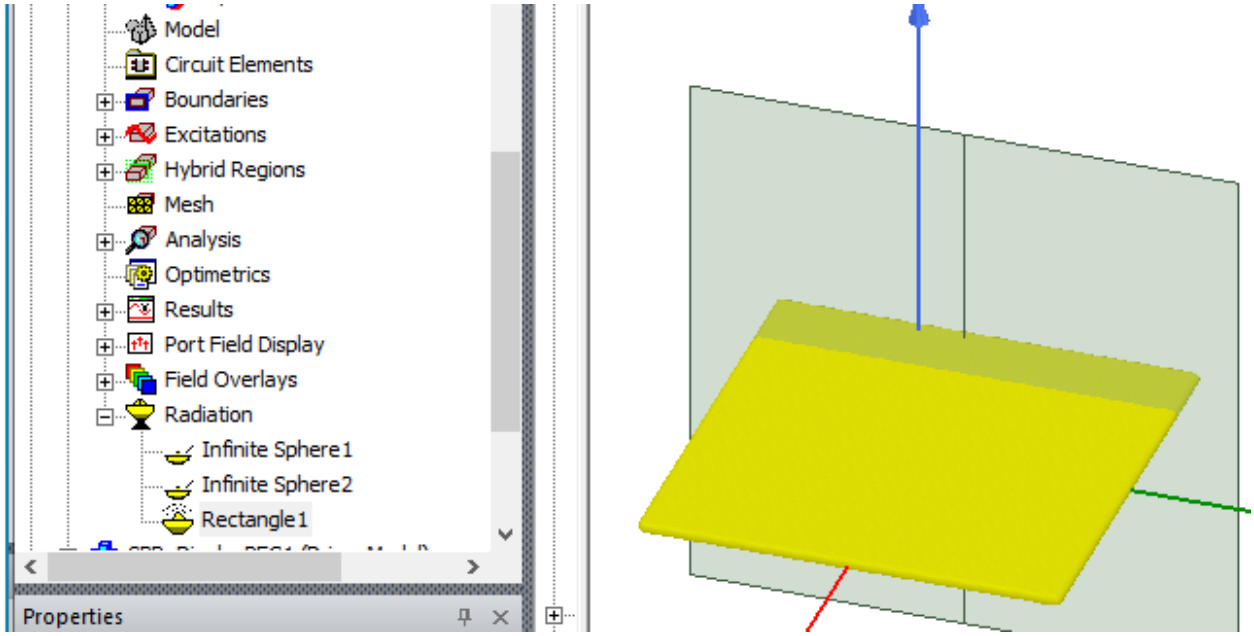
In the dialog to create the **Rectangle** setup, enter the desired parameters.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



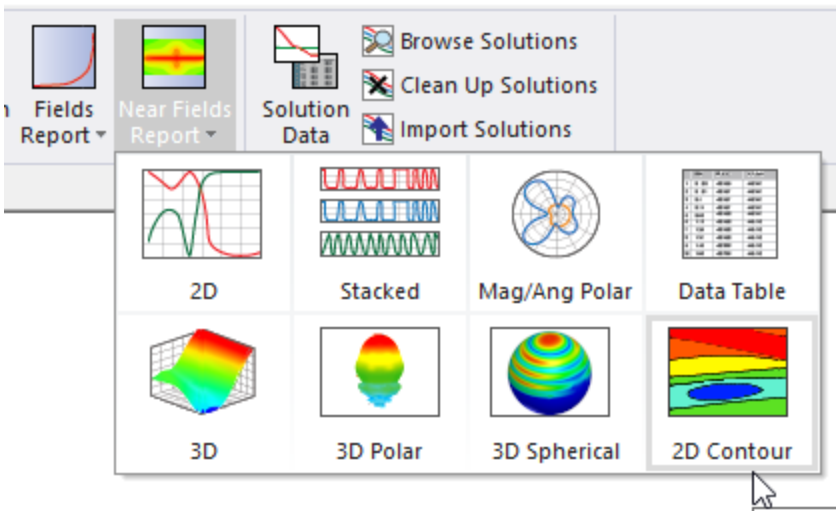
Select the newly created Radiation setup in the project tree to visualize the sample points in the 3D model window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



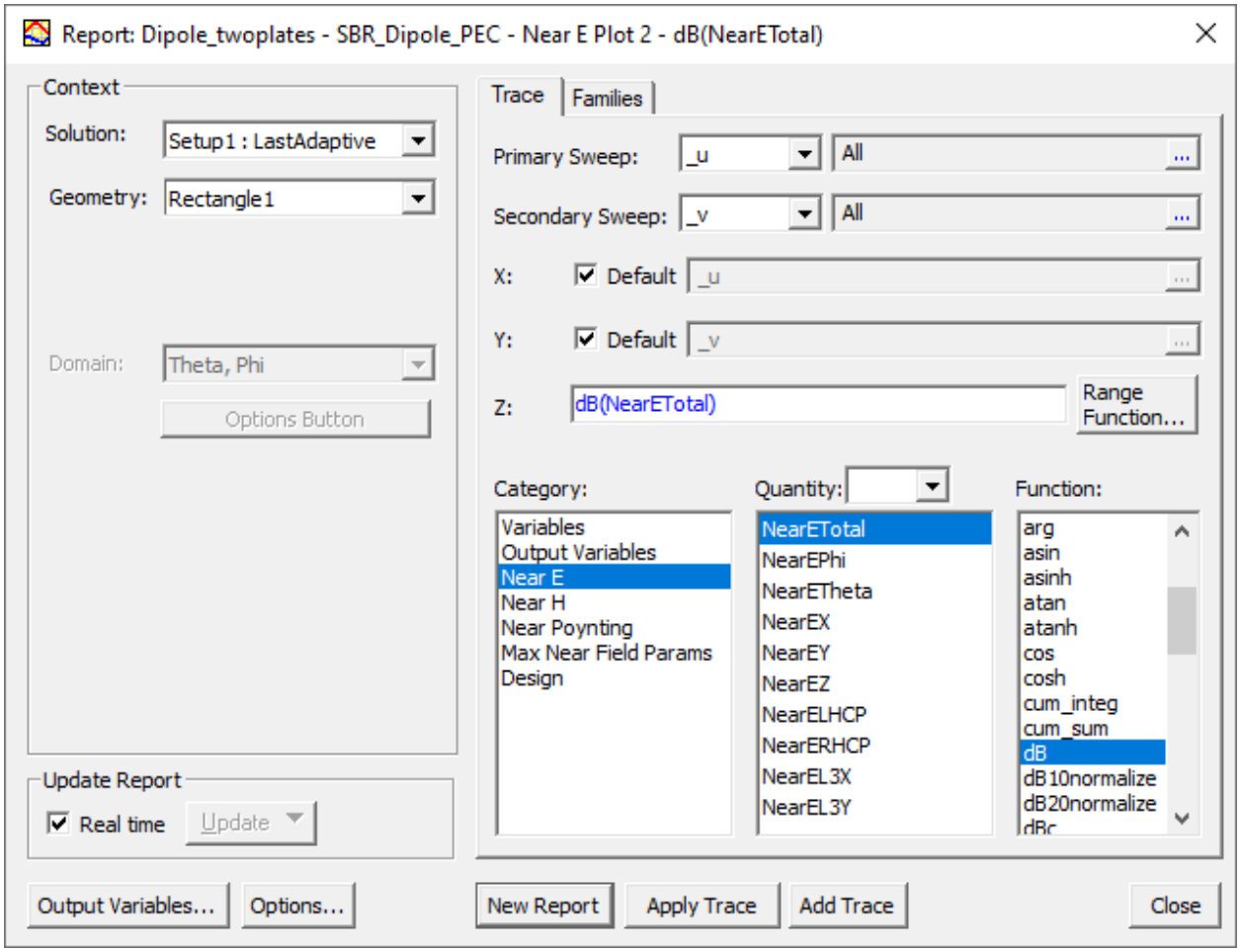
7. Run simulation
8. Generate a Rectangular Contour Plot of the Near Field.

Select **Results>Create Near Fields Report...>Rectangular Contour Plot** to bring up the **Reporter** dialog box, or on the **Results** tab of the ribbon, select 2D Contour form the Near Fields Report drop down.



In the **Report** dialog box, select the Near Field radiation setup for the report Context, and the desired Quantity and Function.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

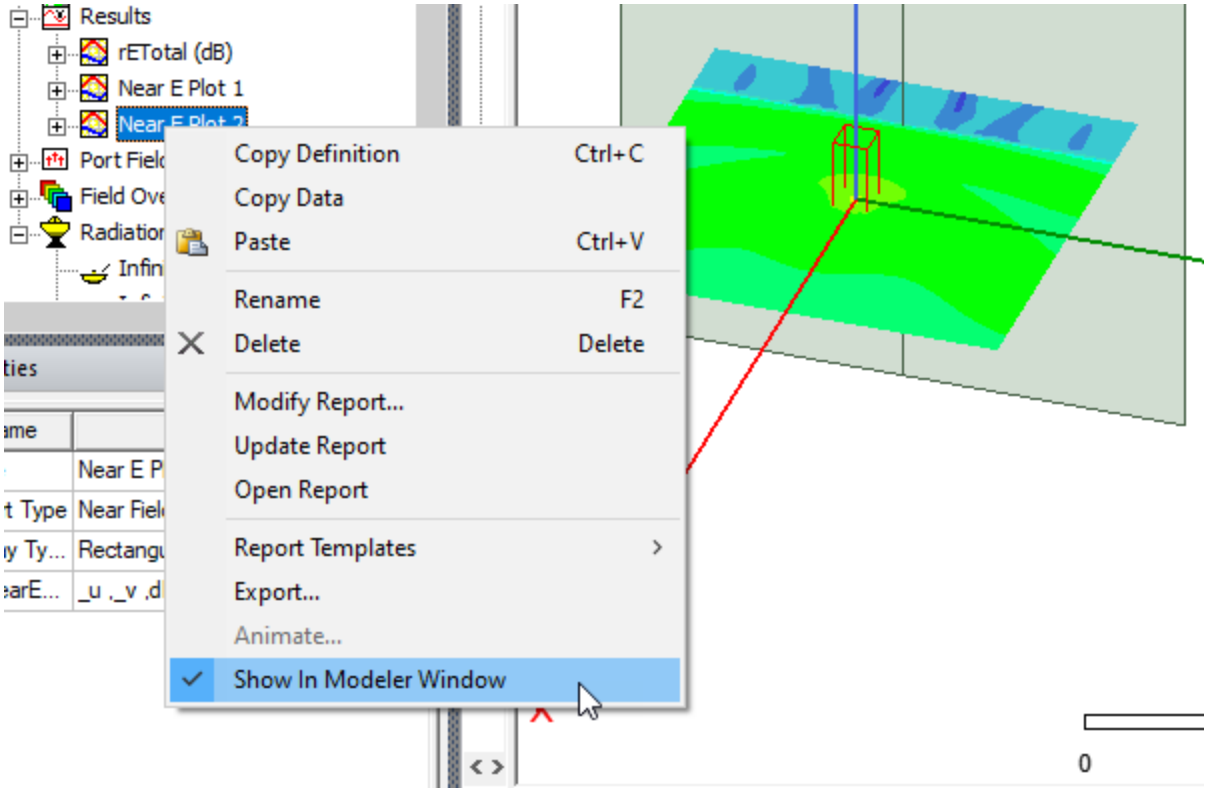


Select the Near Field radiation Field Observation Domain as the Geometry for the Context. Select Near E as the Category, NearETotal as the Quantity, and dB as the Function., and click **New Report**. This creates the Near E Rectangular Contour plot.

Be aware of that the point locations will be defined relative to the _u (X) and _v (Y) axes of the local relative coordinate system while the reported polarized field components are always aligned with global coordinates.

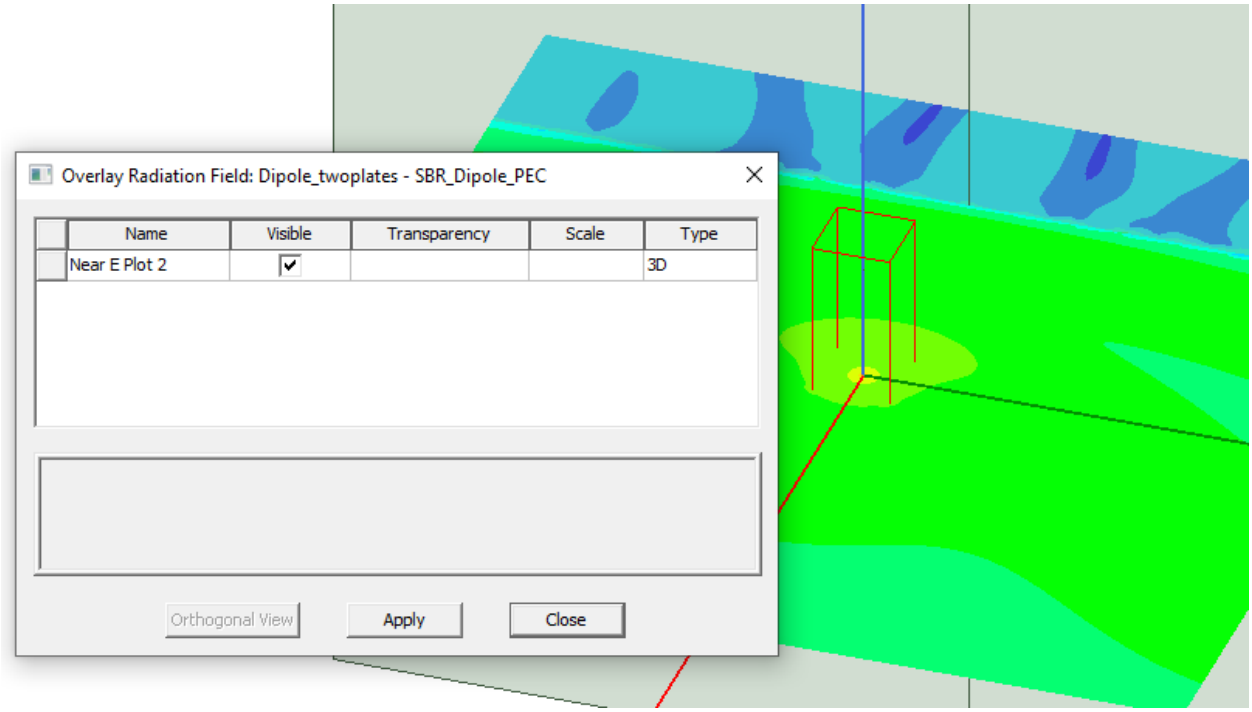
9. To overlay the near field report contour plot in **Model** window, right click on the Plot in the **Project** tree and select **Show in Modeler Window**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also right click in the Modeler window and select **Plot Fields...>Radiation Field...** to open the **Overlay Radiation Field** dialog box, and check for the plot to be visible. If more than one plot exists, you can overlay any or all of them that have current solution data.

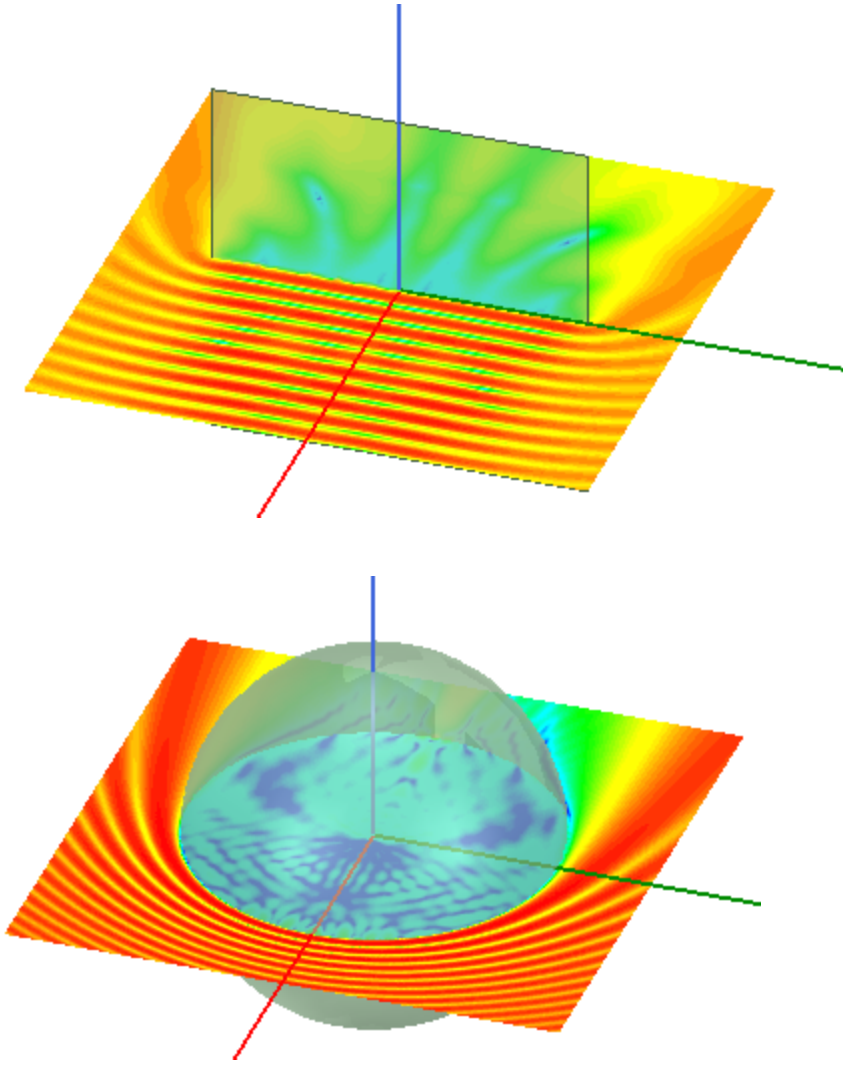
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Near Field Analysis using Incident Plane Wave in SBR+ Design

SBR+ designs and HFSS designs with SBR+ Hybrid regions can provide near field analysis and overlay rectangular near field plots on the modeler window. For SBR+, the near field setup must be defined before simulation, as is currently done for an SBR+ far field setup. The near field setups allow for a 3-D grid of sample points to be defined.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This chapter describes the process flows for Near Field Analysis using Incident Plane Wave in SBR+ designs, and for HFSS Terminal or Modal designs with SBR+ Hybrid Regions.

Prerequisites

- Create an HFSS design with SBR+ regions, either driven model or terminal solution type or an SBR+ solution type.
- Near Field analysis with SBR+ supports antenna port excitations and supports Incident Plane wave sources.
- You must specify Tx or Tx/Rx roles for antenna ports, or Incident Plane wave source to make Near Field analysis available.
- **Note:** When there are any encrypted components in the design, Near Field Analysis will be aborted.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Technical Notes

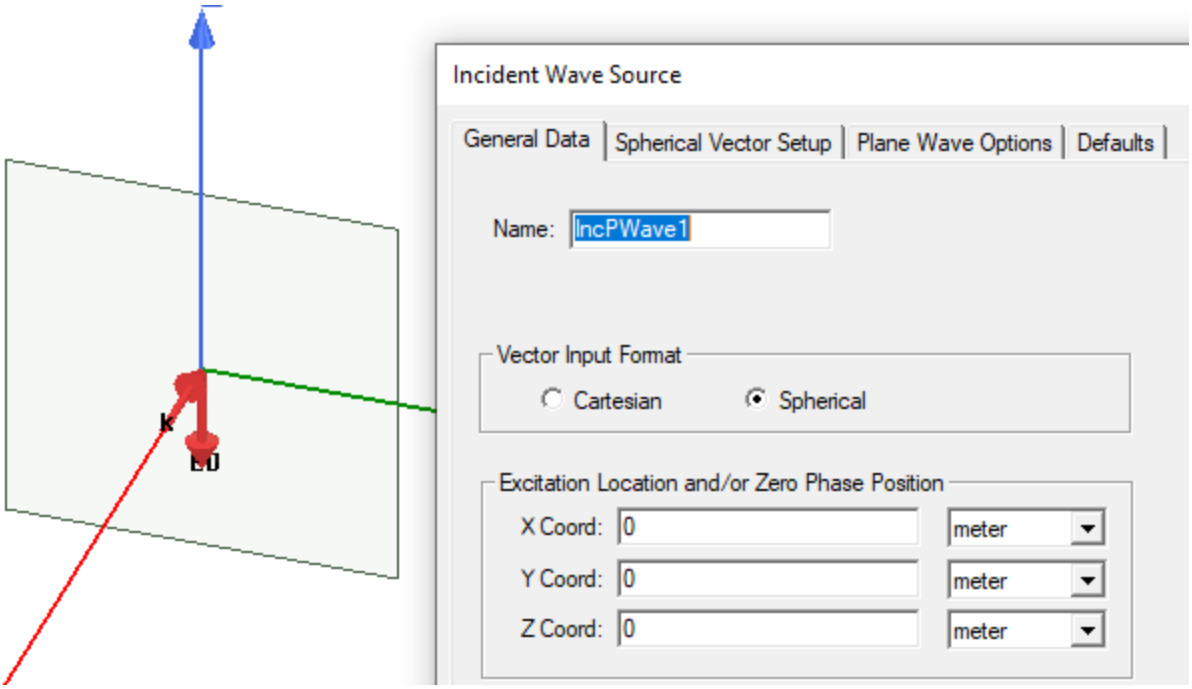
- Placing near-field observation points very close to an SBR+ region surface can increase SBR+ simulation time, sometimes significantly; when a near-field observation point is near an SBR ray footprint, the solver needs to finely subdivide the ray footprint to maintain accuracy in its scattered field contribution to that observation point."
- In the SBR+ methodology, final (total) field results are always coherently assembled upon the base of an incident field from the source radiation radiating in free space. That means the fields start strong everywhere before the SBR rays are launched. Any reduction of this initial field due to structural blockage is achieved through destructive interference with a comparably strong scattered field arising from SBR currents painted by the rays hitting the blocking structure.

  Under conditions of strong shadowing, as in the extreme example of volume enclosed by a metal surface, one expects very weak or zero fields in the shadowed region. Destructive interference is rarely perfect, and one can generally expect some finite field at every near-field observation point. Field extinction (or nulling) in an enclosed volume is only achieved to the extent of predicting currents with high fidelity upon the enclosing surface. If you observe unrealistically large fields in a deeply shadowed or enclosed region, try increasing SBR Ray Density until the results converge. You can also try deploying SBR+ physics enhancements, such as UTD Edge Diffraction Rays and, if the source is an antenna mounted on a smooth metal surface with convex curvature, Creeping Rays. However, SBR+ remains an asymptotic (high-frequency) approximation best suited for electrically large scattering geometry, so the accuracy of painted currents and therefore field extinction in shadowed spatial regions is ultimately limited by the accuracy of multi-bounce geometric optics (GO), UTD edge diffraction, and UTD creeping-wave formulations.

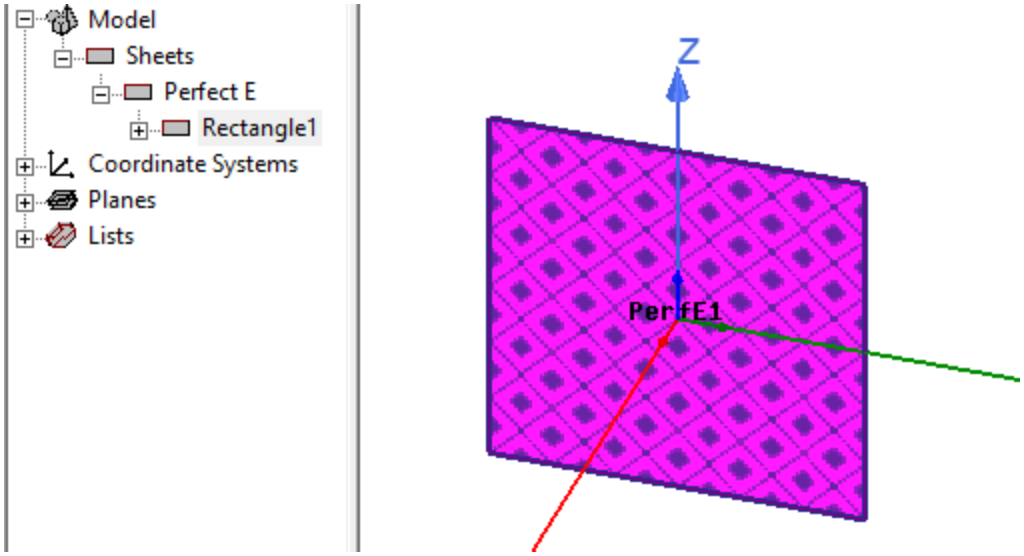
Process Flow for Near Field Analysis in SBR+ Design

1. For this design with SBR+ solution type, create an Incident Plane Wave as the example below shows.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



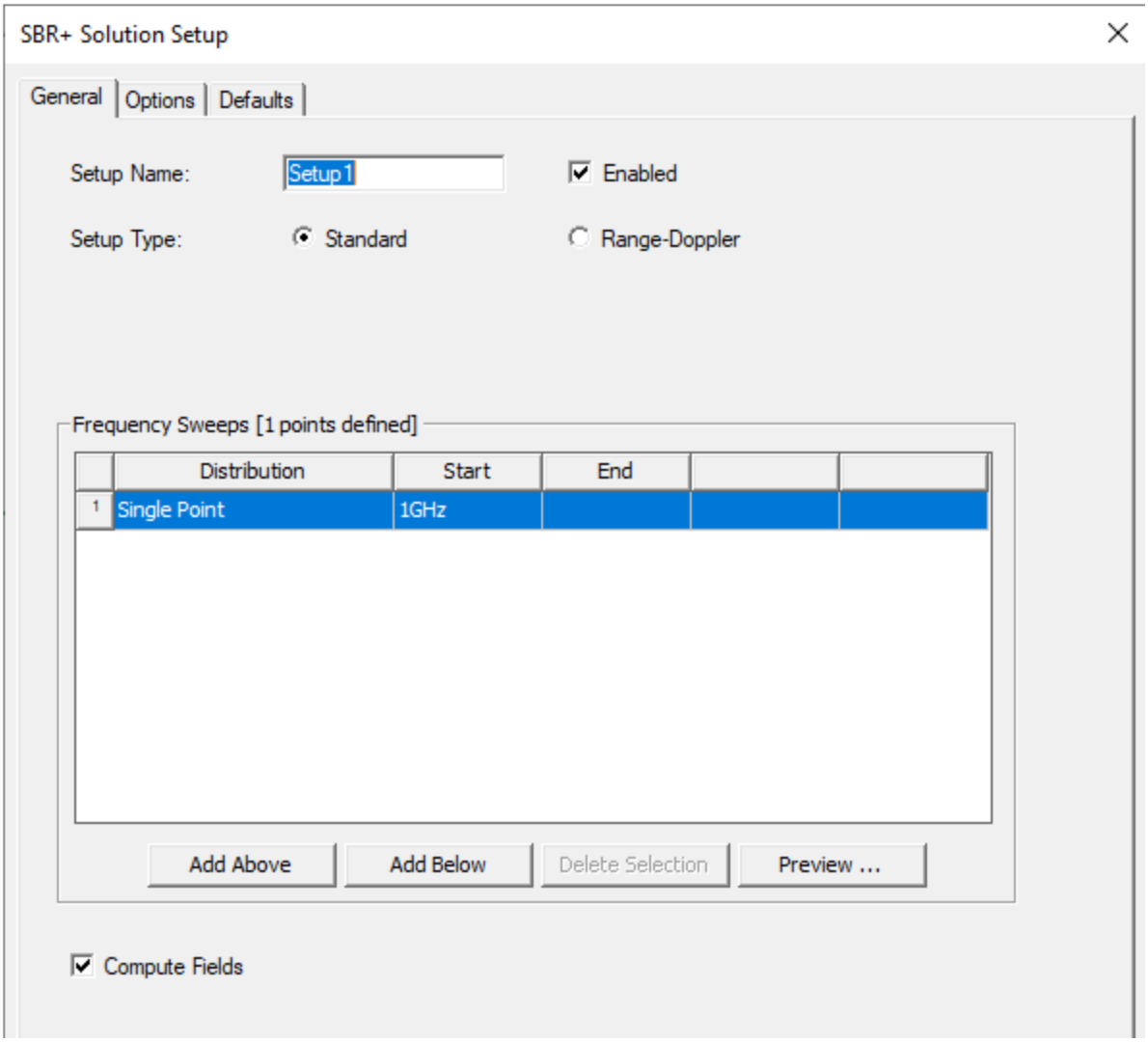
2. Add scene geometry components. This example contains a rectangle assigned a Perfect E boundaries.



3. Right-click on the **Analysis** icon in the Project tree and select **Add Solution Setup...**, or select the **Simulation** tab of the ribbon and select the **Setup** icon.

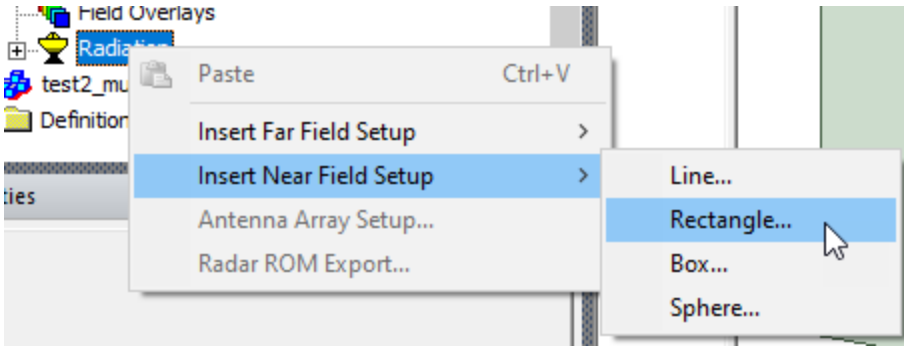
The **SBR+ Solution Setup** dialog displays. On the **General** tab check the box to **Compute Fields**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

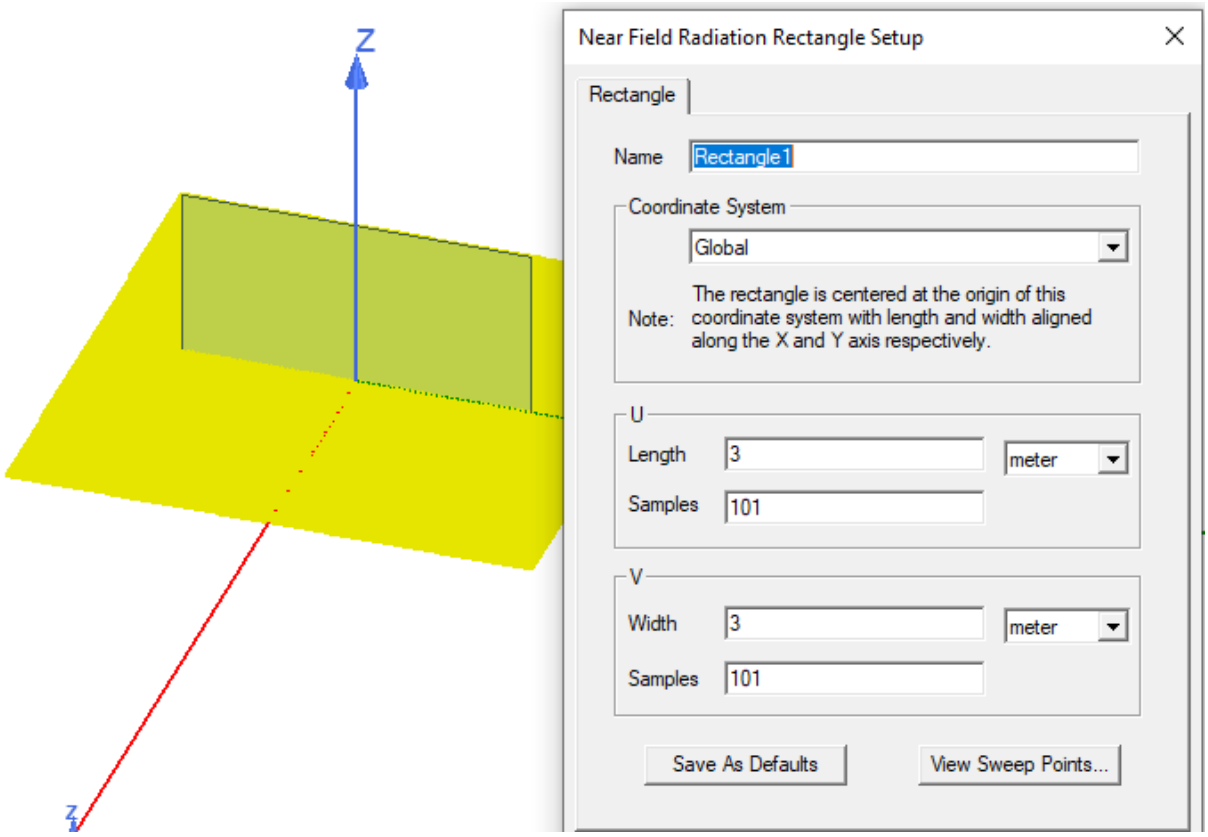


4. Next we need to create a Near Field observation domain (a Near Field Radiation Setup) and assign it to the solution setup. Right-click on the Radiation icon in the **Project** tree and select **Insert Near Field Setups>** and select **Rectangle...** from the menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



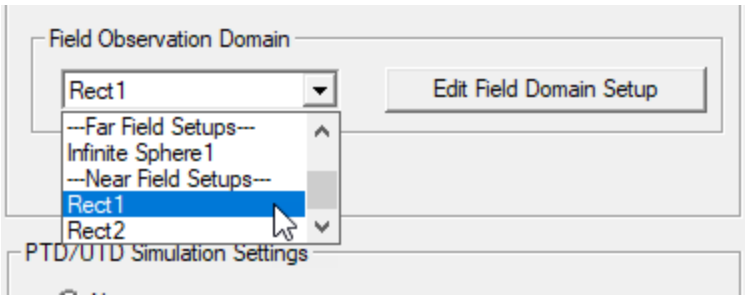
This opens a **Near Field Radiation Rectangle Setup** dialog box. You specify the name, Coordinate System, U and V length, units, and number of samples. To move or re-oriented the near-field sampling grid, set the Coordinate System to one with the desired translation and rotation, first creating a new coordinate system for this purpose if needed. This first example shows a setup based on the Global Coordinate system.



Returning to the **Analysis Setup** dialog box, the **Options** tab contains the Field Observation Domain configuration. Any Near Field or Far Field setups that you create, or that exist for the

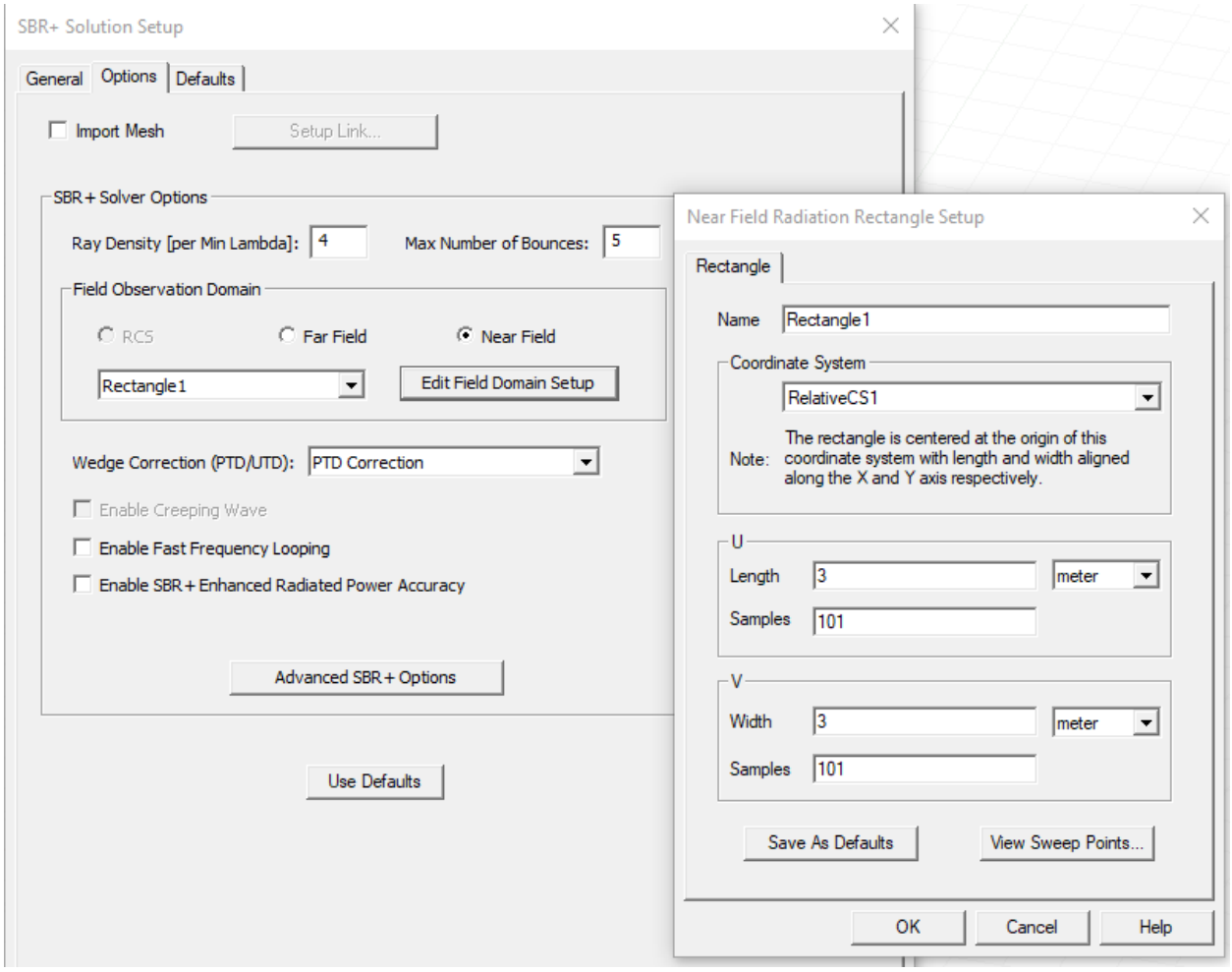
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

design, are listed in the **Field Observation Domain** drop down menu of **Options** tab of the **SBR+ Solution Setup**.

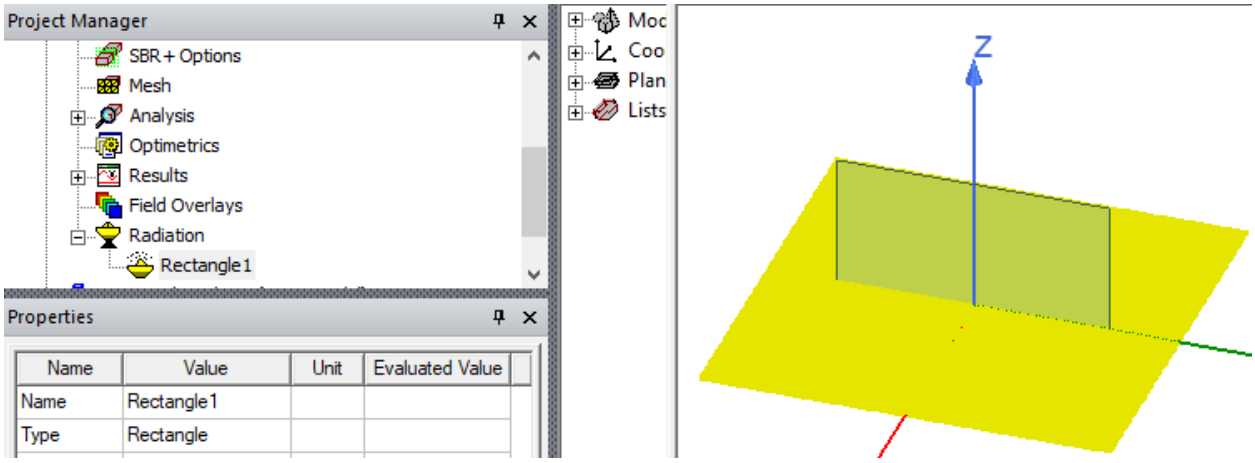


The **Field Observation Domain** drop down menu can also be used to directly create a new domain by selecting **"Create Near Field..."** The setup dialog for the **Near Field Radiation Rectangle** is then used to create/configure the Rectangle. Closing the dialog returns to the Analysis setup. This is made available as a shortcut, so note that the full set of Near Field Radiation setups is not available here. Those setups (Box, Line, etc.) can be created via the Radiation object in the Project tree as previously described. Note this example also assigns PTD Correction to include diffraction of the sharp plate edges in the simulation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



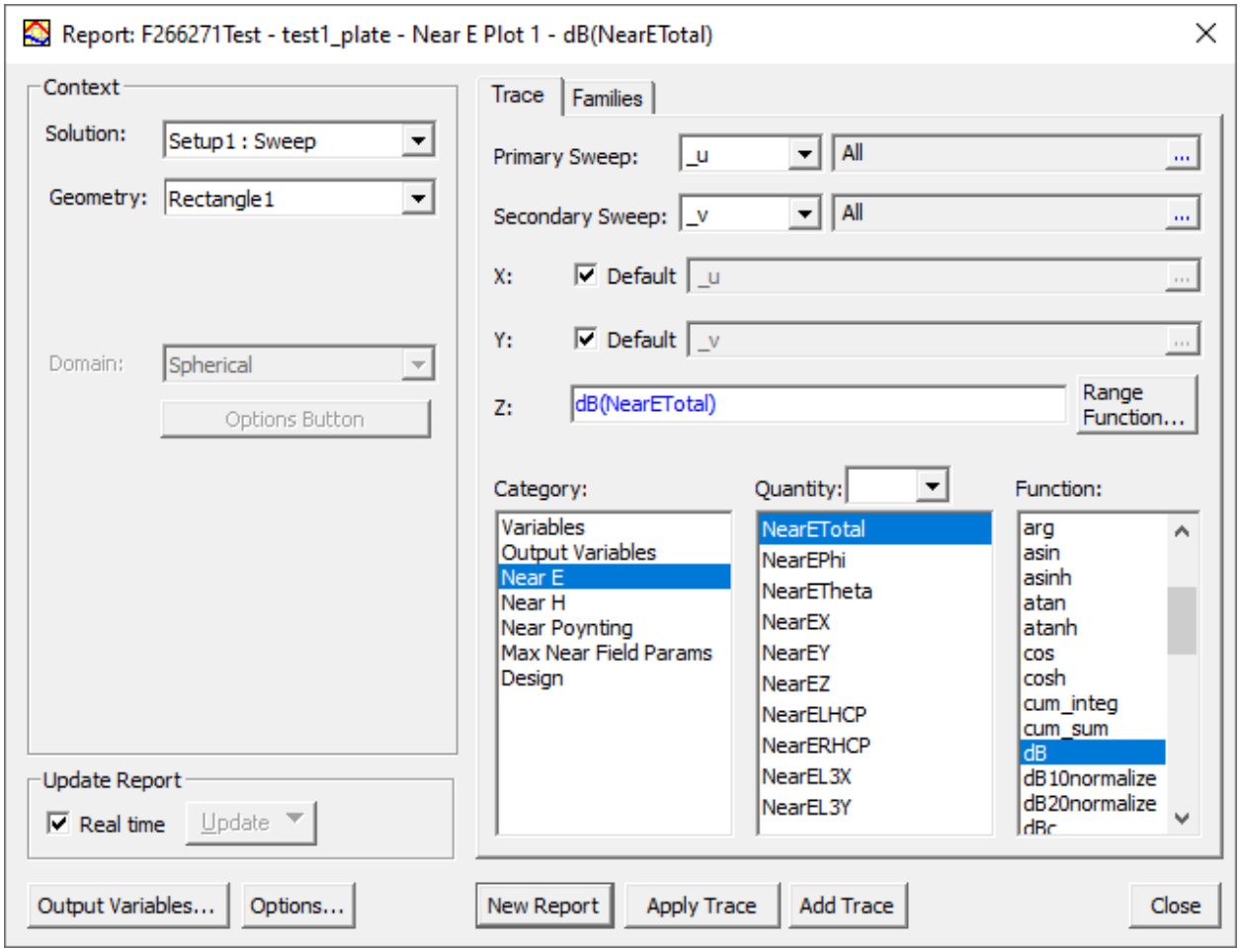
5. To visualize the sample points in the Modeler window, select the Radiation setup in the Project tree.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Right click on the **Solution** setup icon the **Project** tree and click **Analyze** to run the simulation.
7. Generate a Rectangular Contour Plot of the Near Field.

Right-click on the **Results** icon in the **Project** tree and select **Results>Create Near Fields Report...>Rectangular Contour Plot**, or select the **Results** tab of the ribbon, and click on the **Near Fields Report** icon and select the 2D Contour icon. These bring up the **Reporter** dialog.

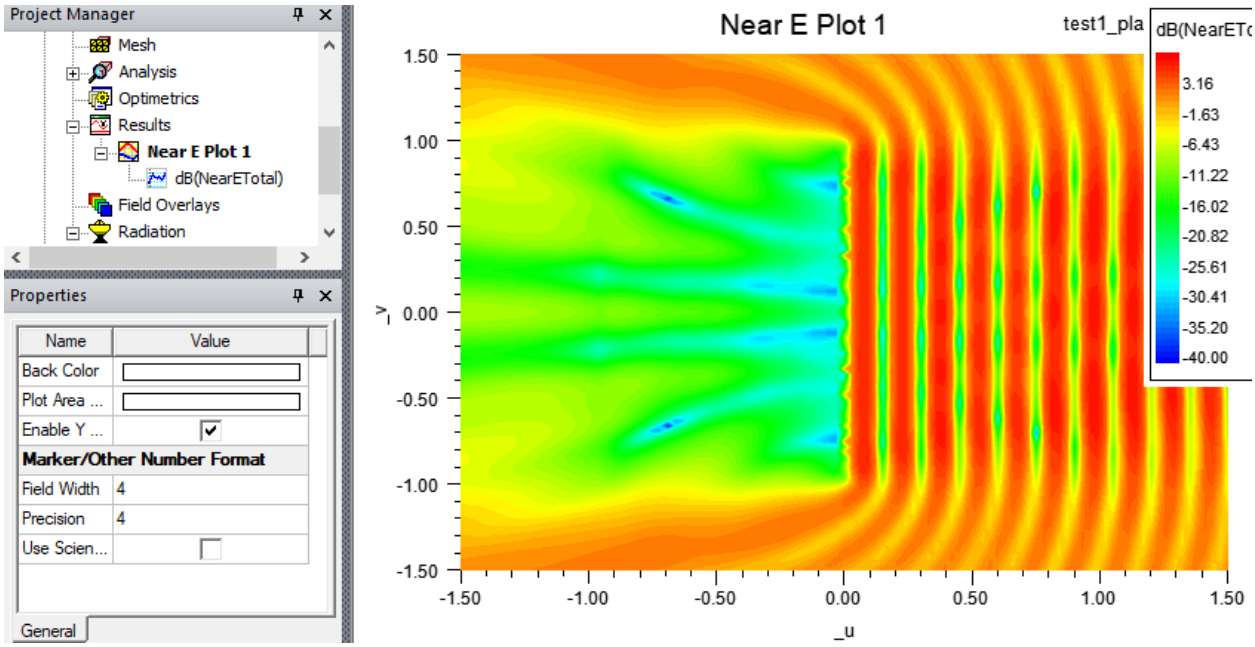


Select the Near Field radiation Rectangle1 as the for the Context. Select Near E as the Category, NearETotal as the Quantity, and dB as the Function., and click **New Report**. This creates the Near E Rectangular Contour plot.

Note that these plots are associated with the local coordinate system for positions and global coordinate system for field polarizations. Point positions of Sweeps _u and _v are associated with X and Y of the local Relative CS in which the near-field sample points are defined, while the reported polarized field components are associated with the orientation of the global

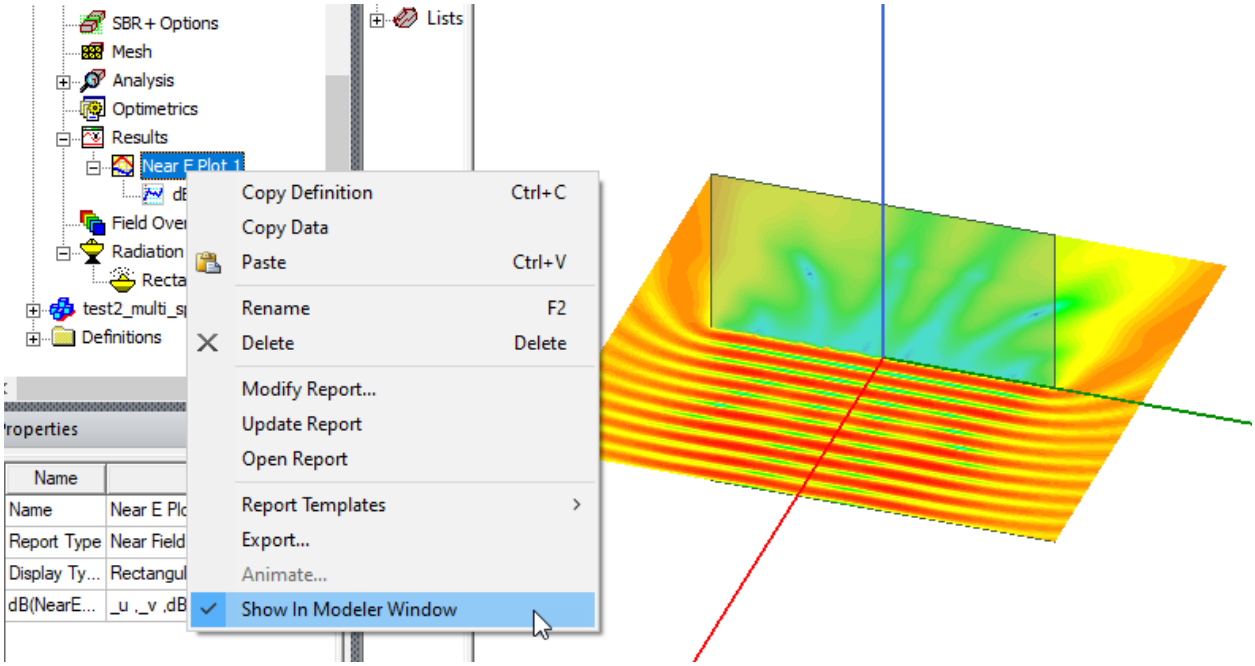
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

coordinate system. For example, suppose we define the Relative CS, named Rect2, for the Rect2 grid is rotated by theta = -90 deg around the Y axis of global coordinates. Thus, the near-field sample points are located in the XY plane of Rect2 Relative CS and also in the YZ plane of global coordinates. Point positions of the sweep _u are associated with the X axis of the local coordinate system, which is the Z axis of the global coordinate system. On the other hand, selecting NearEZ in a field report will show the electric field component oriented along the Z axis of global coordinates, not the Z axis of the Rect2 RelativeCS. It is important to understand that the designated field polarizations are always oriented according to global coordinates.

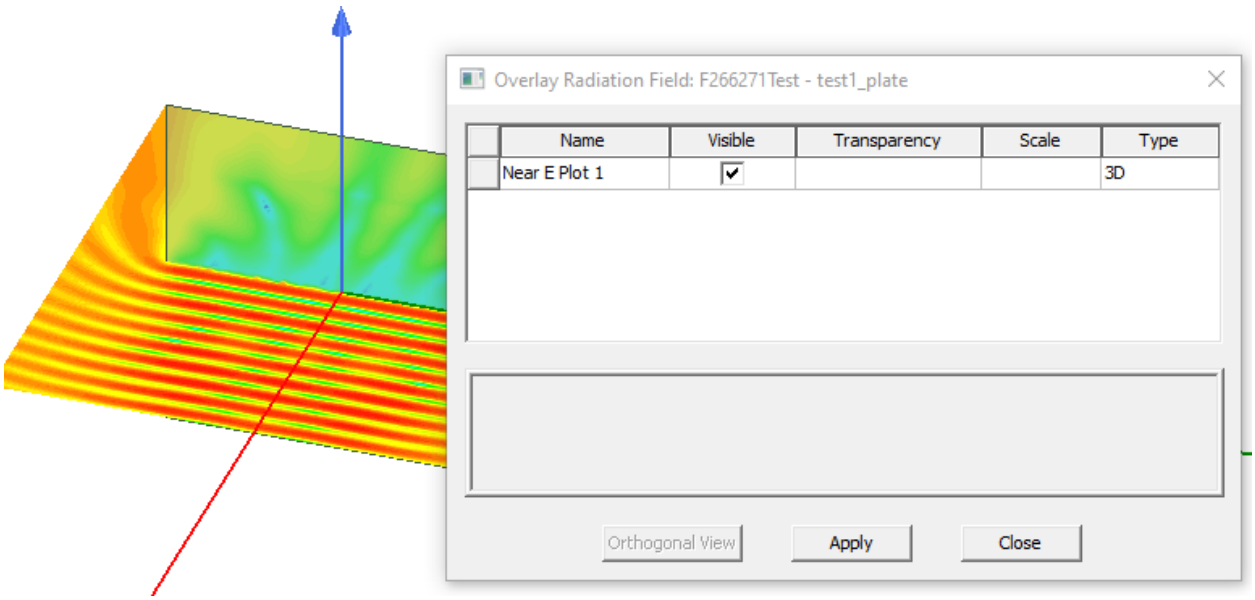


8. To overlay the near field report contour plot in **Model** window, right click on the Plot in the **Project** tree and select **Show in Modeler Window**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



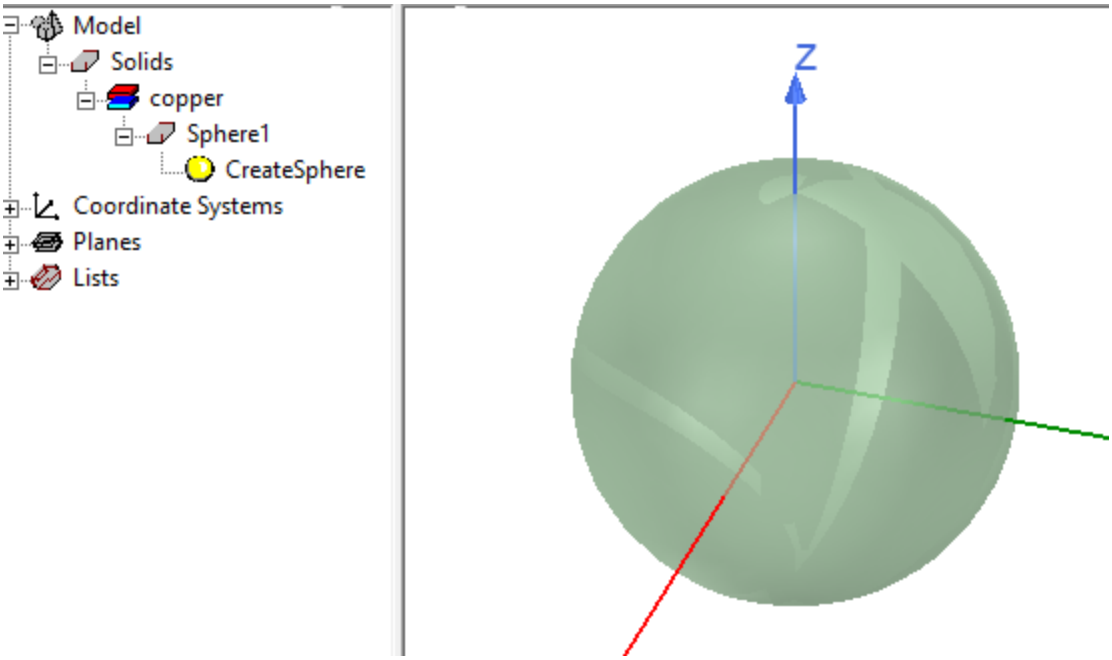
You can also right click in the Modeler window and select **Plot Fields...>Radiation Field...** to open the **Overlay Radiation Field** dialog box, and check for the plot to be visible. If more than one plot exists, you can overlay any or all of them that have current solution data.



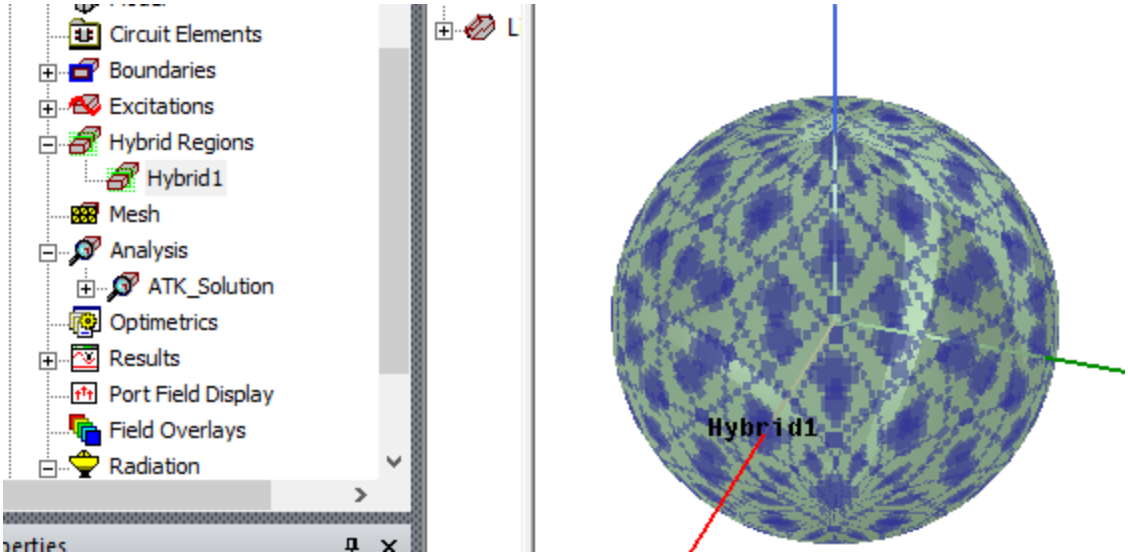
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Process Flow for Near Field Analysis in HFSS Modal or Terminal Design with SBR+ Hybrid Regions

1. Create an HFSS design of HFSS with Hybrid and Arrays solution type as either Modal or Terminal.
2. Add scene geometry components. For this example, we add a copper sphere.

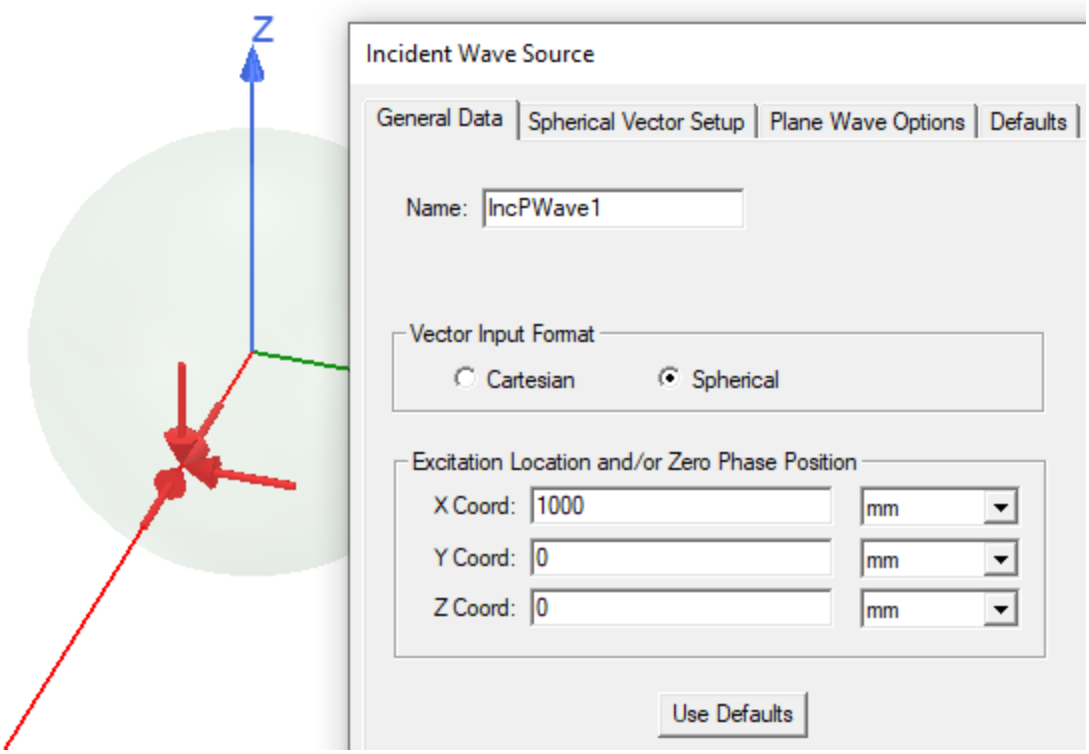


3. Assign SBR+ hybrid region(s). In this example, assign the and SBR+ Hybrid Region to the sphere.

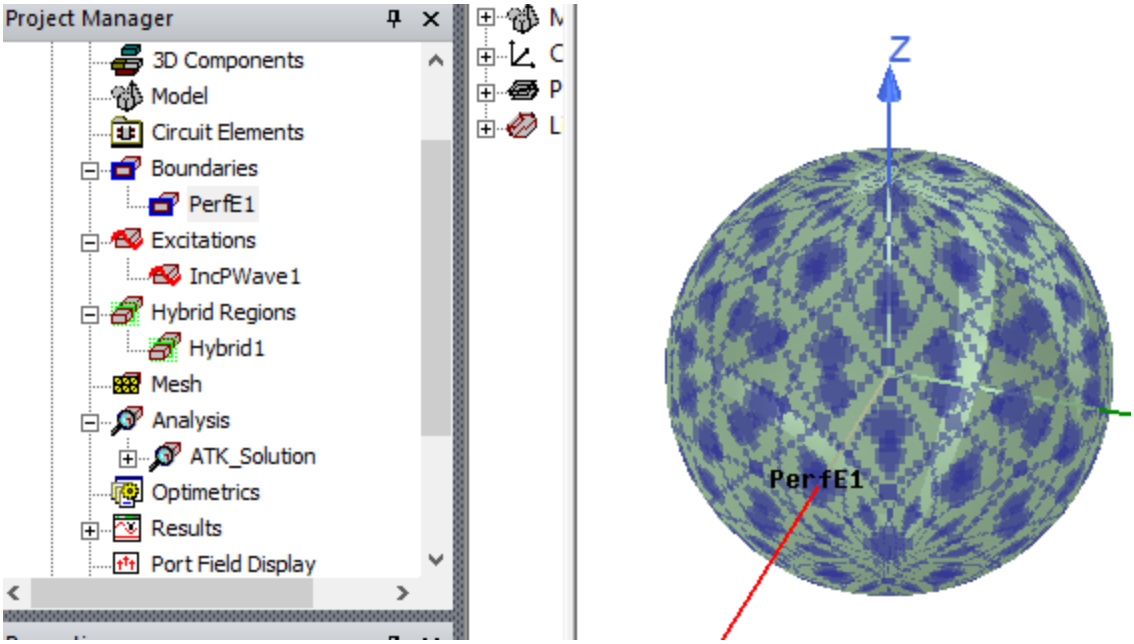


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Add an Incident Plane Wave source excitation.



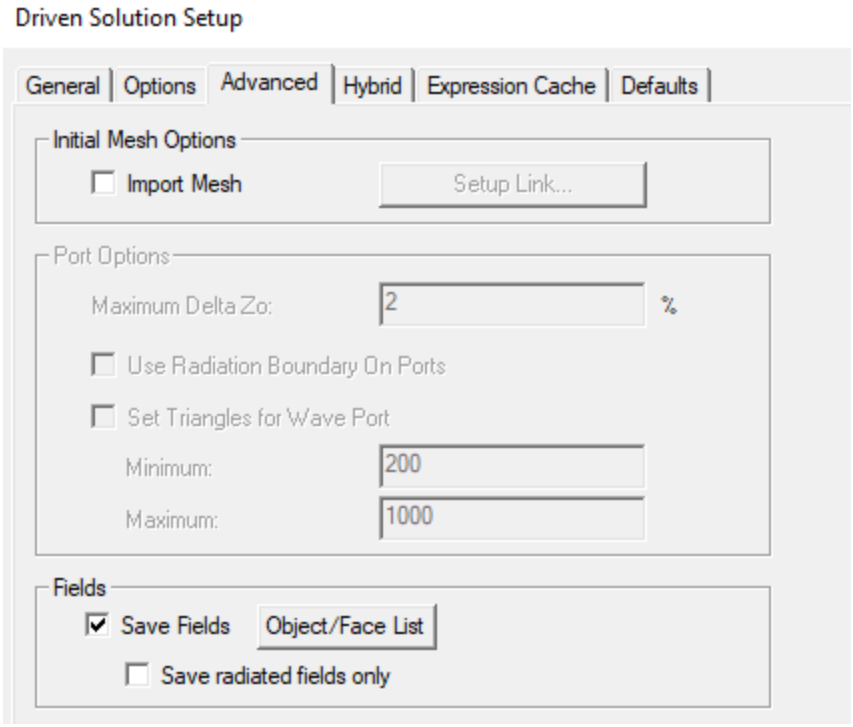
5. For driven Modal/Terminal solution types, the antenna geometries with the excitation must be surrounded by a FE-BI, a Radiation boundary, or a Perfect E boundary. In this example, we assign the sphere a Perfect E boundary.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

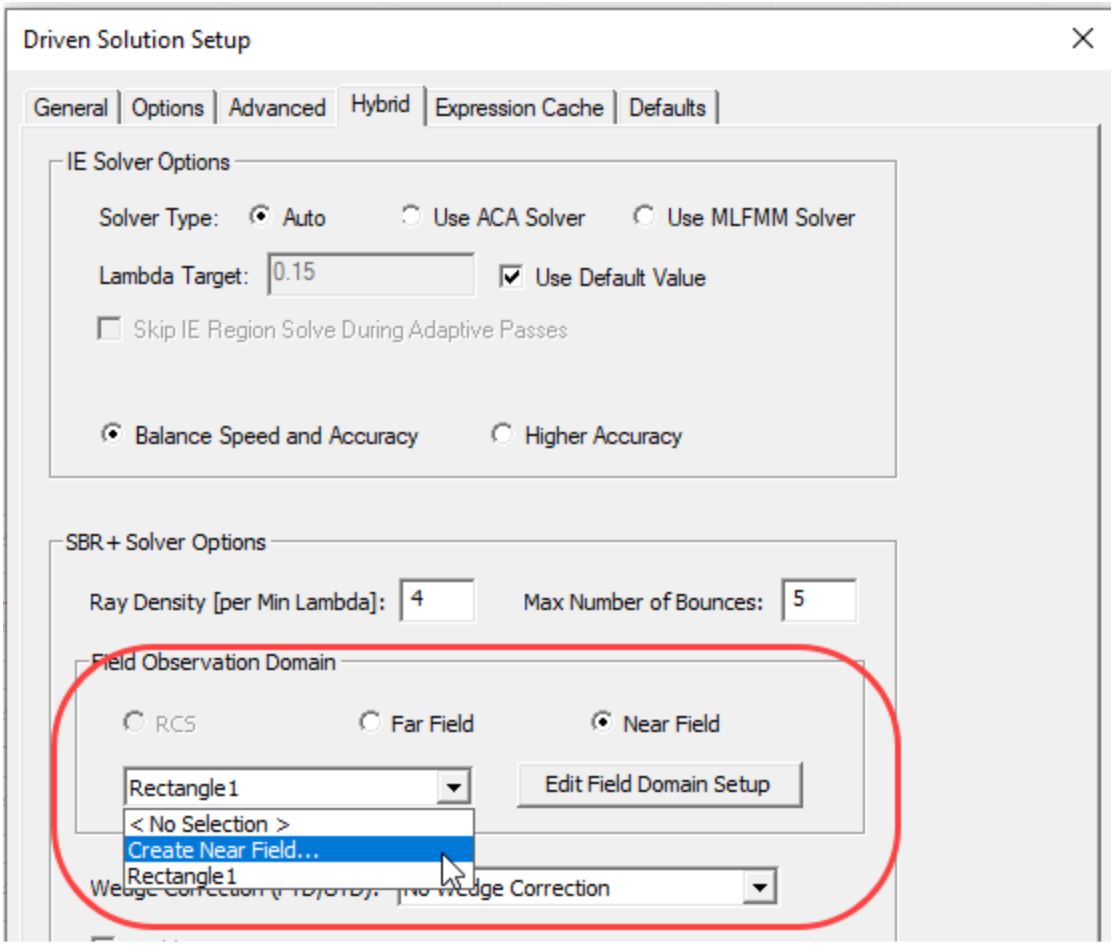
6. Add an Analysis Setup.

On the **Advanced** tab check the box to Save Fields.



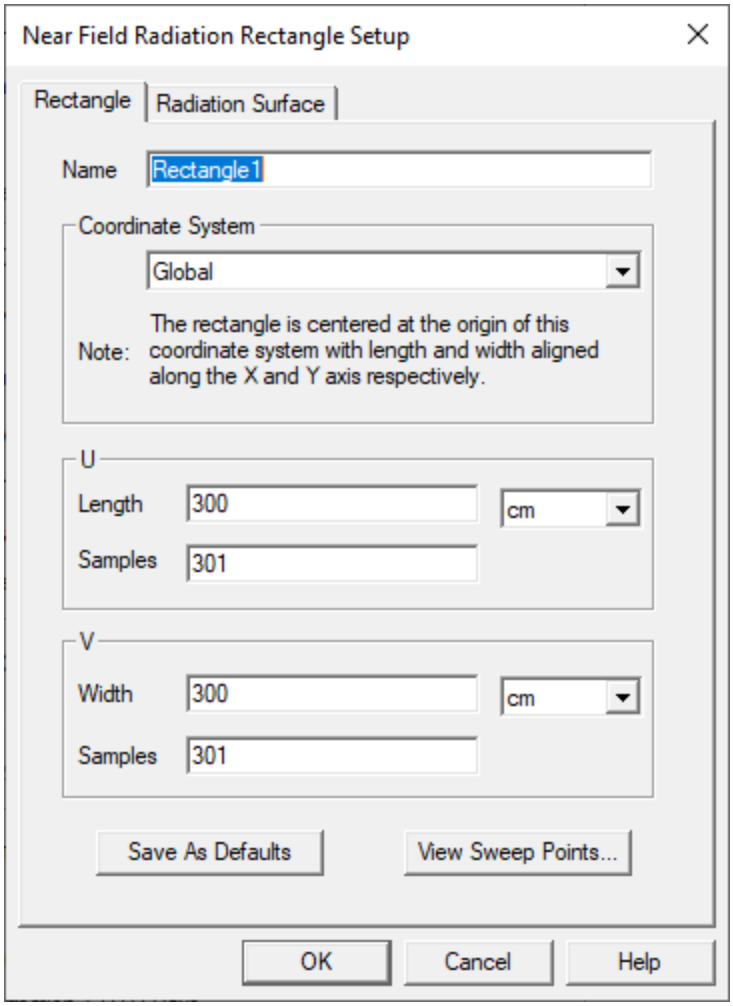
On the **Hybrid** tab, select a Field Observation Domain by clicking on the dropdown list and selecting **Create Near Field...** (or select an existing one, if already exists).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



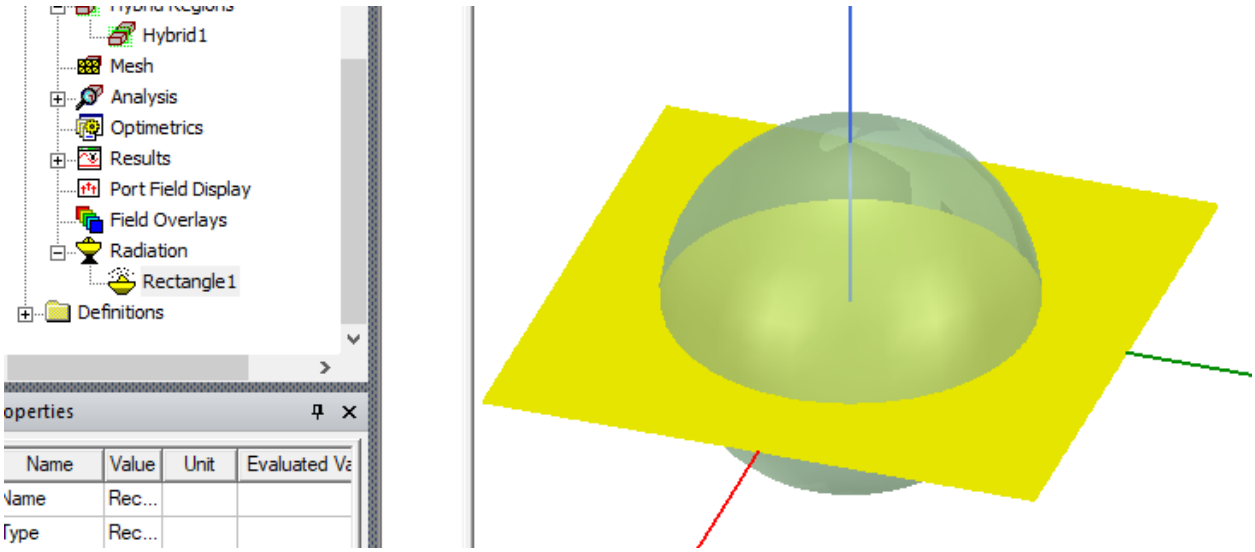
In this example, we **Enable Creeping Wave**. In the dialog to create the **Rectangle** setup, enter the desired parameters.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



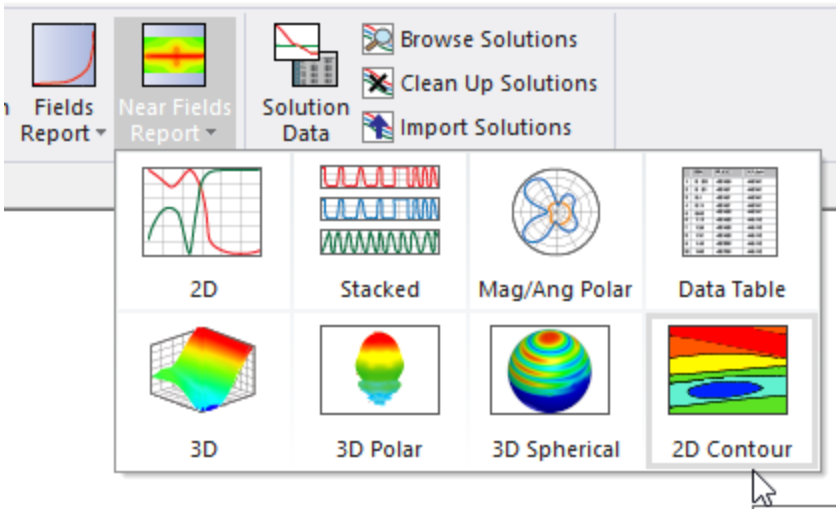
Select the newly created Radiation setup in the project tree to visualize the sample points in the 3D model window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



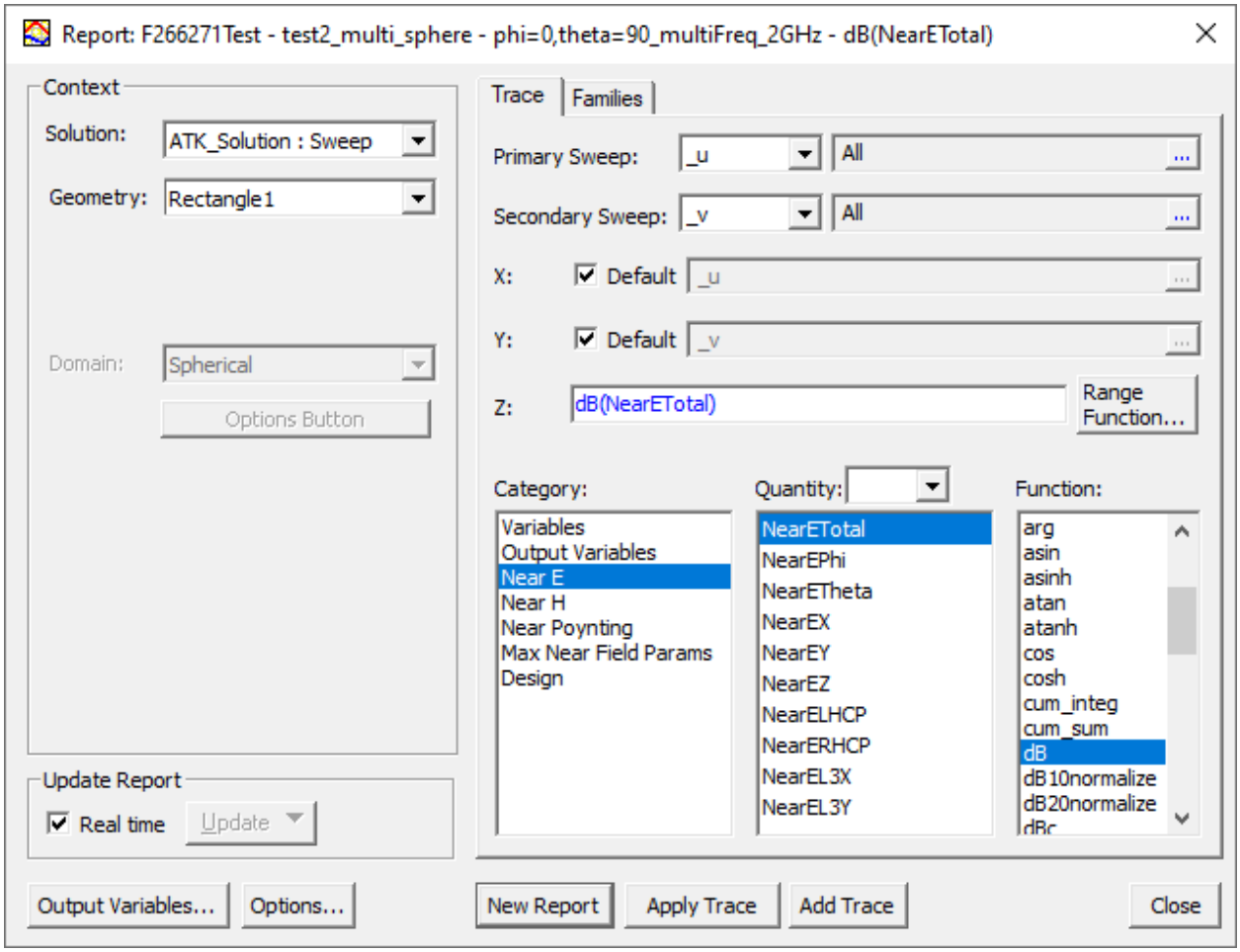
7. Run simulation
8. Generate a Rectangular Contour Plot of the Near Field.

Select **Results>Create Near Fields Report...>Rectangular Contour Plot** to bring up the **Reporter** dialog box, or on the **Results** tab of the ribbon, select 2D Contour form the Near Fields Report drop down.



In the **Report** dialog box, select the Near Field radiation setup for the report Context, and the desired Quantity and Function.

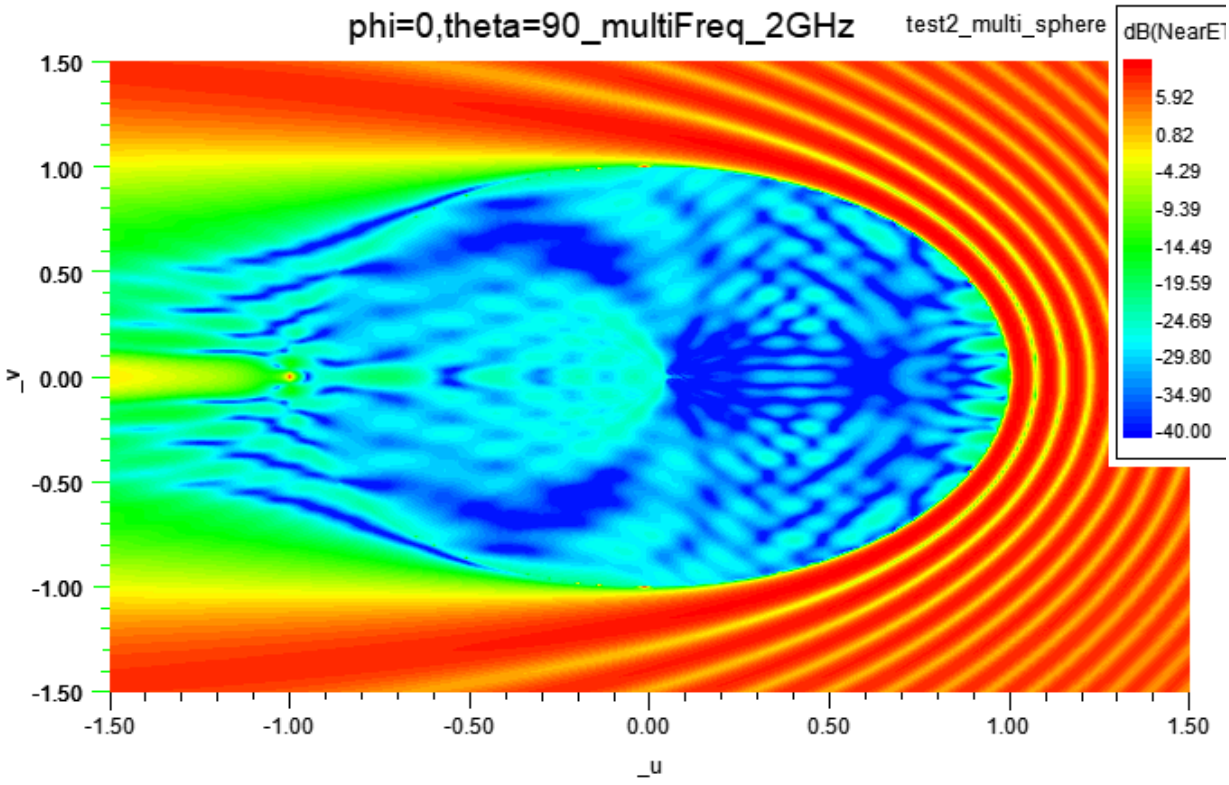
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



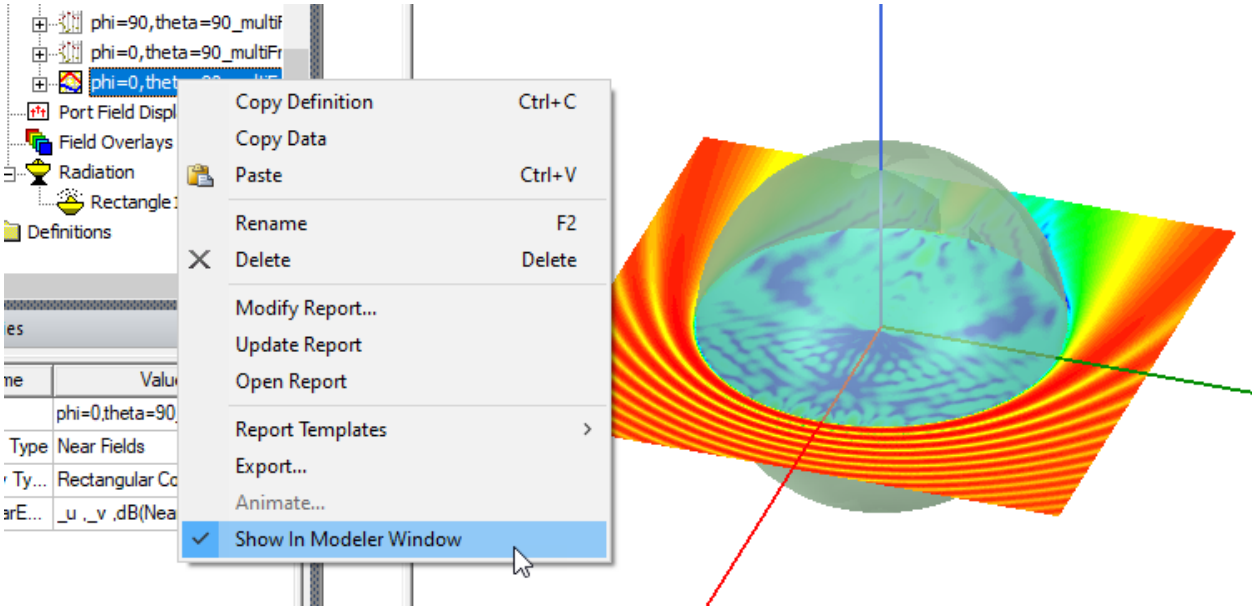
Select the Near Field radiation Field Observation Domain as the Geometry for the Context. Select Near E as the Category, NearETotal as the Quantity, and dB as the Function., and click **New Report**. This creates the Near E Rectangular Contour plot.

Be aware of that the point locations will be defined relative to the _u (X) and _v (Y) axes of the local relative coordinate system while the reported polarized field components are always aligned with global coordinates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

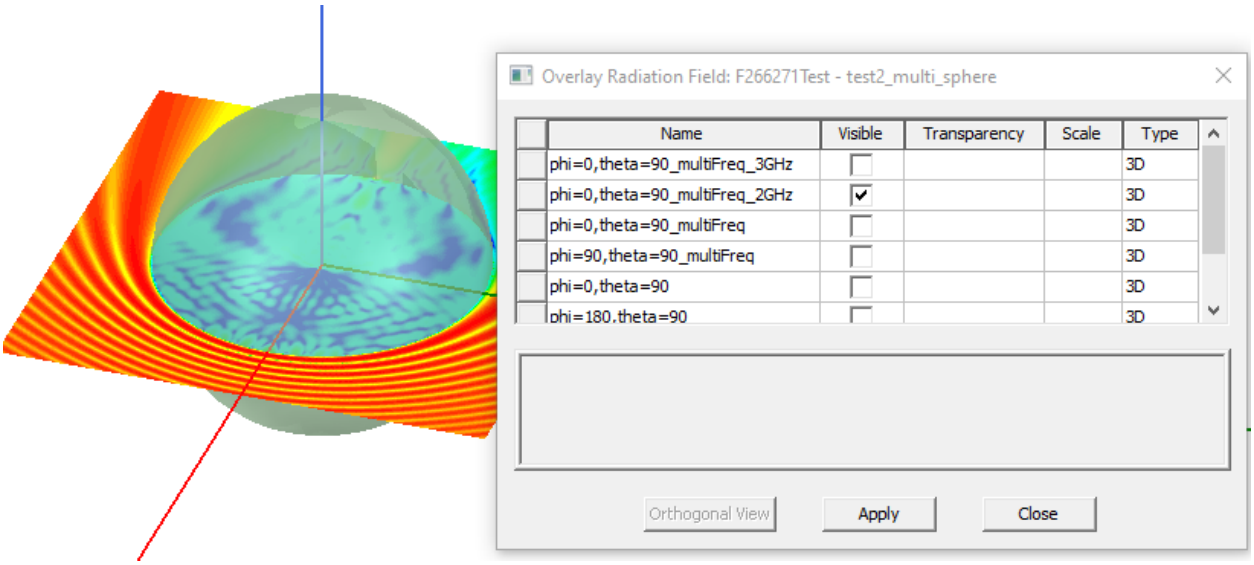


9. To overlay the near field report contour plot in **Model** window, right click on the Plot in the **Project** tree and select **Show in Modeler Window**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also right click in the Modeler window and select **Plot Fields...>Radiation Field...** to open the **Overlay Radiation Field** dialog box, and check for the plot to be visible. If more than one plot exists, you can overlay any or all of them that have current solution data.

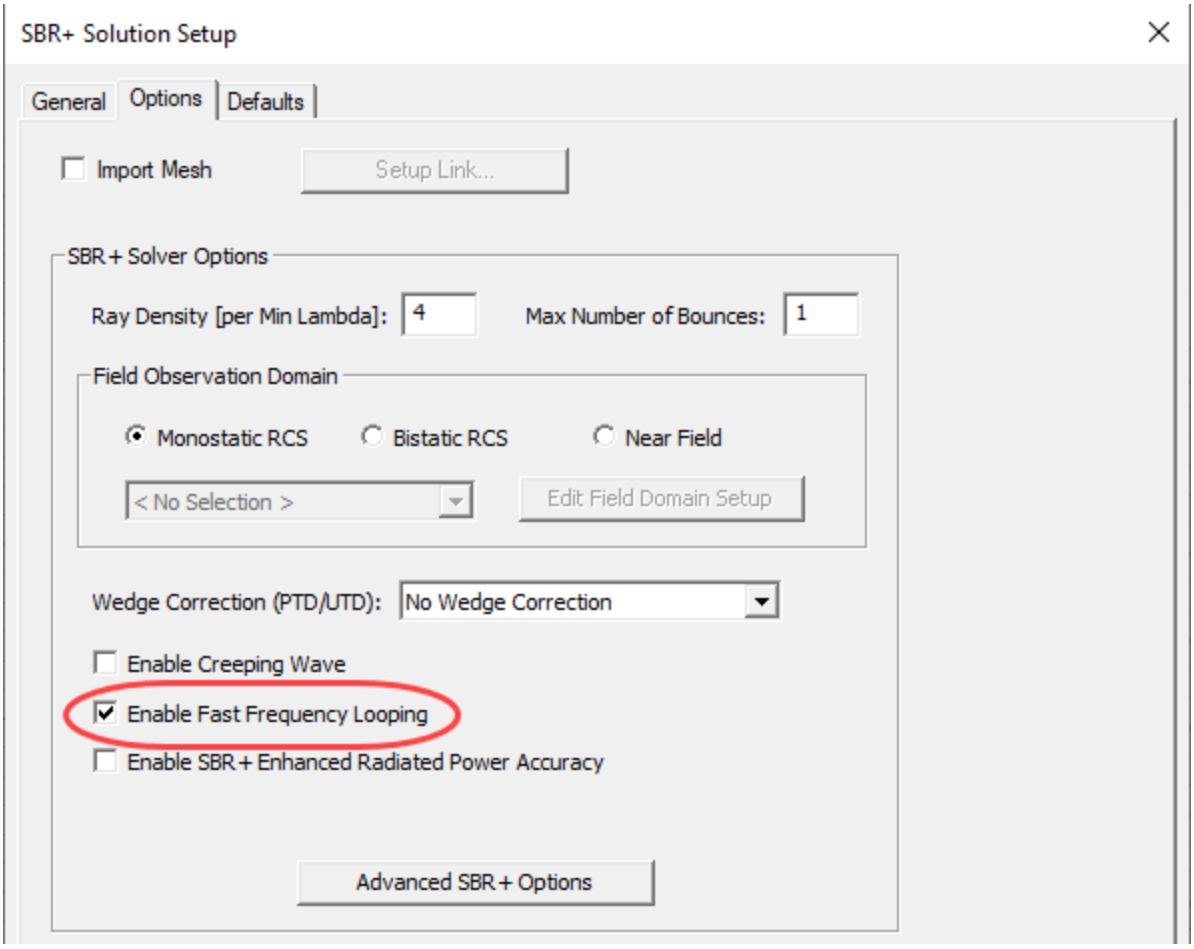


SBR+ Fast Frequency Looping for High Density Frequency Sweeps

The SBR+ Fast Frequency Looping feature significantly reduces simulation time for high-density frequency sweeps with tens to thousands of uniform frequency samples. The workflow change is simple. In the SBR+ solution type, if you have configured a uniform frequency sweep with at least 2 frequencies, you can switch on the Fast Frequency Looping option in the simulation setup dialog box.

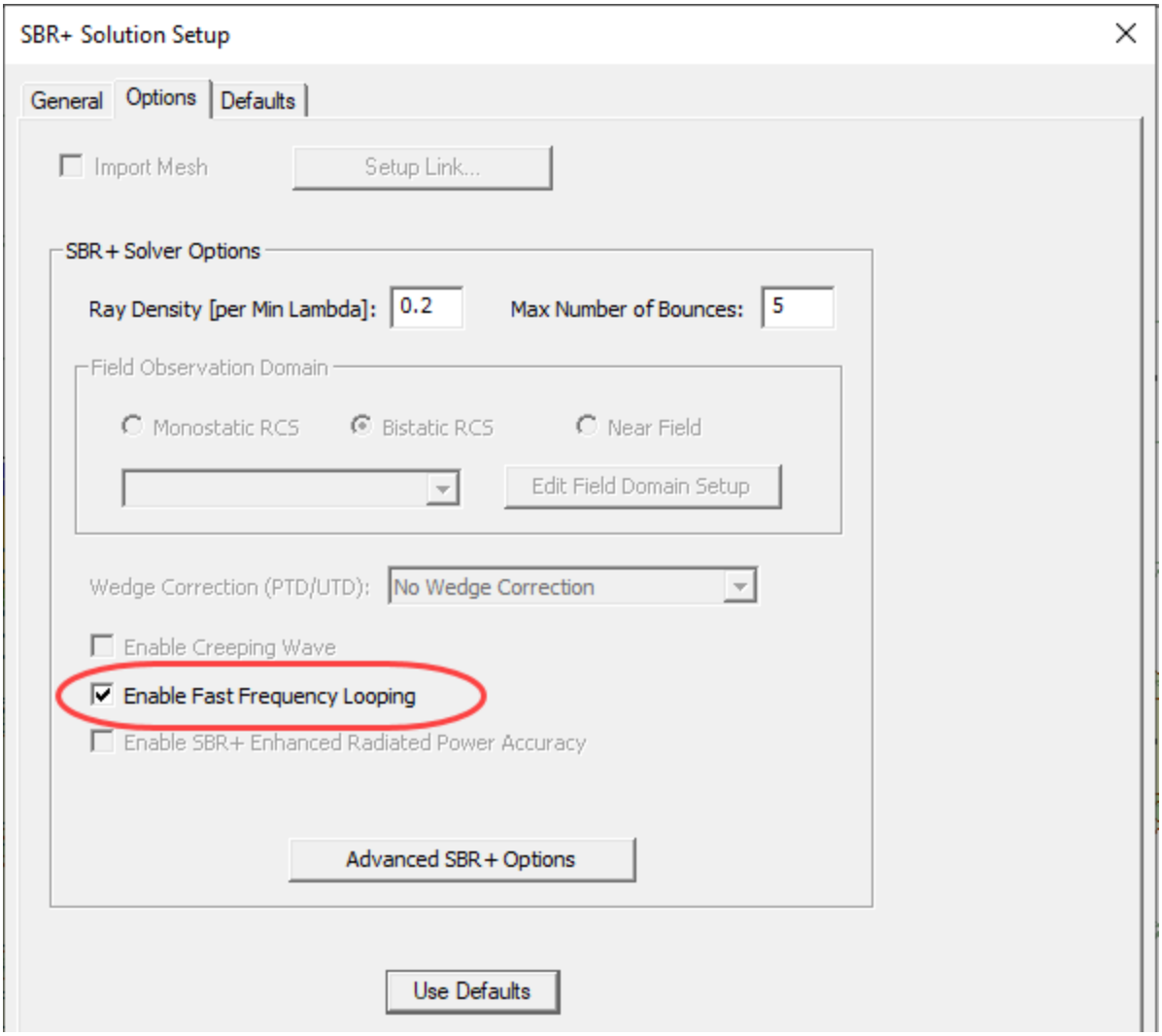
For a standard SBR+ setup:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For a Range Doppler setup:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



From there, the acceleration is handled automatically, with no other workflow changes in problem setup or post-processing. The acceleration engages for all SBR+ field and signal output types: S-parameters, far-field antenna patterns, near-field E/H-field spatial distributions. Responses from such sweeps are often post-processed outside of Desktop with IFFTs to yield a response as a function of time (or distance) instead of a response vs. Frequency.

Technical Background

An important class of SBR+ applications require high-density, uniform frequency sweeps. These include (inverse) synthetic aperture ((I)SAR) imaging, automotive range-Doppler radar sensing, and 5G coverage fading analysis. More generally, users are interested to Fourier process (IFFT) frequency-domain responses to obtain results in the time domain, which often provides a more intuitive/actionable representation of field and Rx signal behavior. The time-domain representation may be of direct interest, or users may want a corresponding “space-domain”

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

representation by scaling the time axis according to the speed of light. An example of the latter is an mono-static RCS range profile, where the time axis is scaled by half the speed of light (c/2) to correlate scattering features in the response with physical features of the scattering geometry.

Fast Frequency Looping (FFL) significantly reduces simulation time for high-density frequency sweeps where there are tens, hundreds, or even thousands of frequency samples. High sampling density is needed to control aliasing in the time responses. Time (space) resolution improves linearly (Δt and ΔR get smaller) with increasing frequency bandwidth, but if this is not compensated with additional frequency samples to maintain Δf, the uniform interval between frequency samples, aliasing gets correspondingly worse. The aliasing period is inversely proportional to Δf (i.e., smaller Δf and more samples is better).

Like the HFSS FEM and IE solvers, SBR+ is a time-harmonic frequency-domain solver, so frequency samples are added by looping the overall simulation, and run time grows in proportion to the number of frequencies. SBR+ has an important advantage in this regard since the geometric ray paths are frequency independent. For a high-density frequency simulation, the rays are only shot once. Nonetheless, the rays are still processed N_f times in an internal frequency loop. In an SBR+ simulation, one typically observes very little run-time impact in going from 1 to 2 frequencies. However, once N_f reaches 10 or 20 samples, the ray-processing time overtakes geometric ray tracing time, and run time grows linearly with N_f.

TD-SBR+ exploits geometric information in the rays to reduce the computation cost of a high-density frequency sweep to that of simulating at just one frequency. Internally, the long-form SBR+ simulation is performed at just the center frequency. Then, using cached information from each ray path and FFTs, this single-frequency result is efficiently extrapolated to all other frequencies of the uniform sweep. The payoff for hundred-plus frequency samples is typically 5x – 10x.

The workflow change is simple. In the SBR+ solution type, if you have configured a uniform frequency sweep with at least 2 frequencies, you can enable Fast Frequency Looping in the simulation setup dialog box. From there, the acceleration is handled automatically, with no other workflow changes in problem setup or post-processing. The acceleration engages for all SBR+ field and signal output types: S-parameters, far-field antenna patterns, near-field E/H-field spatial distributions.

Because FFL only performs the long-form SBR+ simulation at the center frequency (f_c), it is no longer necessary for excitations to be defined over the entire the entire frequency sweep of the solution setup. For example, imported far-field FFD pattern files only need to be defined at the center frequency, as only that frequency sample will be consulted during the simulation. Likewise, linked sources only need to be simulated at the center frequency in the source design, which can therefore reduce the overall simulation time.

IMPORTANT: When taking advantage of this source simplification while using FFL, it is vital to understand how FFL determines center frequency f_c in order to ensure the source includes the required frequency sample, either as an actual frequency sample in its definition or within the lower and upper extents of the source's own frequency sweep. *The center frequency f_c of the*

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FFL simulation is always one of the frequency samples of the configured uniform frequency sweep, which is not necessarily the average of the lower and upper frequency sample of that sweep. If the number of frequency samples (N_f) is odd, then f_c is the middle sample of the frequency sweep. However, if N_f is even, then f_c is the first sample in the second half the uniform frequency sweep. For example, if the uniform sweep is configured as **Linear Count** from 9 to 10 GHz and with 100 points, then fc is considered by the FFL simulation to be the 51st frequency sample in that sweep, which is 9.505050505… GHz, not 9.5 GHz. Any source used in that FFL simulation must therefore include 9.505050505 GHz either as an actual frequency sample or within the lower and upper extents of its frequency samples. Otherwise, the simulation will fail with an error message.

In the case of a Range-Doppler setup, the above guidance has a special interpretation. All **Range-Doppler** setups require a **Center Frequency input**. SBR+ uses this and other information in the **Range-Doppler** setup to internally generate a uniform frequency sweep for the simulation. The SBR+ **Range-Doppler** setup interprets **Center Frequency** in the exact same manner as FFL. If the number of frequency samples is even, then the user-provided **Center Frequency** becomes the first frequency in the second half of the uniformly spaced frequency samples. For example, if you configure a **Range-Doppler** setup with **Center Frequency** set to 76.5 GHz, and you also switch on **Enable Fast Frequency Looping** in that setup, then all radar antennas used in that simulation must either have a frequency sample at 76.5 GHz or lower and upper sweep definition that encompasses 76.5 GHz.

Limitations on the Fast Frequency Looping Feature

This feature is:

- Only available in SBR+ Solution Type.
- Limited to uniform frequency sweeps: Linear Step or Linear Count.
    - This includes the ADP Range-Doppler setup option, which is implicitly uniform frequency sampling.
    - Switching on **FFL** with a non-uniform frequency sweep will result in a validation error.
- Not recommended when relative bandwidth (center frequency divided by absolute bandwidth) exceeds 10 %.
    - The extrapolation is necessarily approximate and only focuses on the impact of changes in the electrical lengths of ray paths and how that causes ray contributions to coherently interfere. This is the typically the dominant source of frequency dependence in response computed by SBR+, but the extrapolation ignores all other sources of frequency dependence:
        - Source antenna pattern or current distribution
        - Distributed sources (arrays and large current-source) with a spread-out time response relative to the time-resolution (bandwidth) of the simulation will not be accurately captured by this acceleration feature, and it is not

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

recommended under those conditions (i.e., if the speed-of-light divided by the absolute bandwidth is smaller than the physical size of the FEBI antenna or the array).

- Material dispersion: implicit multi-bounce in layered impedance boundaries will only be captured in a composite sense at the center frequency, and multiple internal reflections that would be resolved in the time-domain response at high enough absolute bandwidth will not be resolved when TD-SBR+ is engaged, even if the relative bandwidth remains small.
- The accelerated **FFL** results will show increasing deviation from unaccelerated SRB+ results with increasing separation from the center frequency. In many cases, these can appear significant, yet the resulting time-responses after post-processing will remain excellent.

- Simulations that already benefit from GPU acceleration may not show much or any benefit when this feature is engaged. This is because the GPU parallelizes the frequency loop. This feature will engage with or without the GPU processing option, but when run on the GPU, the upshot may be reduced GPU utilization and only slightly faster run time.
- **FFL** acceleration is well suited to range-Doppler setups. These typically involve hundreds of frequency samples spanning a relative bandwidth that is well under 10%. Under these conditions, FFL will provide 5x – 10x or greater speed-up with little loss of accuracy."
- **FFL** requires 4x to 8x more memory for intermediate results when the feature is engaged. If the system does not have enough memory, this can be mitigated by reducing the number of CPU cores while still realizing a net acceleration benefit.

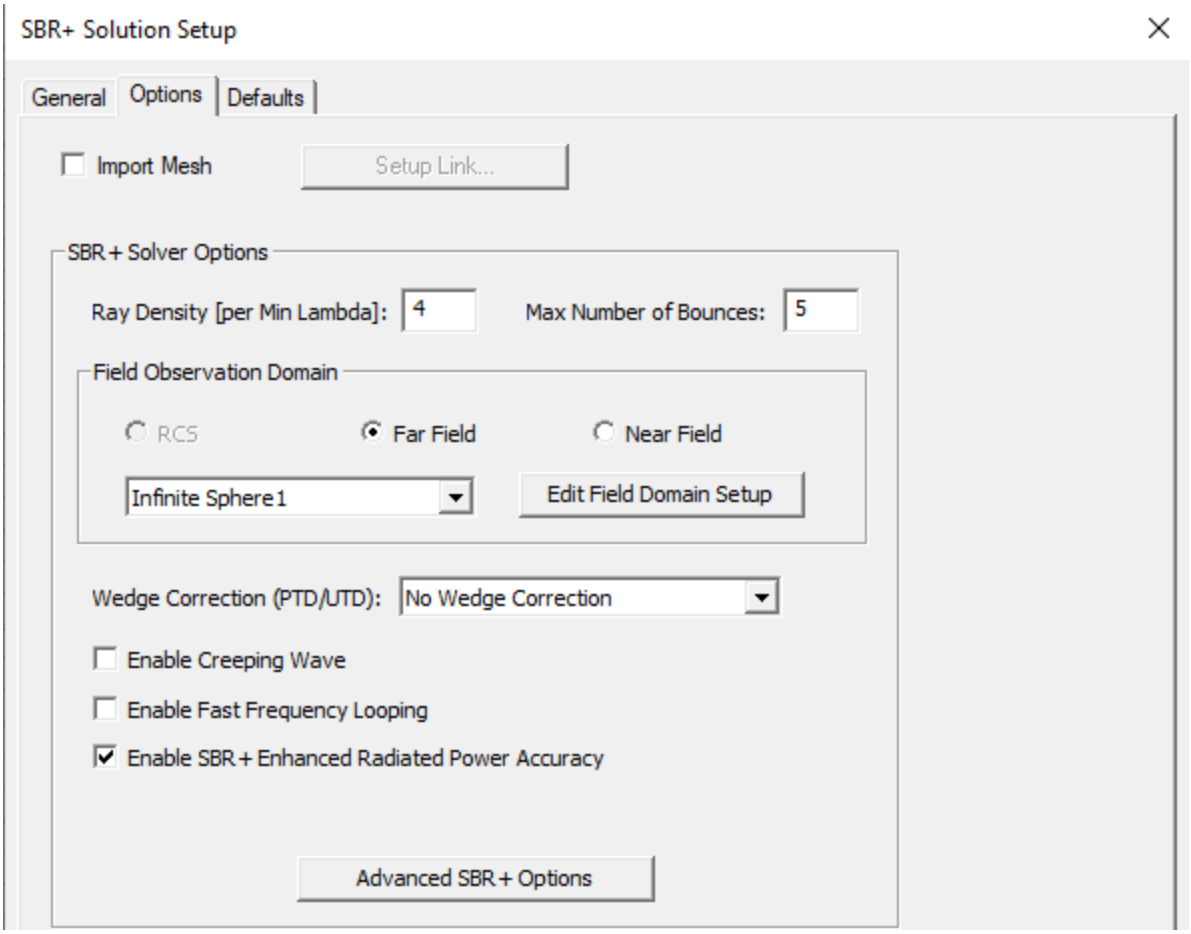
Summary Workflow:

1. Create an HFSS design and set the Solution Type to SBR+
2. Configure the SBR+ design in the usual fashion: sources, excitations, boundaries, materials, near-field and far-field observer domains, etc.
3. In the General tab of the SBR+ Solution Setup dialog
    a. Set the Setup Type to Standard (default) and configure a Linear Count or Linear Step frequency sweep with a large number of frequency samples, OR
    b. Set the Setup Type to Range-Doppler and configure the Range Resolution to be a small fraction of the Range Period (which is typical for such setups). The implied number of frequency samples is Range Period divided by Range Resolution.
4. In the Options tab of the SBR+ Solution Setup dialog box, switch on Enable Fast Frequency Looping
5. Run the simulation
6. Post-process results in usual fashion

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

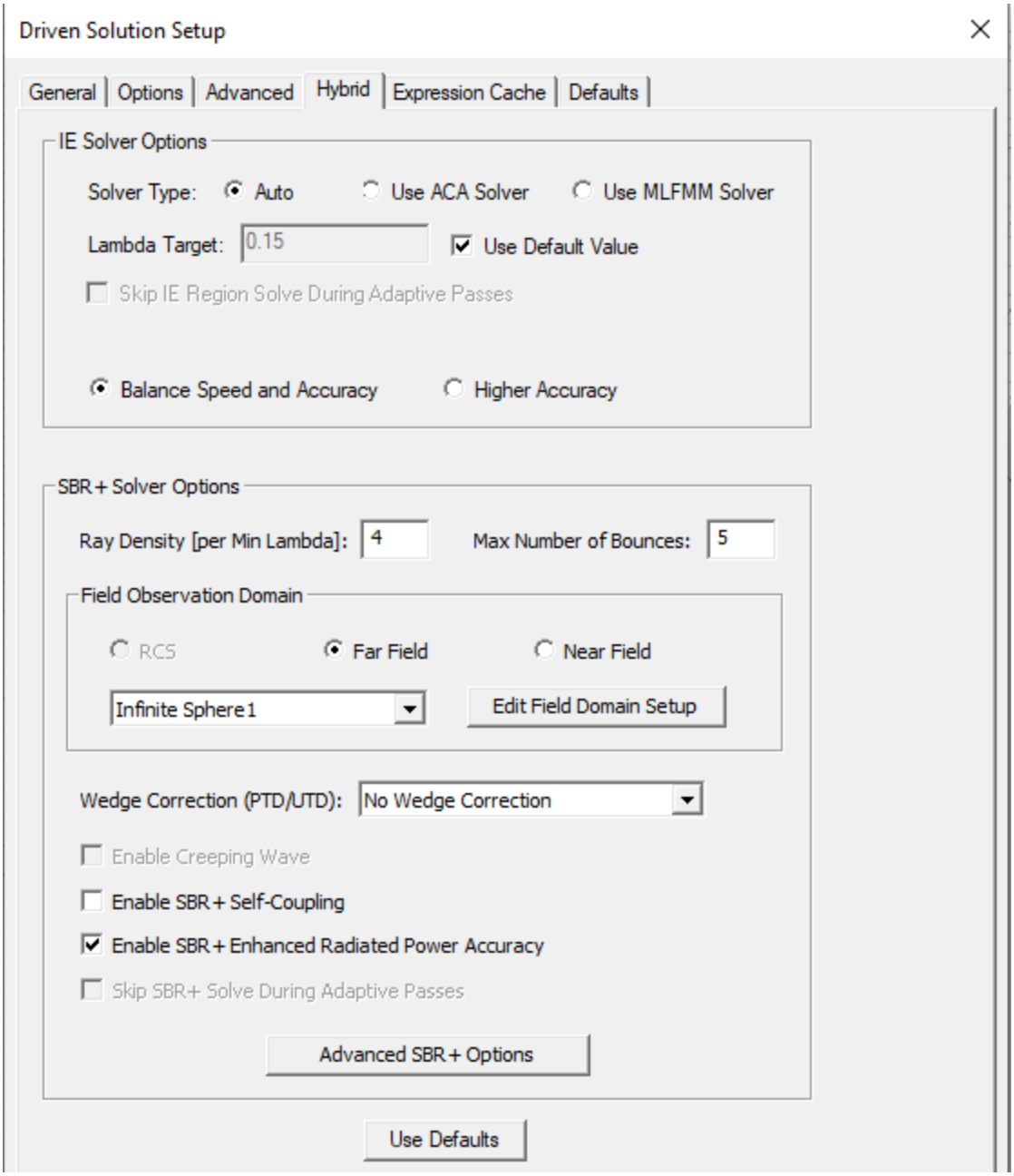
SBR+ Enhanced Radiated Power Accuracy

Radiated power is usually computed without considering scattering from the SBR+ region. Incident power contributions from all linked and parametric antennas are first combined the using Edit Source configuration, and then power loss is subtracted. To account for impact of scattering by the SBR+ region (including lossy materials such as non-penetrable coatings or half-space dielectrics), select Enable SBR+ Enhanced Radiated Power Accuracy on the **Options** tab of an **SBR+ Solution Setup**.



The same feature can be set on the on the **Hybrid** tab of an **HFSS with Hybrid and Arrays Solution Setup**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

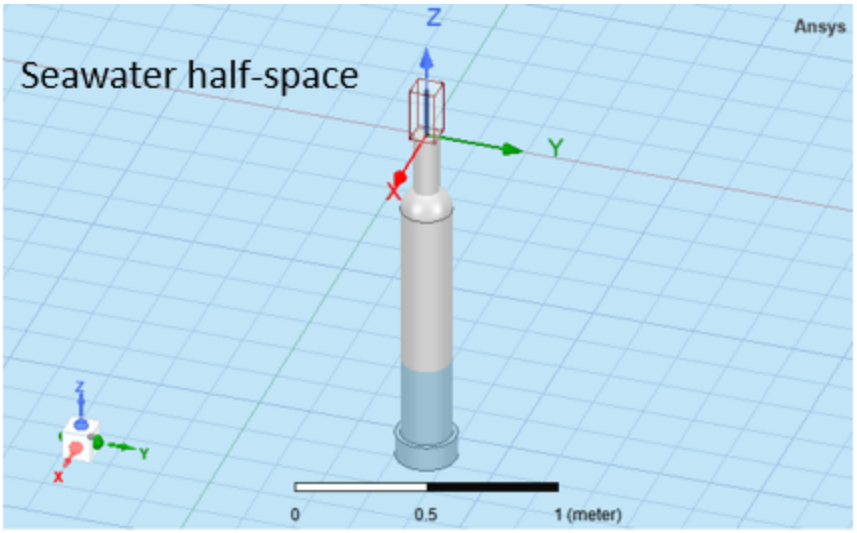


With Enhanced Radiated Power Accuracy enabled, the field scattered by the SBR+ region is included in the radiated power calculation. For each excitation, the SBR+ solver performs an additional simulation to compute the total field across the entire far-field sphere. HFSS then recombines the fields based on theEdit Sources configuration and computes the total radiated power, which is then used to compute radiated power, directivity and other power related quantities. Users should consider whether including the SBR+ region in the radiated power calculation is appropriate for their problem.

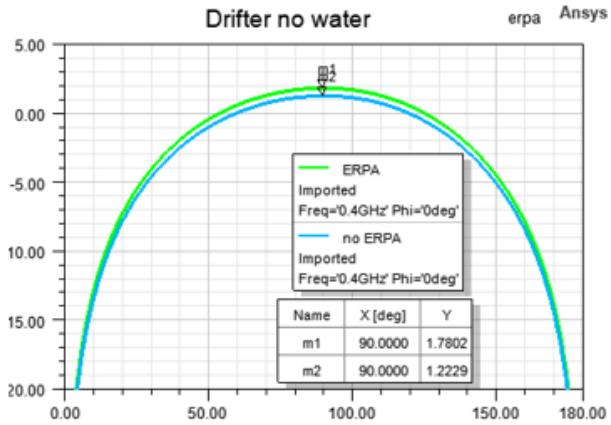
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

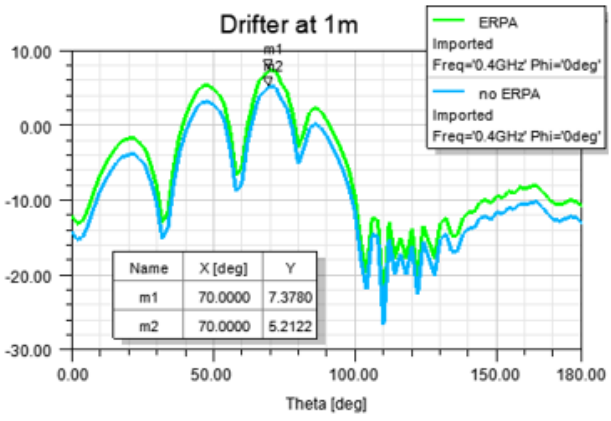
Note: Because the angular resolution of the far-field sphere is determined by the electrical size of the platform, this can be computationally expensive. Its use should be limited to situations where accurate SBR+ power calculations are needed, such as scenarios with large areas of lossy materials in the SBR+ region. Also, current source thinning will artificially reduce the power radiated by the SBR+ region. It should be used judiciously when accurate SBR+ power calculations are desired.

Consider for example a simulation of drifting sensors in the oceans. The drifter communicates with a satellite using a 400MHz monopole antenna.



The following plots show the differences in plots with and without Enhanced Power Accuracy enabled.





After solving the design, radiation parameters can be evaluated by clicking **Radiation** <*Far field name*> **> Compute Antenna Parameters...** Here radiated power quantities, including

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

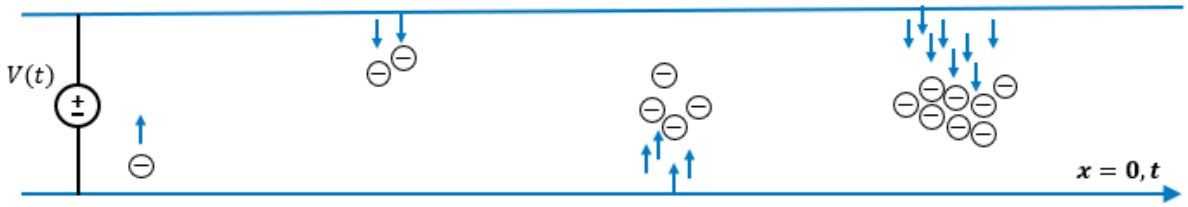
directivity, radiated power and radiation efficiency, are displayed. You can also create a directivity field plot, under **Results >Create Far-Fields report**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

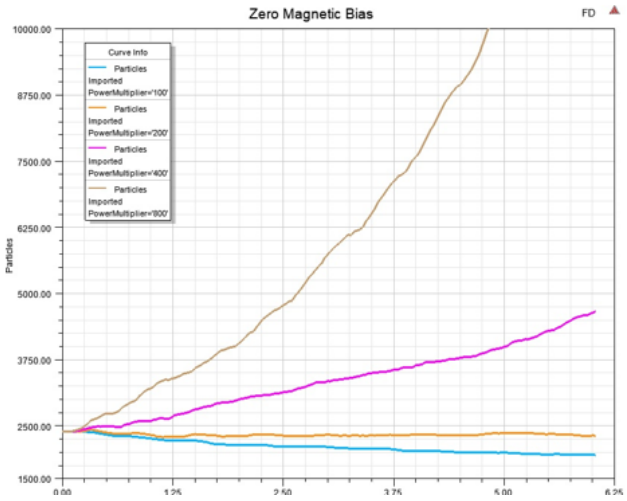
14 - HFSS Multipaction Analysis

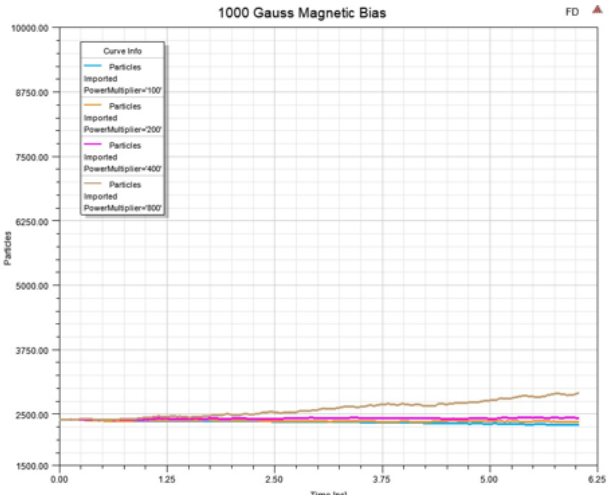
HFSS Multipaction provides a finite element approach to simulating vacuum multipaction in space-born communications systems and vacuum electron devices. When space charges are accelerated by time-harmonic fields in a confined space surrounded by materials, the charges can knock electrons off the material surfaces. If the transit time is in sync with the time-harmonic fields, the electron multiplication can be sustained and will eventually lead to an electron avalanche.



Detecting multipaction through measurement is difficult and expensive for a large antenna feed network on a satellite. Failure is also not an option for a mission-critical space project. Therefore, there is a big incentive to run computer simulation for identifying components prone to multipaction before launching a satellite into space. HFSS can guide the design of multipaction suppression measures.

HFSS Multipaction supports multipaction on both metallic and dielectric surfaces. You can set up a uniform DC bias field as part of HFSS Multipaction, or use Maxwell near field links to set up non-uniform DC bias fields. This is helpful in design and simulation because multipaction is less likely to happen after applying magnetic bias.





Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

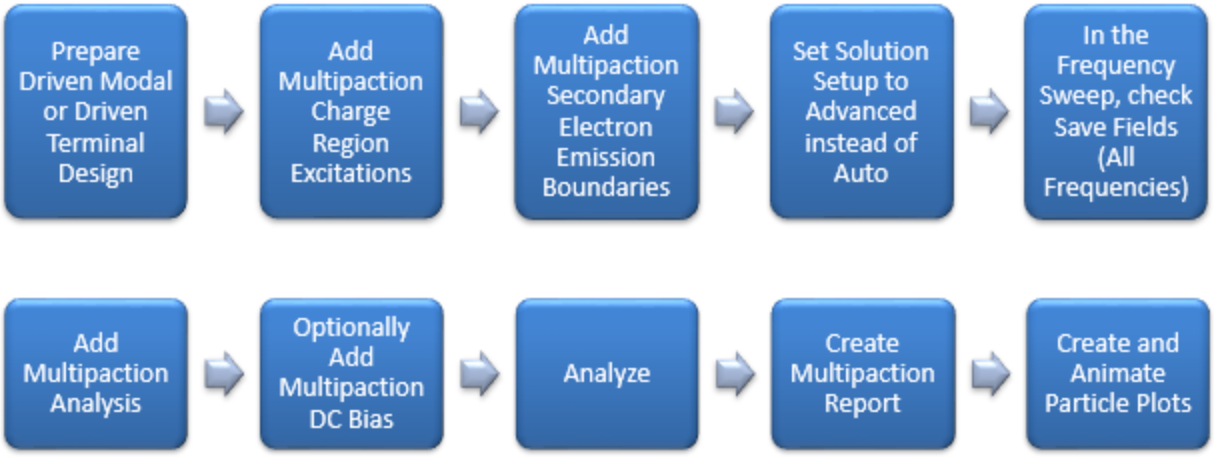
Optionally, you can select the **Charge distribution** option in the **Multipaction Analysis** setup to output particle and geometry mesh information used to produce particle plots and animations.

Prerequisites

- A multipaction analysis requires the time-harmonic fields being saved for a driven-terminal or driven-modal solution.
- While HFSS Multipaction can simulate uniform bias, to simulate the effect of non-uniform bias fields, use Maxwell near-field links.

General Multipaction Analysis Process

The process flow for running Multipaction Analysis follows:



If you want to creation particle animations, you must select the **Charge distribution** option in the **Multipaction Analysis** setup to output particle and geometry mesh information.

Prepare a Driven Modal or Driven Terminal Design

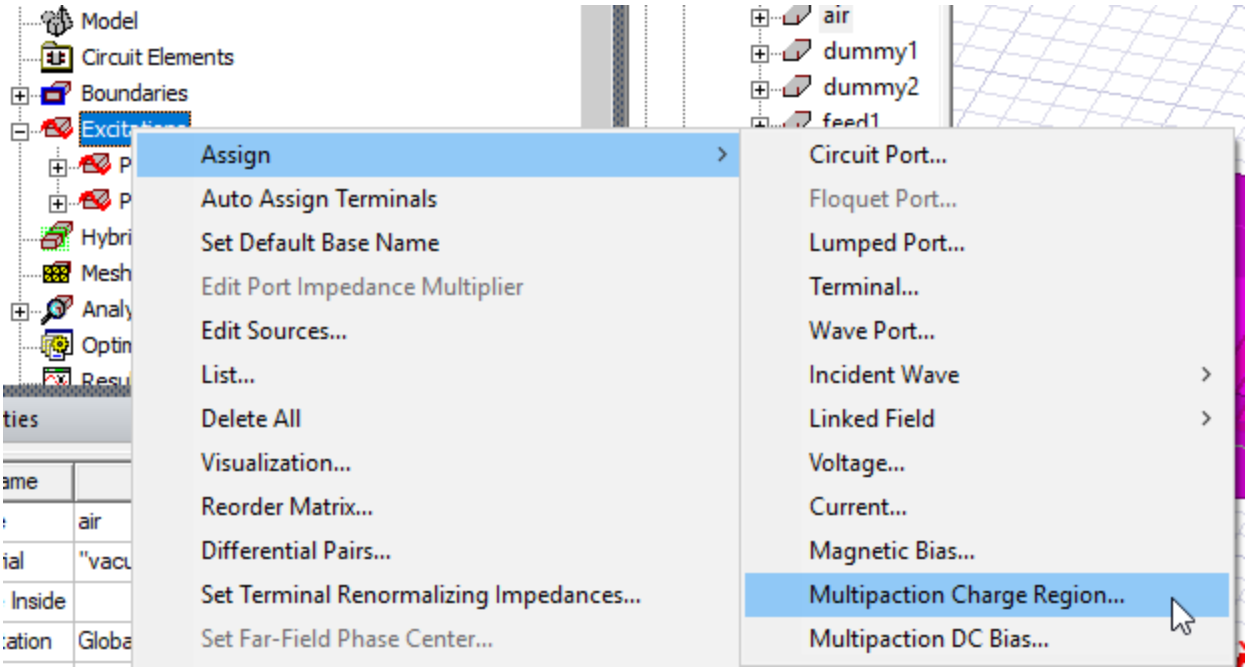
As stated above, a multipaction analysis requires the time-harmonic fields being saved for a driven-terminal or driven-modal solution. Therefore, the multipaction setup and associated boundary conditions apply only to these types of HFSS projects. Other solution types, such as transient, eigenmode, characteristic mode, and SBR+, do not support the multipaction analysis. In addition, a multipaction simulation currently does not support renormalization or deembedding of ports.

Add Multipaction Charge Region Excitations

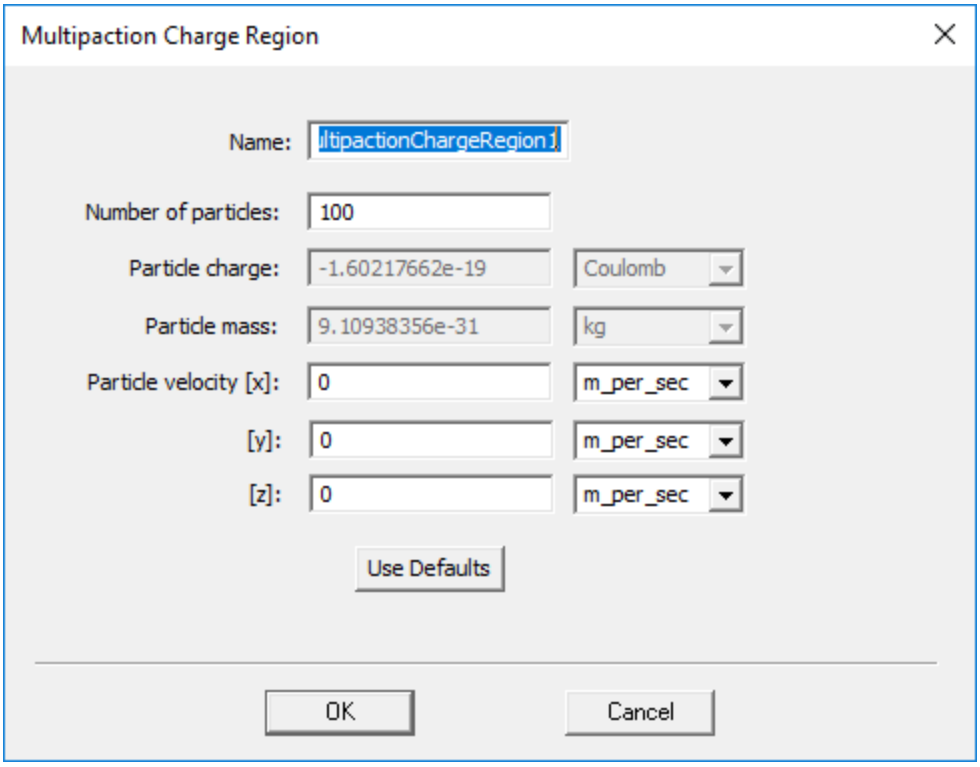
In HFSS Multipaction, charges are added to the simulation domain as excitations. A Multipaction Charge Region consists of either multiple objects or multiple faces, depending on whether the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selection Mode is Objects or Faces.



This opens the **Multipaction Charge Region** dialog box.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For each charge region, we can specify the number of electrons and their initial velocities. It is not necessary to seed charges in the entire simulation domain if the regions prone to multipaction can be easily identified, such as small gaps, coaxial feeds, parallel plates, and regions with strong fields. The simulation time can be significantly reduced by seeding charges only in critical regions.

The statistical variation of multipaction simulation can be reduced by using more particles, but the simulation time will increase as well. Moreover, using too few particles (less than 1000 for all charge regions) will introduce noises to the results and may trigger an early termination of the simulation.

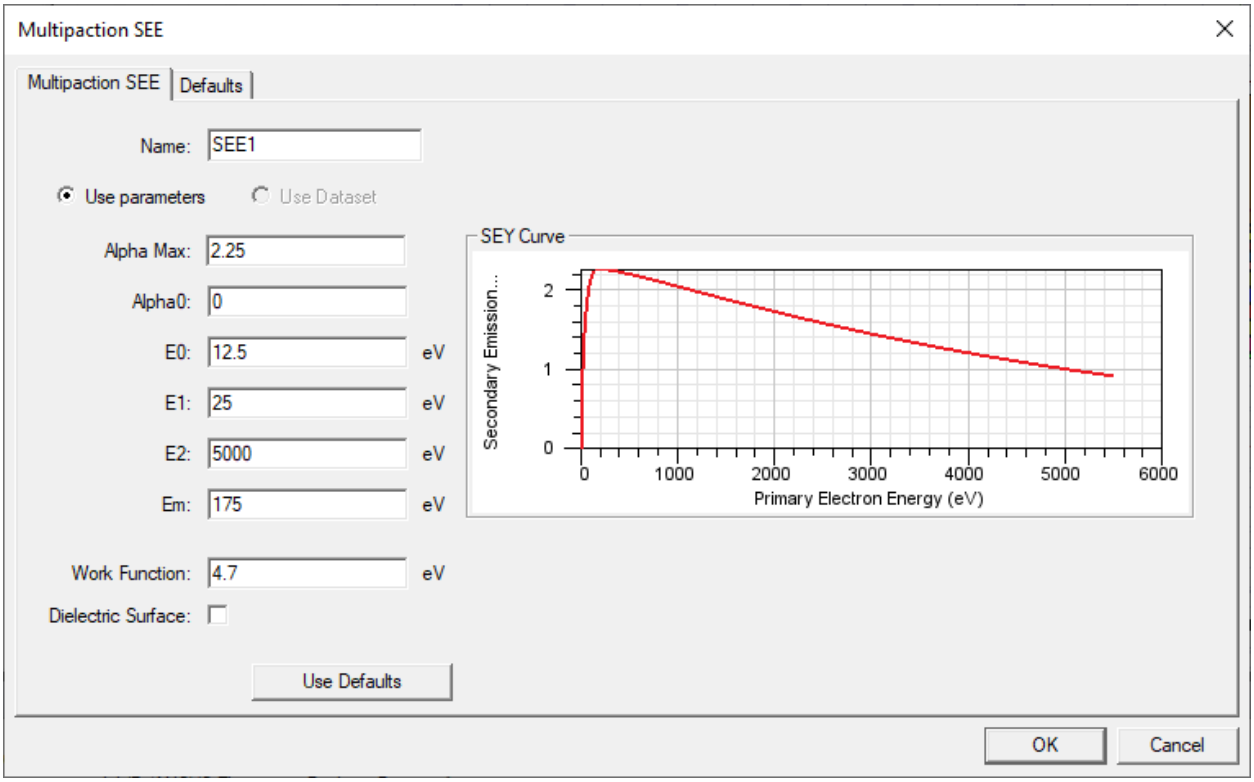
Adding Multipaction Secondary Electron Emission (SEE) Boundaries

The Multipaction SEE boundaries should be added to vacuum-material interfaces where secondary electrons will be generated. Typically, those interfaces are associated with PEC, impedance, or finite-conductivity boundaries in electromagnetic simulation. If a Solve Inside object contacts the vacuum region, you should add SEE boundaries on their interface to avoid charged particles penetrating the object.

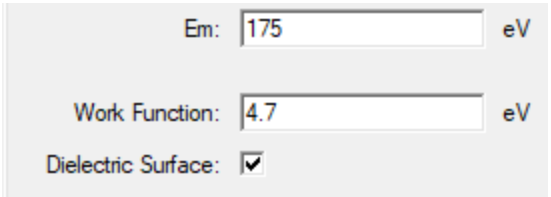
Charged particles will be entirely absorbed by surfaces in contact with vacuum but not covered by SEE boundaries. For example, port faces should not be covered by SEE boundaries because charged particles should be allowed to leave the simulation domain through ports.

Select the faces or objects to which you want to assign SEE boundaries, right-click on **Boundaries** in the Project Manager, and select **Assign> Multipaction SEE...** This command opens the **Multipaction SEE** dialog box:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



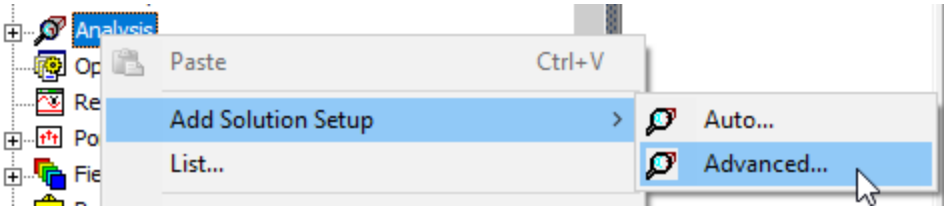
The secondary emission yield (SEY) depends on the impacting energy of the primary electron. We adopt an enhanced Vaughan model where the SEY curve will fall exactly on the low and high cutoff energies E1 and E2, where SEY is equal to 1. SEY reaches the maximum $Alpha_{Max}$ for an energy Em in between E1 and E2. For low-energy primary electrons, SEY is equal to $Alpha_0$ when the impacting energy is between 0 and E0. The work function is needed for calculating the energy of secondary electron emission by the Chung-Everhart distribution. All energies related to the SEE boundary are measured in electron volts (eV). You must check the **Dielectric Surface** checkbox for an insulator-vacuum interface, which allows the accumulation of positive surface charges after secondary electrons leave the interface, or the accumulation of negative surface charges after the interface absorbs primary electrons.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

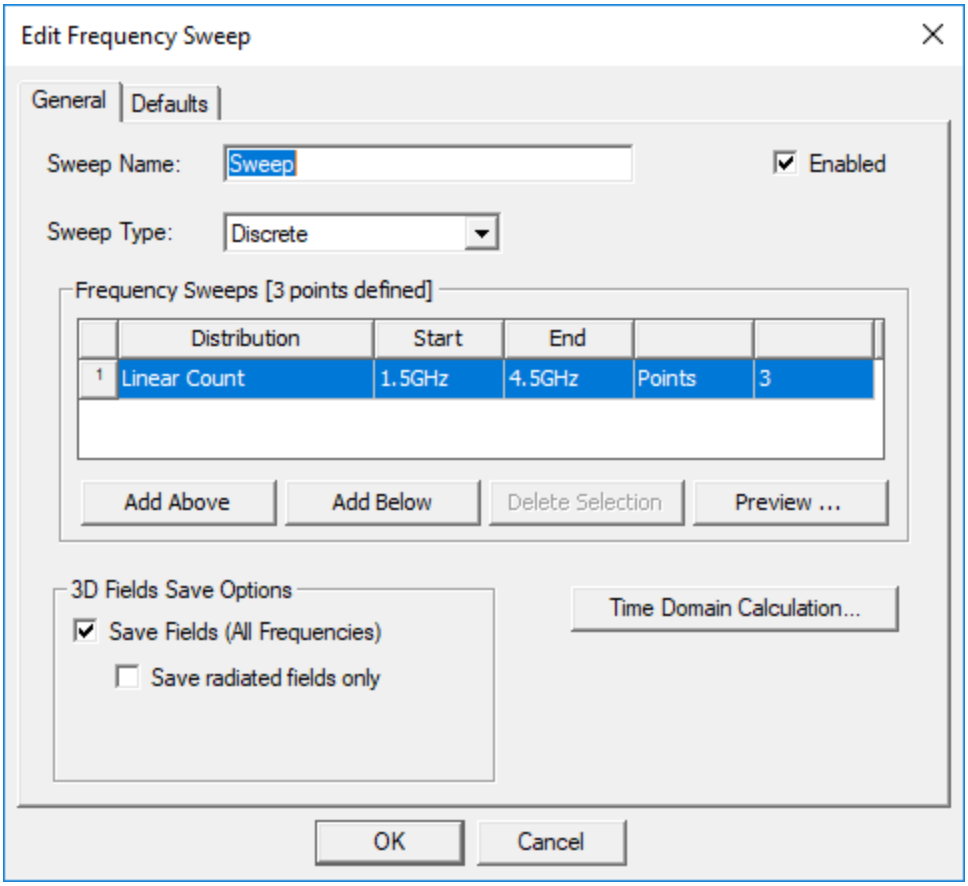
Adding a Multipaction Setup

At least one charge region and one SEE boundary must exist before you can add a Multipaction Analysis. To add a Multipaction Analysis, you must add a Solution Setup and a Frequency Sweep. You then add the Multipaction Setup to the Sweep. When you add the Solution Setup to an Analysis in HFSS, select **Advanced** instead of **Auto**.

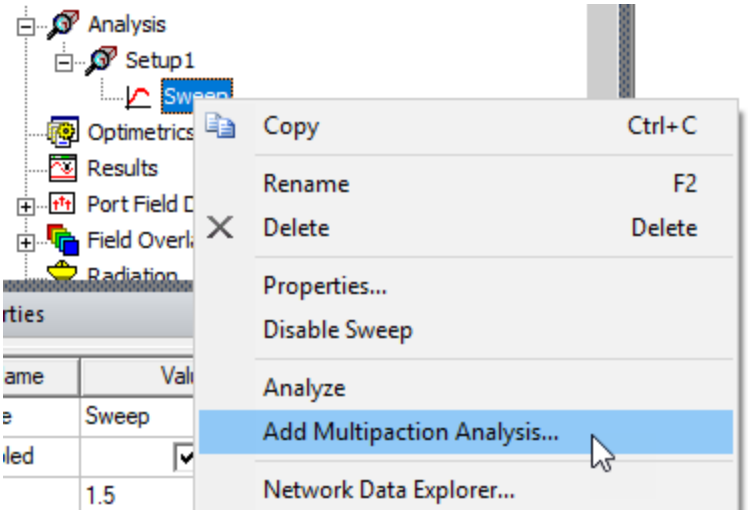


The frequencies specified in the frequency sweep will be reused for setting up the multicarrier signal for a multipaction analysis. You must select the **Save Fields (All Frequencies)** checkbox while editing the frequency sweep, otherwise **Add Multipaction Analysis** will be grayed out on the menu. There will be no multipaction if there is no time-harmonic field to sway the charged particles in the simulation domain.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

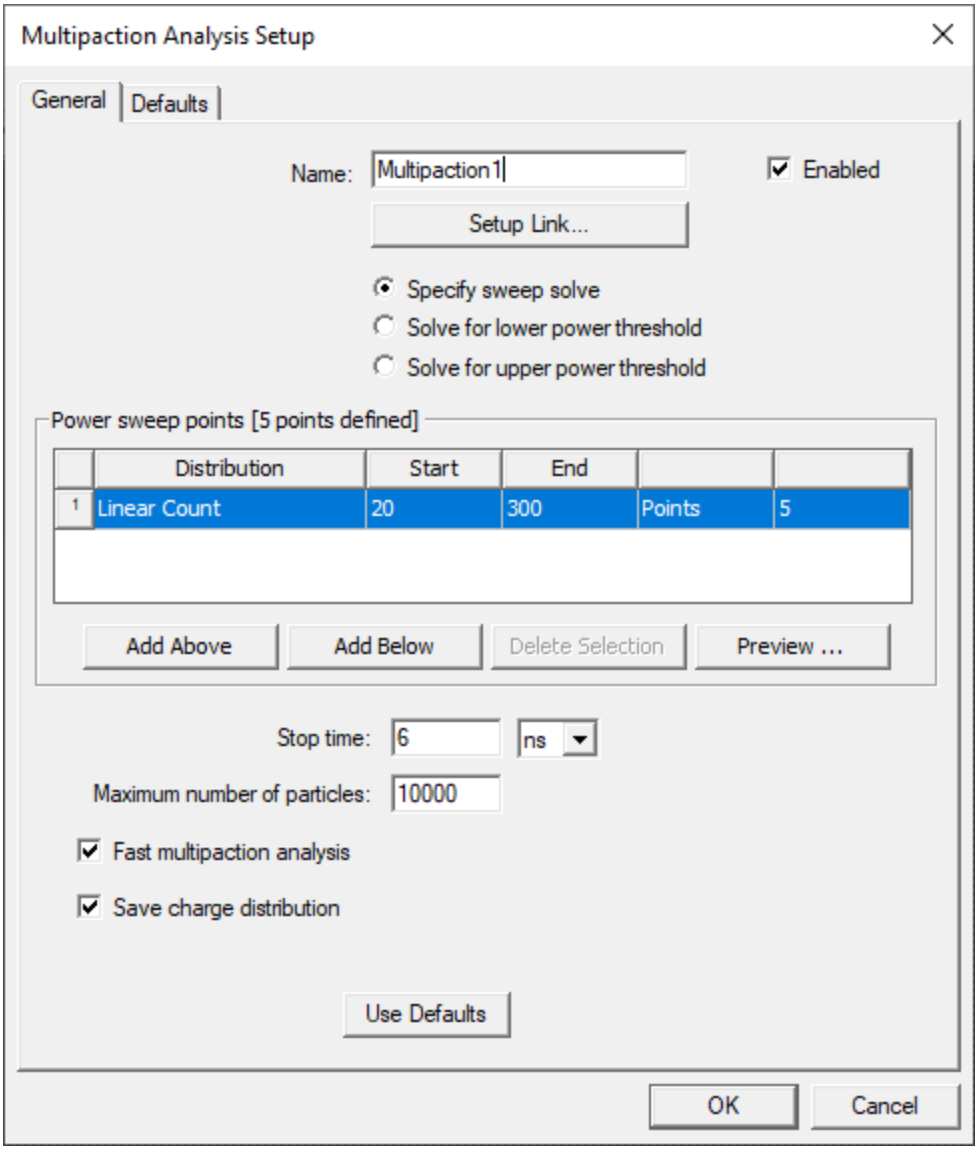


Once the above procedures are completed, you can right-click the frequency sweep and select **Add Multipaction Analysis...**



This opens the **Multipaction Analysis Setup** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

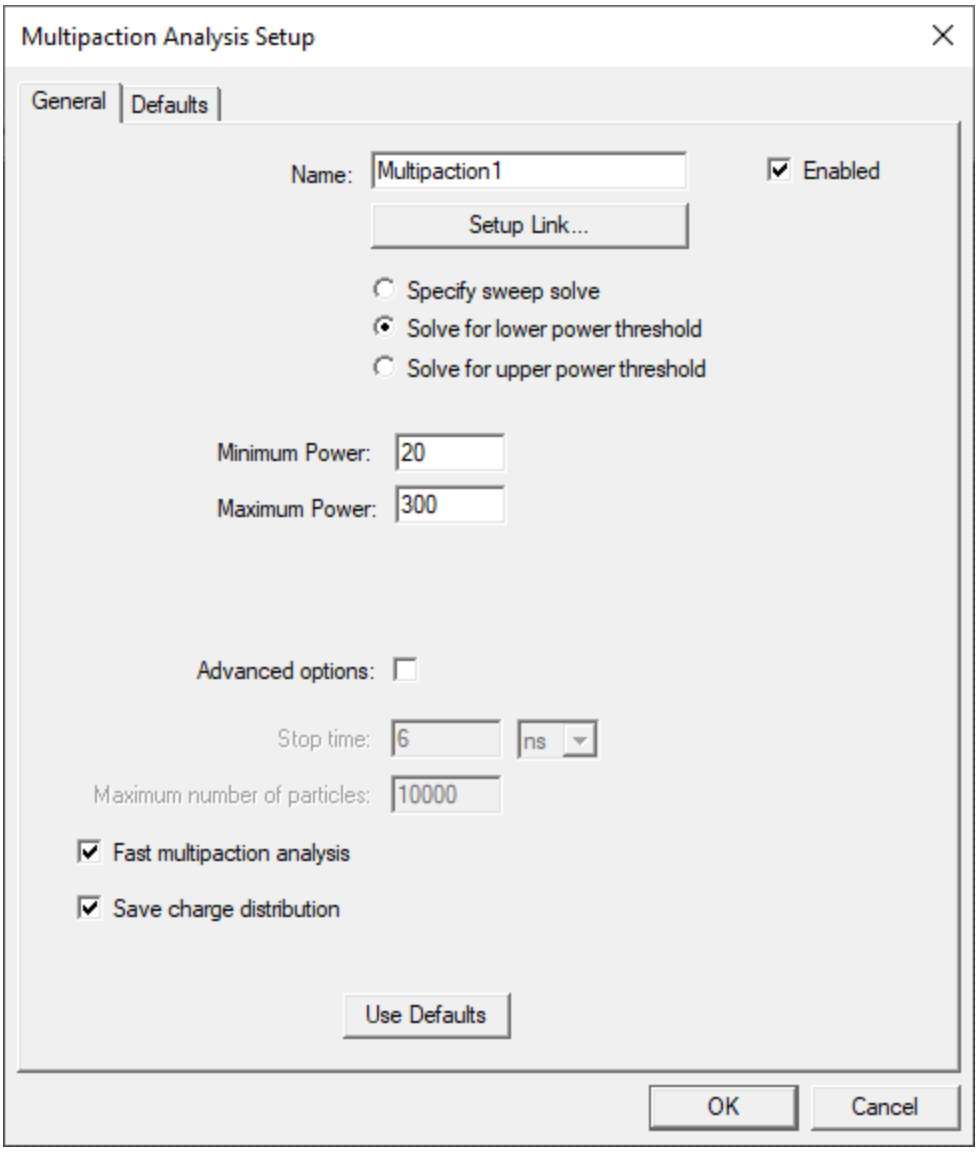


The **Solve for lower power threshold** button allows automatic determination of the lower power threshold. It is the power threshold where the breakdown status transits from no multipaction to multipaction with increasing input power. Similarly, the **Solve for upper power threshold** button allows automatic determination of the upper power threshold. It is the power threshold where the breakdown status transits from multipaction to no multipaction with increasing input power.

Given the power range provided by the user, the software determines which power multiplier to simulate next based on previous multipaction history, instead of reading preset power multipliers from a table. By default, the advanced options are off. The lowest power threshold is found using a stop time ranging from 80 to 120 RF cycles and no more than twice the initial number of

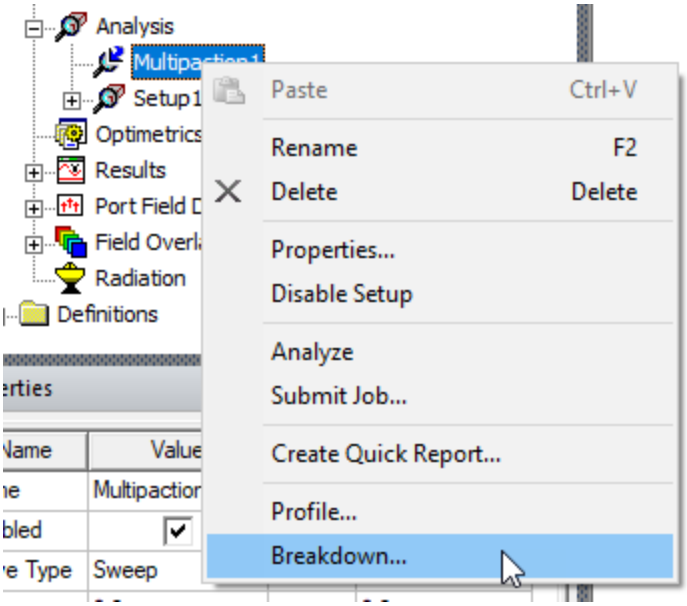
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

particles. The default settings can be overridden by checking advanced options if users prefer to set up the parameters by themselves



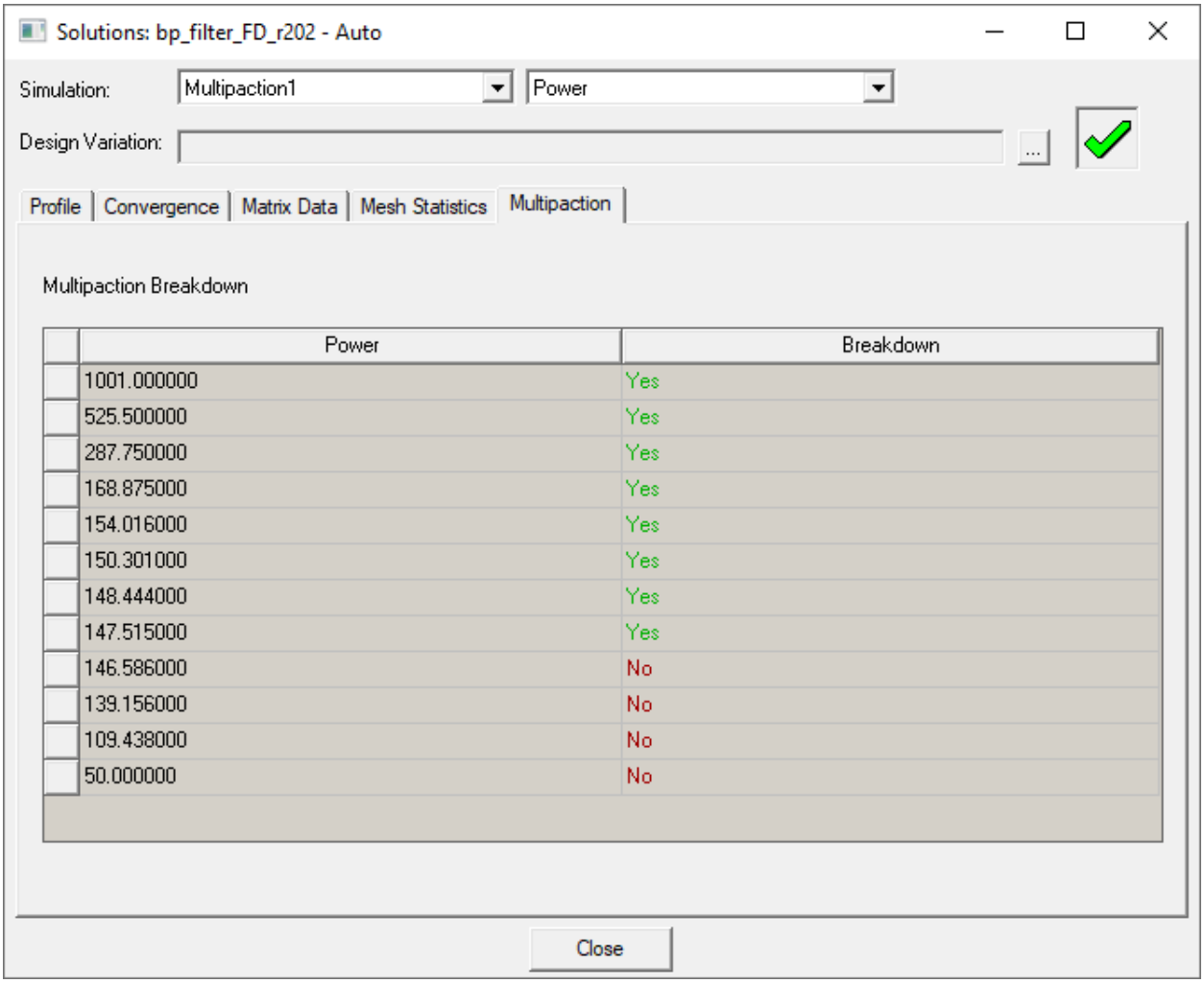
While the automatic simulation is running, you can check the breakdown status of simulated power multipliers by clicking **Breakdown...** under the multipaction setup shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

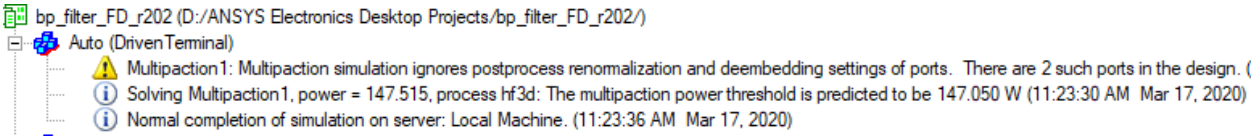


This opens a **Solutions** dialog box with a **Multipaction** tab that provides Power and Breakdown information as follows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



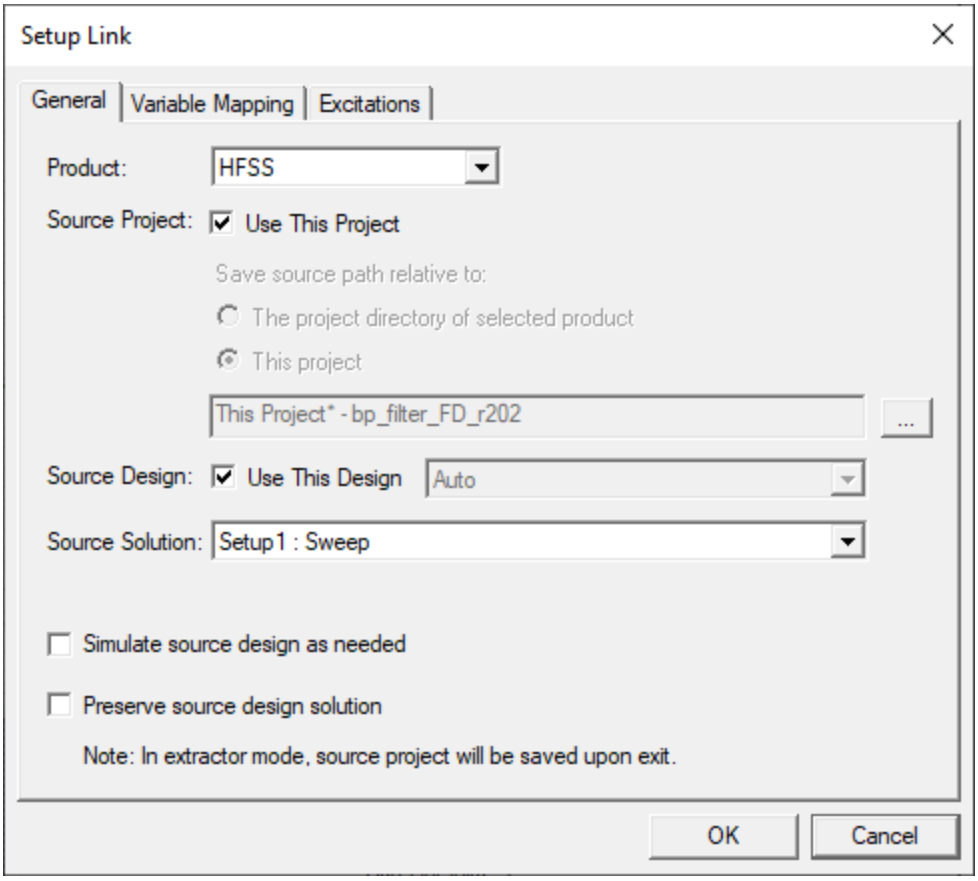
When the Auto Solve converges, there will be a message showing the lowest multipaction power threshold in the Message Manager window.



If the Auto Solve does not converge, it means the structure is free of multipaction for given geometry, material, and simulation setups.

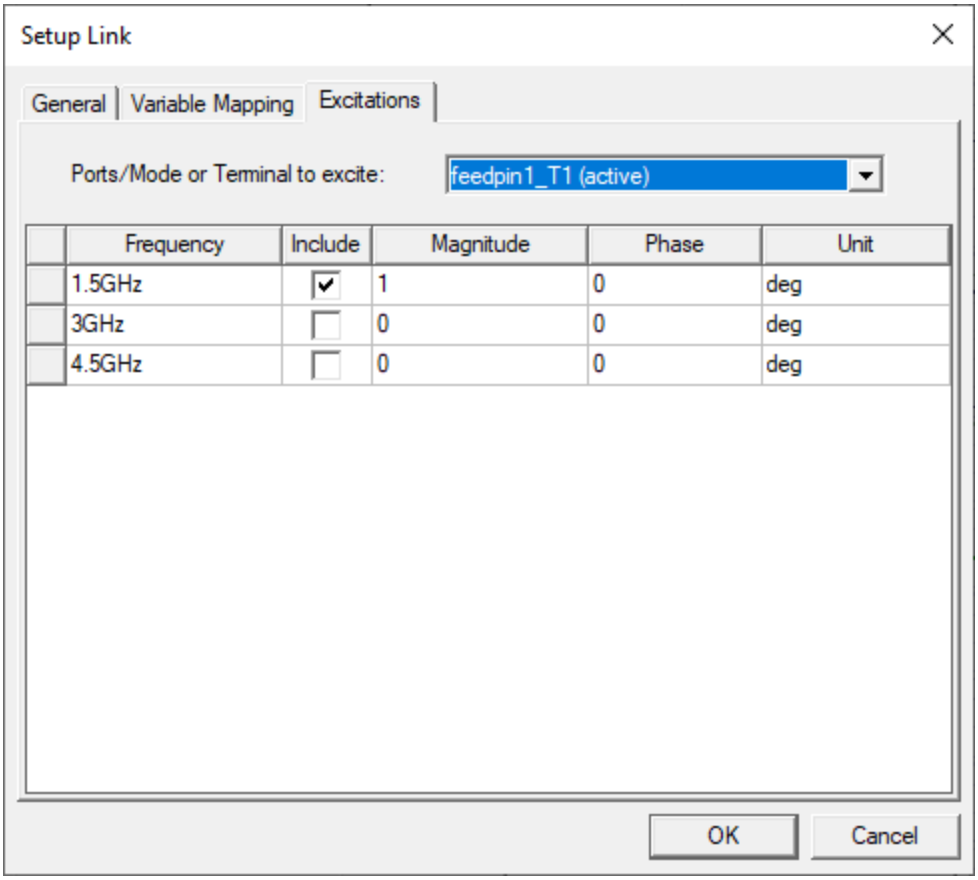
Whether you choose **Sweep Solve** or **Automatic Solve**, you have to click **Setup Link…** to link the multipaction analysis with a frequency sweep before running the multipaction simulation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



By default, the link points to the current project, design, and sweep. After setting up the source, click the **Excitations** tab so you can specify the excitations or the DC bias fields for the multipaction analysis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



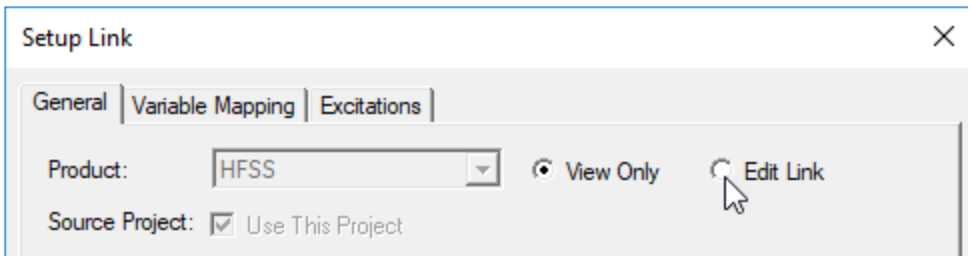
You must include at least one terminal with a positive magnitude. The input signal for each mode or terminal is a linear combination of multi-carrier signals multiplied by the square root of the power multiplier:

$$V(t) = \sqrt{p}\sum_{n=1}^{N} V_i^{1W} A_i \sin\left(2\pi f_i t + \phi_i\right),$$

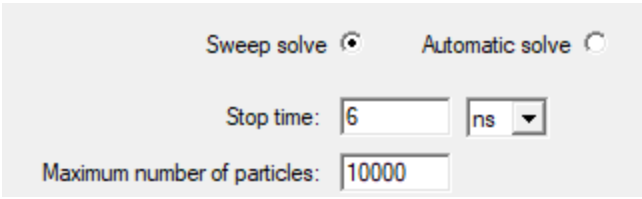
where V_i^{1W} is the port voltage of the *i*-th carrier with one-watt input power in driven-modal solutions. For driven-terminal solutions, the input power of a port is not one-watt. Therefore, scaling must be done before applying the above formulation for calculating the port voltage.

After setting up the excitations, press **OK** to complete the link setup. Afterward, you can only edit the **Setup Link** if you click **Edit Link**. Otherwise, the Setup Link dialog box will be in read-only mode.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

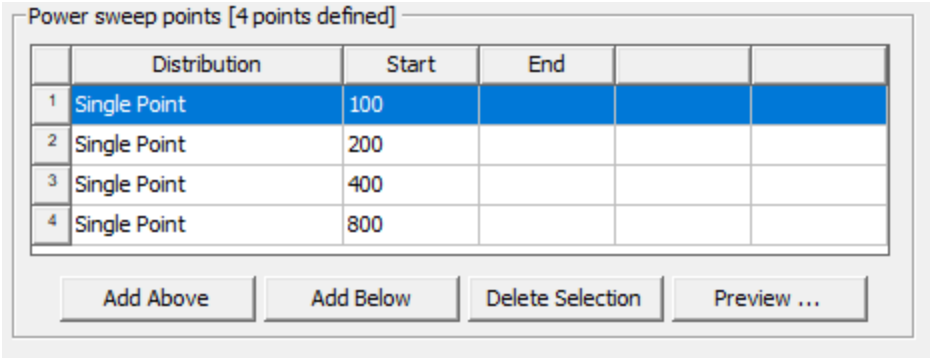


The second step in the **Multipaction Analysis** dialog box, if you have selected **Sweep solve**, is to set up the stop criteria by a stop time and the maximum number of particles.



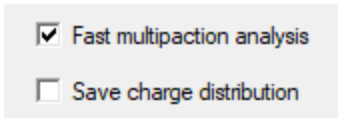
The multipaction simulation will be terminated according to the stop time, which is usually set to 20 cycles of the lowest frequency of the multicarrier signal. The simulation also terminates when the number of particles reaches zero or exceeds the maximum number of particles.

The third step for **Sweep solve** is to specify power sweep points or multipliers for scaling the power of each port mode or terminal voltage. Each mode in a driven-modal design, or each terminal in a driven-terminal design has one-watt input power. When there is only one mode or one terminal active, the power multiplier is essentially the input power of the multipaction analysis.

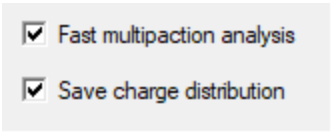


The fourth step for both **Sweep solve** and **Automatic solve** is to select whether the multipaction analysis type will be fast or not. The fast simulation is less compute-intensive but also less accurate when the structure to be analyzed has complex geometry and fine details. However, our tests show that the fast simulation is satisfactory for most applications.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

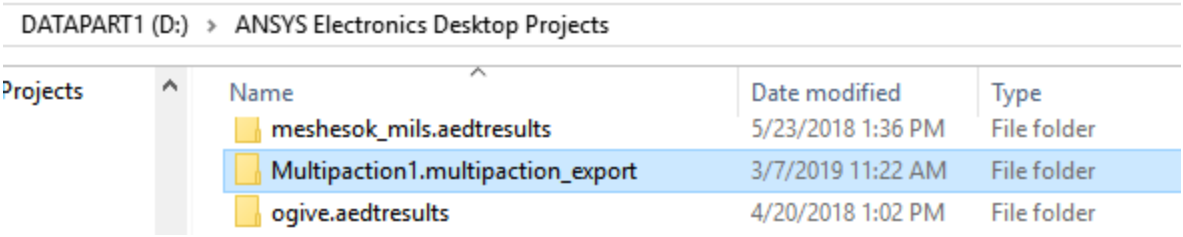


The last step is to choose whether to write out Charge distribution and model geometry. This data can be displayed as particle overlays and animations in Electronics Desktop.



Saving charge distribution involves writing to a hard drive and will slow down the simulation. If the goal is to predict the multipaction power threshold, the overhead can be avoided by unchecking the option.

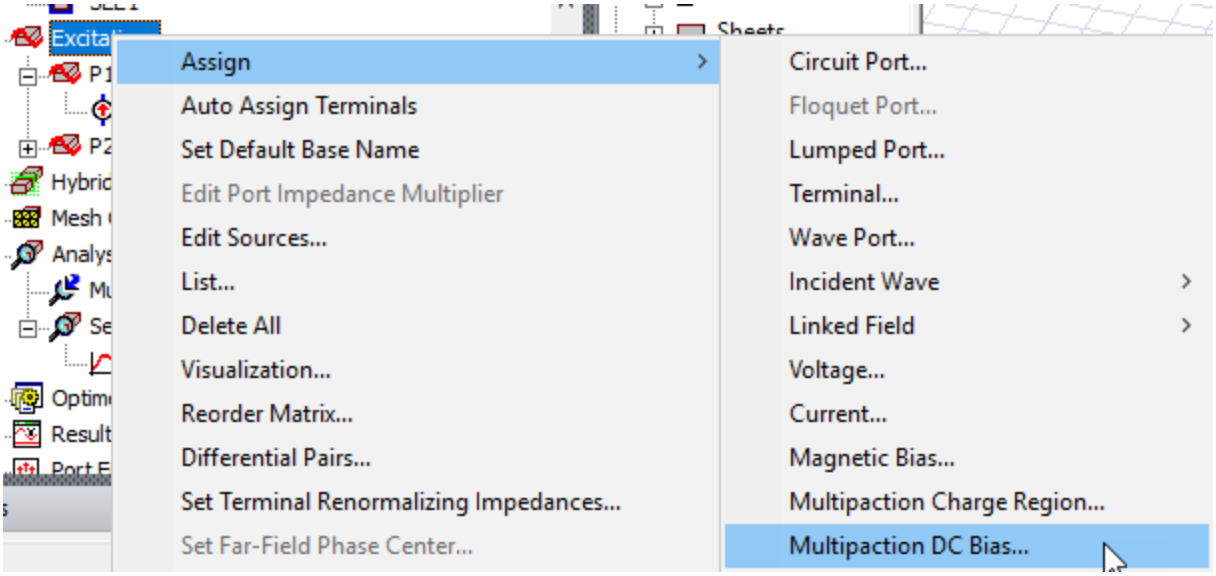
The model and particle files (*.vtu) will be written to a directory at the same level as the HFSS project:



Optional Multipaction DC Bias

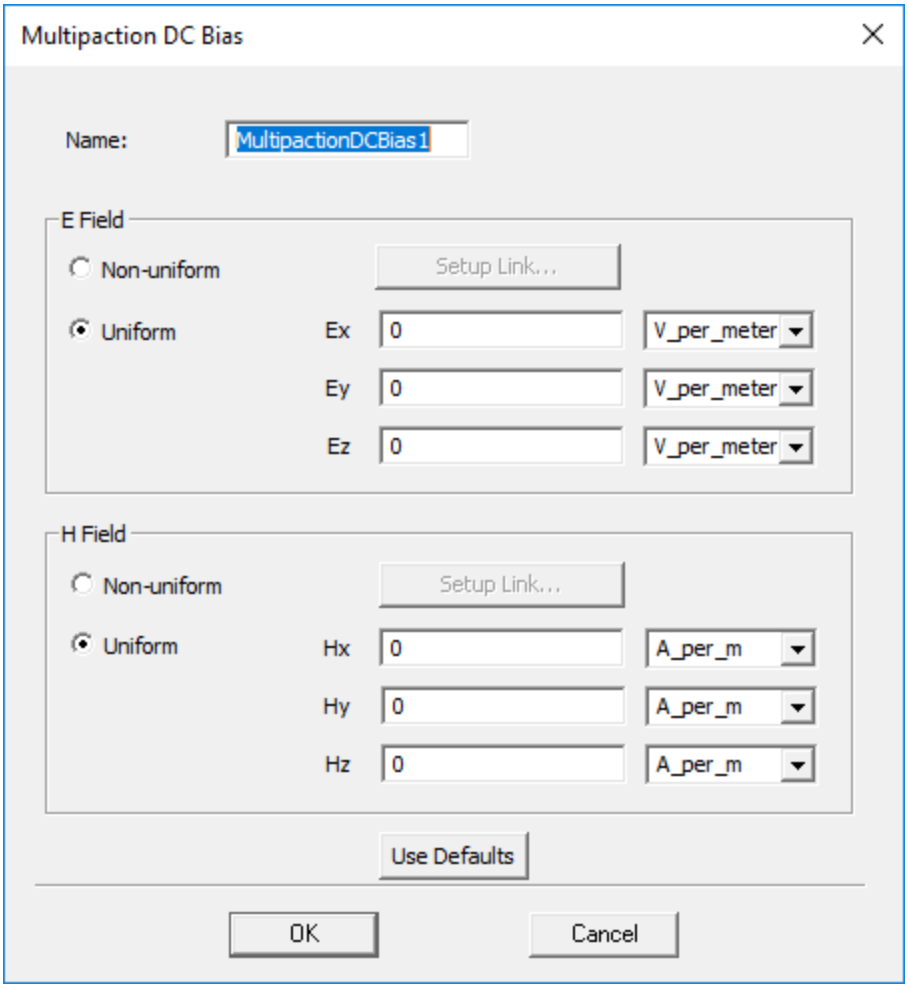
It is optional to set up DC electric or magnetic bias fields for a multipaction analysis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This opens the Multipaction DC Bias dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

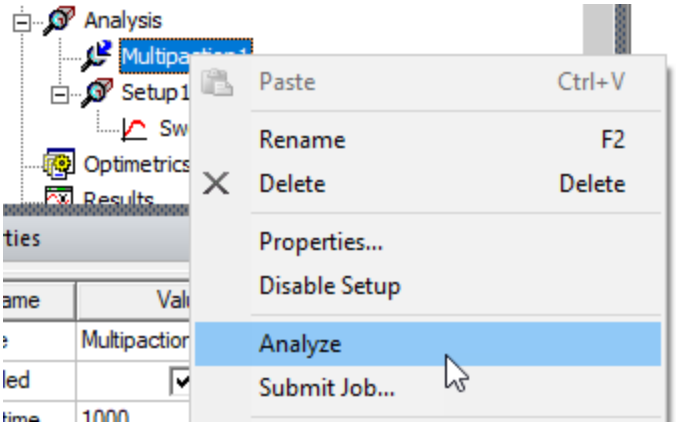


The type of bias can be either uniform or nonuniform. For a uniform bias, the field values are the same throughout the entire simulation domain. The non-uniform bias fields are specified through Maxwell near field links.

Running the Multipaction Simulation

Once the multipaction setup is completed, right-click the multipaction analysis and select **Analyze** to run the multipaction simulation.

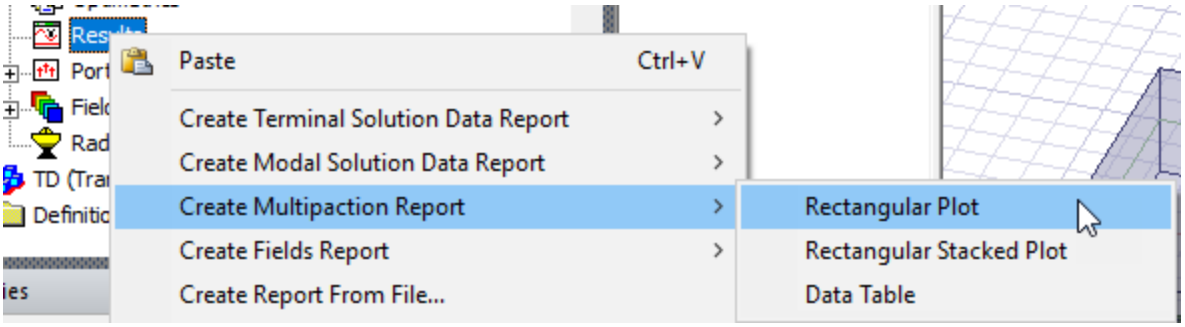
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



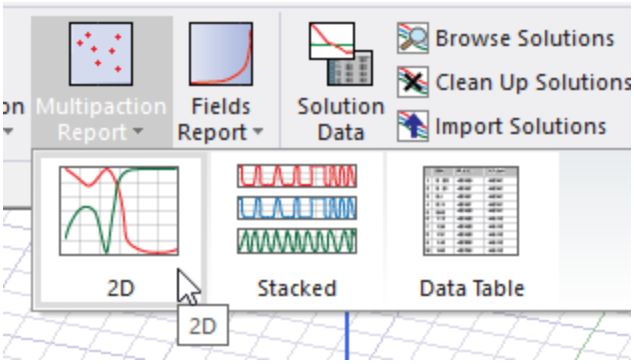
If the 3D field solutions of the linked frequency sweep and the Maxwell projects are not available, additional HFSS and Maxwell simulations will be invoked as shown in the Progress window.

Plotting the Charge Evolution

The final step is to plot the charge evolution through multipaction plots. You can right-click on results, select **Create Multipaction Report**, and select the report type.

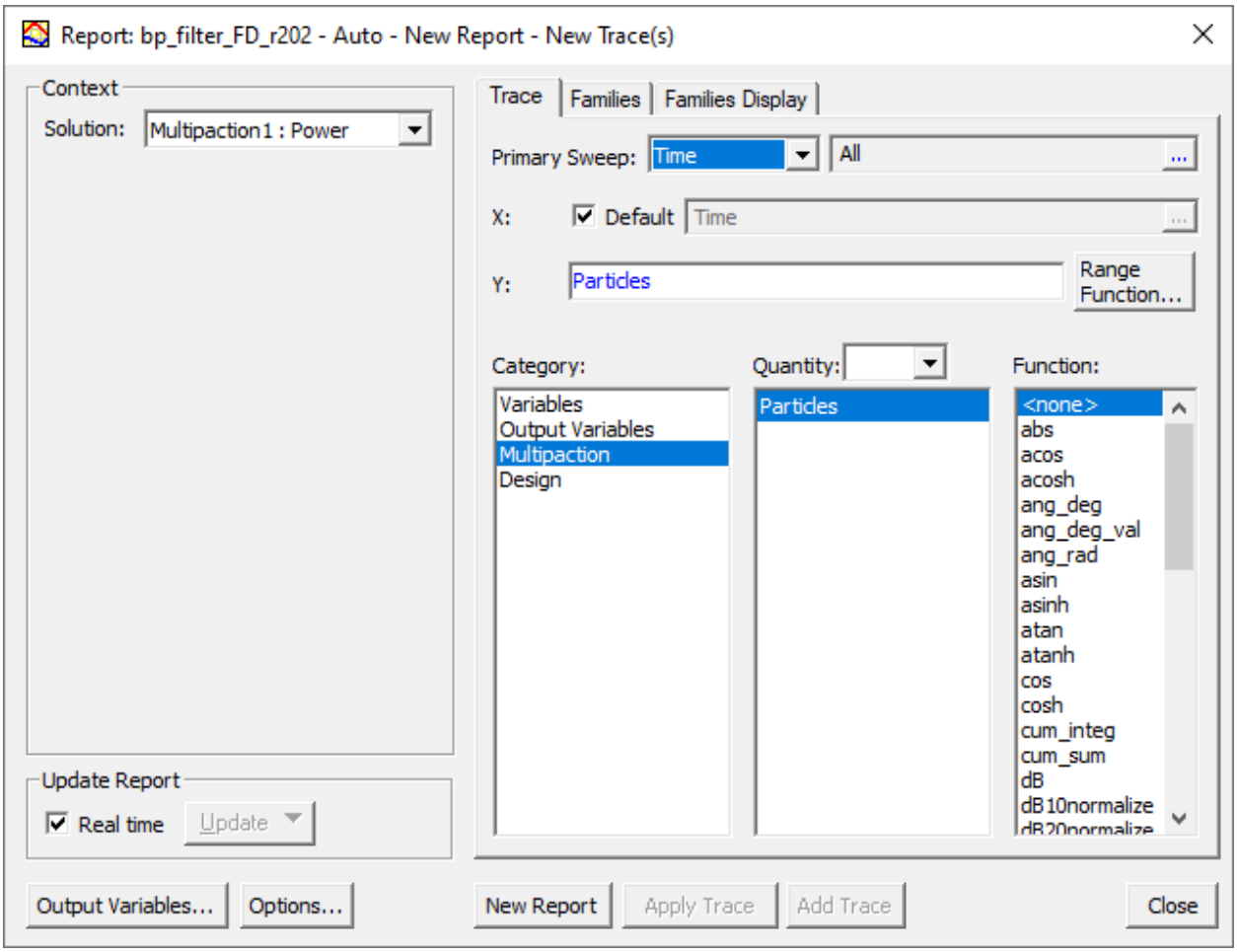


You can also access the Multipaction Reports on the **Results** tab of the ribbon.



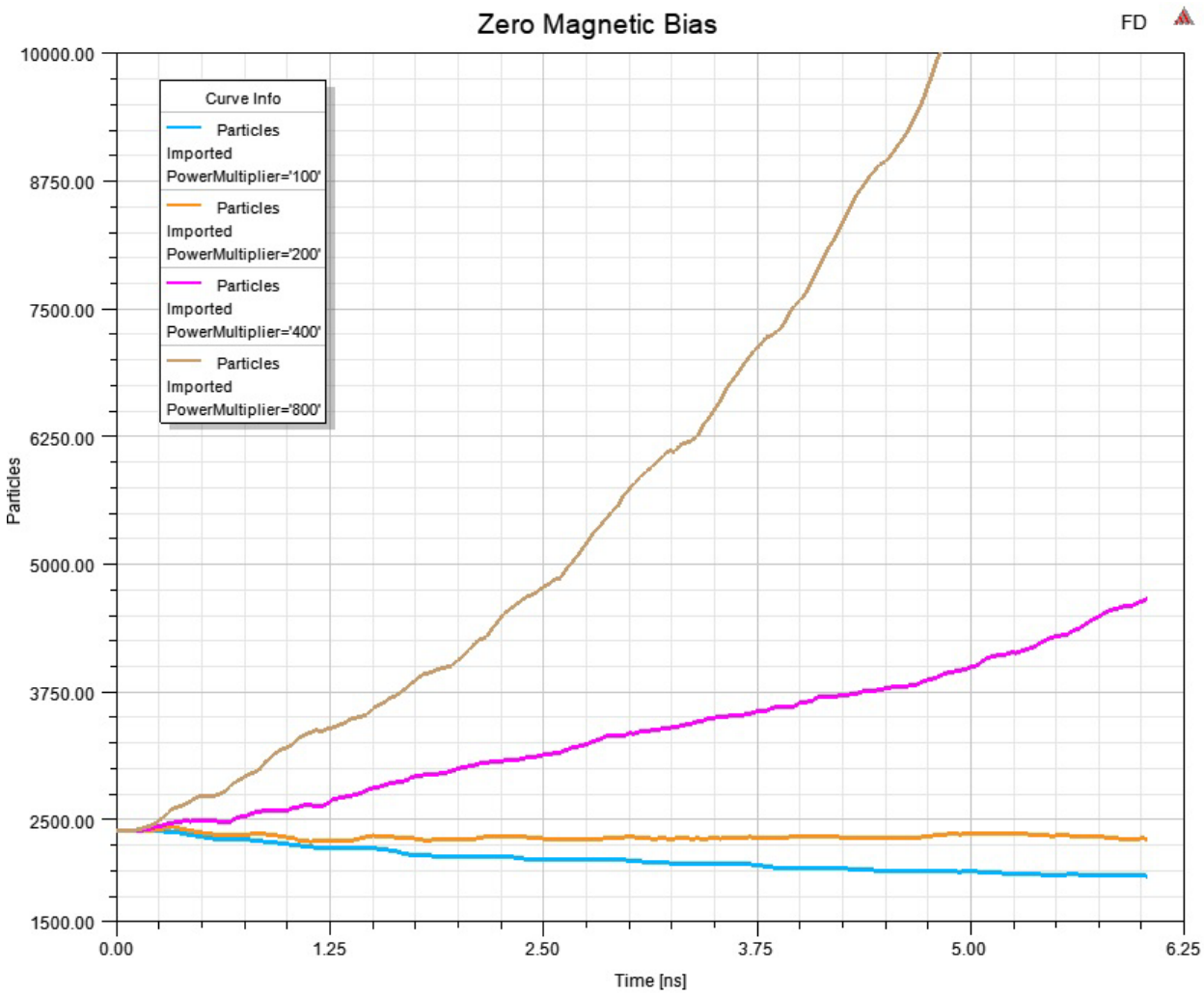
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Report** dialog box, select Multipaction as the Category and Particles as the Quantity.



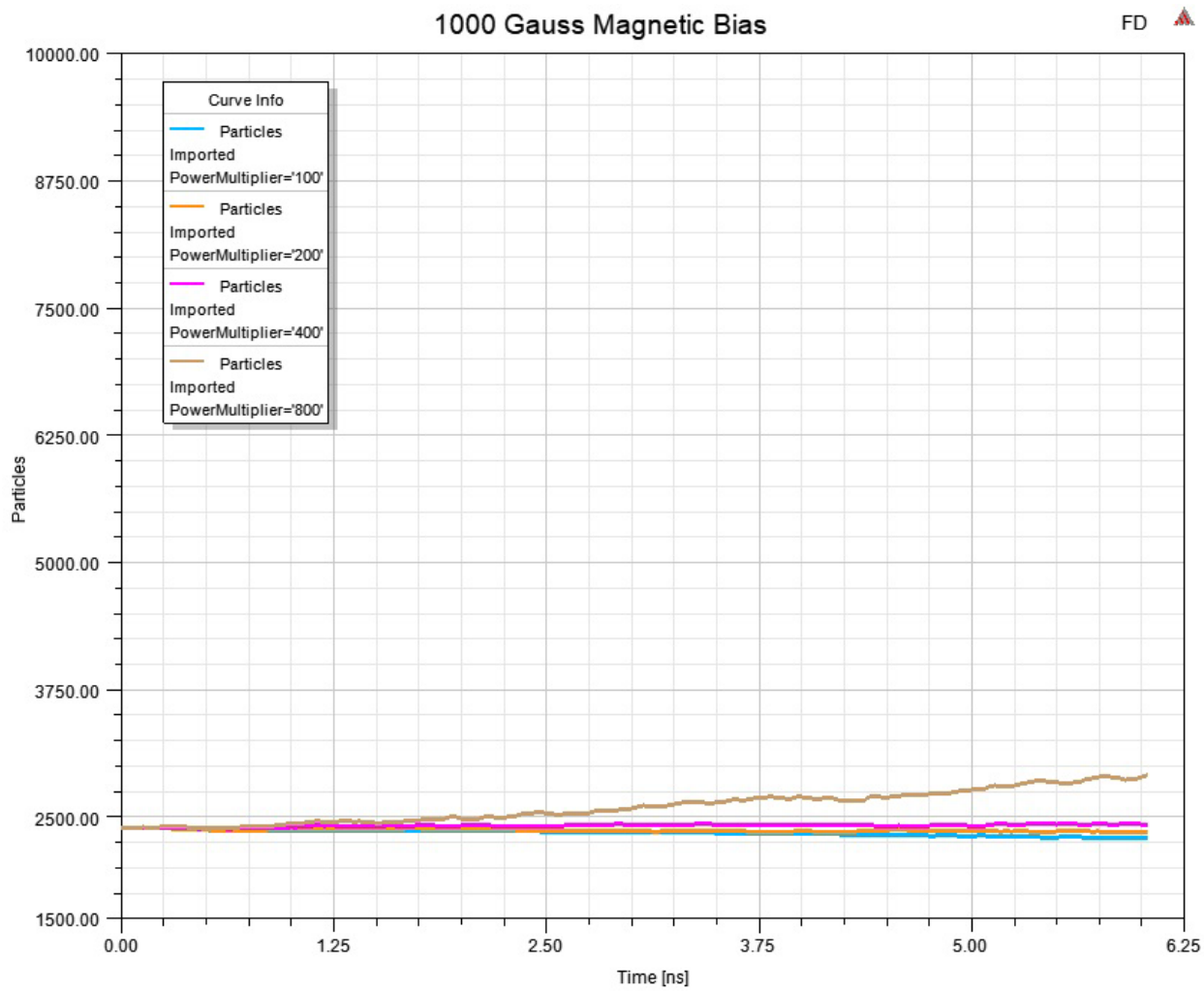
The following example report shows the results with different PowerMultipliers and Zero Magnetic Bias.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



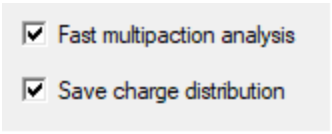
The following example shows a range of power multipliers with a 1000 Gauss Magnetic Bias assigned.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



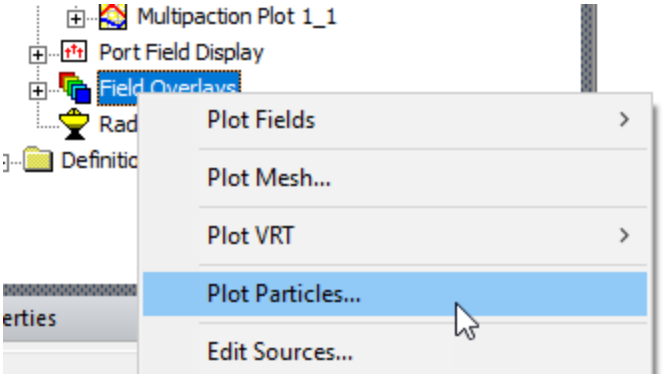
Displaying and Saving Charged Particle Animations

In order to display a charge particle animation, you have to select **Charge Distribution** in the **Multipaction Analysis Setup** dialog box.

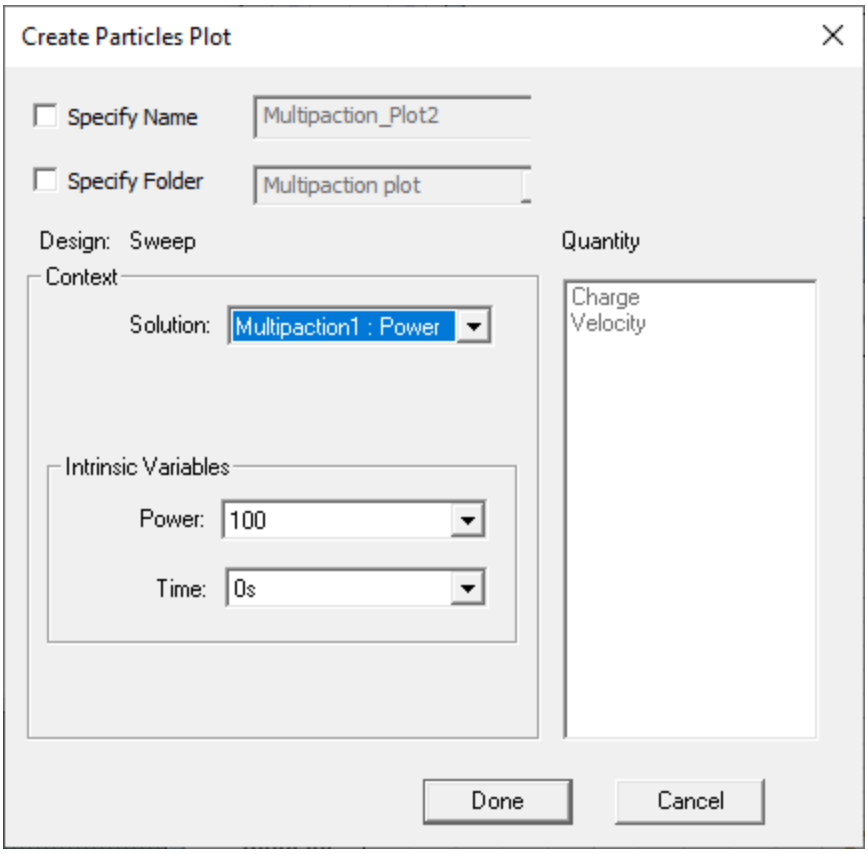


Once the multipaction simulation is finished, you can right-click **Field Overlays** and select **Plot Particles...**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

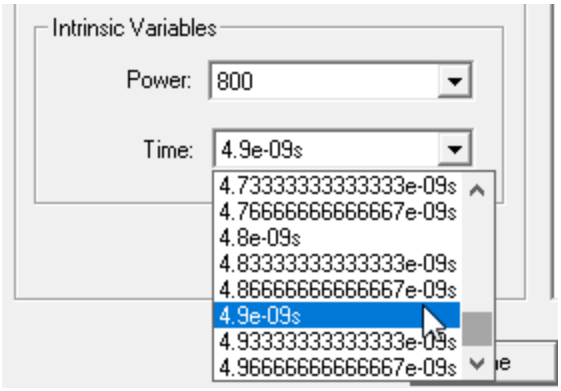


This opens the **Create Particles Plot** dialog.

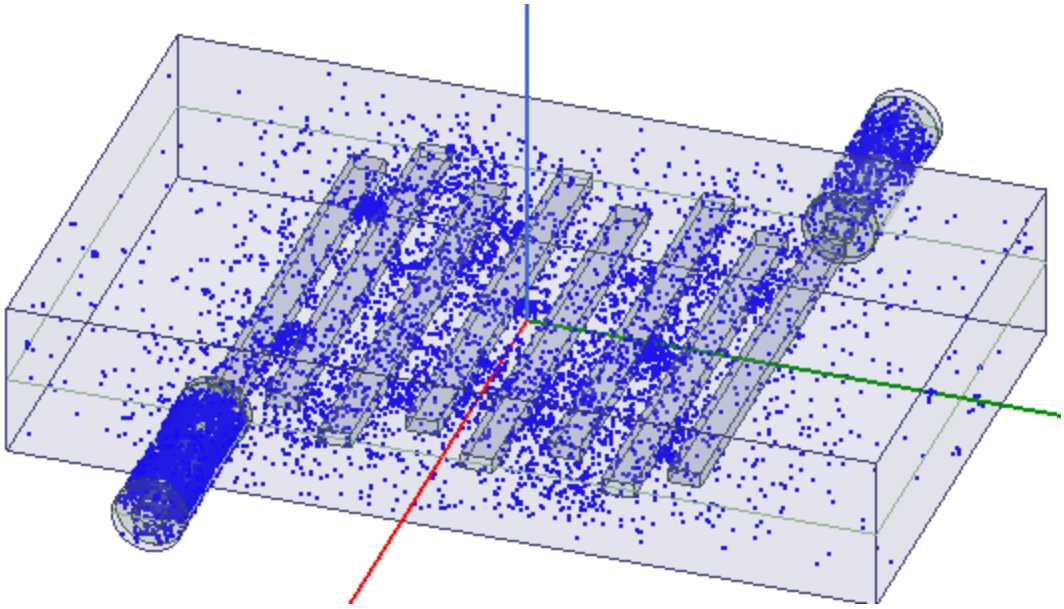


When there are multiple multipaction setups and power multipliers, the dialog allows you to select specific Solution and Power to plot the charged particle distribution and animation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

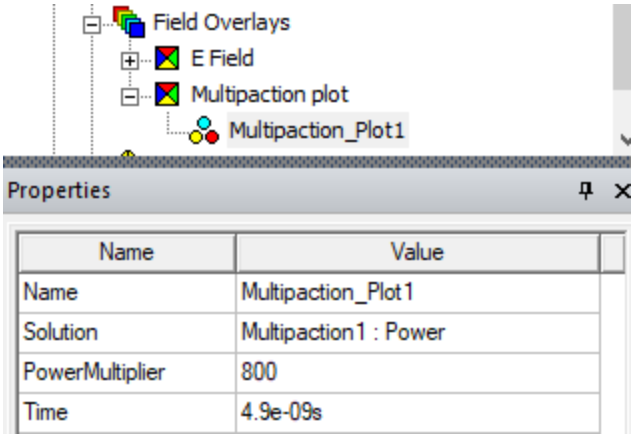


Click **Done** to cause a Particle plot to appear in the Modeler window.

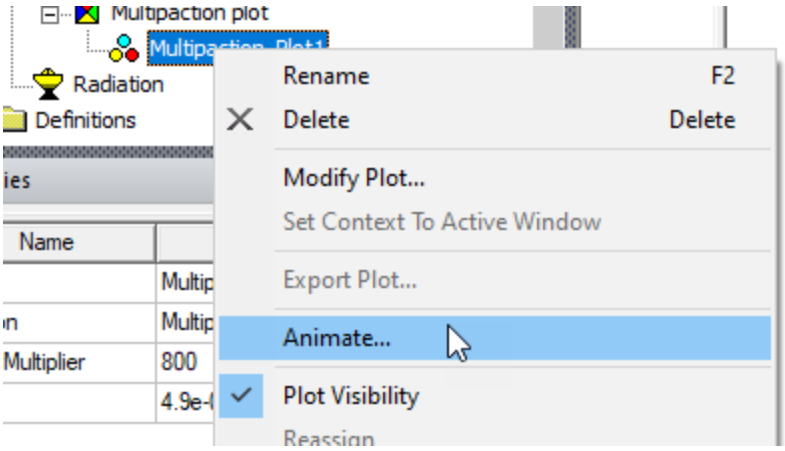


The properties of the particle plot are shown in the Properties window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

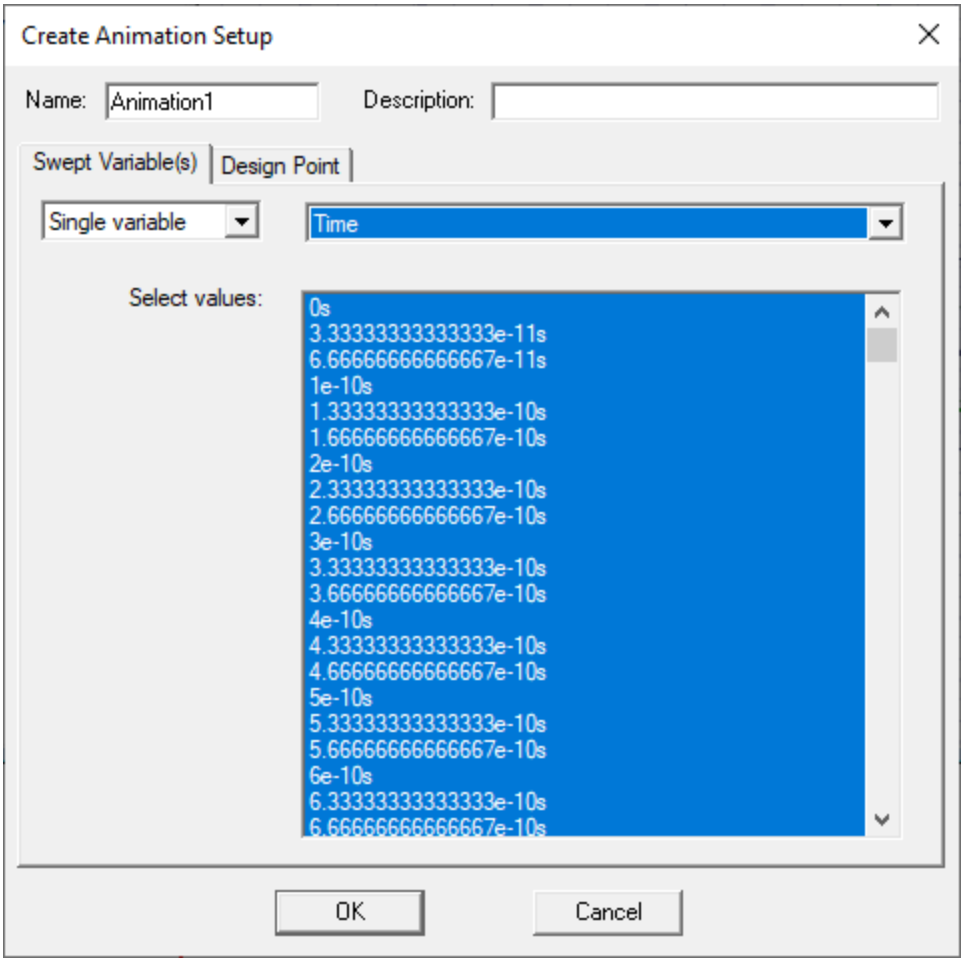


Once you have generated the Multipaction Plot in the Field Overlays, you can create an animation of the movement of the charged particles. Right-click the Multipaction Plot and select **Animate...** from the shortcut menu.



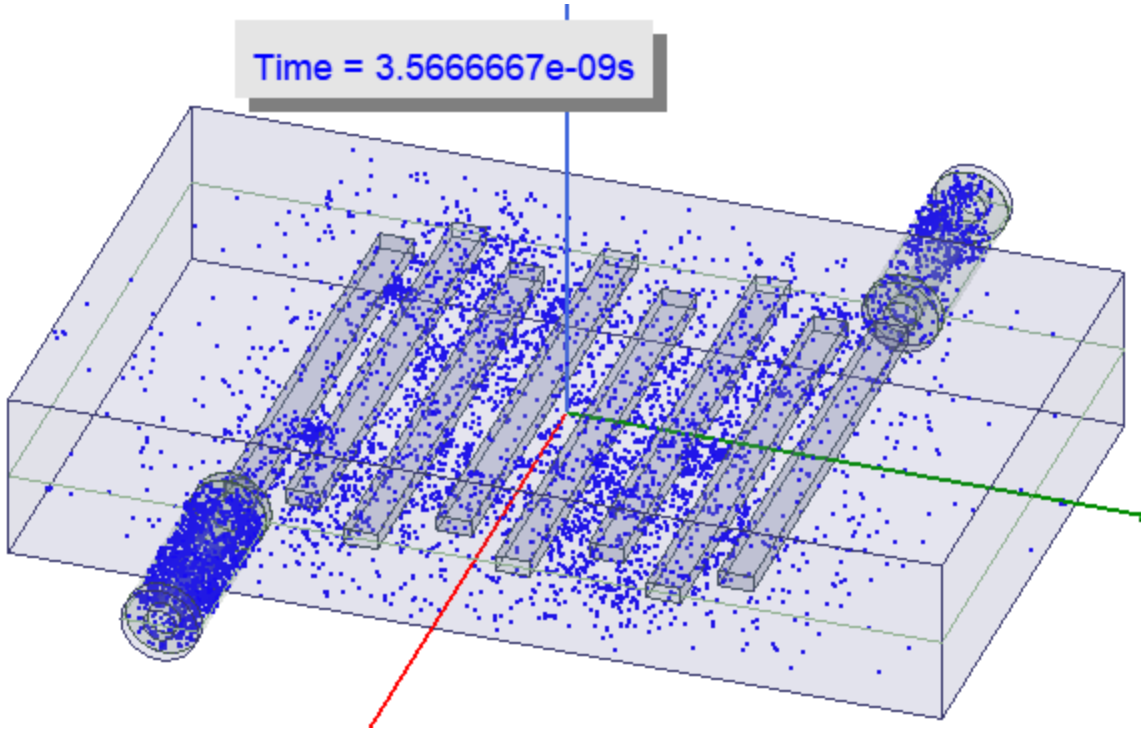
If you have previously created an animation for the plot, the **Select Animation** dialog box will open. You can select an animation to view, Edit, or Delete, or select **New...** to open the **Create Animation Setup** dialog box. If you have not previously created an animation, the Create Animation Setup dialog opens automatically.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

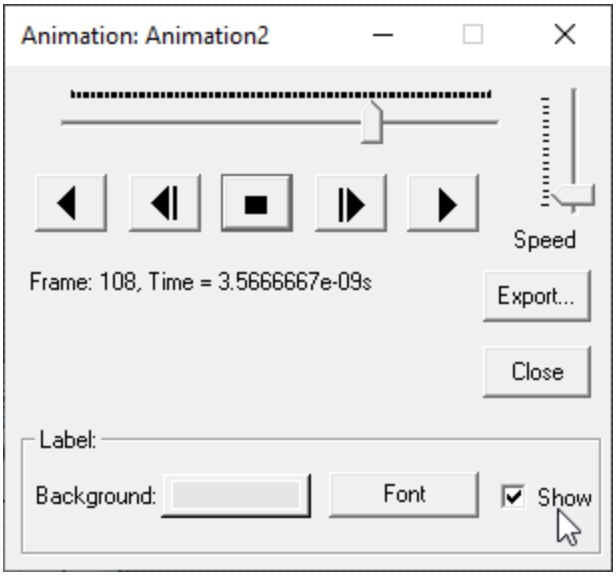


By default, the charge distribution for all time steps will be animated. The animation comes with a label showing the current time.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

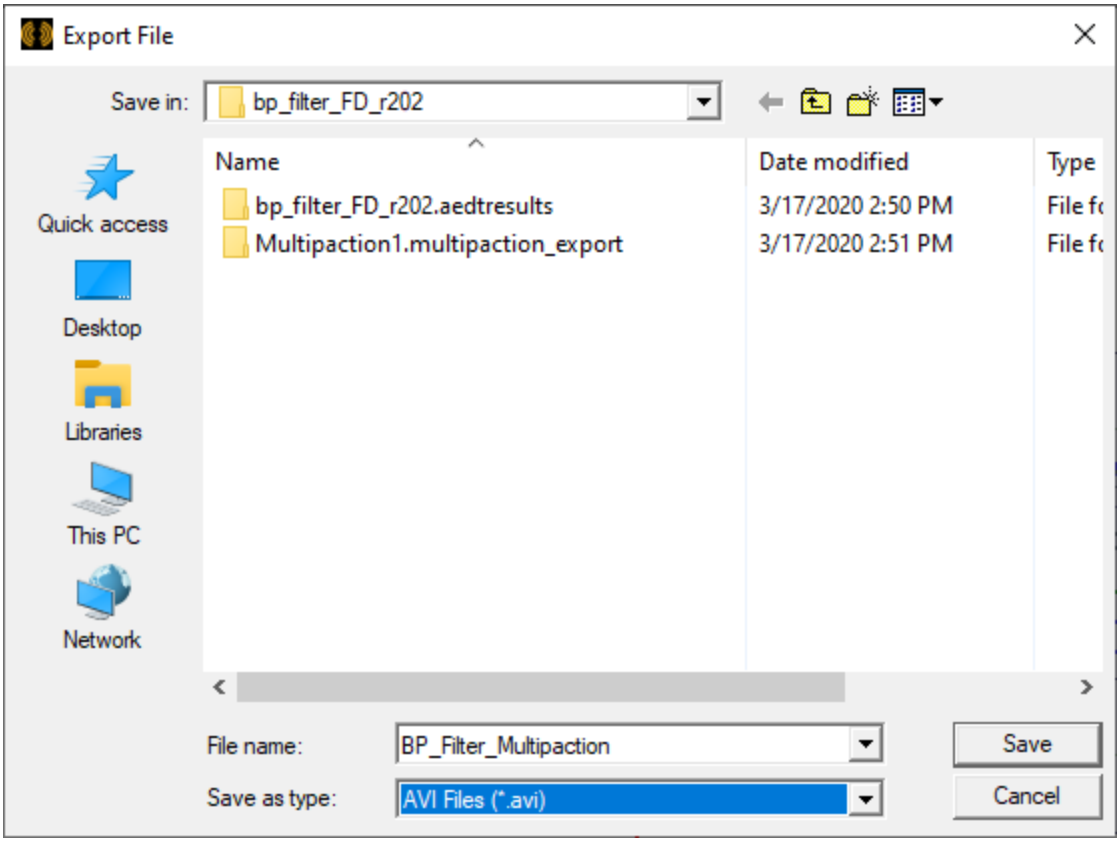


You can remove the label by unchecking Show in the Animation control panel.



To save an animation file, click **Export...** and choose the path and file type of the animation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Files can be AVI Files (*.avi), Animated GIF (*.gif), or Web M Files (*.webm).

To Store and Obtain Power via a Script

You may want to get the power threshold for each autosolve analysis without reading the entire breakdown status table and figuring out the power threshold through a complicated script. You can access the power threshold through a single line of script. Although the power threshold should not be time-dependent, with the current implementation we can sample the value at t = 0.

```
# ----------------------------------------------
# Script Recorded by Ansys Electronics Desktop Version 2025.1.0
# 10:55:07 Jun 05, 2024
# ----------------------------------------------
import ScriptEnv
import ctypes
ScriptEnv.Initialize("Ansoft.ElectronicsDesktop")
oDesktop.RestoreWindow()
oProject = oDesktop.SetActiveProject("JPL_coax_power_threshold_r251")
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
oDesign = oProject.SetActiveDesign("Coaxial_7-8")
oModule = oDesign.GetModule("OutputVariable")
oModule.CreateOutputVariable("GetPowerThreshold", "PowerThreshold",
"100MHz_low : Power", "Multipaction", [])
PowerThreshold = oModule.GetOutputVariableValue
("GetPowerThreshold","Time='0","100MHz_low : Power", "Multipaction",
[])
ctypes.windll.user32.MessageBoxW(0, str(PowerThreshold), "Power
Threshold", 1)
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15 - HFSS RF Discharge

HFSS RF Discharge targets applications related to aerospace and satellite communications where electrical breakdown in gases is a concern. A small number of free electrons naturally exist in gases due to photoionization and irradiation. When the electrons are accelerated by RF/microwave waves, they gain enough kinetic energy to ionize neutral gas atoms or molecules. The ionization is easier to happen at high altitudes or a space environment due to low gas pressures and abundant cosmic rays. If the number of electrons grows exponentially due to cumulative ionization, it can eventually trigger a discharge to damage RF/microwave circuits and sources. At high frequencies, the discharge mainly depends on the electron transport properties of the gases rather than the electron emission properties of the electrodes. This tool solves for the condition which leads to the onset of gas discharge not completely bridging material walls.

An RF discharge analysis is useful for predicting electrical breakdown in the following applications:

- TT&C (Telemetry, Tracking, and Command) subsystem switched on during satellite launching
- Uplink transmission to communication satellites
- Outgassing during the lifetime of satellites or space probes
- Re-entry vehicles and interplanetary missions
- Airborne radars and communication systems

It requires the time-harmonic fields being solved for a driven-terminal or driven-modal projects. Therefore, the RF discharge setup and associated DC magnetic bias (possibly nonuniform through Maxwell link) should be added to aforementioned types of HFSS projects. You can also perform an RF Discharge simulation on an Eigenmode solution. Other solution types, such as Transient, Characteristic Mode, and SBR+, do not support the RF discharge analysis. By option, you can save and plot electron density data.

Prerequisites

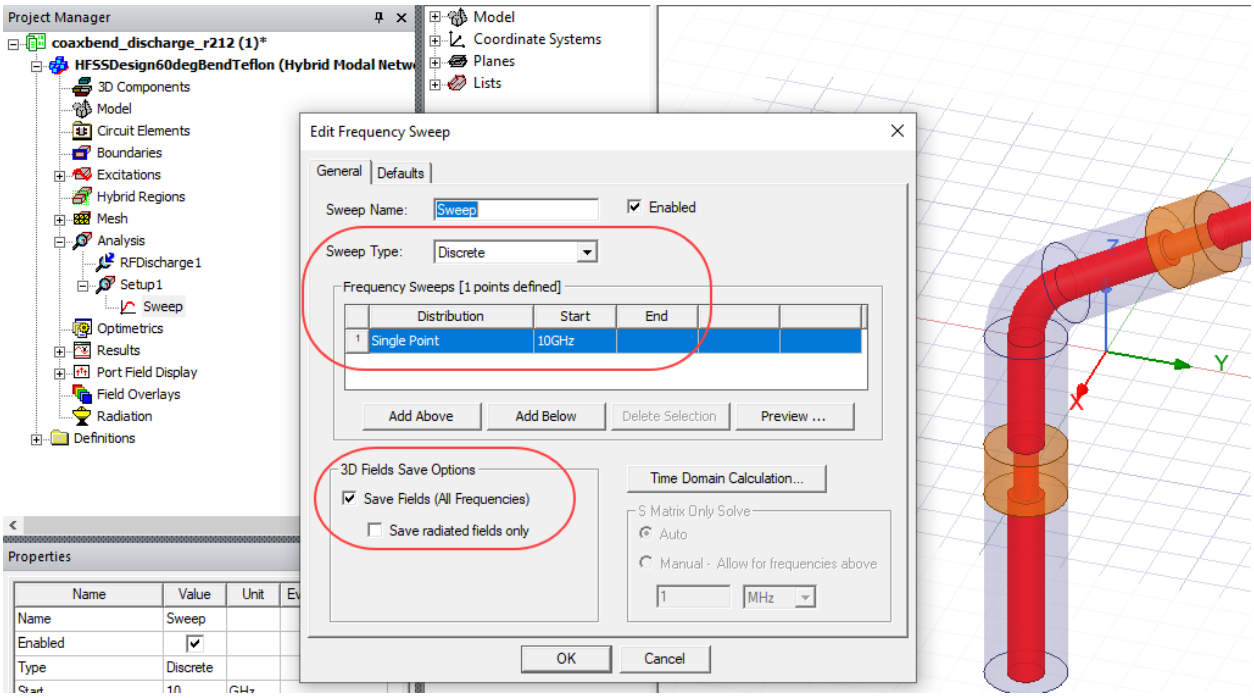
The Solution type must be Driven Terminal or Driven Modal or Eigenmode.

Save Fields for a Driven Terminal or Driven-Terminal Project

An RF discharge simulation requires time-harmonic fields as excitations. You must edit the frequency sweep of a driven-modal or driven-terminal project to save fields. However, storing fields takes time and disk space, especially when the size of the project and the number of frequency points are large. In addition, the RF discharge solver will solve one Paschen curve for each frequency. To reduce the simulation time and avoid clogging the RF discharge plot, should

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

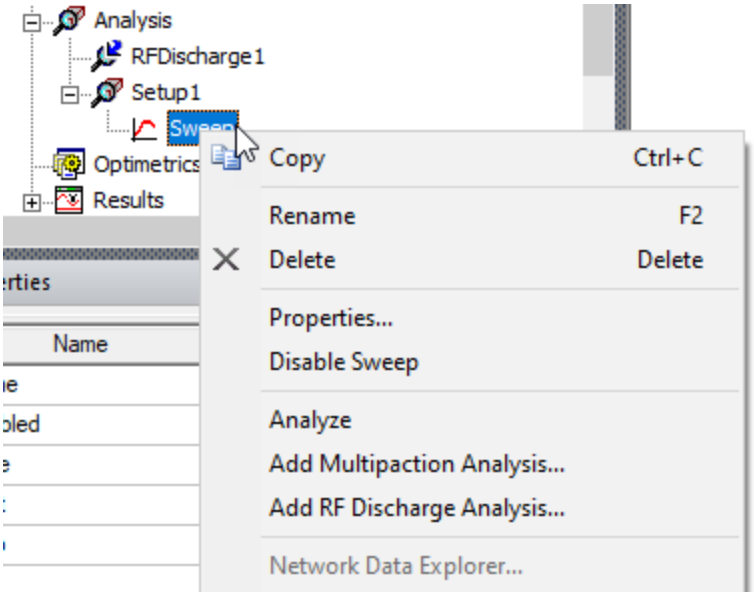
create a frequency sweep with few frequency points for linking with an RF discharge analysis. A frequency sweep for interpolating sweep of S-parameters over hundreds of frequency points is not suitable for linking with an RF discharge analysis.



Add an RF Discharge Analysis to a Driven Solution

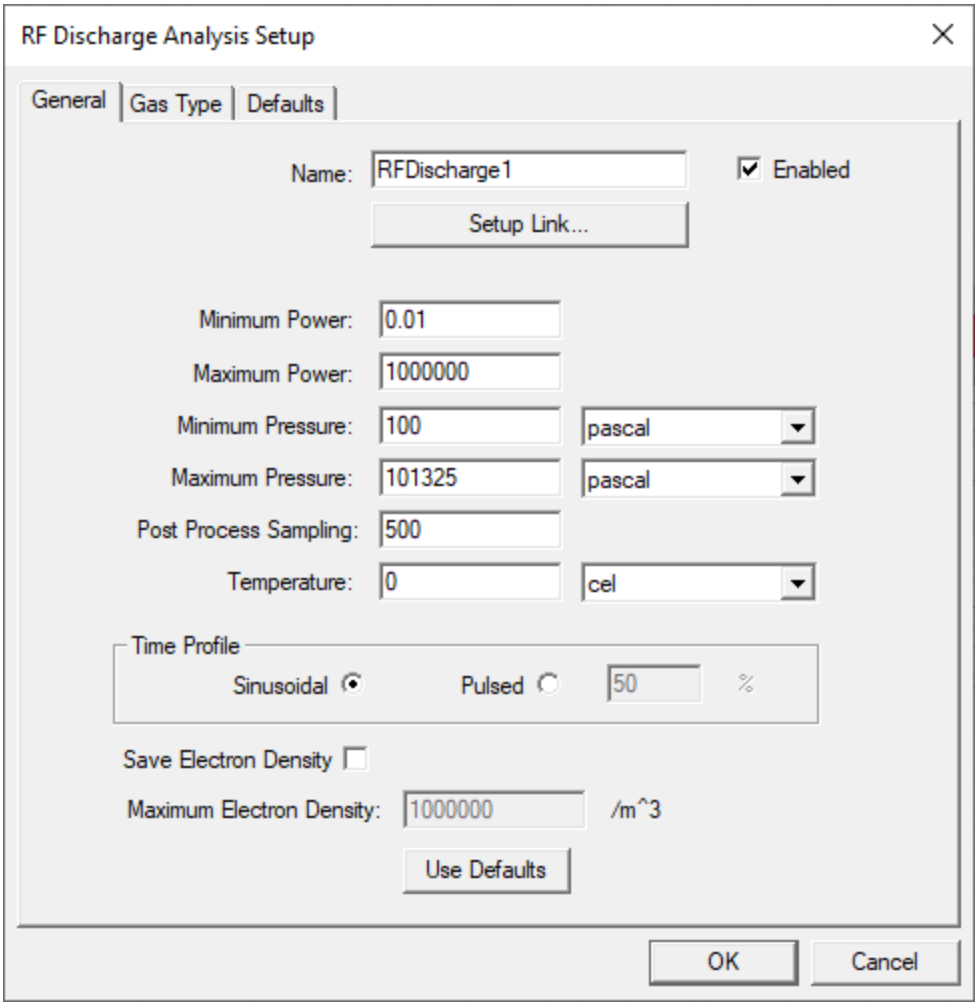
An RF discharge analysis can be created only when there is a frequency sweep with **3D Fields Save Options** checked. Otherwise, the featured is grayed out. Right click the frequency sweep to **Add RF Discharge Analysis**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **RF Discharge Analysis Setup** dialog lets you specify General parameters for Maximum and Minimum Power and pressure, as well as sampling and temperature values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

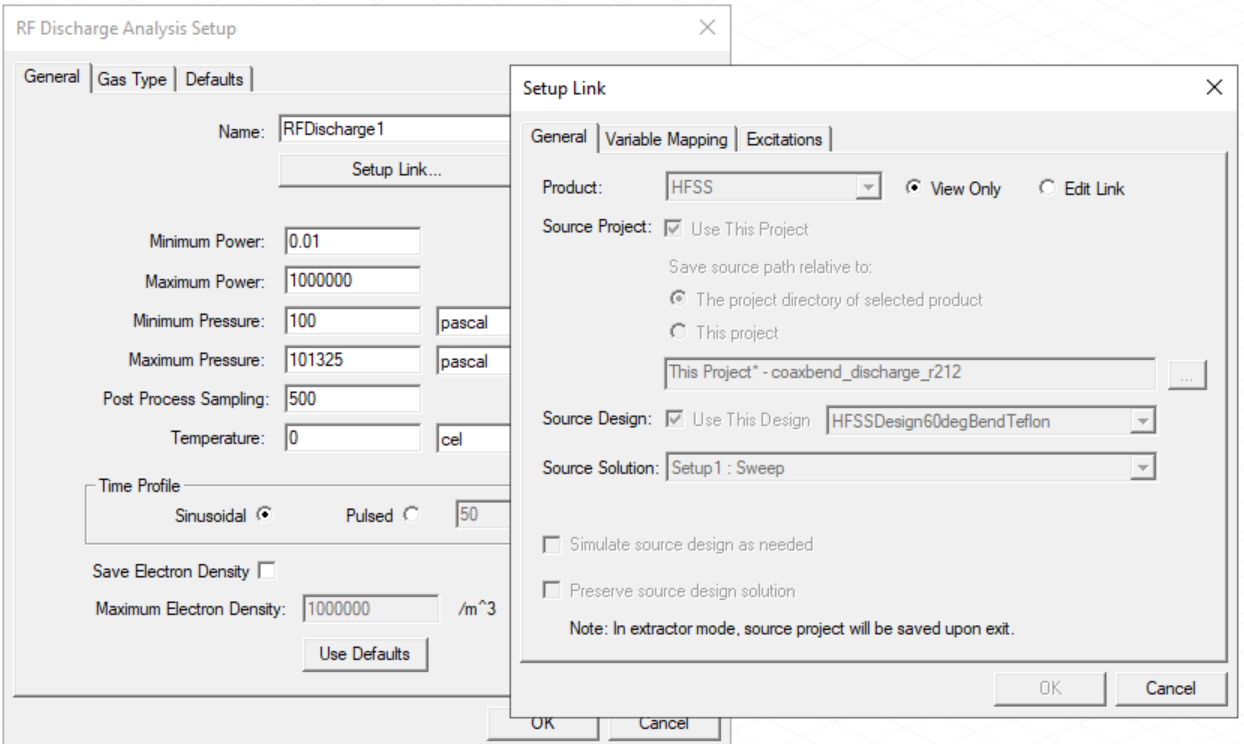


By default, the pressure ranges from 0.1 to 101.325 kPa. You set appropriate power and pressure bounds according to the operating range of your designs.

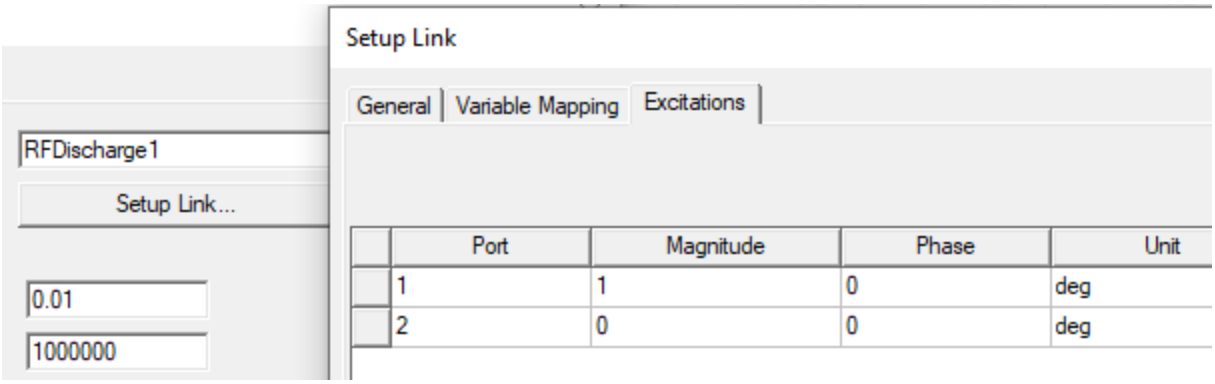
Select the Source and Port Excitations

A driven design must contain at least one active frequency sweep with at least one active port for you to set up the source link. When the solution type is Modal Network or Terminal Network, the total field used in discharge simulation is a linear combination of field patterns of individual ports weighted by the complex multipliers (phasors) listed in the table. You set the input power of each active port on the **Excitations** tab of the **Setup Link** dialog to one Watt.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



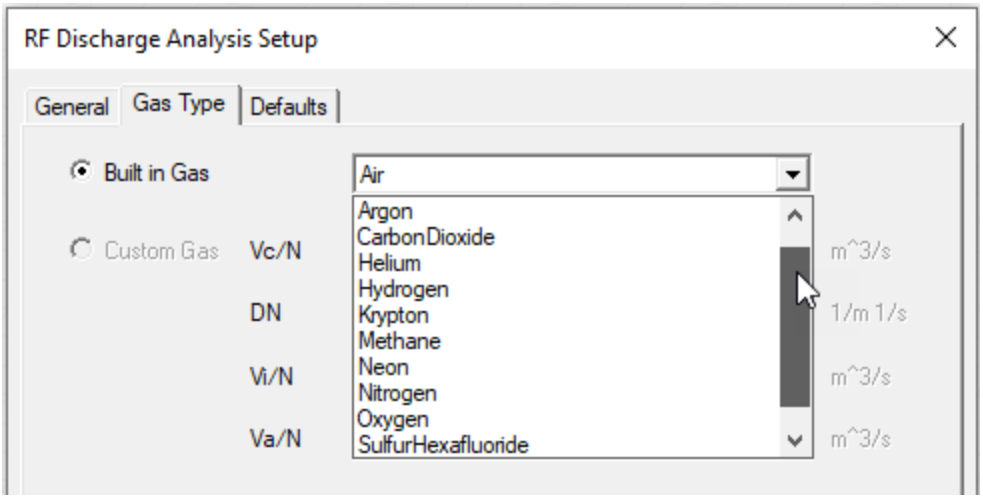
The total field used in discharge simulation is a linear combination of field patterns of individual ports weighted by the complex multipliers (phasors) listed in the table. You set the input power of each active port on the **Excitations** tab of the **Setup Link** dialog to one Watt.



OK the **Setup Link** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Gas Type for RF Discharge Analysis



Click the **Gas Type** tab to select either Built-in or Custom gases. There are 12 built-in gases in the default gas library: Air, Argon, Carbon Dioxide, Helium, Hydrogen, Krypton, Methane, Neon, Nitrogen, Oxygen, Sulfur Hexafloride, and Xenon. Use the arrows to scroll through the drop-down list. Non-conductive materials in the simulation domain with relative permittivities no more than 1.02 and relative permeabilities in between 0.99999 and 1.00001 will be replaced by the gas selected from the list. For example, air and vacuum in the HFSS material library will be replaced according to the above rule.

A Custom Gas can be specified by four datasets of the electron collision frequency (V_c/N in m^3s^{-1}), the diffusion coefficient (DN in $m^{-1}s^{-1}$), the ionization rate (V_i/N in m^3s^{-1}), and the attachment rate (V_a/N in m^3s^{-1}) as functions of the reduced electric field E/N (Td). If the project or the design does not have any dataset defined, the Custom Gas parameters will be grayed out on the **Gas Type** tab of the dialog.

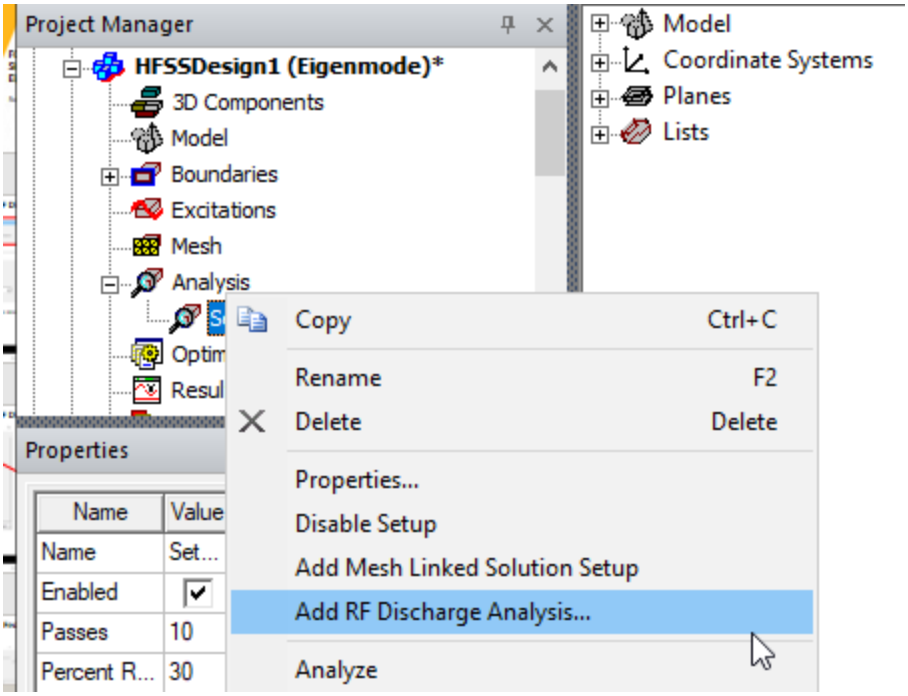
Click **OK** to complete the setup.

Please note that the gas parameters of built-in gases are based on measured data with E/N between 0.1 and 5000 Td. The limitation may cause simulation results to deviate from ideal Paschen curves especially at low pressure.

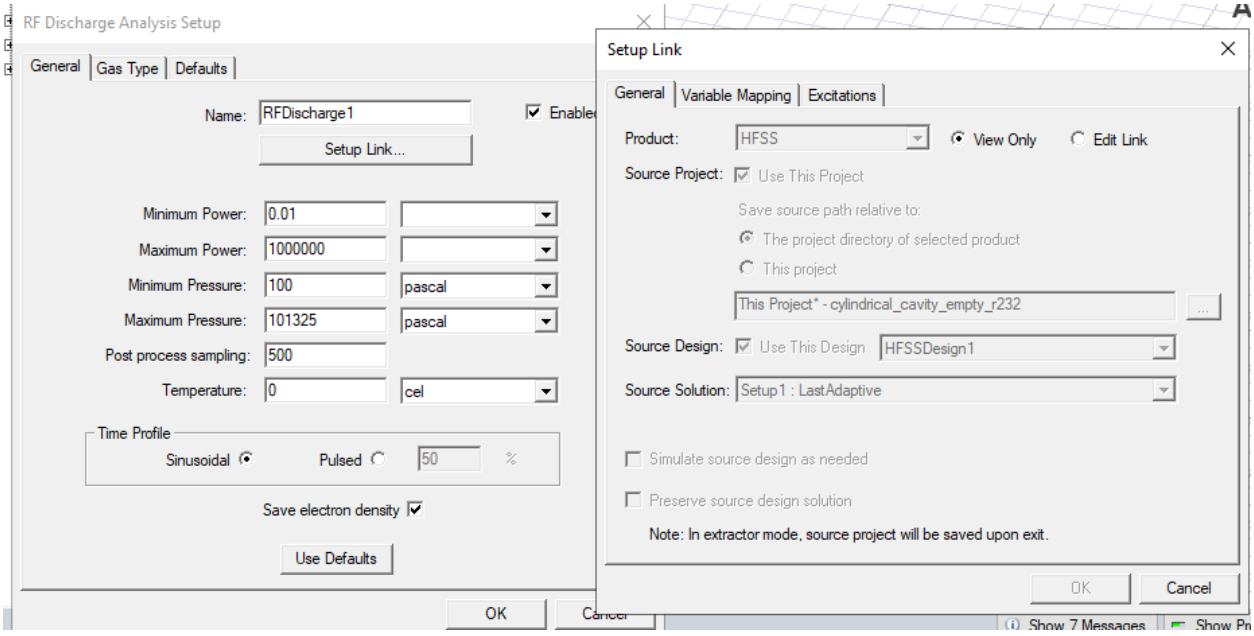
Add an RF Discharge Analysis to an Eigenmode Solution

You can add an RF Discharge analysis to Eigenmode solution.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



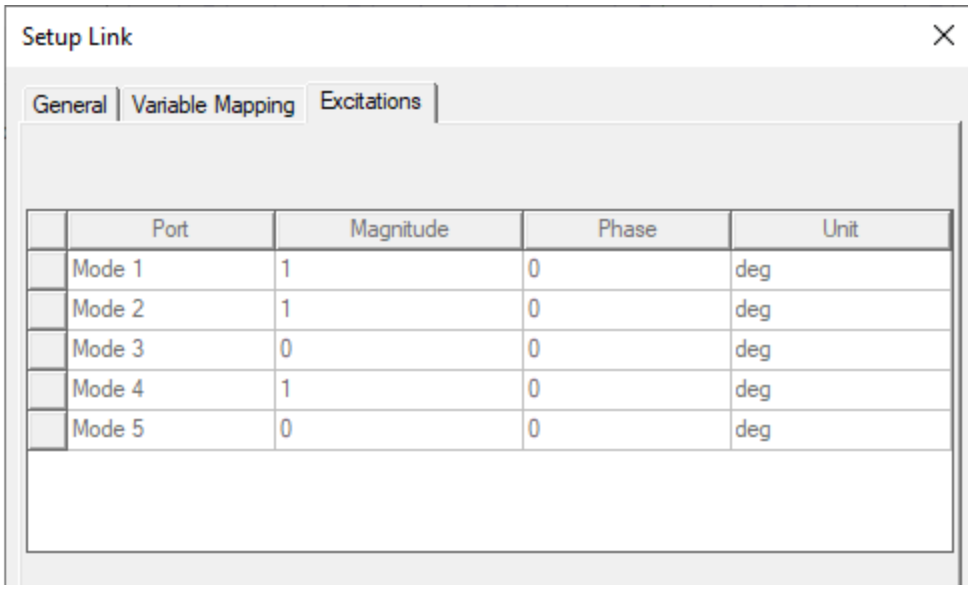
Use the **Add RF Discharge Analysis...** command to set parameters in the and **Setup Link...**



When the solution type is Eigenmode, the fields used in discharge simulation are individual field patterns of each mode weighted by the complex numbers (phasors) listed in the table. In the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

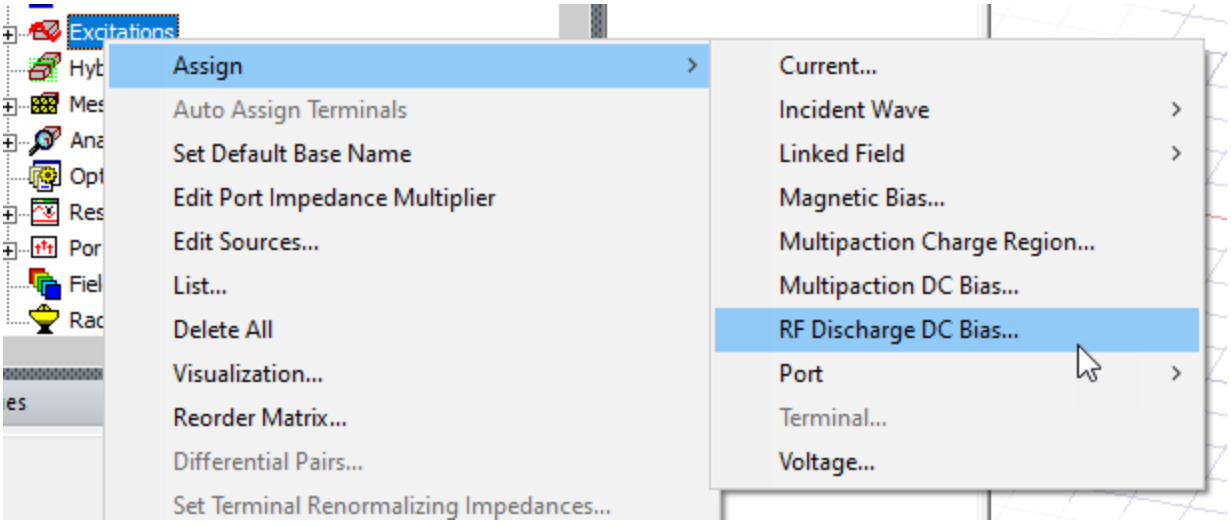
following example setup, you set the stored power of Mode 1 on the **Excitations** tab of the **Setup Link...** dialog to be one Watt.



Please note that there will be one Paschen curve for each mode. The excitation is not a linear combination of eigenmodes in RF discharge simulation.

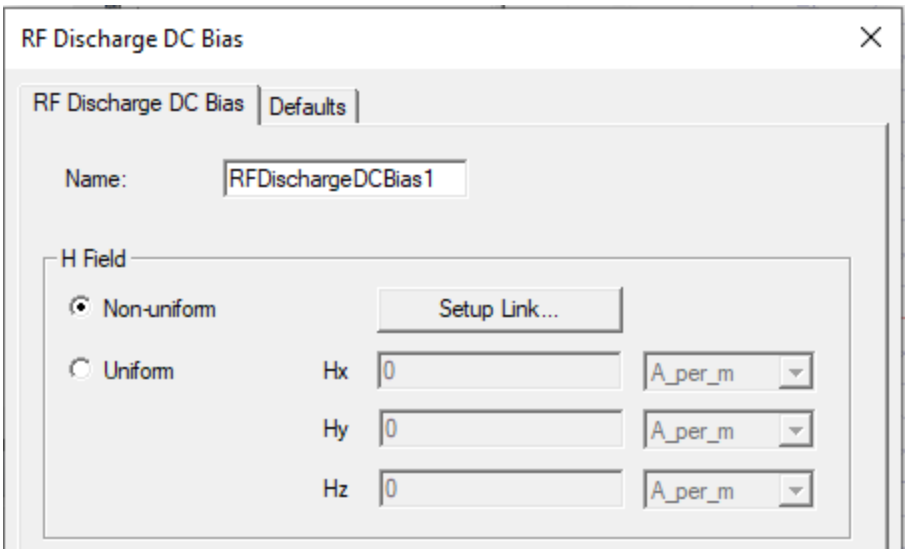
(Optional) Assign DC Magnetic Bias

To assign a DC magnetic bias in an RF discharge simulation, the first set the Model Window Selection Mode must be changed to Object. Then select the relevant object in the Model window or History tree, and right-click a solid object to assign an RF Discharge DC Bias.

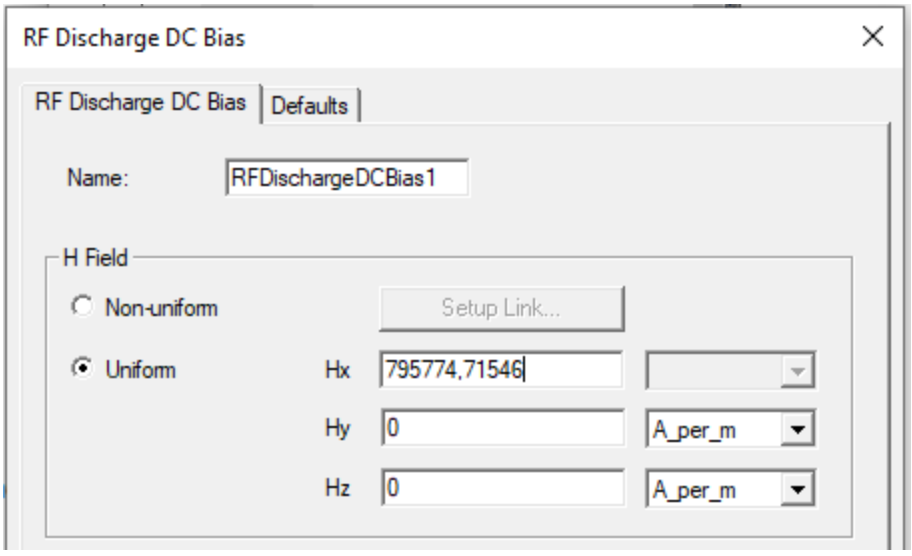


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **RF Discharge DC Bias** dialog opens. You can specify a uniform bias, or setup a link to a Maxwell project for a non-uniform bias mapped to the selected region.



For example, in the following figure, the region is assigned a uniform magnetic bias of one Tesla (T). The conversion is through $Bx\ (T) = \mu 0\ (Tm/A)\ Hx\ (A/m)$, where $\mu 0 = 4\pi \times 10^{-7}$ is the free space permeability.

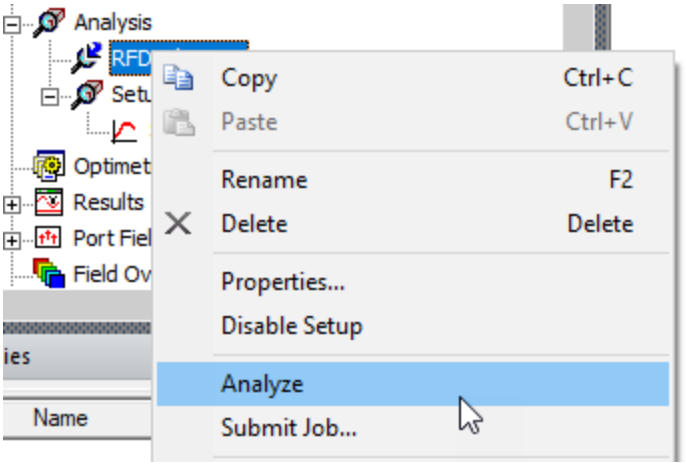


You can also assign a nonuniform magnetic bias through Maxwell link where the solution of the linked Maxwell project is mapped to the selected region in the RF discharge simulation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Run RF Discharge Simulation

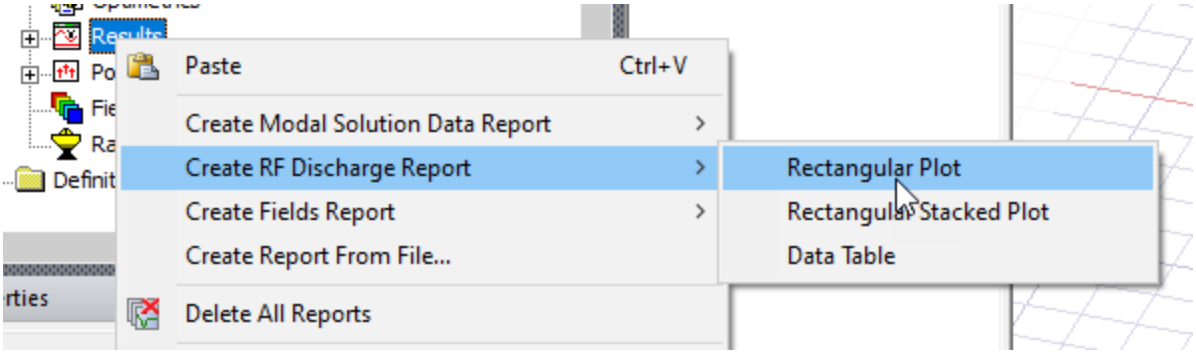
Once the analysis setup has been completed and the magnetic bias has been added, click **Analyze** to run the RF discharge simulation.



The Frequency Sweep will run first, then the RF Discharge simulation. You watch these occur through the Progress window. At completion, a message appears in the Message window.

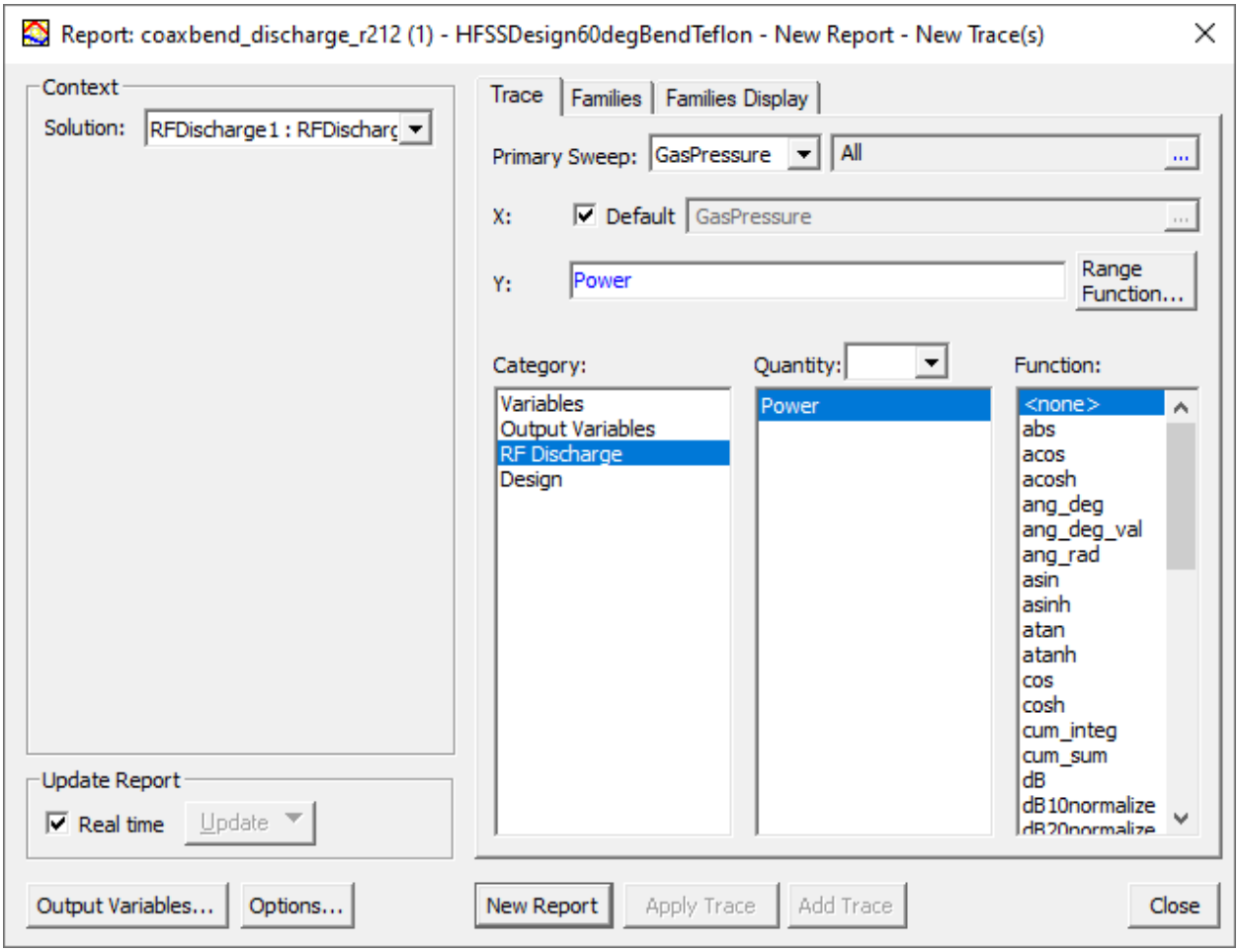
Create an RF Discharge Report

Right click on the **Results** icon and select **Create RF Discharge Report** and the type of plot.



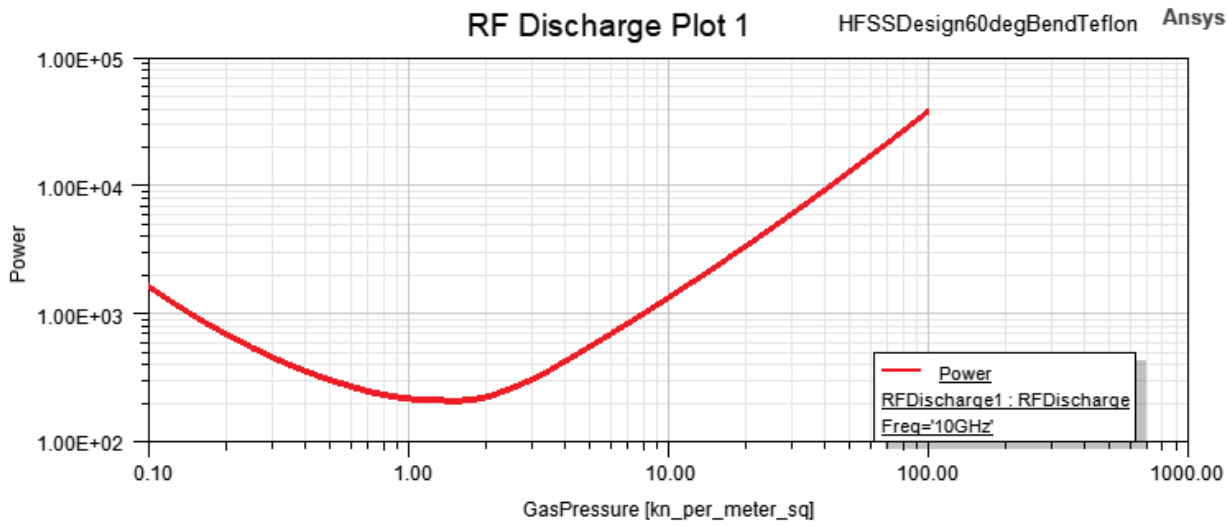
The **Report** dialog opens up. Select **RF Discharge** as the Category and **Power** as the Quantity.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Paschen curve shown in the plot represents the breakdown power as a function of the gas pressure. In this example, circuit designers mainly care about the minimum breakdown power and the critical pressure where the coaxial bend is most prone to microwave discharge during satellite launch or spacecraft re-entry.

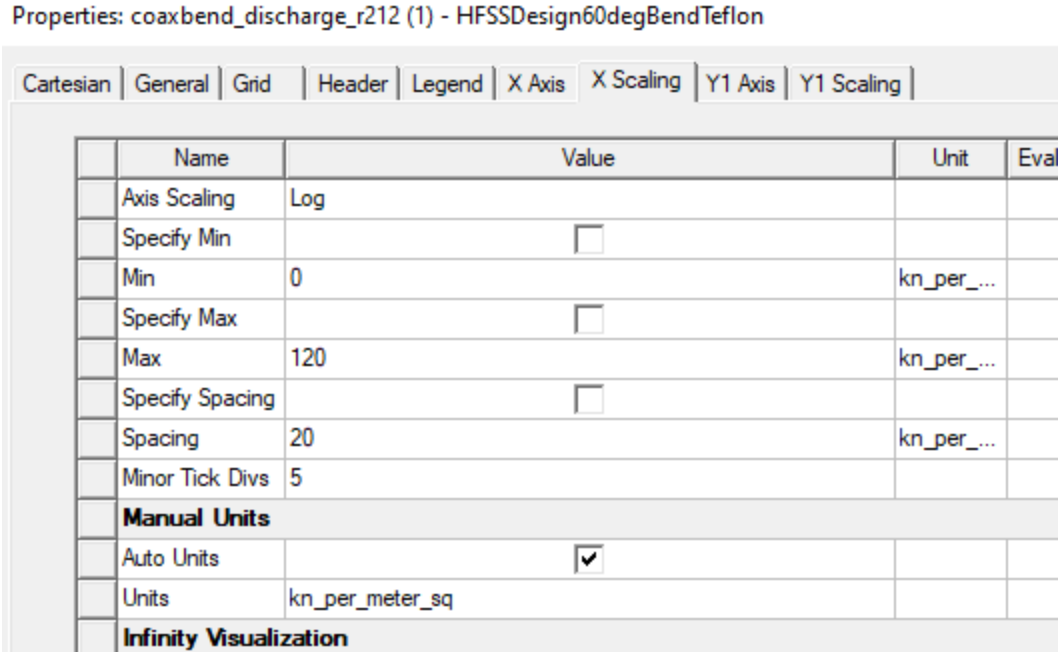
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For an eigenmode solution, the breakdown stored energy is equal to the breakdown power divided by the eigen frequency.

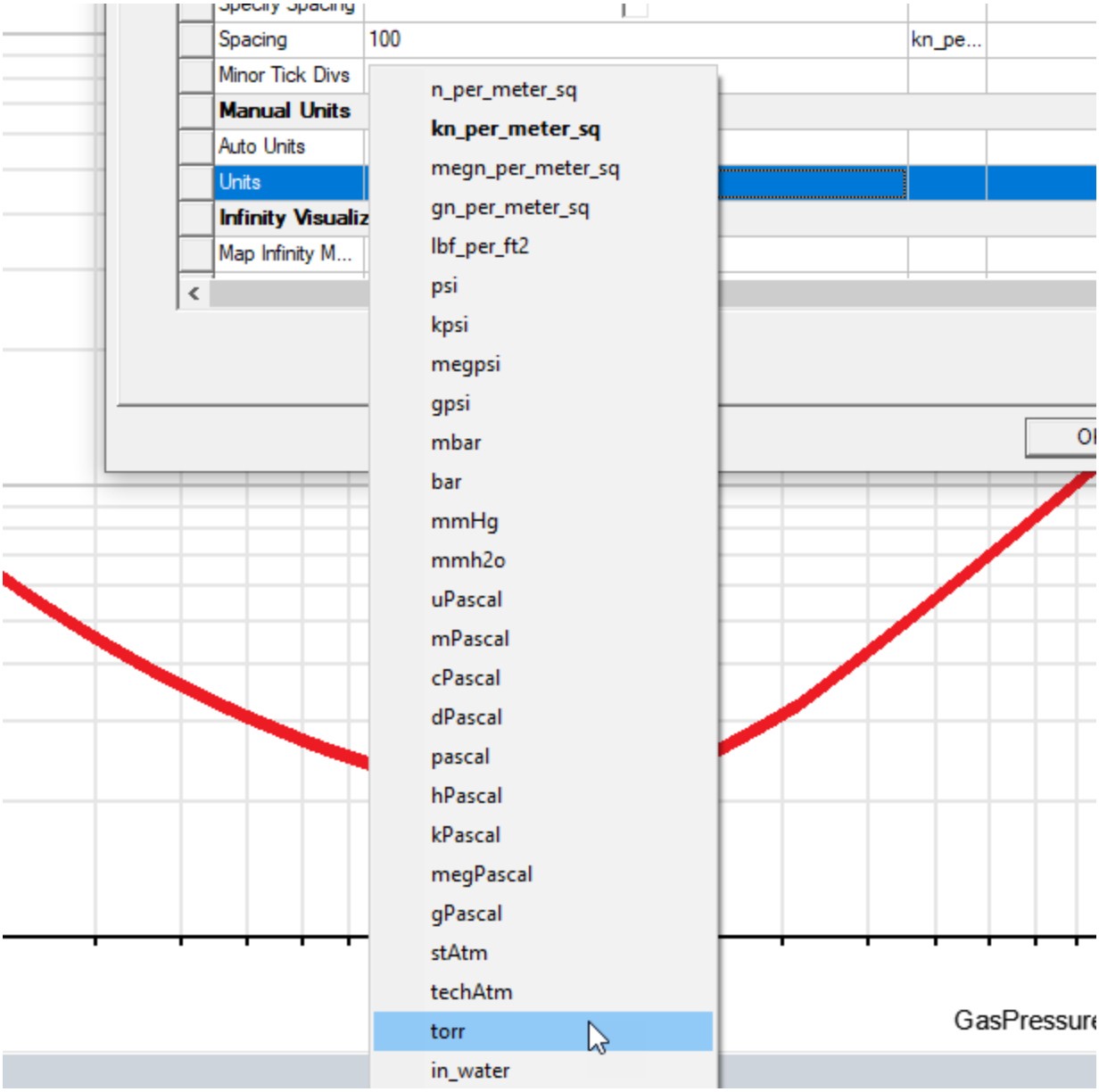
(Optional) Change the Pressure Unit

To change the pressure unit, double click Curve Info to open the Properties dialogue and select the X Scaling tab.



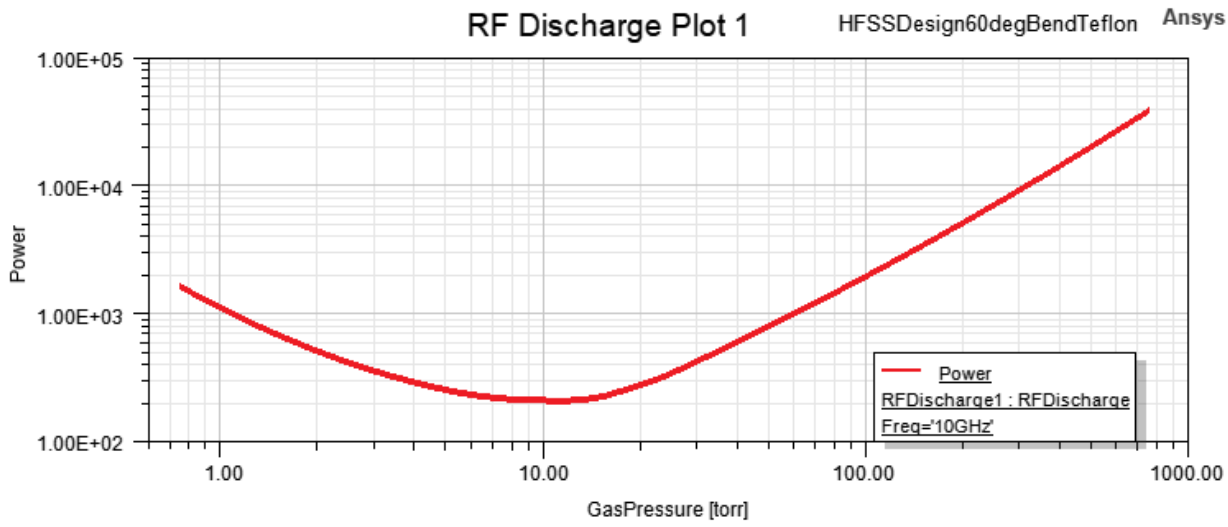
Uncheck Auto Units and on the Units line, use the drop-down menu to select torr.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The plot will become the following by adjusting the bounds of X and Y values to fit the curve in the window.

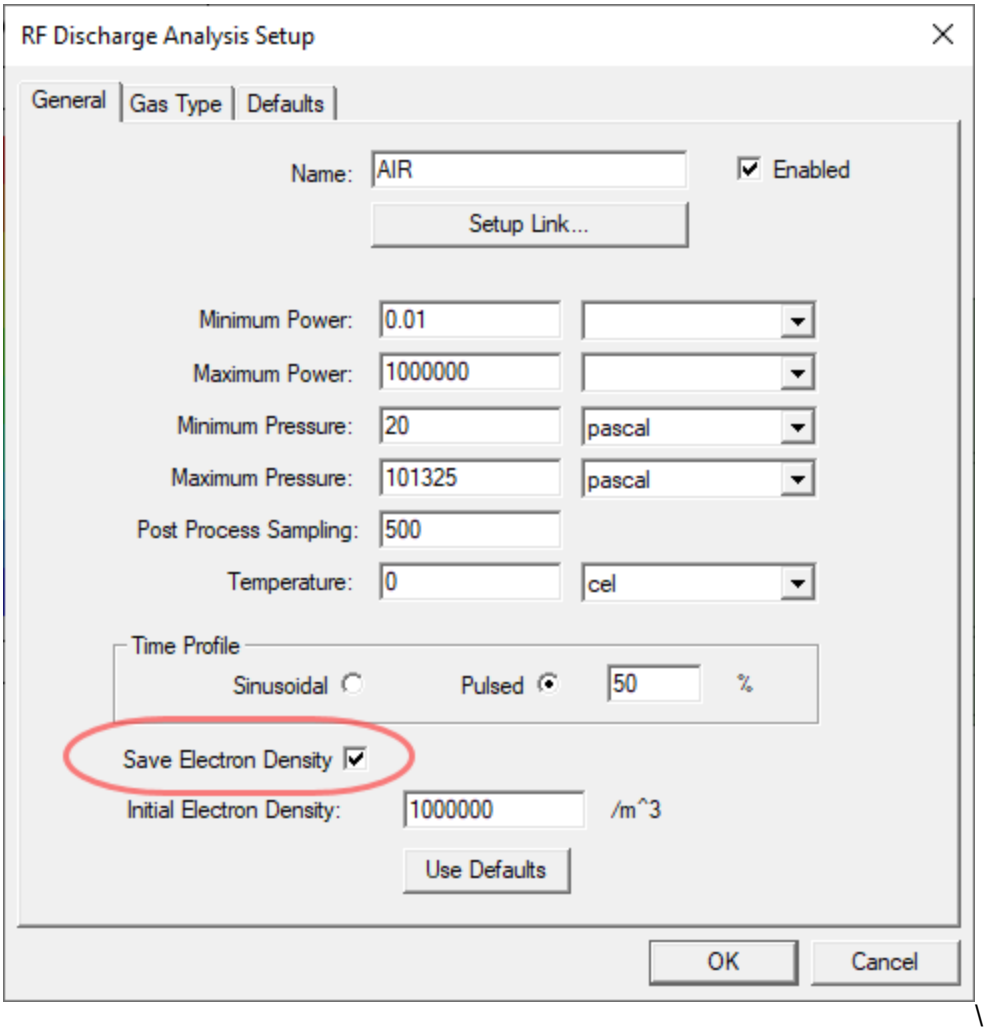
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



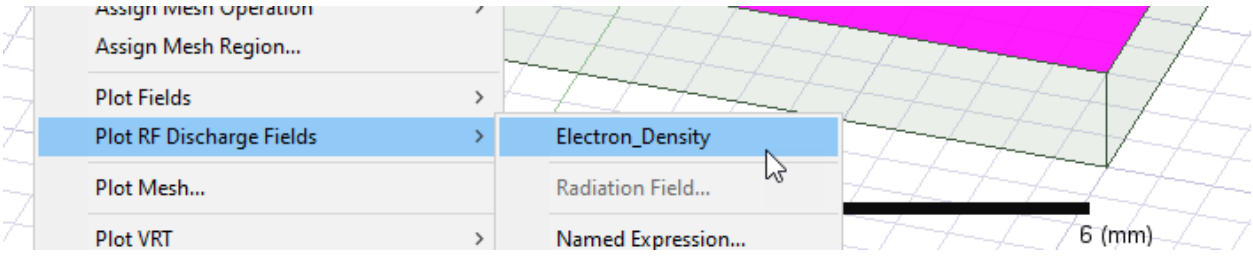
Create an Electron Density Plot or the RF Simulation

Enable the data for an electron density plot in an **RF Discharge Analysis Setup** checking **Save electron density**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

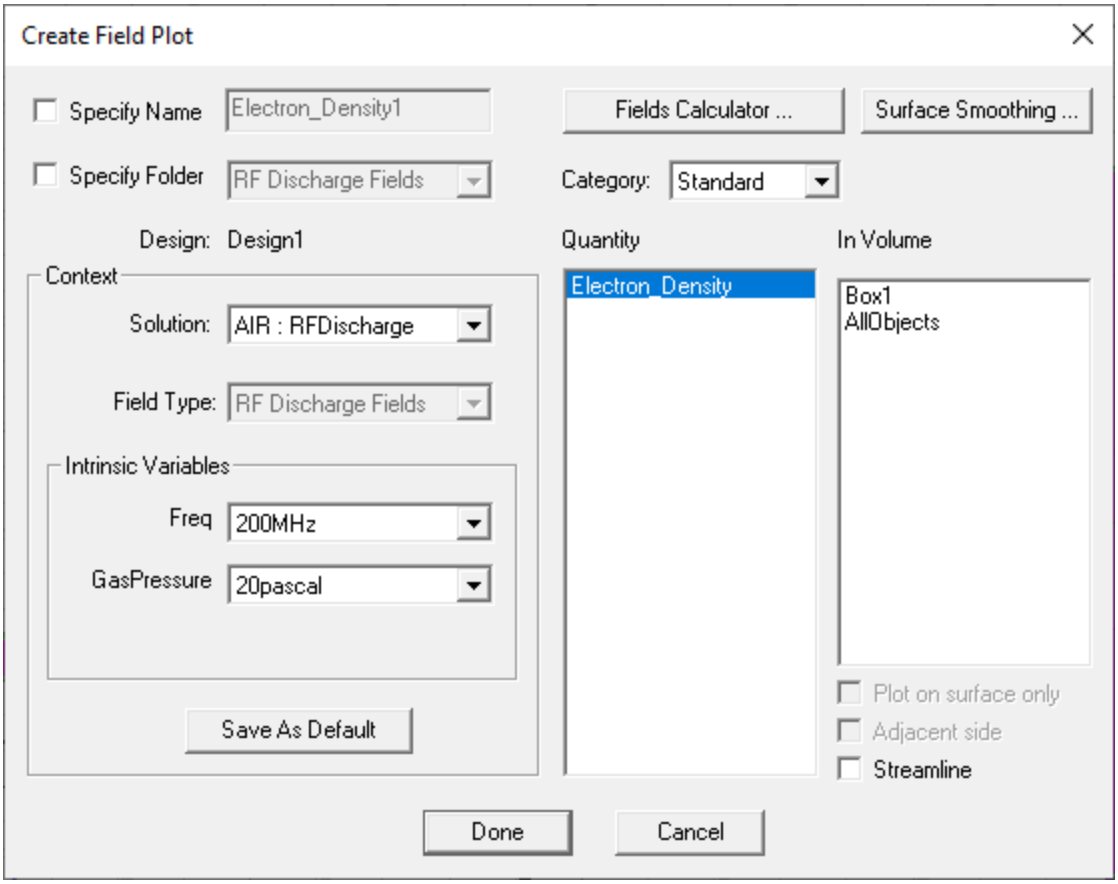


After you solve with this checked, select a face of the object for which you want to plot fields. Then right-click in the **Modeler** window and select the enabled **Plot RF Discharge Fields>Electron Density** menu.



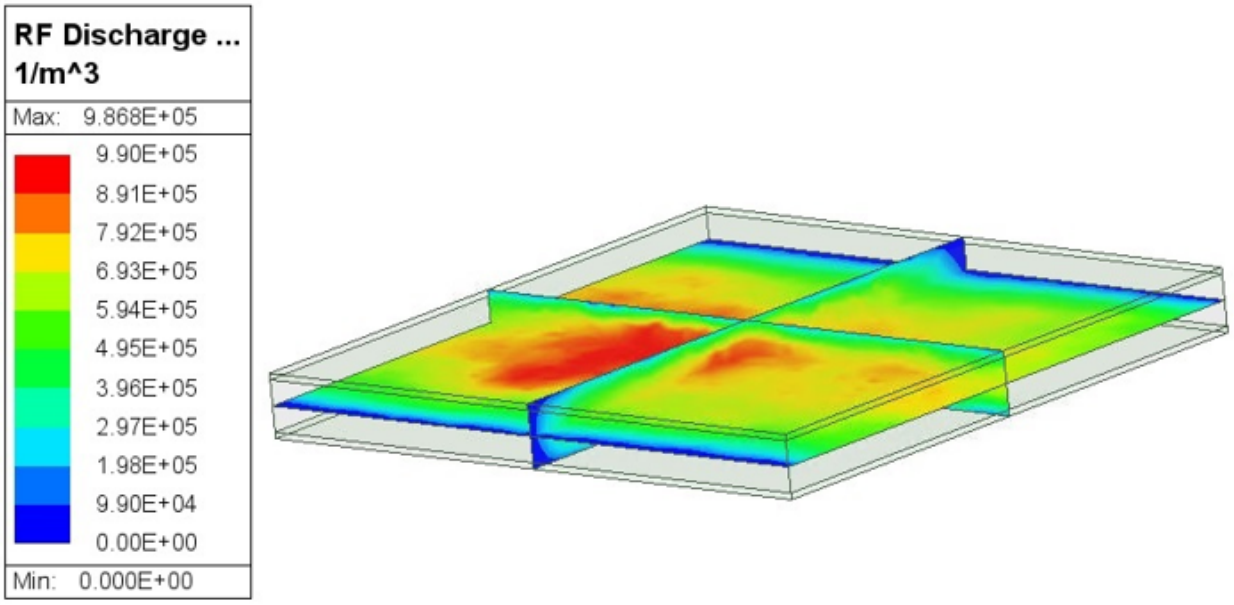
Specify parameters in the **Create Field Plot** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

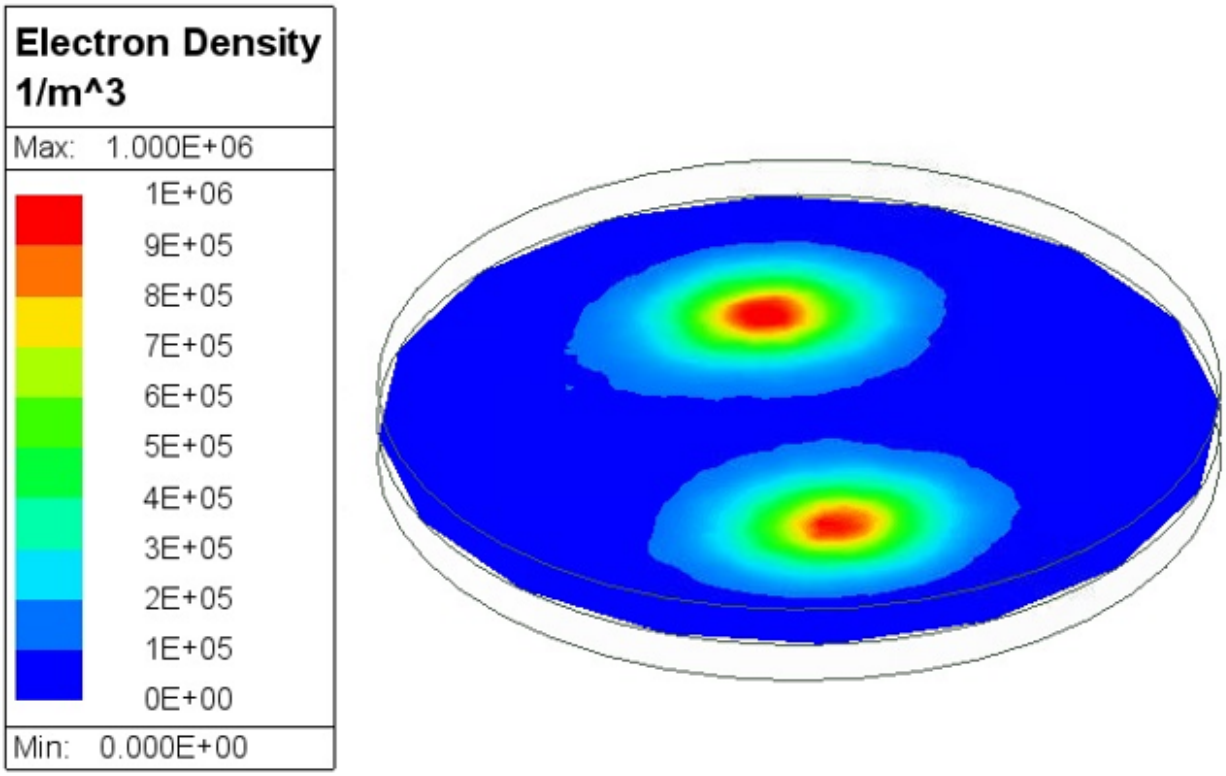


Make Variable and In Volume Selections and click **Done**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also create an electron density plot for an Eigenmode simulation.



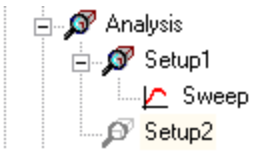
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For getting smoother electron density plots, it is often required to use mesh seeding to increase uniformity and density of the mesh. With an increasing mesh density, we can select low order basis functions in Mesh/Solution Options to save simulation time.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

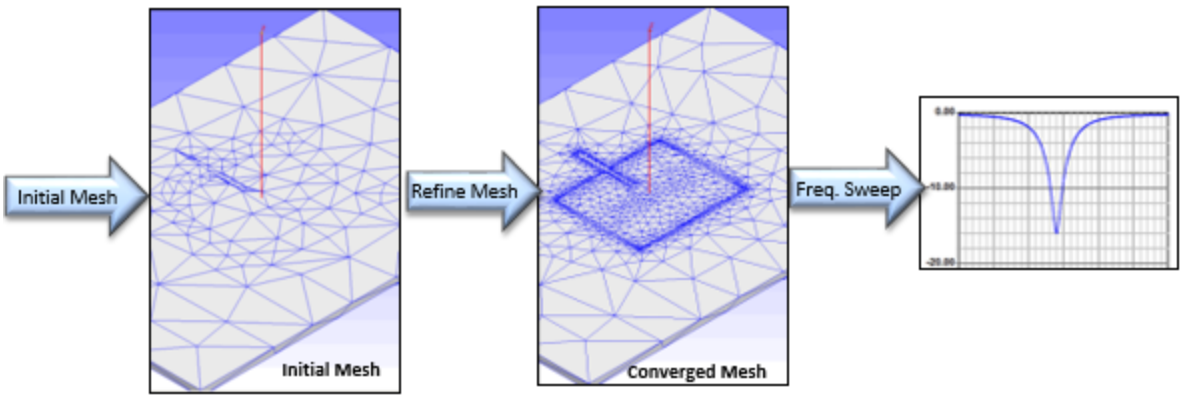
16 - Specifying Solution Settings

Specify how Ansys Electronics Desktop will compute a solution by adding a solution setup to the design. You can define more than one solution setup per design.



Each solution setup includes the following information:

- General data about the solution's generation.
- Adaptive mesh refinement parameters, if you want the mesh to be refined iteratively in areas of highest error.
- Whether to **Save Fields** (the default) and if so, whether to save **Save radiated fields only**. You must save fields for fields post processing, plotting and animating plots.
- If you want to solve of a range of frequencies, you can add a frequency sweep.

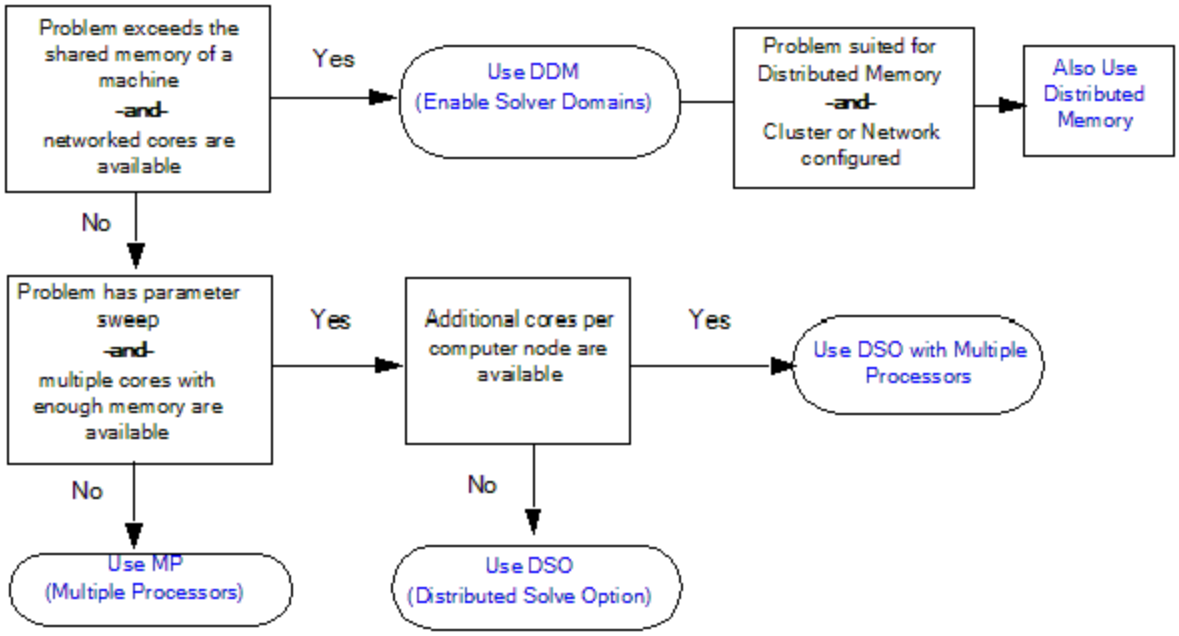


What do you want to do?

- Add a Solution Setup to an HFSS Design
- Add a Solution Setup to an HFSS Design with Hybrid and Arrays
- Add a Solution Setup to an HFSS Transient Design
- Add a Solution Setup for an SBR+ Design
- Add a Solution Setup for an Eigenmode Design
- Add a Characteristic Mode Analysis Setup and Generate Reports

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For large problems, depending the whether the solution involves a parameter sweep and your resources of shared machine memory, networked cores and cores per computing node, you can configure for different high performance computing solutions:



Additionally, a command line Large Scale DSO is used for 'large scale parallel' jobs, which either fail or scale poorly as Regular DSO jobs. A Large Scale DSO job does not support the output of full parametric results, but produces 'reduced' datasets corresponding to predefined Rectangular plots.

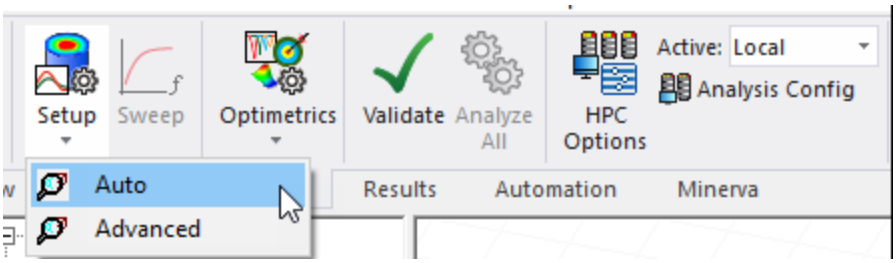
Adding a Solution Setup to an HFSS Design

If you have an existing setup, you can **Copy** and **Paste** it, and then edit parameters. If you have already created a solution and you want to use an existing mesh, you can click **Add Dependent Solve Setup**. To add a new solution setup to an HFSS Solution Type design:

1. Select a design in the project tree.
2. Click **HFSS**>**Analysis Setup**>**Add Solution Setup**, or right-click **Analysis** in the project tree, and then click **Add Solution Setup** on the shortcut menu, or select the **Simulation**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tab on the ribbon, and click the **Setup** icon:

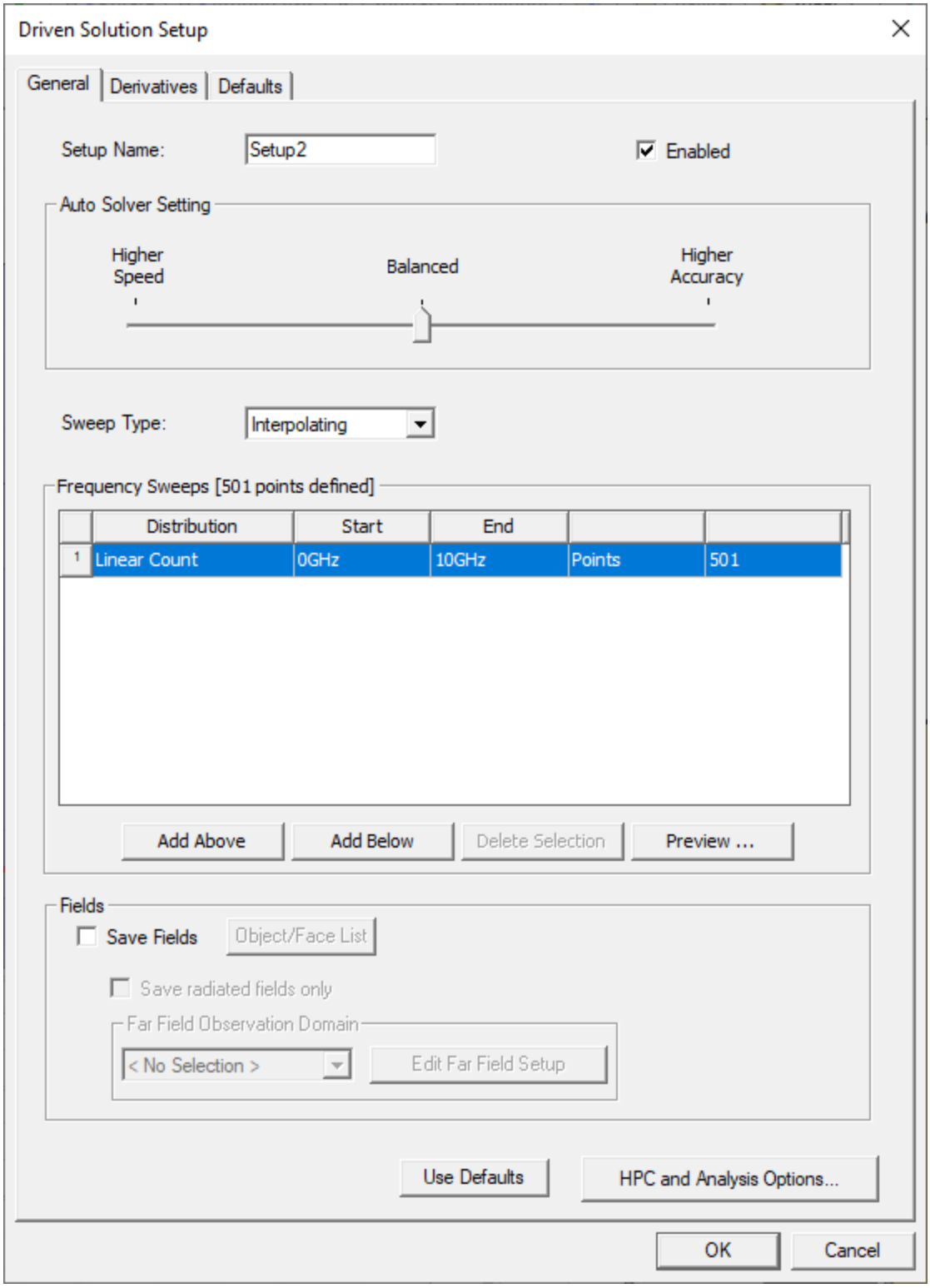


For HFSS designs you can choose **Auto** or **"Advanced Solution Setup" on page 16-7**.

Auto Solution Setup

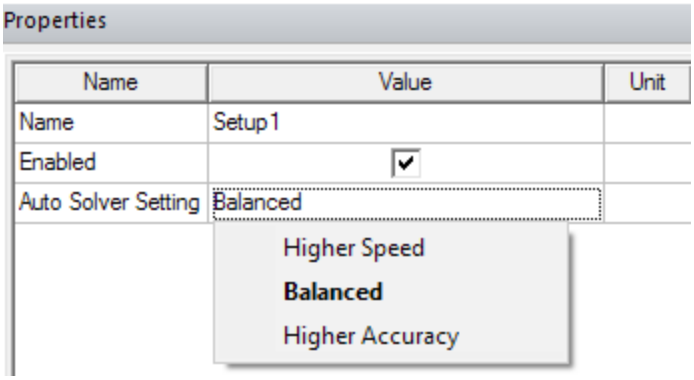
In order to use Auto setup, "Use Automatic Settings" under Analysis Configuration must be checked. **Auto** features a simplified dialog with a slider bar that you can adjust from Higher Speed (which sacrifices accuracy to achieve optimum speed), to Balanced (which is more accurate but faster than the highest accuracy setting) to Higher Accuracy (which takes the time to ensure optimal accuracy).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



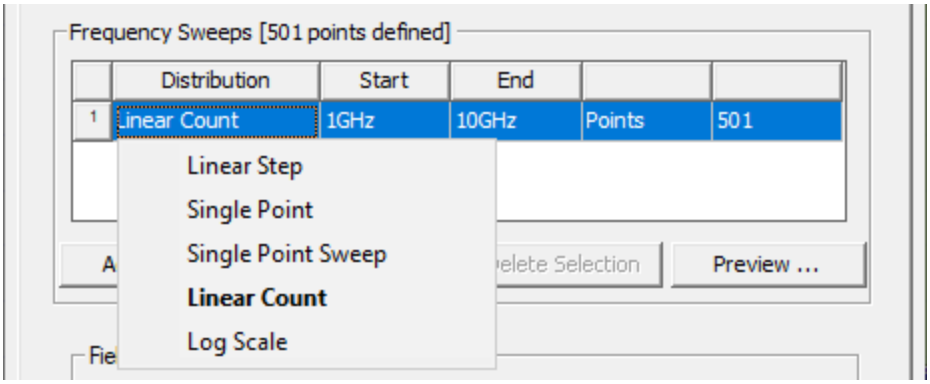
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Properties for the Auto Solver Setting includes the same choices.



The Auto setup includes a single sweep, for which you can edit the Distribution, Start, and End. If the design does not include a port, the sweep type can only be Discrete. If one or port ports exist, the sweep can be Interpolating or Discrete.

- You specify sweeps in terms of Distribution type, which can be Linear Step, Linear Count, Log Scale, Single Point , or Single Point Sweep, which adds a set of 10 Single Point Sweeps, defaulting from 1 GHz to 10 GHz in increments.. The **Add Above**, and **Add Below** buttons permit you to add additional sweeps, including mixed sweep types. This feature provides flexibility. For example, you can define sweeps with log scale at lower frequencies, and linear step at higher frequencies.



If you add more sweeps, you can **Delete** a selected distribution.

The **Preview** button displays the sweep(s) as currently defined.

The **Auto** setup includes tabs for **General**, **Derivatives**, and **Defaults**.

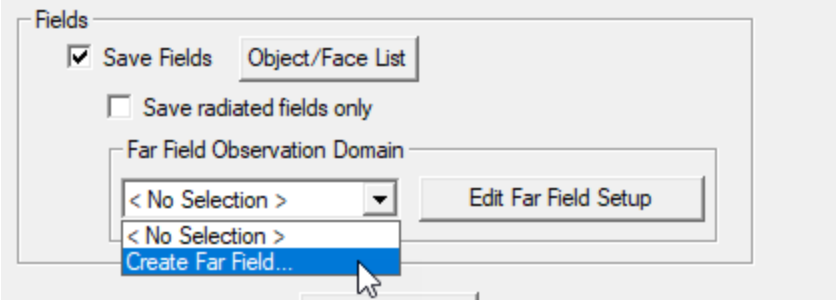
General Includes general solution settings. You can specify the name, change the slider bar to do different solver settings, choose sweep type, set frequency range, choose **Save Fields** and **Save radiated fields only** options, (required if you intend to Plot Field

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Overlays) and do **HPC** settings.

Includes settings for mesh linking, mesh assembly for designs that include 3D components, absorbing boundaries on ports, wave port adapt options, and whether to **Save Fields** and **Save radiated fields only**. You must save fields if you intend to plot or post process fields.

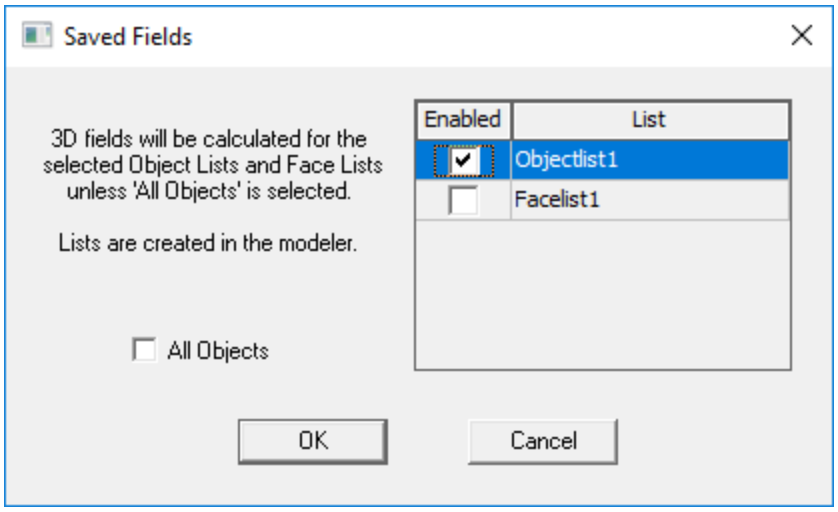
For HFSS solutions the Fields area contains a Far Field Observation Domain area that allows you to specify a Far Field sphere or edit an existing setup. To evaluate radiated fields in the far-field region, you must check Saved Fields in the solution and sweep setups, and set up an infinite sphere that surrounds the radiating object. You can set up the infinite sphere by selecting **Create Far Field...** from the menu. For the procedure, see Setting up a Far Field Infinite Sphere.



If you do so, fields are generated during the solve and for any dependent frequency sweeps, and written as .ffd files rather than as a post processing step. After the simulation, editing or deleting the selected far field setup invalidates the last adaptive solution of the driven setup as well as its dependent sweep's solution.

For very large designs, saving fields can potentially result in very large files. In such cases, you can use the **Object/Face List** button to open a **Saved Fields** dialog to select from object or face lists you have predefined. You would then save 3D fields only on certain objects or faces, which could save disk space. The default is All Objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The selection in the solve setup will dictate the behavior of the frequency sweep solve. Changing the selected lists will invalidate the last adaptive solution. If you subsequently create a Near Field or Far Field Radiation setup and select a list there, you receive a warning if your selections do not match those selected for the solve.

The selection in the solve setup will dictate the behavior of the frequency sweep solve. Changing the selected lists will invalidate the last adaptive solution. If you subsequently create a Near Field or Far Field Radiation setup and select a list there, you receive a warning if your selections do not match those selected for the solve.

Derivatives If your design contains variables, they are listed here. HFSS can calculate derivatives for your variables.

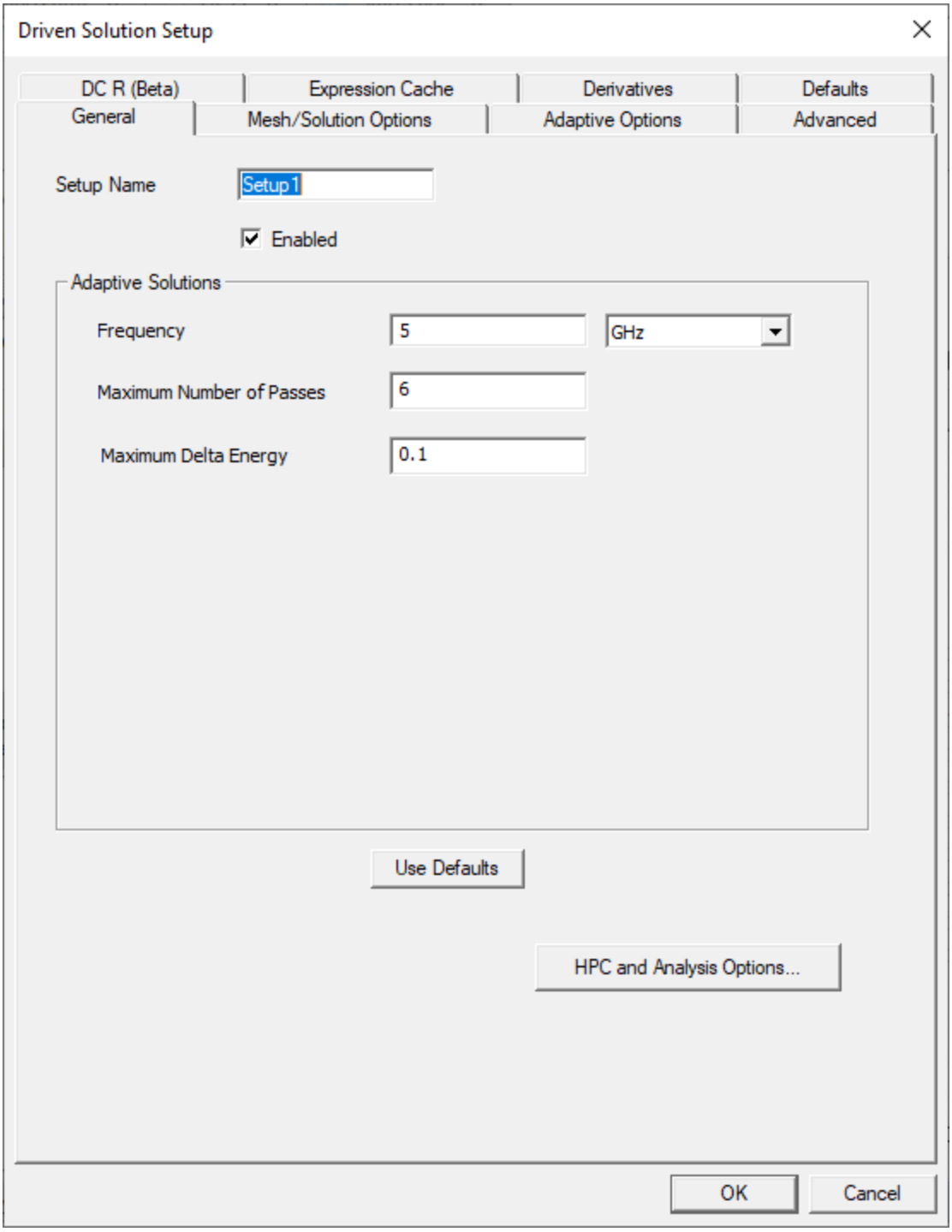
Defaults Enables you to save the current settings as the defaults for future solution setups or revert the current settings to HFSS's standard settings.

Advanced Solution Setup

The Advanced Solution Setup lets you have more control over the settings, and provides more options. For all HFSS solution types, if you choose **Advanced**, the **Solution Setup** dialog box appears. The appearance differs depending on the Solution Type. For example for

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Driven Solution Terminal Types, the DC R (Beta) tab also appears.



It is divided among the following tabs:

General Includes general solution settings, including frequency and S-parameter

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

settings.

Mesh/Solution Options

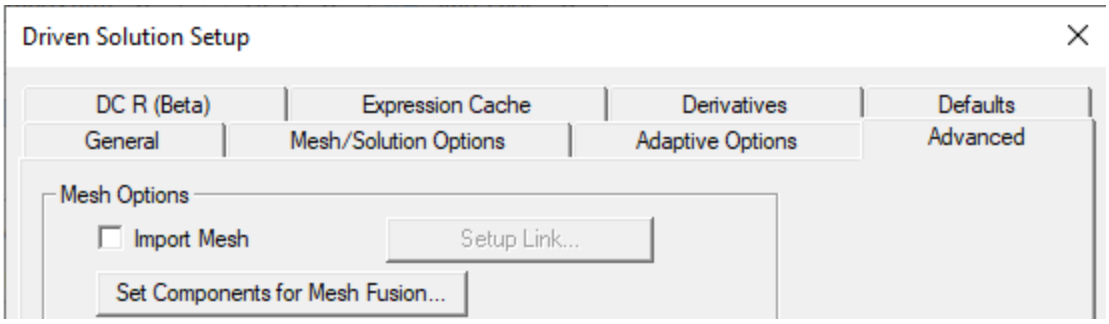
Includes Initial Mesh option settings for lambda refinement.

The **Solution** options include the Order of Basis setting, whether to use the direct solver or iterative solver, whether to enable the use of solver domains (if available for the solution type, and for designs containing lumped ports and/or circuit ports as sources), and whether to use **Enhanced low frequency accuracy**.

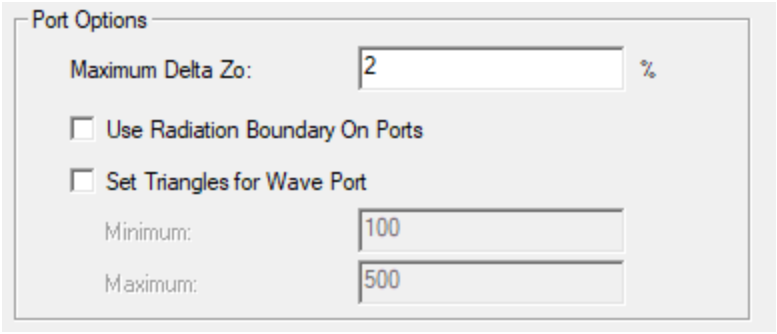
When **Enhanced low frequency accuracy** is enabled, the solver is tuned to reliably solve lumped ports at low frequencies. In addition, for designs with only lumped ports and/or circuit ports, interpolating sweeps are tuned to solve more low frequency points in order to accurately represent very low frequency results.

Advanced

Includes settings for mesh linking, and for designs that include 3D components, Mesh Fusion.



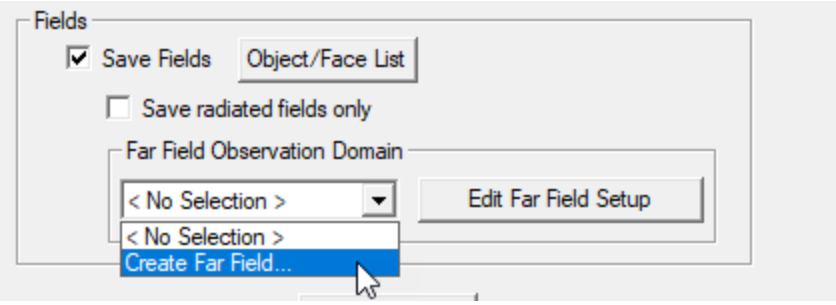
For assigned Ports includes areas that are active if you have Use Radiation Boundaries on Ports, and wave port adapt options.



The Fields pane lets you specify whether to **Save Fields** and **Save radiated fields only**. You must save fields if you intend to plot or post process fields.

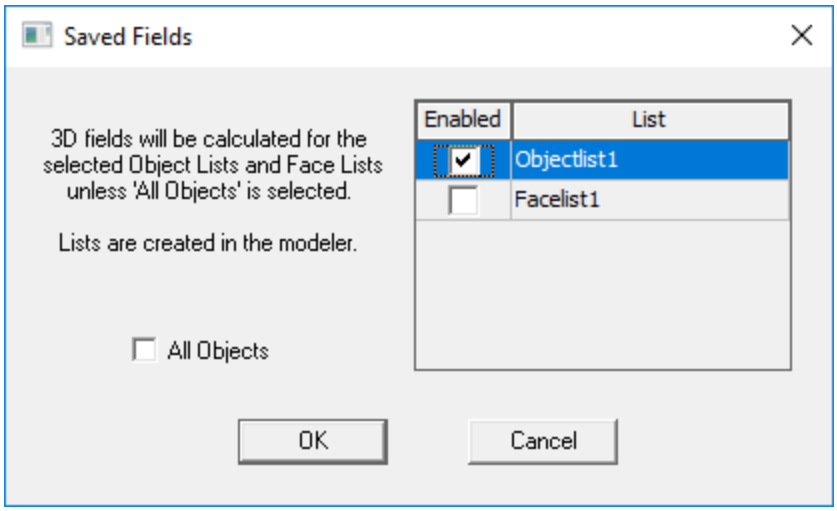
For HFSS solutions the Fields area contains a Fields Observation Domain area that allows you to specify a Far Field sphere or edit an existing setup. If you do so, fields are generated during the solve,

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you do so, fields are generated during the solve and for any dependent frequency sweeps, and written as .ffd files rather than as a post processing step. After the simulation, editing or deleting the selected far field setup invalidates the last adaptive solution of the driven setup as well as its dependent sweep's solution.

For very large designs, saving fields can potentially result in very large files. In such cases, you can use the **Object/Face List** button to open a **Saved Fields** dialog to select from object or face lists you have predefined. You would then save 3D fields only on certain objects or faces, which could save disk space. The default is All Objects.

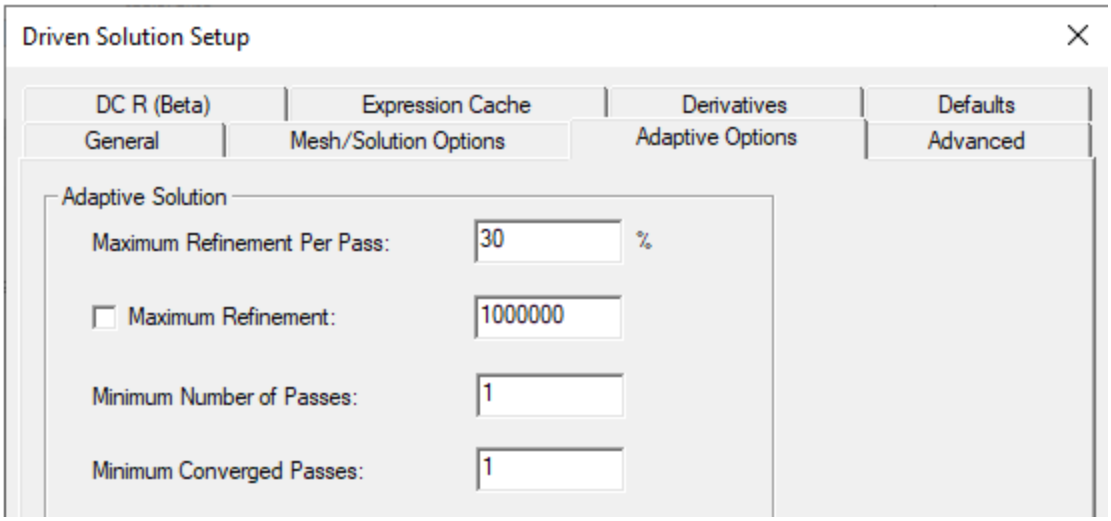


The selection in the solve setup will dictate the behavior of the frequency sweep solve. Changing the selected lists will invalidate the last adaptive solution. If you subsequently create a Near Field or Far Field Radiation setup and select a list there, you receive a warning if your selections do not match those selected for the solve.

Adaptive Options

The Adaptive Options include Maximum Refinement per pass, Maximum Refinement, and Minimum Number of Passes, and Minimum Converged Passes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



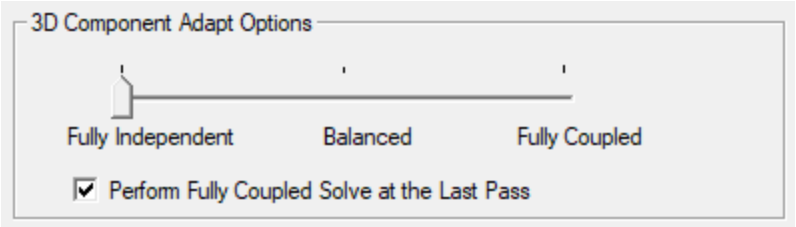
Maximum Refinement Per Pass

Maximum Refinement

Minimum Number of Passes

Minimum Number of Converged Passes

If you have set the Beta Option for Parallel Component Mesh Adapt, and the design contains 3D components, the **Adaptive Options** tab also lets you select 3D Component Adapt Options.

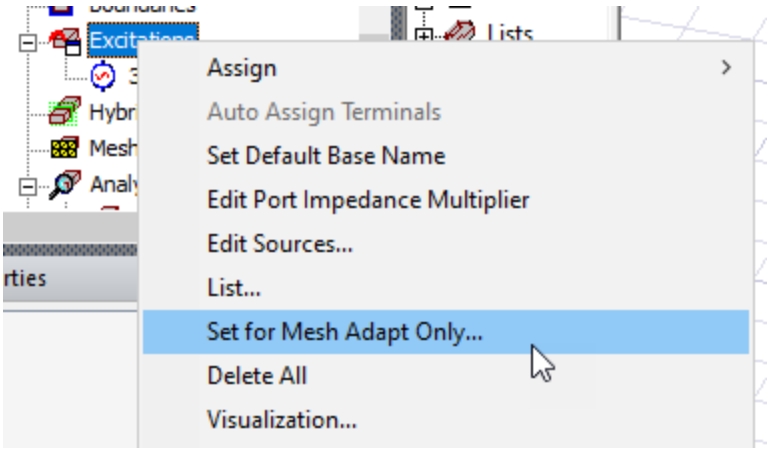


Parallel component adapt for mesh fusion workflow in MCAD including both 3D component array and general mesh fusion. During the component adapt, the multiple components are solved in parallel during each adaptive pass. You can select from Fully Independent adapt (the default for a Mesh Fusion project), Balanced adapt, and Fully coupled adapt. If you select Fully Independent or Balanced, you can specify that the last adaptive solve performs a Fully Coupled Solve, or if unchecked, a memory estimation.

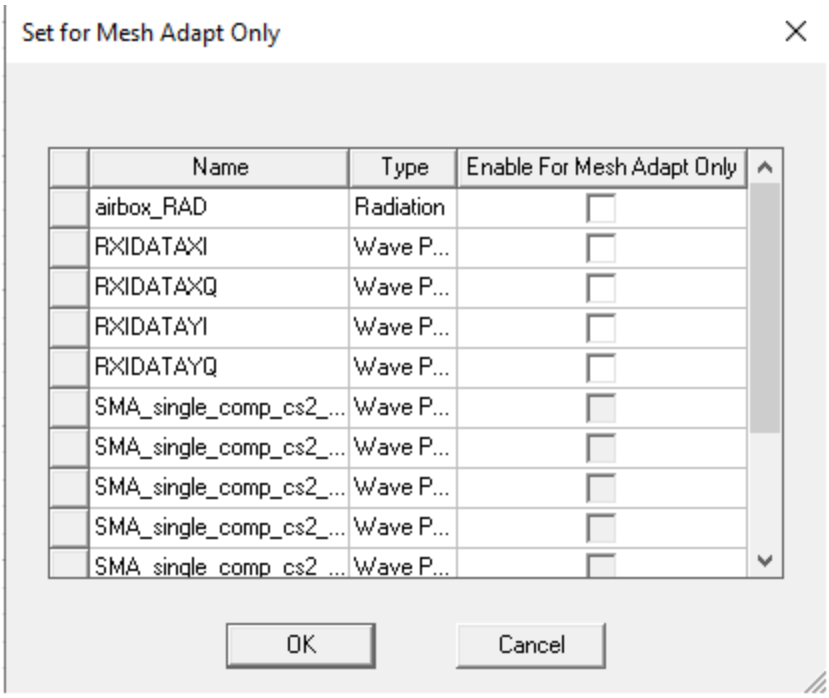
You can assign certain boundaries and ports to improve adaptive accuracy.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, for component without any ports, certain ports together with a PEC boundary can be added to drive the adaptive pass. These boundaries and/or ports are present during adaptive pass only and will be removed automatically during full design solve. To do so, right-click on the Excitations icon and select **Set for Mesh Adapt Only...**



This opens a dialog listing the ports/boundaries.

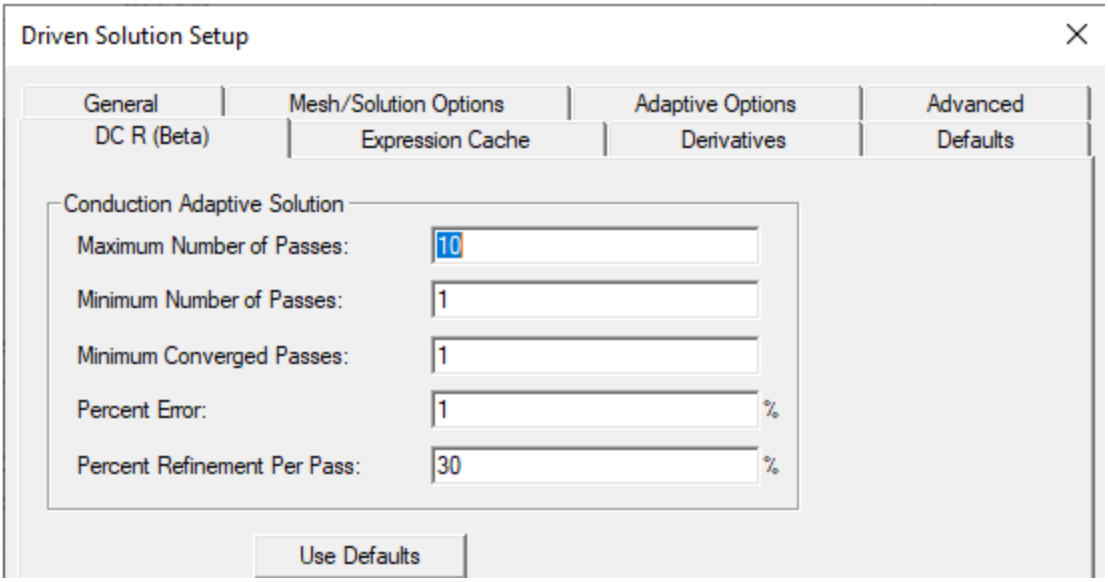


When an adapted port is present, you have an option to display s-parameter results that do or do not include adapted ports (full s-matrix or reduce s-matrix)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for adaptive solves. Ports selected here do not appear in the Edit Sources dialog. Because there is no LastAdaptive solution if you turn off “Perform Fully Coupled Solve at the Last Pass”, the far field plot will be blank.

DC R The ability to Use Q3D for DC Simulation in HFSS causes an additional **DC R (Beta)** tab to appear in the **Driven Solution Setup** dialog for HFSS Network Analysis Terminal solution types.



The fields allow you to adjust the values for DC point calculation in HFSS. Also see Options for Interpolating Sweeps, and Options for Discrete Sweeps where you can select and option to **Use Q3D to Solve the DC Point**.

Expression Cache Includes a list of expressions (including post processing variables) that you can use for convergence for adaptive analysis.

Derivatives If your design contains variables, they are listed here. HFSS can calculate derivatives for your variables.

Defaults Enables you to save the current settings as the defaults for future solution setups or revert the current settings to HFSS's standard settings.

3. Click the **General** tab.
4. The **Enabled** check box on **General** tab permits to you to disable a setup so that it does not run when you select Analyze All.
5. For Driven solution types, do the following:
   a. Select whether you want to solve for Single frequency, Multi-Frequency, or Broadband, and enter the appropriate **Solution Frequency** data and select the frequency units from the pull down list.
   b. Optionally, select **Solve Ports Only**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. If you are performing an adaptive analysis, enter **2** or more passes in the **Maximum Number of Passes** box, and then specify the remaining adaptive analysis parameters for HFSS.
7. The lower right corner also contains a button for HPC and Analysis options. Here you can select or create an analysis configuration.
8. Click **OK**.
9. Optionally, add a frequency sweep to the solution setup.

Add Dependent Solve Setup

For driven setups (not Eigenmode), to apply all settings from an existing setup to a child setup:

1. Select an existing setup in the project tree.
2. Right-click the setup in the project tree, and then click **Add Dependent Solve Setup** on the shortcut menu.

A dependent setup icon appears, which has an altered graphic to distinguish it from the parent setup icon. The child setup name is "*parent_setup name*_1." All of the settings from the parent setup are copied to the child setup. The dependent setup uses the mesh from the parent setup. This is shown under the **Advanced** tab of the **Solution Setup** dialog box, **Specifying a Source** for the initial mesh. You can add a dependent setup to another dependent setup, and form of the name shows the hierarchical dependence by appending "_1" to show further dependence.

If you intend to change any of the settings, you do this just as you would for a new setup.

The **Enabled** check box on General tab permits to you to disable a setup so that it does not run when you select Analyze All. If a solve is disabled, the **Analyze** command on the shortcut menu for the Project tree is disabled and the **Enable Setup** command is available.

Renaming a Solution Setup

Do the following to rename a solution setup:

1. In the project tree, under **Analysis**, right-click the setup you want to rename.
   A shortcut menu appears.
2. Select **Rename** from the shortcut menu.
   The setup name text is highlighted in the project tree.
3. Type the new name for the setup, and press **Enter**.

You can also rename the solution setup by changing the text in the Name text box of the appropriate **Solve Setup** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copying a Solution Setup

Solution setups may be copied and pasted within a design or across designs of the same type. This is beneficial for setups having a large number of parameters to specify, or where minor changes to a setup are being evaluated.

Do the following to copy a solution setup:

1. In the project tree, under **Analysis**, right-click the setup you want to copy.

   A shortcut menu appears.

2. Select **Copy** from the shortcut menu.

   The setup parameters are copied to the clipboard.

3. In the project tree, right-click the **Analysis folder to receive the copied setup**.

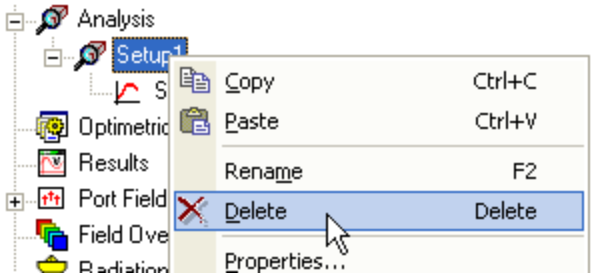
   A shortcut menu appears.

4. Select **Paste** from the shortcut menu.

   The setup parameters are copied to the **Analysis** folder as a new setup.

Deleting a Solution Setup

To delete an existing solution setup, select it in the Project tree, right-click to display the shortcut menu and select **Delete**.

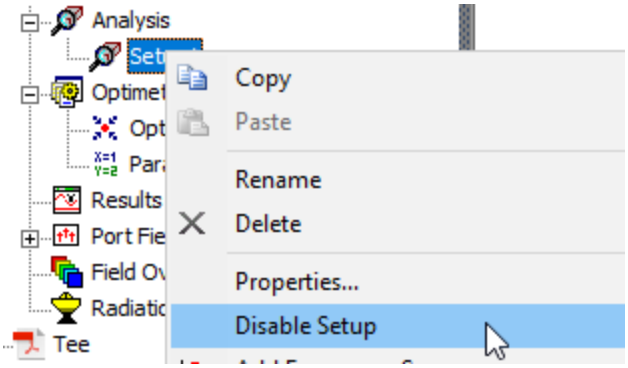


Disabling and Enabling an Analysis Setup

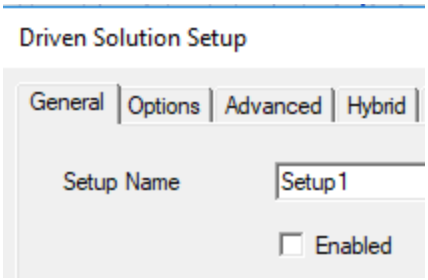
To disable an Analysis setup definition without deleting it:

1. Expand the tree hierarchy under the **Analysis** icon in the Project Manager.
2. Right-click the icon for the setup definition. In the shortcut menu that appears, click **Disable Setup**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also disable an analysis setup by selecting **Properties** from the shortcut menu to open the **Edit Properties** window. In the upper right corner of the **Edit Properties** window, uncheck the **Enabled** box. Then click **OK**.



To reactivate the analysis setup, open the shortcut menu again and select **Enable Setup**.

You can also reactivate the analysis setup by selecting **Properties** from the shortcut menu for the sweep to open the **Edit Properties** window. Check the **Enabled** box in the upper right corner, and click OK to apply the change and close the window.

Setting the Solution Frequency

For Driven solution types.

For every Driven solution setup, specify whether to solve a **Single Frequency**, **Multi-Frequency**, or **Broadband**, and provide the **frequency** information and units at which to generate the solution. Each method changes the fields shown in the dialog to those needed for the selection. If you enable HPC, multiple frequencies will be distributed.

If a single frequency sweep is solved, an adaptive analysis is performed only at the solution frequency. If you want to solve over a range of frequencies, define a frequency sweep or a Mult-Frequency or Broadband solve. You must first assign an excitation for the Mult-Frequencies and Broadband options to appear on the setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In multiple frequencies solve, port adapt is at the highest frequency and lambda refine is based on an algorithm that uses the highest and lowest user frequencies to allow maximum reuse of solved data. The lambda refinement frequency is the "nominal frequency". The mesh at each pass is solved at multiple frequencies and energy error from all frequencies points are used to drive the mesh refinement for the next pass. The adaptive solution is converged when the simulation reaches the maximum number of passes or when all frequency points reach the targeted maximum delta S. The minimum converged passes and minimum number of passed parameters apply as the total requested passes.

Single Frequency

If you select Single for solution frequency, you specify a value and units with a maximum delta S (or Matrix Convergence), plus the maximum number of passes.

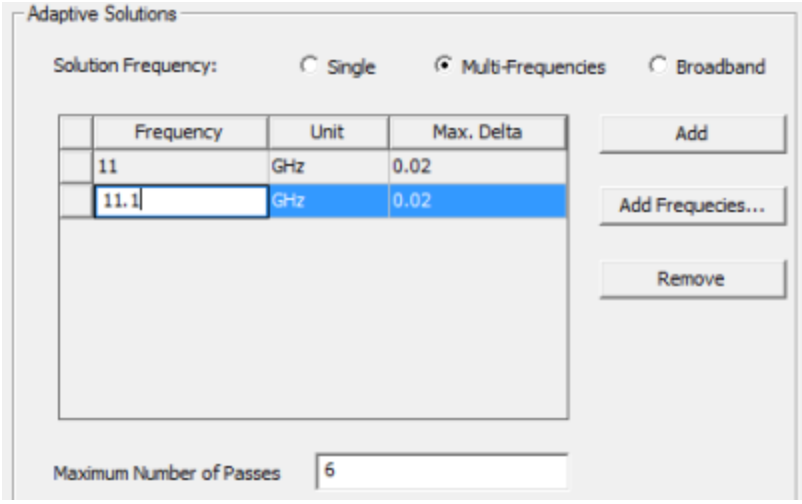
- Enter a value for **Solution Frequency** and select the frequency units from the pull down list.
- Specify the **Maximum Number of Passes** and Maximum Delta Energy.

> **Note:**
>
> For Fast sweeps, HFSS uses the solution frequency as the center frequency if it is within the frequency range (greater than the start frequency and less than the stop frequency.) Otherwise the middle of the frequency range is used as the center frequency.

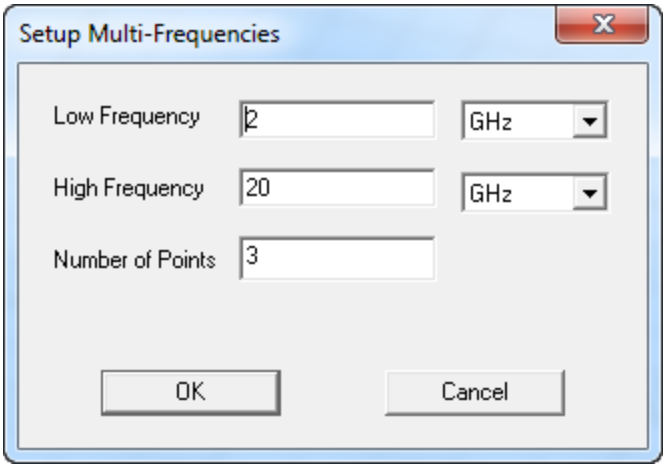
Multi-Frequency

If you select **Multi-Frequencies**, you can click **Add** to add additional frequencies, and edit the value, units, and Max. Delta for each frequency.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can use the **Add** button to specify one frequency at a time or click **Add Frequencies…** to use the **Setup Multi-Frequencies** dialog to populate a range of frequencies. The frequencies added by the Setup Multi-Frequencies matches the first batch of frequencies solved in an interpolating sweep



When you OK the dialog box, rows for the frequencies across the number of points appear in the **Solution Setup** dialog.

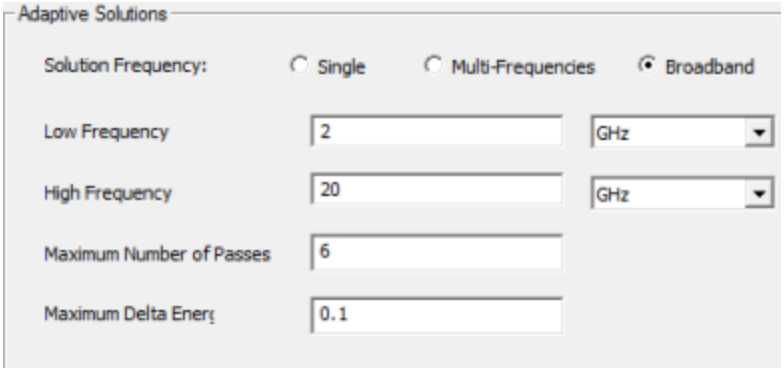
You can use **Remove** to remove any selected items in the Frequency Sweeps table.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

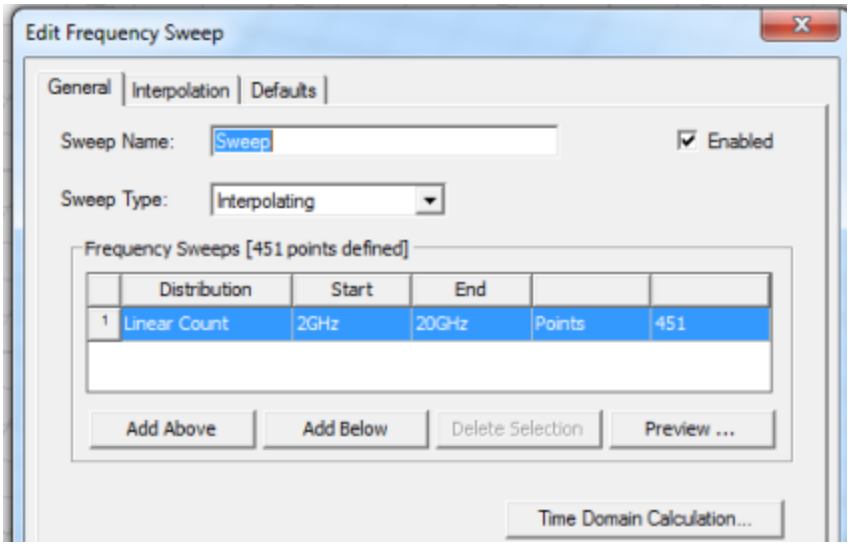
> **Note:**
>
> If you want to create a Frequency animation of field results, you must specify the frequencies here in the Multi-Frequency **Solution** setup.

Broadband

If you select Broadband, you define the frequency band by providing the low and high frequency, with a maximum delta S (that is applied to all solved frequencies), plus the maximum number of passes. The meshing frequencies include the Low and High with the remaining points determined automatically by HFSS. When the user requests DC as the low frequency, the lowest frequency solved by the adaptive meshing process is determined by an algorithm that looks at the range of frequencies you entered.



When you OK the Solution Setup for Broadband, you are automatically presented with a dialog for specifying an Interpolating sweep.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Simulation and Convergence

In multiple frequencies modes, when frequency point p convergences prior to frequency point q, keep adapting at p until q and all other points achieve convergence.

The convergence data in the solution data dialog for single frequency mode is calculated in the same way as before. In multiple frequencies mode, the convergence plotted on the convergence display item corresponds to the global maximum delta S. Convergence data such as “Max. Delta S” appear in the reporter as standard solution quantity so that you can easily plot the maximum delta S for each frequency point vs. pass.

Solution Reporting

For multiple frequencies adapted solutions, the S-Matrix solution display let you to view matrix data across different frequencies of its AdaptivePass and LastAdaptive solutions.

In “Broadband” mode, the nominal frequency is solved and its results are available for all post processing, Optimetrics and expression cache setup. In this mode, results of other solved frequencies will be shown in the S-Matrix solution display but will not be available in reporter, Optimetrics and expression cache setup. In “Muti-Frequencies” mode, all post postings of the AdaptivePass and LastAdaptive will be extended to support results at all user-specified frequencies. Creating a report with solution across multiple frequencies will be the same as creating such report with a sweep solution.

Solving for Ports Only

For Driven solution types with ports.

To quickly compute only the 2D excitation field patterns, impedances, and propagation constants at each port:

- Under the **General** tab of the **Solution Setup** dialog box, select **Solve Ports Only**.

This disables the remaining settings for **Maximum number of passes** and **Convergence per pass**.

HFSS calculates the natural field patterns (or modes) that can exist inside a transmission structure with the same cross-section as the port. These 2D field patterns serve as boundary conditions for the full 3D problem.

Setting the Minimum Frequency

For Eigenmode solution types.

For every Eigenmode solution setup, specify the minimum f**requency** at which to search for eigenmodes. HFSS searches for the user-specified number of modes with a higher resonant frequency than the **Minimum Frequency value**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Under the **General** tab of the **Solution Setup** dialog box, type a **Minimum Frequency** and the frequency units.
- You can set the Minimum frequency as a variable by typing a name in the field and pressing Enter. This displays the **Add Variable** dialog for you to enter the value and units. Click OK to close the dialog box. The variable is listed in the Setup and in the Design Properties.

> **Warning:**
>
> Because the minimum frequency is used to normalize some matrices, if the frequency is set too low, HFSS tries to solve a nearly singular matrix, which may erode the accuracy of the calculations. As a general rule, do not enter a frequency less than 0.01 times the suggested, or default, value for **Minimum Frequency**.

Setting the Number of Modes

For Eigenmode solution types.

For every Eigenmode solution setup, specify the number of eigenmode solutions that the solver finds. If you enter **5**, the solver calculates the first 5 eigenmode solutions above the minimum frequency.

The Eigenmode solver can find up to 20 eigenmode solutions.

- Under the **General** tab of the **Solution Setup** dialog box, enter a value for **Number of Modes**.

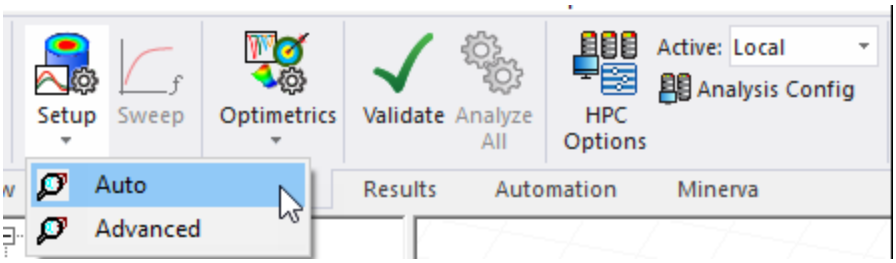
Adding a Solution Setup to an HFSS with Hybrid and Arrays Design

If you have an existing setup an HFSS with Hybrid and Arrays solution type, you can **Copy** and **Paste** it, and then edit parameters. If you have already created a solution and you want to use an existing mesh, you can click **Add Dependent Solve Setup**. To add a new solution setup to a design:

1. Select a design in the project tree.
2. Click **HFSS**>**Analysis Setup**>**Add Solution Setup**, or right-click **Analysis** in the project tree, and then click **Add Solution Setup** on the shortcut menu, or select the **Simulation**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tab on the ribbon, and click the **Setup** icon:

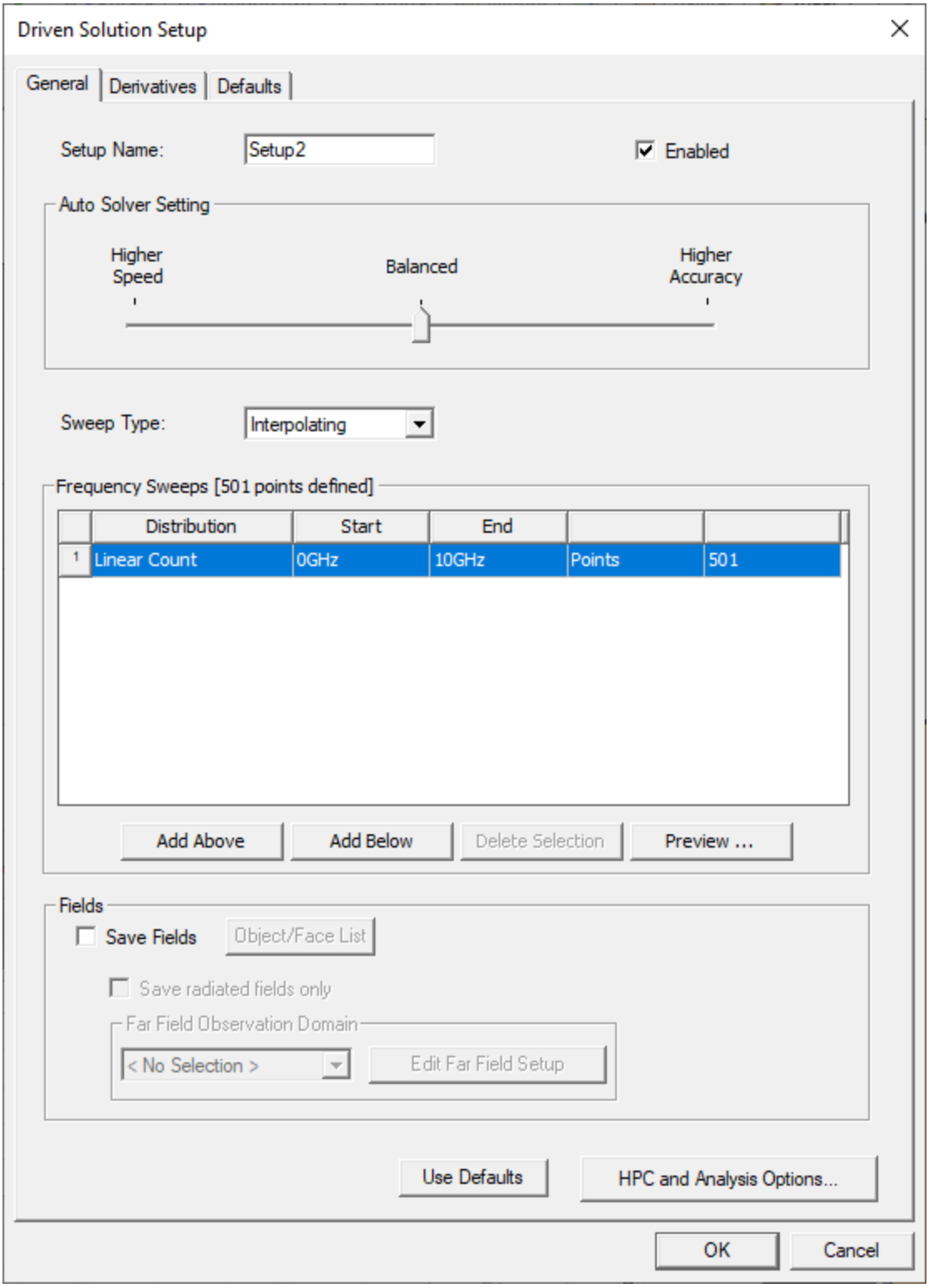


For HFSS with Hybrid and Arrays solutions, you can choose **Auto** or **"Advanced Solution Setup" on page 16-27**.

Auto Solution Setup

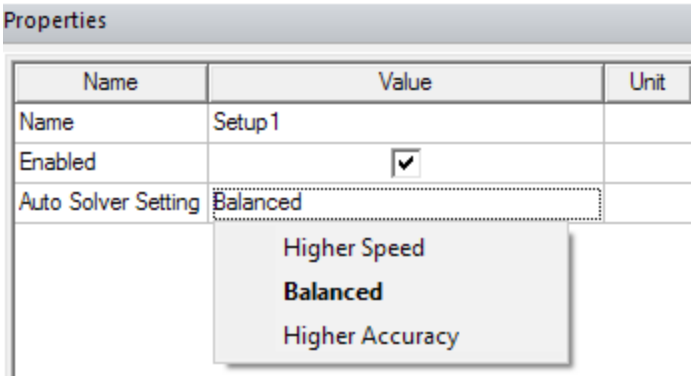
In order to use Auto setup, "Use Automatic Settings" under Analysis Configuration must be checked. **Auto** features a simplified dialog with a slider bar that you can adjust from Higher Speed (which sacrifices accuracy to achieve optimum speed), to Balanced (which is more accurate but faster than the highest accuracy setting) to Higher Accuracy (which takes the time to ensure optimal accuracy). These settings also apply to any FEBI or IE Regions that use the IE Solver in a design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



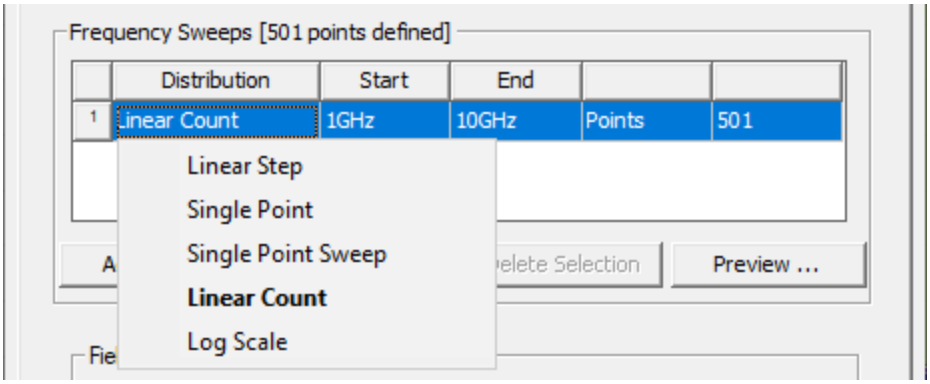
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Properties for the Auto Solver Setting includes the same choices.



The Auto setup includes a single sweep, for which you can edit the Distribution, Start, and End. If the design does not include a port, the sweep type can only be Discrete. If one or port ports exist, the sweep can be Interpolating or Discrete.

- You specify sweeps in terms of Distribution type, which can be Linear Step, Linear Count, Log Scale, Single Point , or Single Point Sweep, which adds a set of 10 Single Point Sweeps, defaulting from 1 GHz to 10 GHz in increments.. The **Add Above**, and **Add Below** buttons permit you to add additional sweeps, including mixed sweep types. This feature provides flexibility. For example, you can define sweeps with log scale at lower frequencies, and linear step at higher frequencies.



If you add more sweeps, you can **Delete** a selected distribution.

The **Preview** button displays the sweep(s) as currently defined.

The **Auto** setup includes tabs for **General**, **Derivatives**, and **Defaults**.

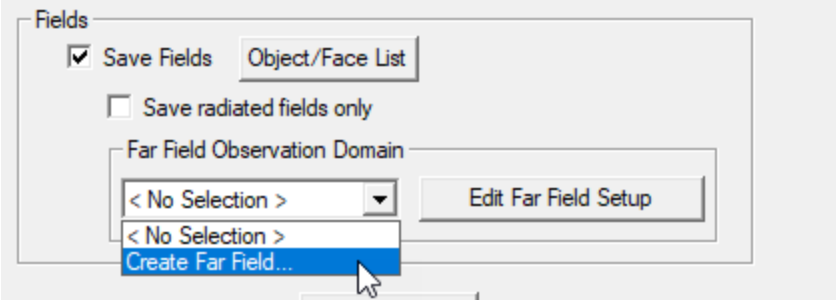
General Includes general solution settings. You can specify the name, change the slider bar to do different solver settings, choose sweep type, set frequency range, choose **Save Fields** and **Save radiated fields only** options, (required if you intend to Plot Field

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Overlays) and do **HPC** settings.

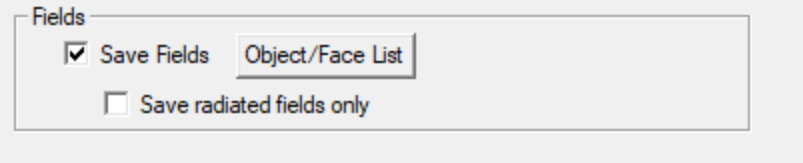
Includes settings for mesh linking, mesh assembly for designs that include 3D components, absorbing boundaries on ports, wave port adapt options, and whether to **Save Fields** and **Save radiated fields only**. You must save fields if you intend to plot or post process fields.

For HFSS solutions without Hybrid Regions, the Fields area contains a Far Field Observation Domain area that allows you to specify a Far Field sphere or edit an existing setup. To evaluate radiated fields in the far-field region, you must check Saved Fields in the solution and sweep setups, and set up an infinite sphere that surrounds the radiating object. You can set up the infinite sphere by selecting **Create Far Field...** from the menu. For the procedure, see Setting up a Far Field Infinite Sphere.



If you do so, fields are generated during the solve and for any dependent frequency sweeps, and written as .ffd files rather than as a post processing step. After the simulation, editing or deleting the selected far field setup invalidates the last adaptive solution of the driven setup as well as its dependent sweep's solution.

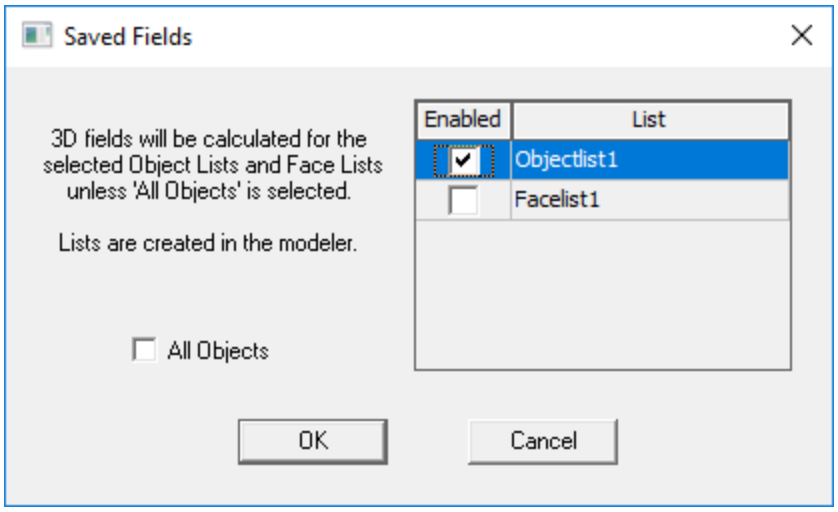
If an HFSS with Hybrid and Arrays project contains an SBR+ Hybrid Region, the Fields area does not contain the Fields Observation Domain area.



For very large designs, especially for finite array designs, saving fields can potentially result in very large files. In such cases, you can use the **Object/Face List** button to open a **Saved Fields** dialog to select from object or face lists you have predefined. You

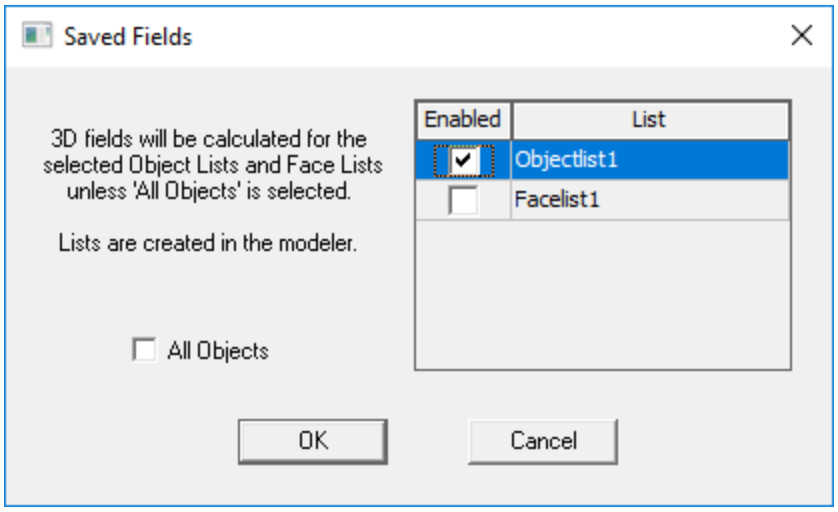
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

would then save 3D fields only on certain objects or faces, which could save disk space. The default is All Objects.



The selection in the solve setup will dictate the behavior of the frequency sweep solve. Changing the selected lists will invalidate the last adaptive solution. If you subsequently create a Near Field or Far Field Radiation setup and select a list there, you receive a warning if your selections do not match those selected for the solve.

For very large designs, especially for finite array designs, this can potentially result in very large files. In such cases, you can use the **Object/Face List** button to select from object or face lists you have predefined. You would then save 3D fields only on certain objects or faces, which could save disk space.



The selection in the solve setup will dictate the behavior of the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

frequency sweep solve. Changing the selected lists will invalidate the last adaptive solution. If you subsequently create a Near Field or Far Field Radiation setup and select a list there, you receive a warning if your selections do not match those selected for the solve.

Derivatives If your design contains variables, they are listed here. HFSS can calculate derivatives for your variables.

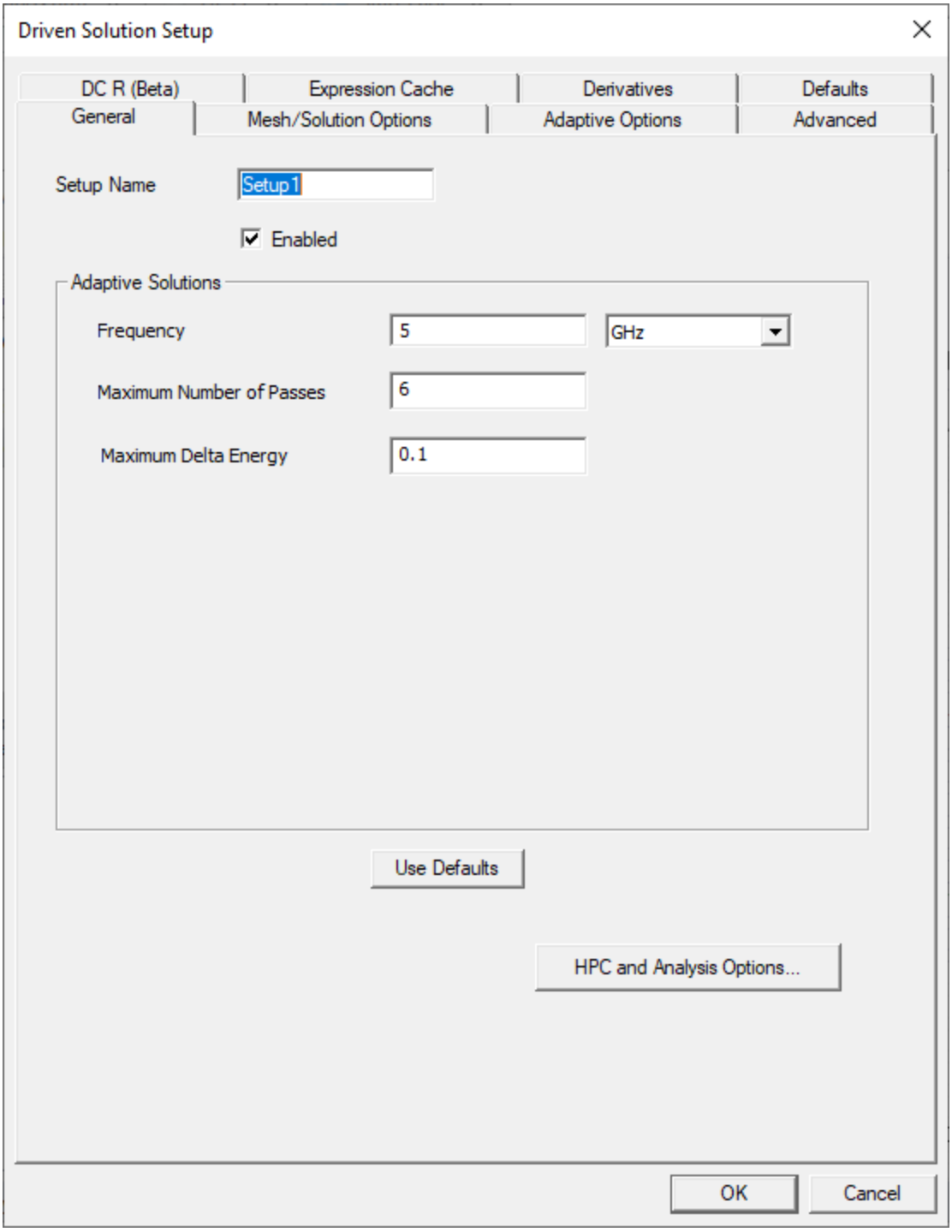
Defaults Enables you to save the current settings as the defaults for future solution setups or revert the current settings to HFSS's standard settings.

Advanced Solution Setup

The Advanced Solution Setup lets you have more control over the settings, and provides more options. For all HFSS solution types, if you choose **Advanced**, the **Solution Setup** dialog box appears. The appearance differs depending on the Solution Type. For example for

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Driven Solution Terminal Types, the DC R (Beta) tab also appears.



It is divided among the following tabs:

General Includes general solution settings, including frequency and S-parameter

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

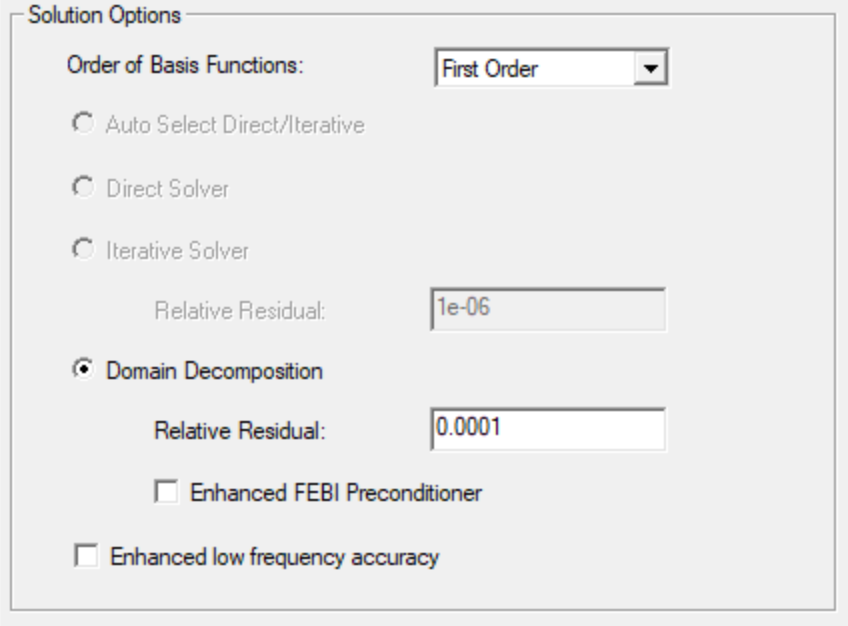
settings.

Mesh/Solution Options

Includes settings for lambda refinement, adaptive analysis and solution options, the Order of Basis setting, whether to use the direct solver or iterative solver, whether to enable the use of solver domains, if available for the solution type, and for designs that contain only lumped ports as sources, or a combination of lumped and circuit ports, whether to use **Enhanced low frequency accuracy**.

In those cases, when you enable **Enhanced low frequency accuracy** the solver is tuned to reliably solve low frequencies. In addition, for designs with only lumped ports and/or circuit ports, interpolating sweeps are tuned to solve more low frequency points in order to accurately represent very low frequency results.

In the case of a design that includes a 3D Component Array and a FE-BI Hybrid Region, the **Mesh/Solution** options includes an option to select **Enhanced FEBI Preconditioner**.



When an HFSS 3D Component Array design has a FEBI region, the setup only allows you to select the Domain Decomposition solver. The **Enhanced FEBI Preconditioner** allows you to achieve faster simulation speed for most cases, at the cost of more memory.

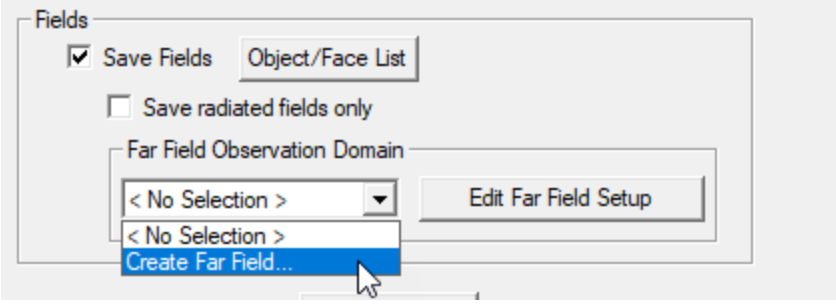
Advanced

Includes settings for mesh linking, mesh assembly for designs that include 3D components, absorbing boundaries on ports, wave port adapt options, and whether to **Save Fields** and **Save radiated fields only**. You must save fields if you intend to plot or post process fields.

For HFSS with Hybrid and Arrays solutions that do not contain SBR+ Hybrid

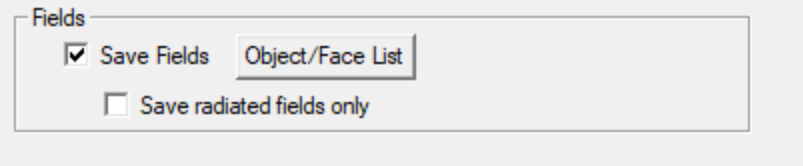
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Regions, the Fields area contains a Fields Observation Domain area that allows you to specify a Far Field sphere or edit an existing setup. If you do so, fields are generated during the solve,



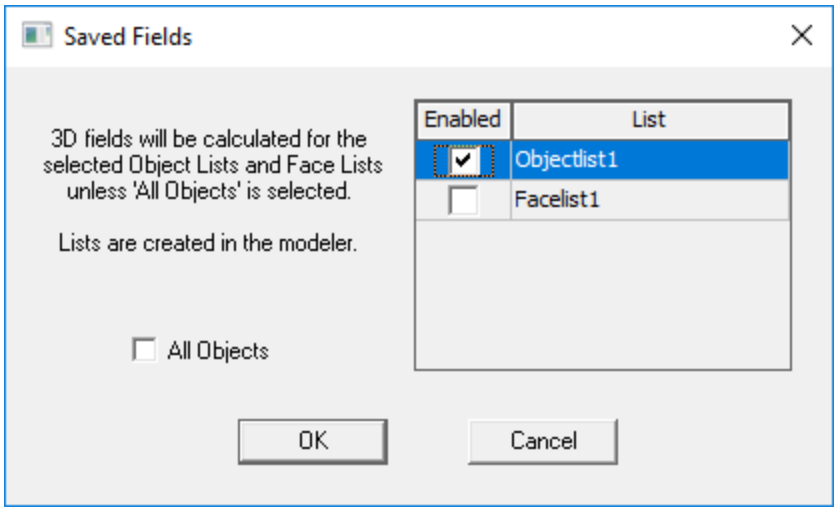
If you do so, fields are generated during the solve and for any dependent frequency sweeps, and written as .ffd files rather than as a post processing step. After the simulation, editing or deleting the selected far field setup invalidates the last adaptive solution of the driven setup as well as its dependent sweep's solution.

If an HFSS project contains an SBR+ Hybrid Region, the Fields area does not contain the Fields Observation Domain area.



For very large designs, especially for finite array designs, saving fields can potentially result in very large files. In such cases, you can use the **Object/Face List** button to open a **Saved Fields** dialog box to select from object or face lists you have predefined. You would then save 3D fields only on certain objects or faces, which could save disk space. The default is All Objects.

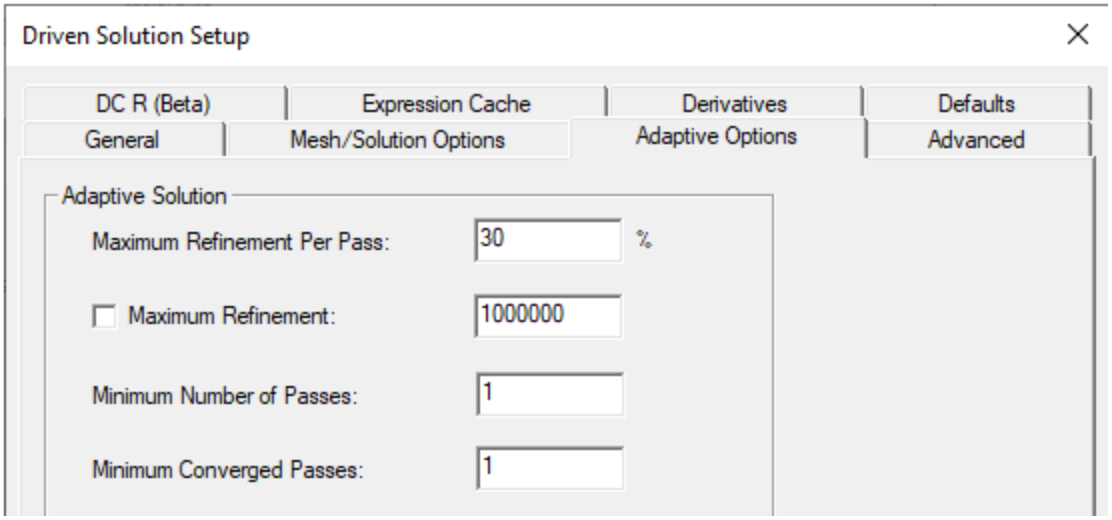
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The selection in the solve setup will dictate the behavior of the frequency sweep solve. Changing the selected lists will invalidate the last adaptive solution. If you subsequently create a Near Field or Far Field Radiation setup and select a list there, you receive a warning if your selections do not match those selected for the solve.

Adaptive Options

The Adaptive Options include Maximum Refinement per pass, Maximum Refinement, and Minimum Number of Passes, and Minimum Converged Passes.



Maximum Refinement Per Pass

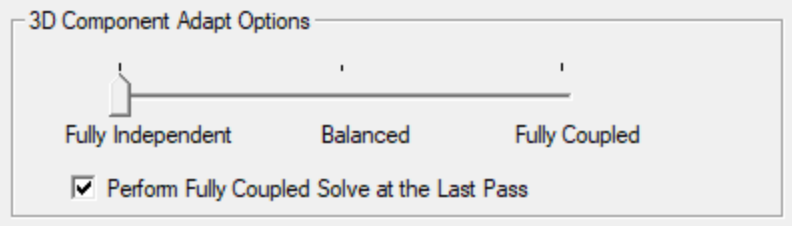
Maximum Refinement

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Minimum Number of Passes

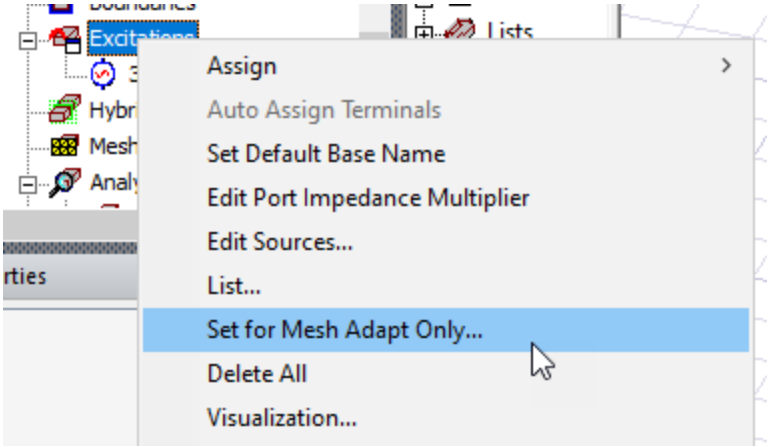
Minimum Number of Converged Passes

If you have set the Beta Option for Parallel Component Mesh Adapt, and the design contains 3D components, the **Adaptive Options** tab also lets you select 3D Component Adapt Options.



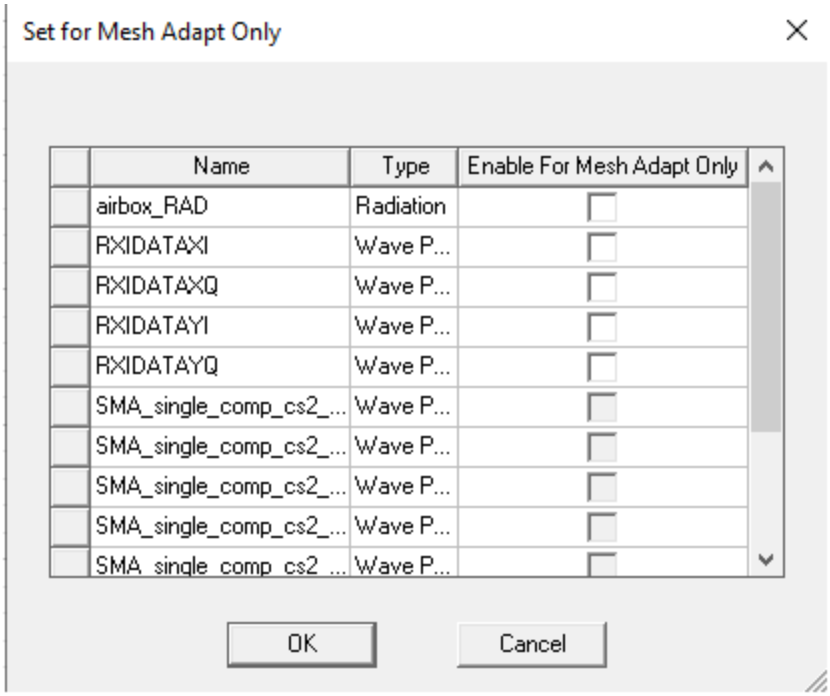
Parallel component adapt for mesh fusion workflow in MCAD including both 3D component array and general mesh fusion. During the component adapt, the multiple components are solved in parallel during each adaptive pass. You can select from Fully Independent adapt (the default for a Mesh Fusion project), Balanced adapt, and Fully coupled adapt. If you select Fully Independent or Balanced, you can specify that the last adaptive solve performs a Fully Coupled Solve, or if unchecked, a memory estimation.

You can assign certain boundaries and ports to improve adaptive accuracy. For example, for component without any ports, certain ports together with a PEC boundary can be added to drive the adaptive pass. These boundaries and/or ports are present during adaptive pass only and will be removed automatically during full design solve. To do so, right-click on the Excitations icon and select **Set for Mesh Adapt Only...**



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

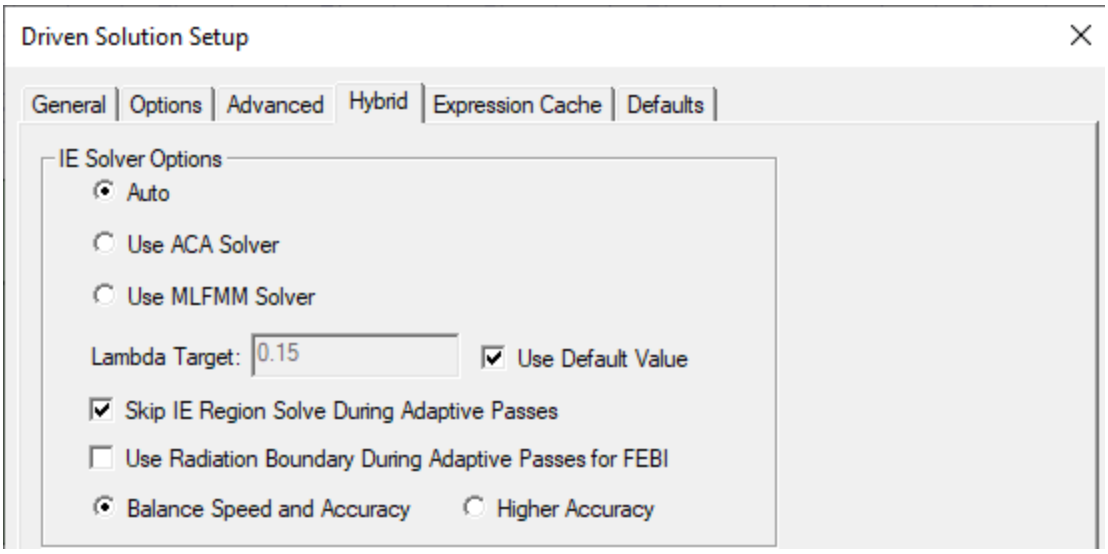
This opens a dialog listing the ports/boundaries.



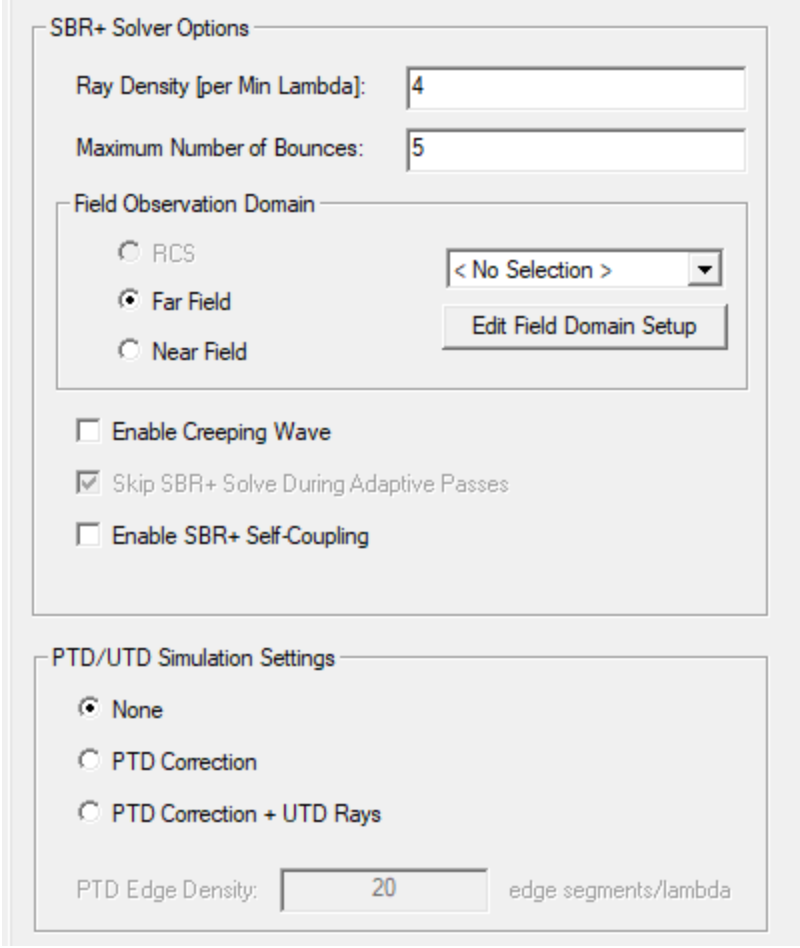
When an adapted port is present, you have an option to display s-parameter results that do or do not include adapted ports (full s-matrix or reduce s-matrix) for adaptive solves. Ports selected here do not appear in the Edit Sources dialog. Because there is no LastAdaptive solution if you turn off “Perform Fully Coupled Solve at the Last Pass”, the far field plot will be blank.

Hybrid If the design includes Hybrid regions, this tab for Setting Hybrid Parameters for HFSS.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If SBR+ Regions are present, you can set SBR+ Solver Options.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

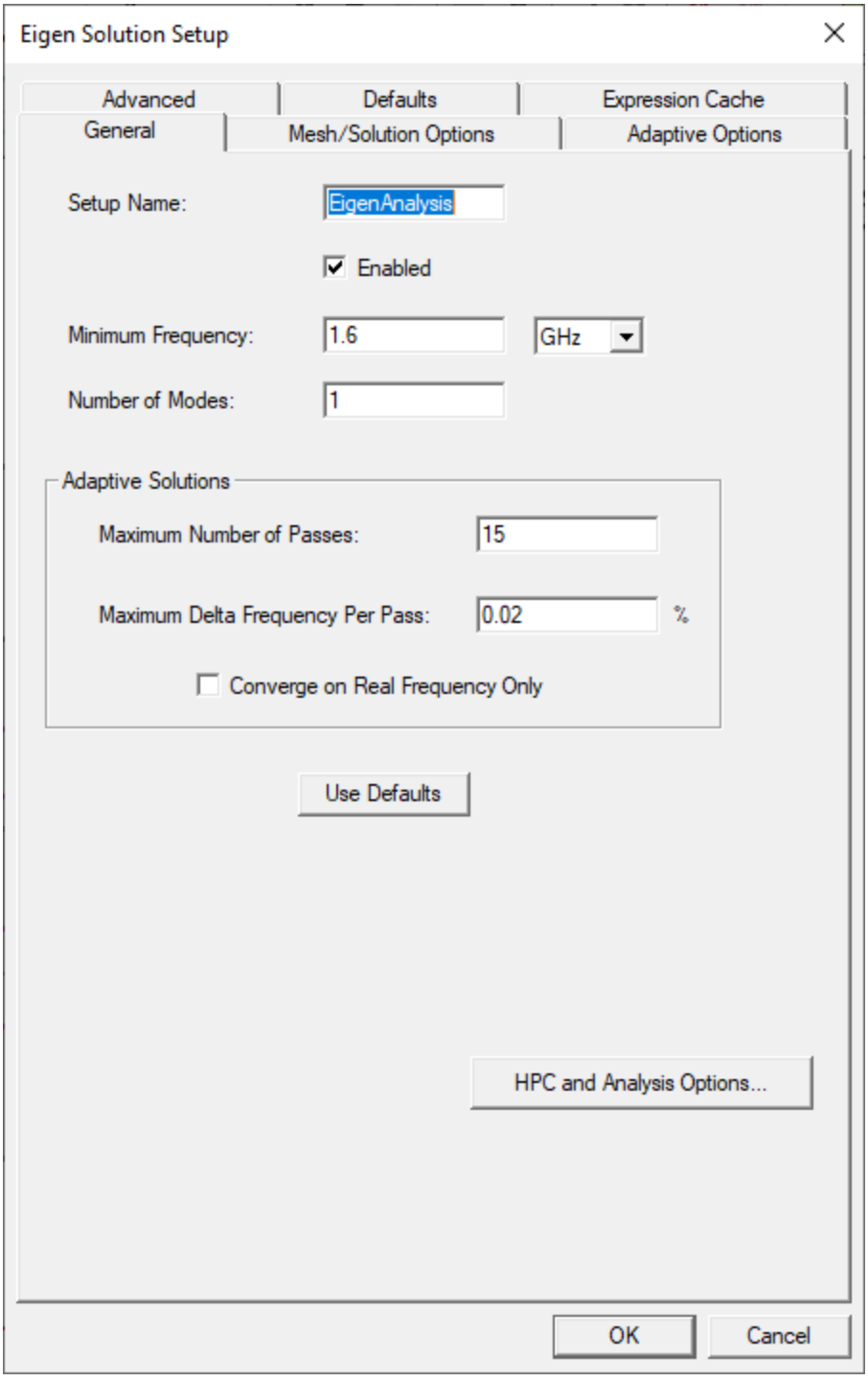
Expression Cache	Includes a list of expressions (including post processing variables) that you can use for convergence for adaptive analysis.
Derivatives	If your design contains variables, they are listed here. HFSS can calculate derivatives for your variables.
Defaults	Enables you to save the current settings as the defaults for future solution setups or revert the current settings to HFSS's standard settings.

3. Click the **General** tab.
4. The **Enabled** check box on **General** tab permits to you to disable a setup so that it does not run when you select Analyze All.
5. For Driven solution types, do the following:
    a. Select whether you want to solve for Single frequency, Multi-Frequency, or Broadband, and enter the appropriate **Solution Frequency** data and select the frequency units from the pull down list.
    b. Optionally, select **Solve Ports Only**.
7. The lower right corner also contains a button for HPC and Analysis options. Here you can select or create an analysis configuration.
8. Click **OK**.
9. Optionally, add a frequency sweep to the solution setup.

Adding an Eigenmode Solution Setup

For calculating the eigenmodes, or resonances, of a structure. The Eigenmode solver finds the resonant frequencies of the structure and the fields at those resonant frequencies. Eigenmode designs cannot contain design parameters that depend on frequency, for example a frequency-dependent impedance boundary condition. For an Eigenmode Solution Type, **HFSS>Analysis Setup>Add Solution Setup...** or right-click on **Analysis** in the Project tree, and select **Add Solution Setup...** from the shortcut menu displays the following dialog:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For Eigenmode solution types, do the following:

a. Enter the **Minimum Frequency** and the frequency units.
b. Enter the **Number of Modes**. The number must be greater than 0 and less than 20.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you are performing an adaptive analysis, enter **2** or more passes in the **Maximum Number of Passes** box, and then specify the remaining adaptive analysis parameters for HFSS.

For Eigenmode solutions, if you are not performing an adaptive analysis, entering **0** will enable you to bypass the adaptive analysis process and just perform a frequency sweep. For driven problems HFSS always requires at least one adaptive pass. Entering **1** will also bypass adaptive analysis, generating a solution only at the solution frequency you specified.

You can check Converge on Real Frequency Only.

The lower right corner also contains a button for HPC and Analysis options. Here you can select or create an analysis configuration.

The **Adaptive Options** tab includes adaptive analysis settings.

- Maximum Refinement Per Pass
- Maximum Refinement
- Minimum Number of Passes
- Minimum Number of Converged Passes

The **Mesh/Solution Options** tab contains the following options: Mesh settings for lambda refinement options, the Order of Basis setting.

The **Advanced** tab contains settings for mesh linking, and whether to **Save Fields** and **Save radiated fields only**. You must save fields if you intend to plot or post process fields.

Adaptive Meshing Methods In HFSS

The adaptive mesh refinement feature in HFSS automatically generates accurate and reliable solutions and lets you focus on setting up your design efficiently instead of spending time in determining and creating the best mesh. The Solution Frequency setup offers the flexibility of adapting the mesh at various frequencies for your design. HFSS offers three Solution Frequency setups for generating adapted meshes:

- Single
- Broadband
- Multi-Frequencies.

> **Note:**
>
> The Single Solution Frequency setup is applicable for all solution types. Broadband and Multi-Frequencies are applicable for modal and terminal solutions only.

Based on the solution frequency setup and the convergence criteria HFSS automatically creates a suitable mesh for your design and generates accurate solutions. Once the final mesh is

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

generated, HFSS also creates a solution across a frequency range of interest provided you defined a frequency sweep.

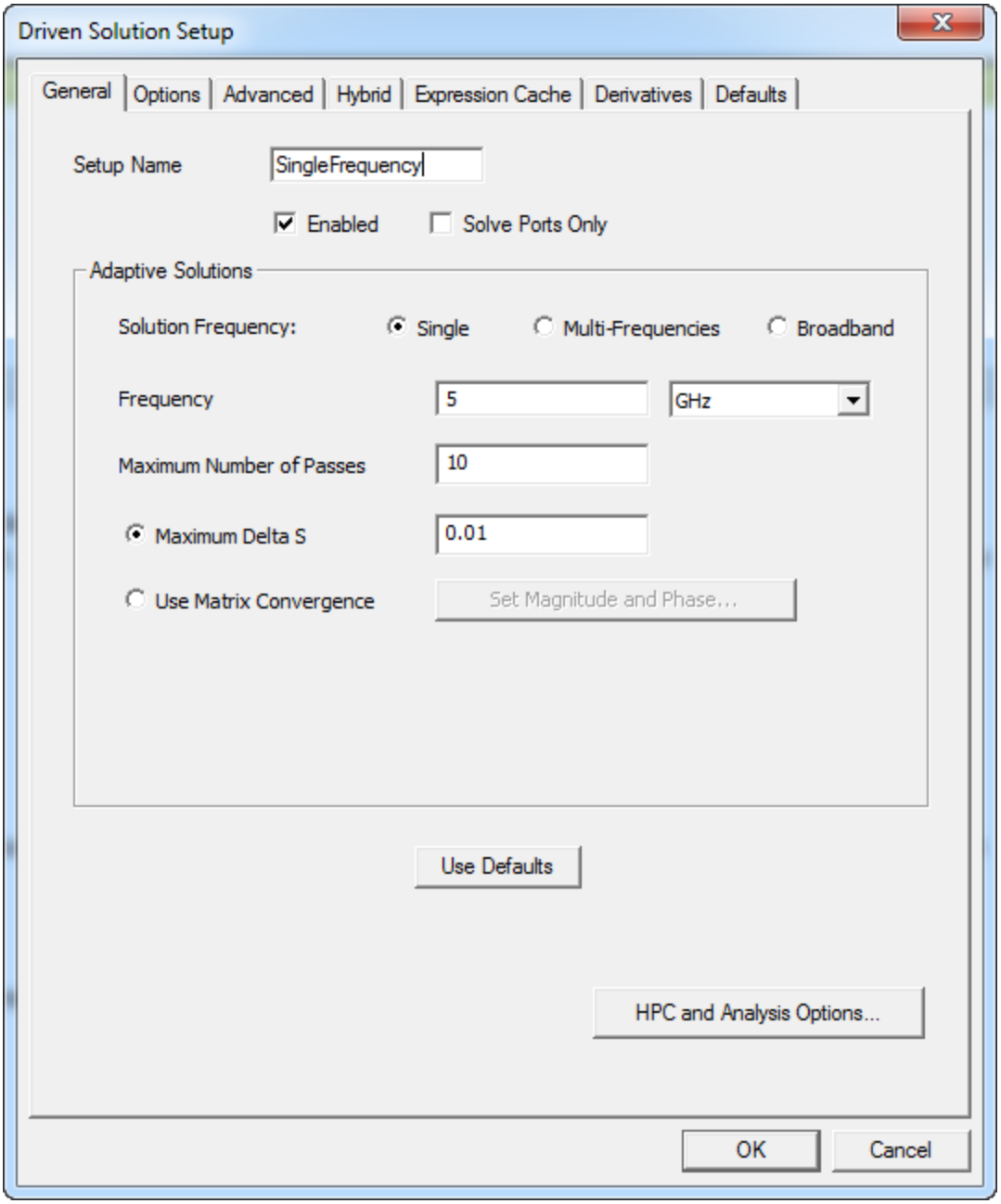
Single Solution Frequency

If you define the Single frequency setup HFSS performs adaptive mesh refinement at that frequency for your design until the convergence criterion is satisfied. The most common convergence criterion is determined by the difference in the S-parameter value between two consecutive solves known as **Maximum Delta S**, which should be less than the specified magnitude. Once the solution converges, the final mesh is used to evaluate the electromagnetic characteristics of the design. If you define a frequency sweep, solution data is generated across this range of frequencies.

To specify the Single Solution Frequency setup, perform these steps:

1. Right-click **Analysis** in the project tree and select **Add Solution Setup**.
2. Select **Single** as the **Solution Frequency**.
3. Specify the **Maximum Number of Passes** and the **Maximum Delta S**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Broadband Frequency

For most problems, specifying the Single solution frequency setup is adequate to obtain accurate results. If you desire increased reliability for broadband devices and more accurate solutions, you can specify the **Broadband** option for the Solution Frequency setup. The Broadband setup enables HFSS to intelligently determine the appropriate frequencies at which to adapt the mesh. The virtue of automatic broadband adaptive meshing lies in eliminating

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

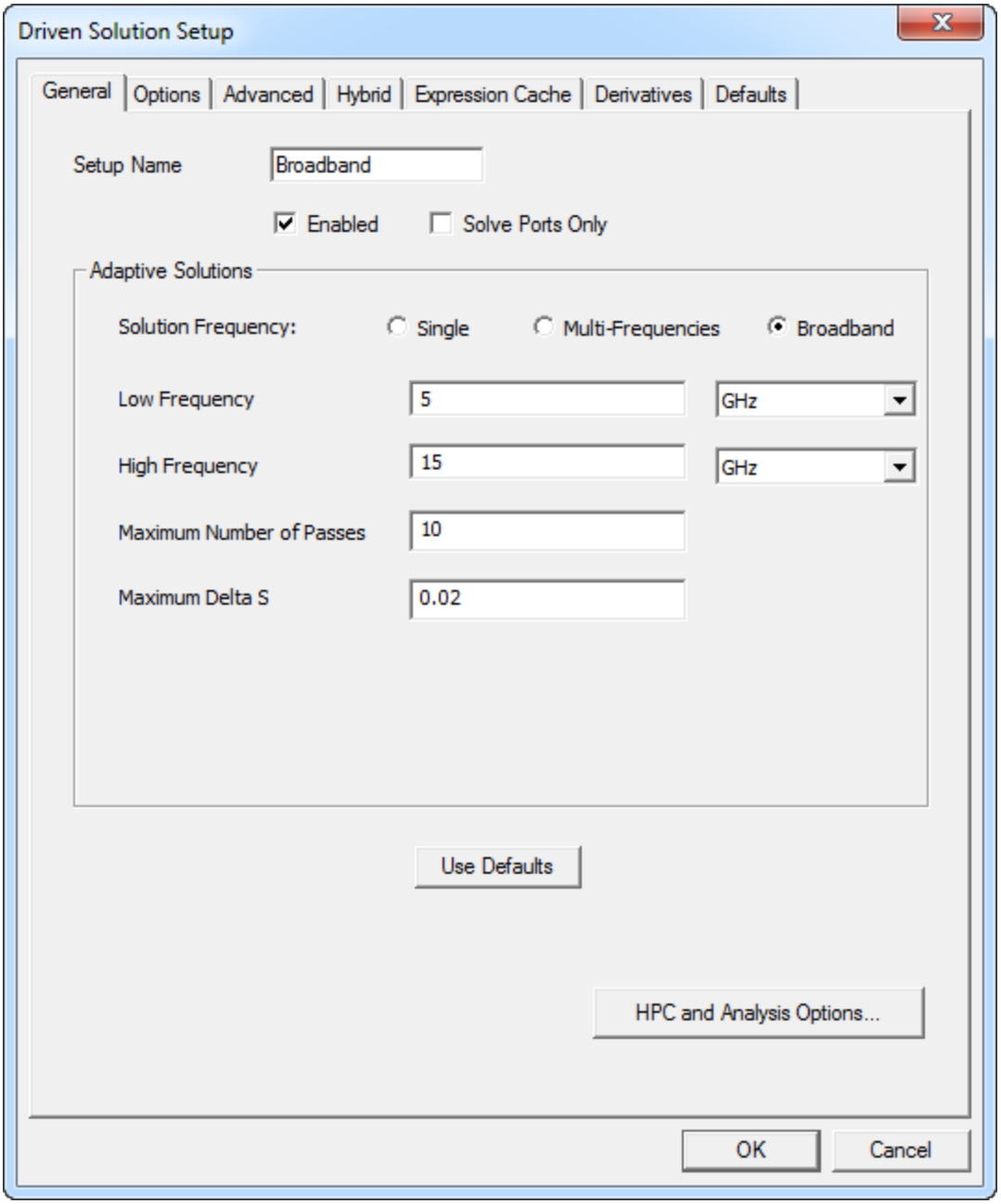
uncertainty in choosing the best frequency for adapting the mesh. You need only specify the highest and lowest frequencies of the range and HFSS determines the frequencies at which to adapt the mesh. The mesh is always adapted for a minimum of three frequencies within the specified frequency range. Adaptive meshing at additional frequencies require enabling the high performance computing (HPC) feature and availability of sufficient computational resources.

The adaptive mesh refinement process occurs at various frequencies within the specified frequency range until the convergence criterion is met. In this case, the values of **Maximum Delta S** between consecutive solves are calculated for each frequency. The weighted average of **Maximum Delta S** across all frequencies must be below the specified magnitude of **Maximum Delta S** for the simulation to converge. You also have the option to set **Maximum Number of Passes** as the stopping criterion.

For Broadband adapt, HFSS prompts you with an interpolation sweep predefined by a range of frequencies identical to the Broadband frequency range. In addition, the algorithm chooses the adapt frequencies in a way that they can also be reused in the frequency sweep. This increases the efficiency of the solution process.

> **Note:**
>
> Broadband frequency setup does not allow viewing the fields for the adapted frequencies. You can define a discrete sweep including the adapt frequencies and solution data will be provided without re-solving.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



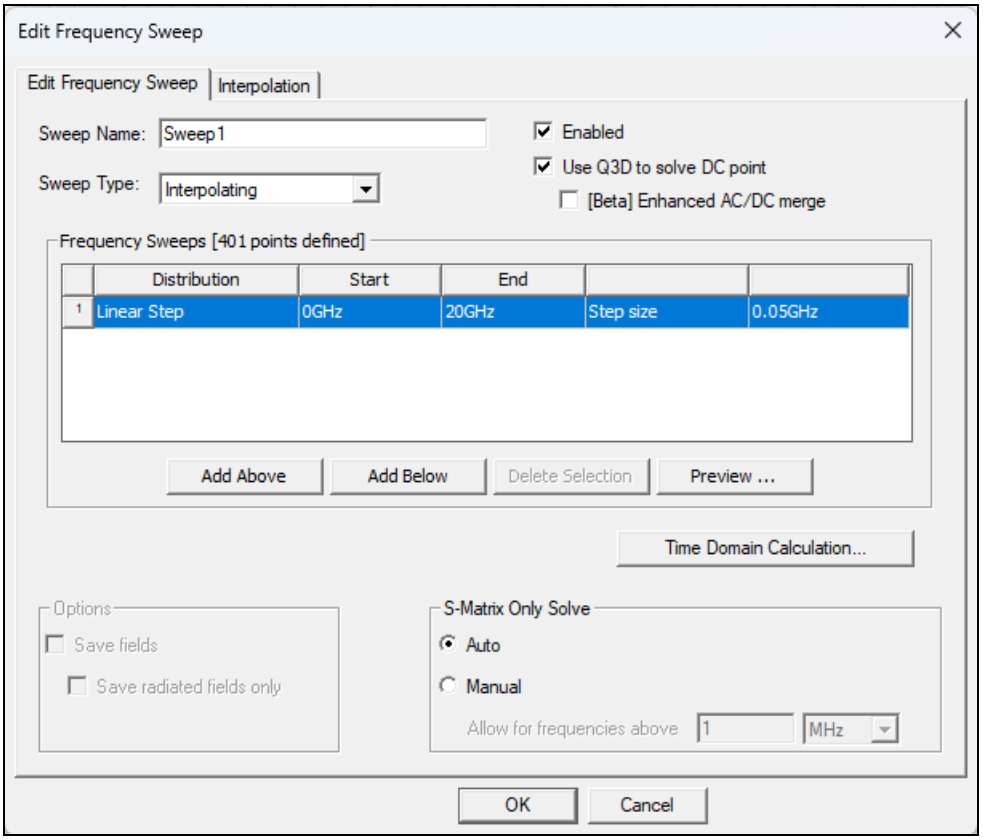
To specify the broadband frequency option, perform these steps:

1. Right-click **Analysis** in the project tree and select **Add Solution Setup**.
2. Select **Broadband** as the **Solution Frequency**.
3. Specify values for **Low Frequency** and **High Frequency**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Specify the convergence criteria that include **Maximum Number of Passes** and the **Maximum Delta S** and click **OK**.

   The **Edit Frequency Sweep** window appears automatically. The interpolating sweep is automatically populated with **Start** and **End** values, which are identical to the low and high frequency values of the broadband frequency range specified in the solution setup window.



Multi-Frequencies

If you know apriori the frequencies at which to adapt the mesh for your design, you can specify the **Multi-Frequencies** setup. You can specify a number of individual frequencies in this setup and adaptive meshing occurs at all the frequencies.

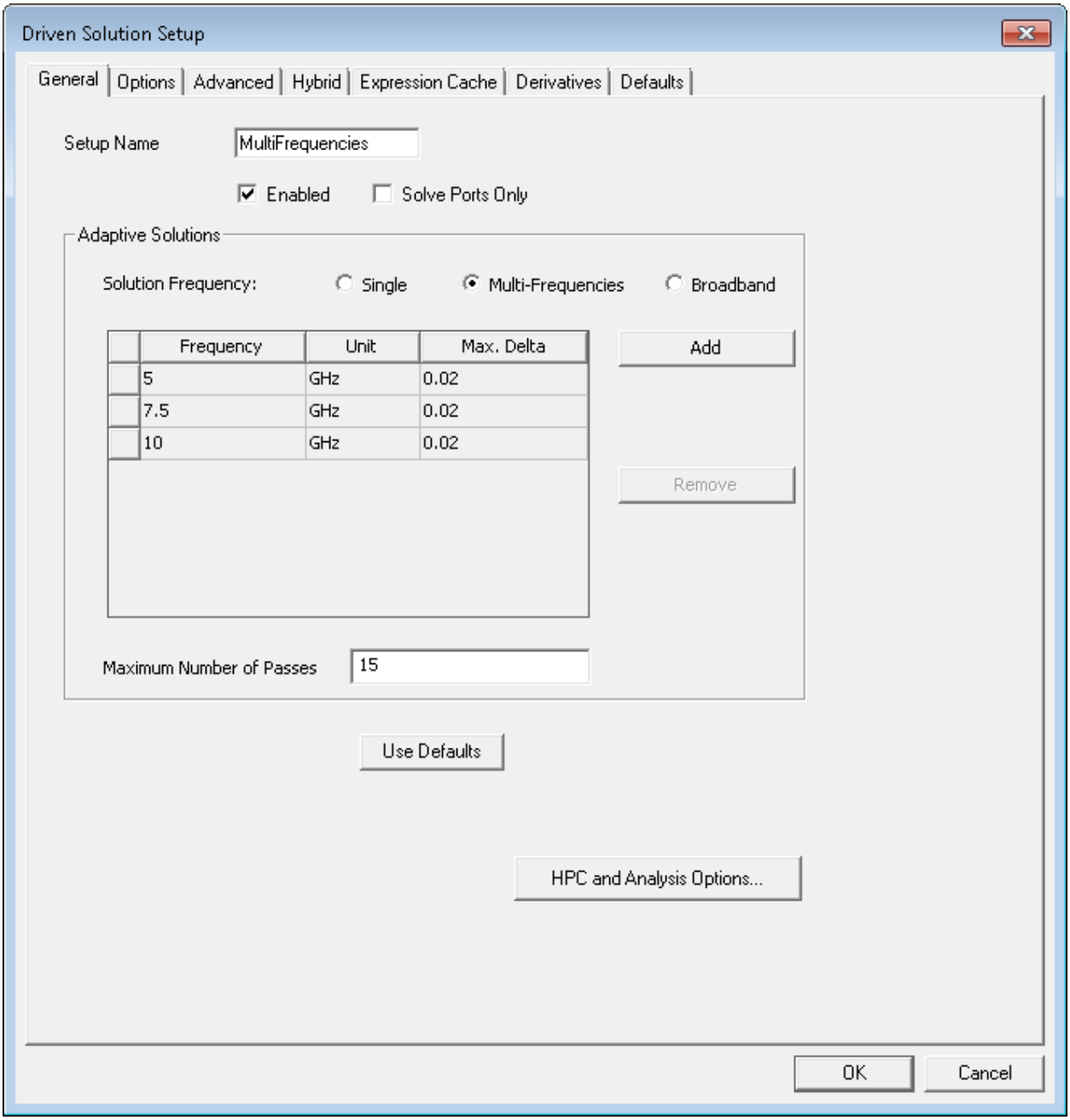
You have the option to set different values of **Maximum Delta S** for the specified frequencies. For each frequency the individual target Delta S value must be reached in order for the entire solution to converge. For instance, if you define 4 frequencies for adapting the mesh, the target value of **Maximum Delta S** for each of the 4 frequencies must be satisfied for the solution to reach convergence.

To specify individual frequencies in the Multi-Frequencies setup, perform these steps:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right-click **Analysis** in the project tree and select **Add Solution Setup**.
2. Select **Multi-Frequencies**.
3. Edit the existing frequency field to a value of your choosing.

To enter additional frequency points, use **Add** and edit the corresponding value in the **Frequency** column.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Adaptive Analysis Parameters for HFSS

When you set up an adaptive analysis, define the following parameters under the **General** tab of the **Solution Setup** dialog box:

- Enter the Solution Frequency and select the frequency units from the pull down list.
- Optionally, select Solve Ports Only.
- Maximum Number of Passes
- Maximum Delta S or Use Matrix convergence (for designs with ports). Here you can set matrix values for convergence, including maximum delta for Mag S and Phase S.
- Maximum Delta Energy for convergence per pass (for designs with voltage sources, current sources, incident waves, or magnetic bias, or for Characteristic Modes solution types).
- For Eigenmode solutions, specify Maximum Delta Frequency Per Pass and, if desired, Converge on Real Frequency Only.
- For Characteristic Modes solutions, Maximum Number of Modes and Minimum Modal significance.

Under the **Mesh/Solution Options** tab of the **Solution Setup** dialog box, you can edit the following settings:

- Lambda Refinement
- Maximum Refinement Per Pass
- Maximum Refinement
- Minimum Number of Passes
- Minimum Number of Converged Passes
- Order of Basis functions
- Enable the Auto Select Direct/Iterative Solver
- Enable the Direct Solver
- Enable Iterative Solver and associated Relative Residual Setting.
- Enable Domain Decomposition

Under the **Advanced** tab of the **Solution Setup**, depending on the solution type, you can edit the following settings.

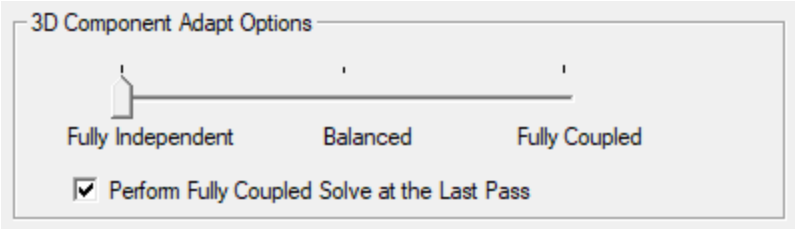
- Initial Mesh Options for mesh linking
- Port options (Maximum Delta Z_o, whether to Use Radiation Boundary on Ports and Min/Max Port Triangle settings)
- Whether to **Save fields**, and/or whether to save radiated fields only. You must save fields to generate field overlays and reports for near and far fields. To view a port field display, you must save fields. Save fields options also occur for Discrete and Fast Sweeps.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Under the **Adaptive Options** tab of the **Solution Setup**, you can edit the following settings:

- Maximum Refinement Per Pass
- Maximum Refinement
- Minimum Number of Passes
- Minimum Number of Converged Passes

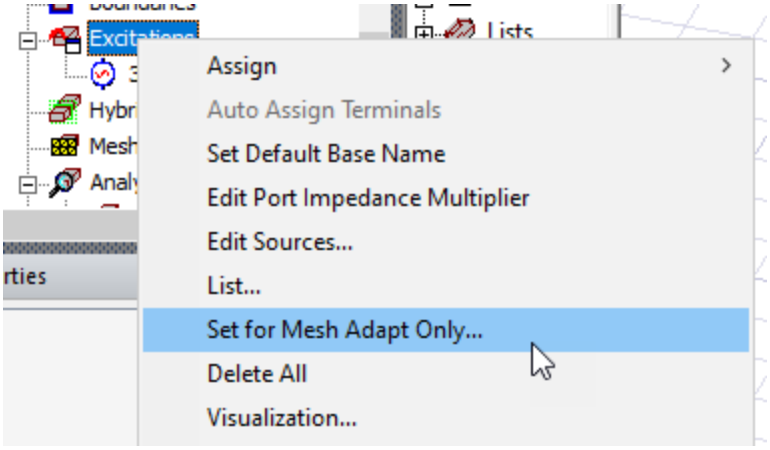
If you have set the Beta Option for Parallel Component Mesh Adapt, and the design contains 3D components, the **Adaptive Options** tab also lets you select 3D Component Adapt Options.



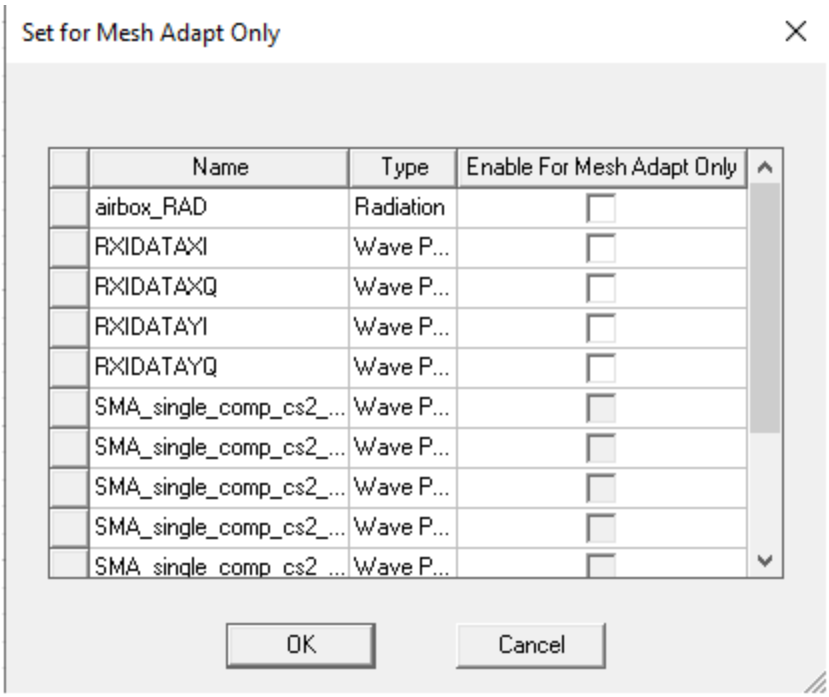
Parallel component adapt for mesh fusion workflow in MCAD including both 3D component array and general mesh fusion. During the component adapt, the multiple components are solved in parallel during each adaptive pass. You can select from Fully Independent adapt, Loosely coupled adapt, and Fully coupled adapt. If you select Fully Independent or Loosely Coupled, you can specify that the last adaptive solve performs a Fully Coupled Solve, or if unchecked, a memory estimation. Because there is no LastAdaptive solution if you turn off “Perform Fully Coupled Solve at the Last Pass”, the far field plot will be blank.

You can assign certain boundaries and ports to improve adaptive accuracy. For example, for component without any ports, certain ports together with a PEC boundary can be added to drive the adaptive pass. These boundaries and/or ports are present during adaptive pass only and will be removed automatically during full design solve. To do so, right-click on the Excitations icon and select **Set for Mesh Adapt Only...**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This opens a dialog listing the ports/boundaries.



When an adapted port is present, you have an option to display s-parameter results that do or do not include adapted ports (full s-matrix or reduce s-matrix) for adaptive solves. Ports selected here do not appear in the Edit Sources dialog.

Under the Hybrid tab of the of the Solution Setup, you can set solve parameters for any Hybrid Regions you have assigned in the design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Under the **Expression Cache** tab of the **Solution Setup**, you can create and manage expressions to use for adaptive convergence.

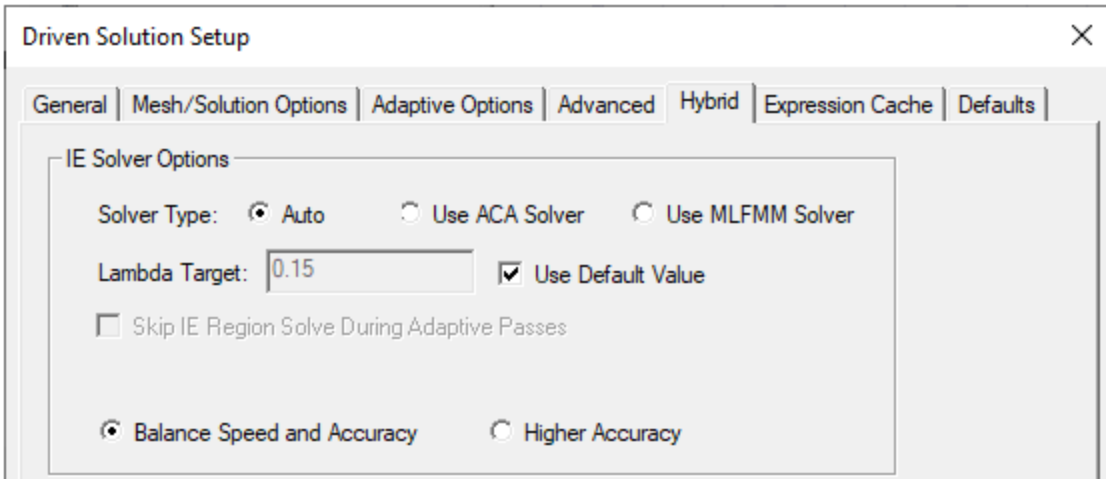
Under the **Derivatives** tab of the **Solution Setup**, you can:

- Specify which variables to use for calculating derivatives.

Setting Hybrid Region Parameters for HFSS

When you set up an adaptive analysis for an HFSS with Hybrid and Arrays solution type, define the following parameters under the **Hybrid** tab of the Advanced **Solution Setup** dialog box:

- **IE Solver Options**. You can specify the IE Solver type as ACA (the traditional method) or as MLFMM, which is superior to ACA for models with large FE-BI surfaces, and also works for IE regions. The default choice is Auto, in which the choice is made based on the characteristics of the design.



Both IE solvers support distributed memory using MPI. The MLFMM solver option provides a more efficient solution to certain classes of scattering problems. The MLFMM solver is typically more efficient (in memory and speed) than the ACA solver for problems having electrically large, mostly smooth, scattering surfaces which are comparable in all three dimensions. For a more detailed discussion, see MLFMM Usage Guidelines.

- **Lambda Target for IE Solvers**. This refers to the background material. For these fields to be active, the Do Lambda Refinement option on the **Options** tab must be enabled. If the Use Default Value check box is checked, the text box is disabled, and the 0.15 Lambda target value is used. If you uncheck Use Default Value, you can specify a target value for the background material for IE regions. This can be useful for designs that include curvilinear elements.
- **Skip IE Region Solve During Adaptive Passes**. If the design contains at least one FE-BI domain and at least one IE region or PO boundary, you can choose this to solve the FE-

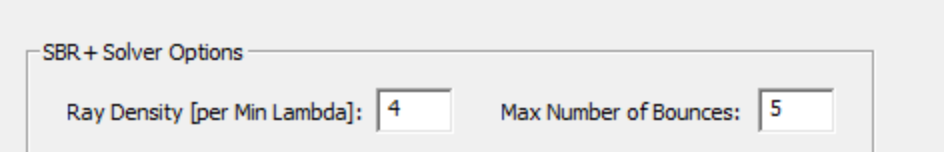
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BI region before solving coupled FE-BI and IE and PO regions. An IE port is not allowed for skipping IE region solve during adaptive passes. If the design also contains SBR+ Regions, the **Skip SBR+ Solve During Adaptive Passes** feature must also be selected.

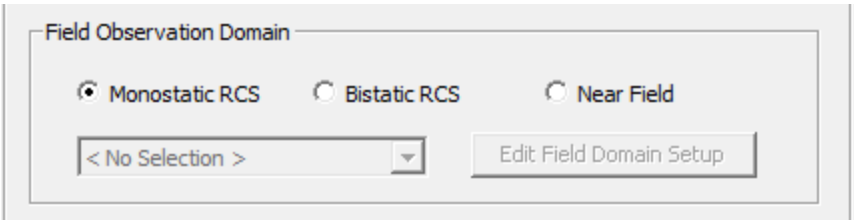
- **Use Radiation Boundary During Adaptive Passed for FEBI**. If selected, the solver uses a radiation boundary instead of FEBI during adaptive passes. When it converges, solver enables the FEBI for the last adaptive pass. If the FEBI surface is sufficiently large, this option can lead to a faster adaptive mesh run.
- **Balance Speed and Accuracy** or **Higher Accuracy**. This refers to the balance of speed and accuracy applied to IE Hybrid Regions. Higher Accuracy corresponds to the Higher Accuracy setting in the Automatic Solution setup. The IE solver is slower and uses more memory when you select this option.

Additional Options for SBR+

- **Ray Density and Maximum Number of Bounces**: If you have assigned an SBR+ Region, you can specify the SBR+ Solver Options for Ray Density (Per Wavelength) and Max Number of Bounces. This lets you balance solution speed and desired accuracy.



- **Field Observation Domain**: If you want to consider Near or Far fields, you can create new Near Field Setup, an new Far Field setup, or edit an existing setup.

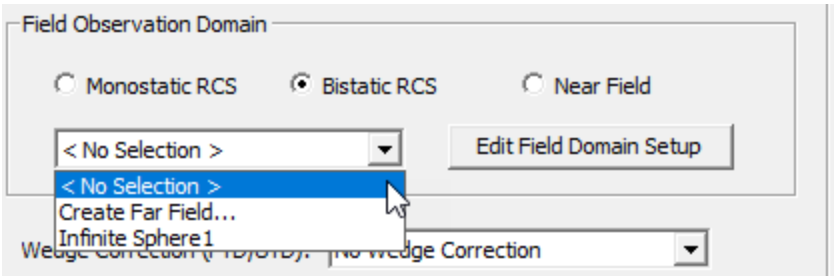


By default, a Far Field Setup is an Infinite Sphere, discussed below. By default the Near Field setup is a rectangle. For setup and use for Near Field setups, see Overlay of Contour Plot of Near Field Rectangle.

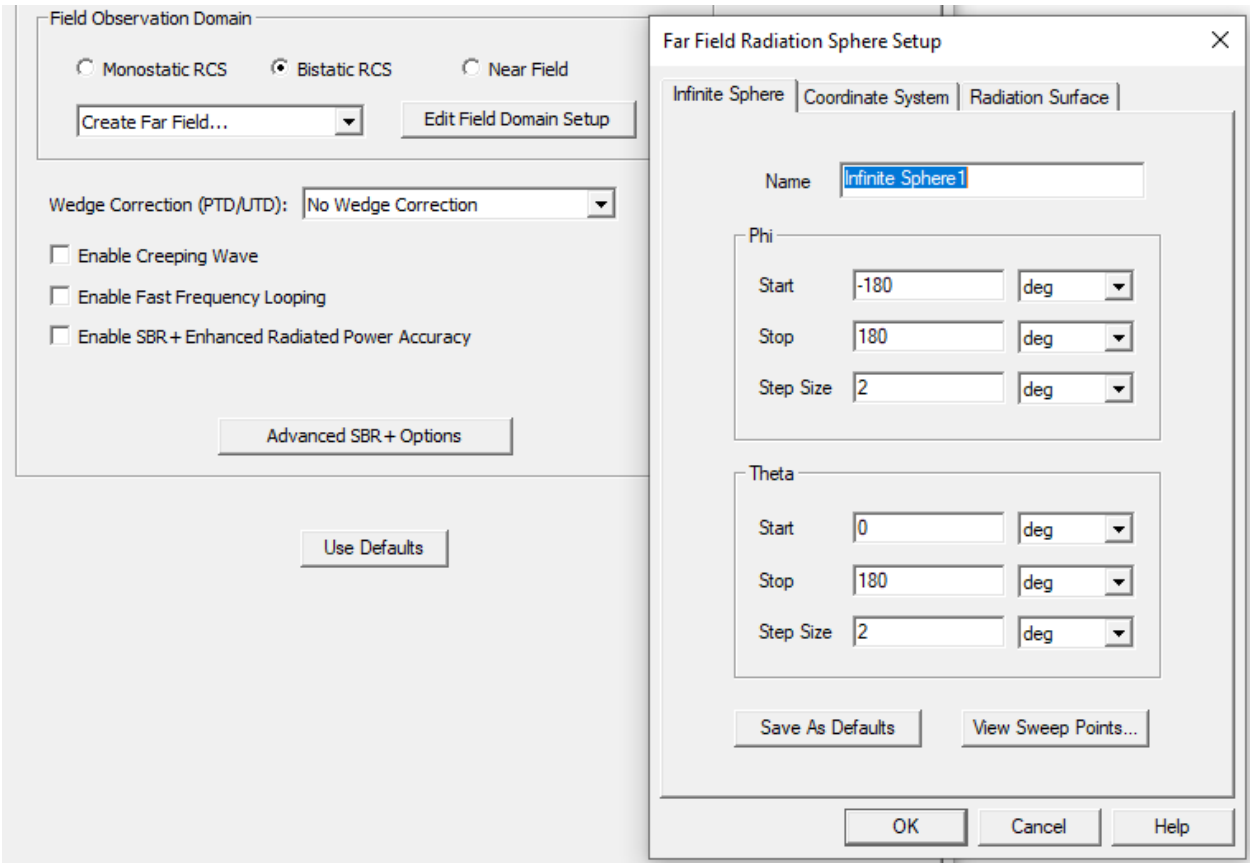
- **RCS Type**: SBR+ supports assigning an Incident Plane wave for calculating RCS. If you have assigned an Incident Plane wave, the options include the RCS Type as Monostatic or Bistatic. If you select Monostatic, you do not need to create an infinite sphere.If you select have assigned both an SBR+ region and an Incident Plane wave, and select Bistatic as the RCS Type, the Far Field Observation domain fields are enabled, and the **Far Field Radiation Sphere Setup** dialog appears as follows, allowing you to create an

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

infinite sphere.

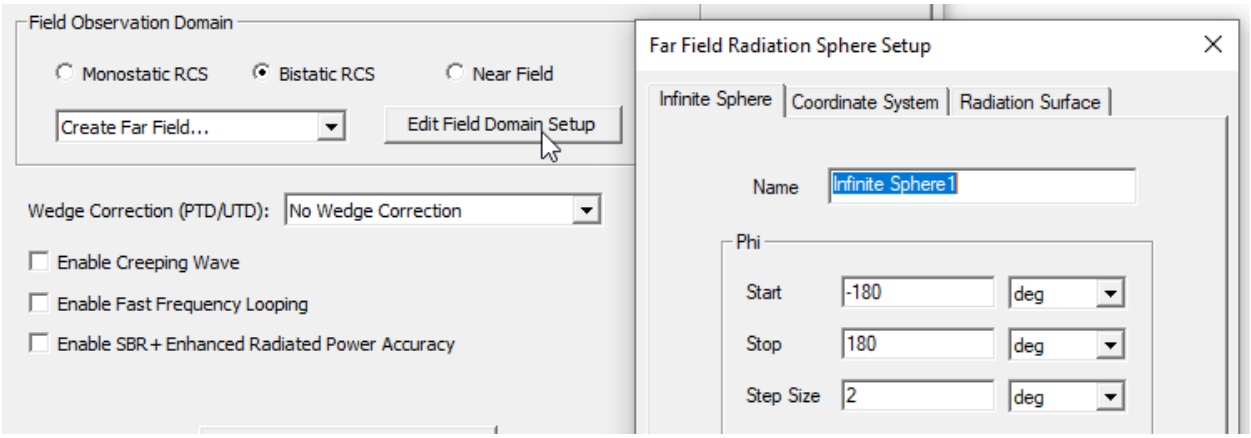


You can create far field, or edit an existing Far Field Infinite Sphere in the drop down list by selecting it and by clicking **Edit Field Domain Setup**.



If you have created one or more infinite spheres, these are listed on the dropdown. You can edit an existing Far Field Infinite Sphere by clicking **Edit Field Domain Setup**.

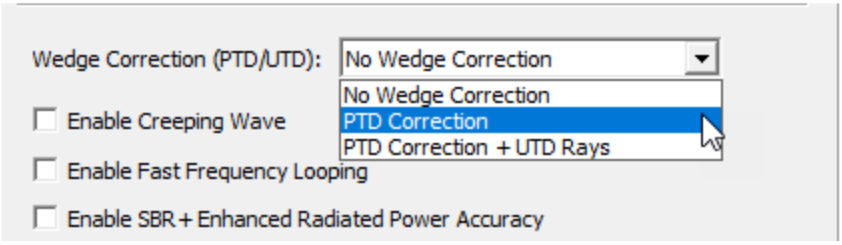
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Enable Creeping Wave**: For a description of the requirements and circumstances when you might enable Creeping Waves, see Creeping Wave for Antenna Placement on Curved Surfaces in SBR+ Simulation or Creeping Wave for RCS Analysis of Curved Surfaces in SBR+ Simulation.
- **Skip SBR+ Solve During Adaptive Passes option**. SBR+ regions are not being mesh adapted and SBR+ solutions have no impact on field solutions on FEM or IE regions. However, SBR+ does impact stopping criteria in some cases such as coupling between two separate source antennas. Therefore, to speed up mesh adaption, you can choose to not solve SBR+ regions until source regions have converged in isolation. However, SBR+ does impact stopping criteria in some cases in the form of cache expression such as coupling between two separate source antennas or far field pattern. In such cases, SBR+ solve could not be skipped. Moreover, SBR+ solve will always be launched when maximum number of passes is reached regardless of source region convergence. If you have selected **Skip IE Region Solve During Adaptive Passes**, you must also select **Skip SBR+ Solve During Adaptive Passes**.

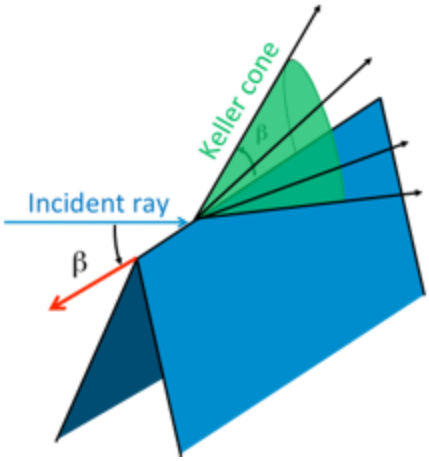
- The **Wedge Correction (PTD/UTD)** simulation settings allow for the inclusion of additional wedge diffraction phenomenology that can improve the accuracy of SBR+ simulations. You can opt out of using the PTD/UTD settings, or select PTD Correction or PTD Correction + UTD Rays. Selecting one of the PTD Correction options enables a field for specifying PTD Edge Density.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



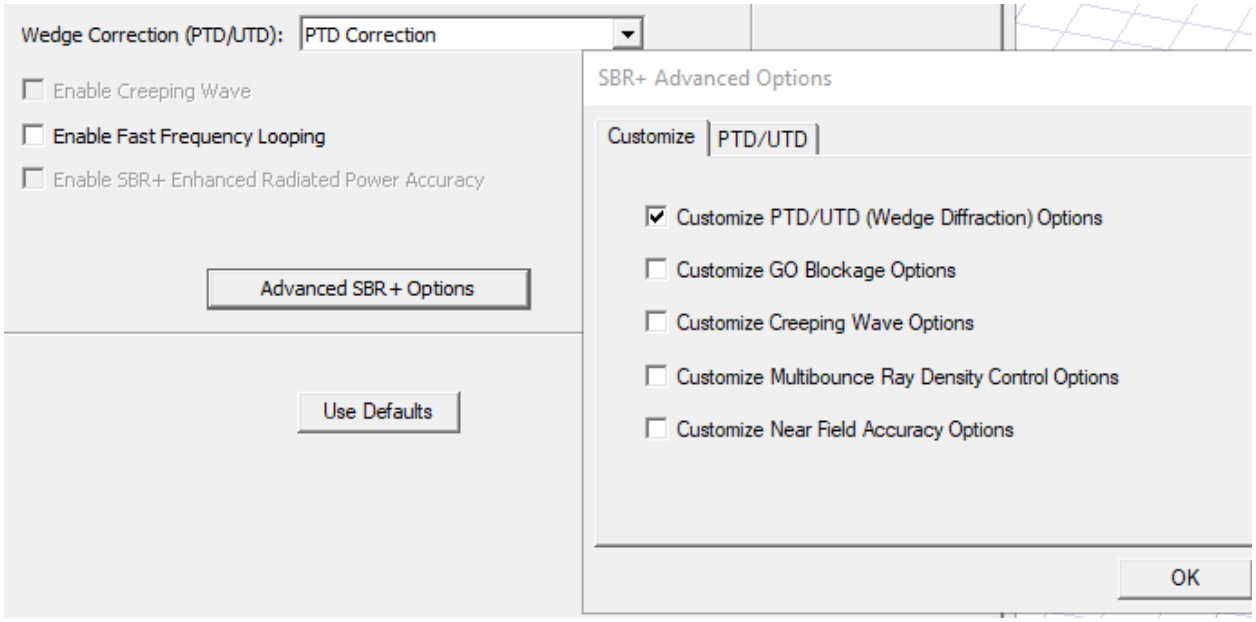
The PTD and UTD wedge features are only deployed for metallic wedges with line-of-sight visibility from the source (Tx) location. If either adjacent surface of the wedge is non-PEC and not within the tolerance for PEC-like, or if the entire edge segment is not visible to the source, the wedge will be ignored in the SBR+ simulation and for visual ray tracing (VRT)

For SBR+ simulations, the Physical Theory of Diffraction (PTD) and Uniform Theory of Diffraction (UTD) features can account for additional phenomenology not well predicted by plain SBR due to truncation of uniform Physical Optics (PO) currents at sharp angular discontinuities ("wedges") on metallic surfaces and blockage of SBR's Geometrical Optics (GO) rays. PTD is a numeric correction to the scattered fields radiated by PO currents near wedges. UTD launches bundles of edge-diffraction rays from directly illuminated portions of each wedge along the Keller cone. Once launched, the UTD rays behave exactly like regular SBR rays, propagating according to GO and painting PO currents at each bounce that contribute to the scattered field. The UTD rays often illuminate portions of the SBR scattering geometry that are never reached by SBR GO rays launched directly from the field source.

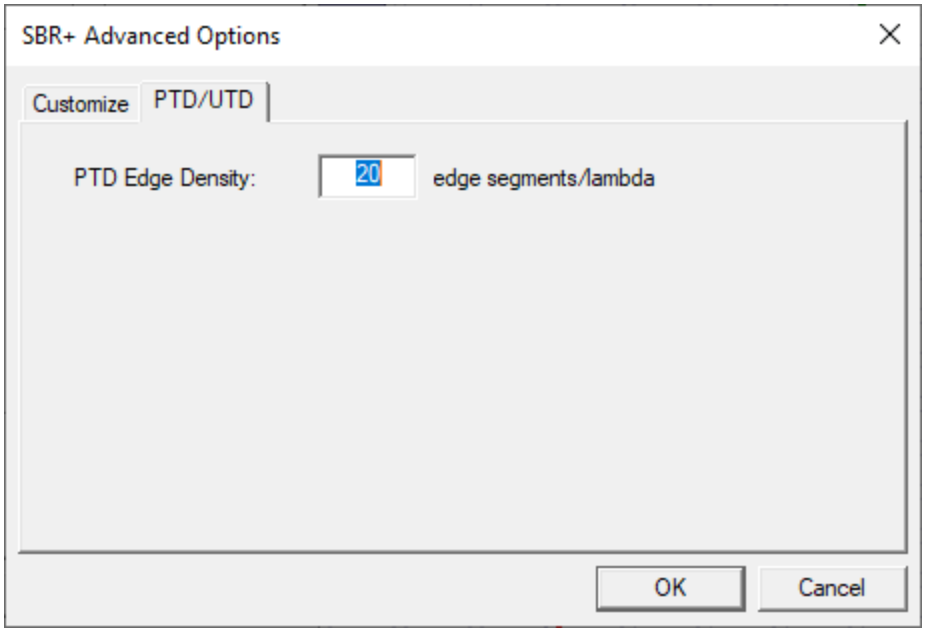


If you select one of the PTD Correction options you can then select the **Advanced SBR+ Options** button to Enable PTD/UTD (Wedge Diffraction) Options.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Enabling PTD/UTD (Wedge Diffraction) Options opens **PTD/UTD** tab for specifying PTD Edge Density.



- Advanced Option for **NF Accuracy Settings** for SBR+

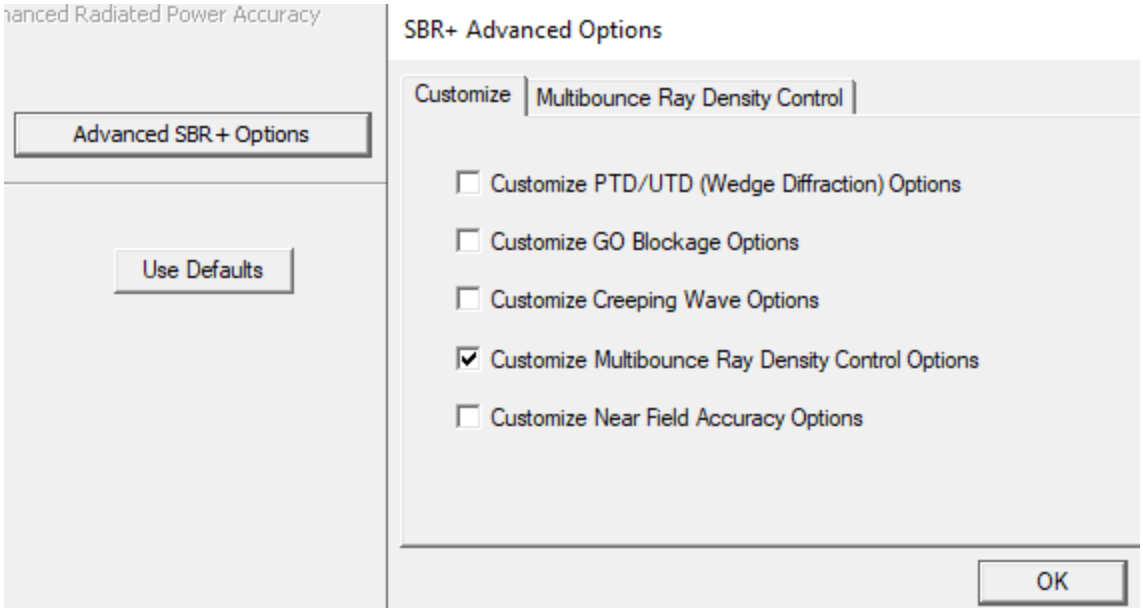
The NF Accuracy settings (Beta feature, NF Accuracy Controls for SBR+) provide access to methodology settings relating to cases where Tx antennas, Rx antennas, and near-field

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

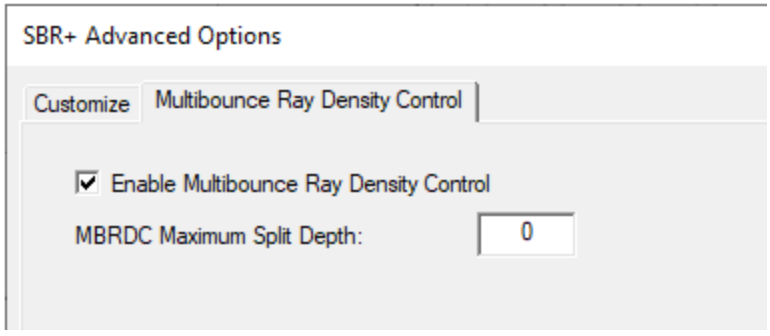
observation points are in proximity to the scattering geometry (i.e., the platform CAD model). When this feature is switched off, HFSS SBR+ uses default near-field accuracy settings that should be adequate for most situations, including close-proximity Tx/observer conditions that this feature is designed to tune.

- Advanced Option for **Multibounce Ray Density**

By option you can select the Advanced SBR+ Options and enable Customize Multibounce Ray Density Control Options.



You can then select the **Multibounce Ray Density Control** tab.



Use the checkbox to **Enable Multibounce Ray Density Control** and enter the maximum number of subdivisions to be used in the solve. Enabling the feature may increase the solve time.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **MBRDC Maximum Split Depth** field value increases the number of spawned ray tracks on each ray bounce.The MBRDC algorithm is designed to achieve the ray density per wavelength criterion at any bounce depth. If a footprint at depth *N* in a ray track does not satisfy the ray-density criterion, the MBRDC algorithm attempts a refined ray shoot from an earlier bounce, just for the ray in question. The size of the triggering footprint informs the level of refinement. If the refined shoot is not successful in achieving the desired footprint size, or new footprints are too large, the MBRDC algorithm attempts further, recursive refinements. The **MBRDC Maximum Split Depth** setting limits how aggressively the refinement algorithm is applied by specifying the maximum number of split recursions. For this reason, linear increases in **MBRDC Maximum Split Depth** can yield a geometric progression in the total number of rays shot and associated solution time.

When both MBRDC and UTD are enabled, first-bounce UTD rays and bright points on wedges are recomputed during the MBRDC splitting process, but only the original raytrack is rendered up to the 1st bounce. Similarly, length-based ray filters are applied to the original UTD initial ray track up to 1st bounce rather than using the information in the additional MBRDC tracks. This is a known limitation.

- Advanced Option for **Geometrical Optics (GO) Blockage**

SBR+ normally applies a physical optics (PO) blockage model where blockage effects emerge from partial cancellation of the incident field by the scattered field. PO blockage not only occurs in connection with the incident field from the Tx antennas, but also within the scattered field itself when reflected fields from the previous bounce are diminished by a blocking surface along the geometrical-optics (GO) reflection path at the current bounce by evaluating the scattering contribution of the incident ray field at that bounce.

In cases of significant blockage by a large obstruction, the PO blockage model requires accurate currents over the extent of the obstruction. Sometimes, this is hard to achieve in the ray tracing approximation. For example, the obstruction may not be well illuminated by direct rays from the Tx antenna or multi-bounce rays after an earlier reflection. An alternative is to use a GO blockage model where scattered field contributions are added to an observation point subject to a line-of-sight (LOS) blockage check performed by the ray tracer. In some cases, this is more accurate than the default PO blockage formulation, while in others it is less accurate. GO blockage also entails a non-trivial computational cost, as the blockage check with the ray tracer must be performed for each observation angle (or point) for each ray hit point.

In addition to GO Blockage for SBR and UTD ray tracks, this feature is also available for PTD and CW.

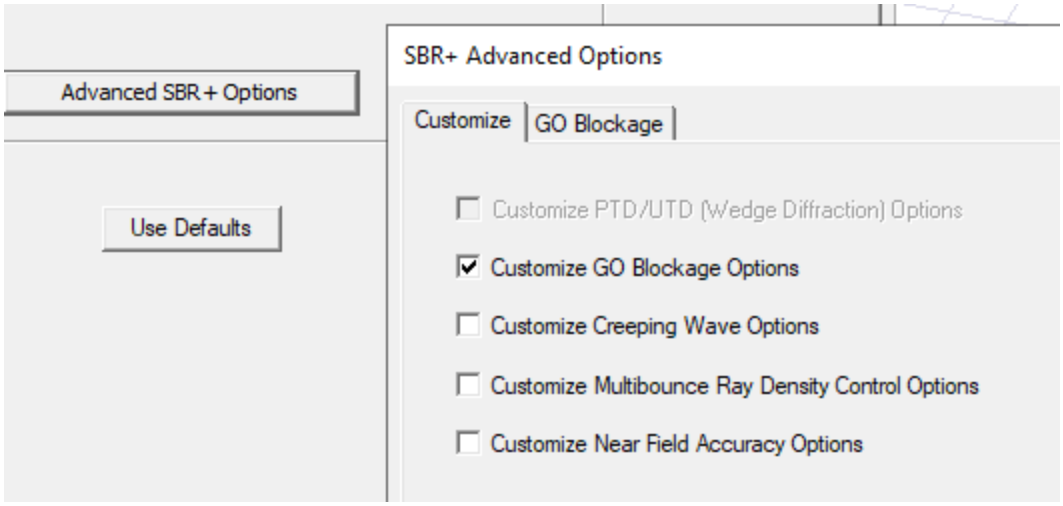
Limitations for GO Blockage:

GO Blockage formulation is suited specifically for SBR+ ray tracks, including UTD ray tracks. While GO Blockage for CW and PTD has been implemented for the first time in this release, results can be surprising and/or inaccurate, as a limitation of the methodology.

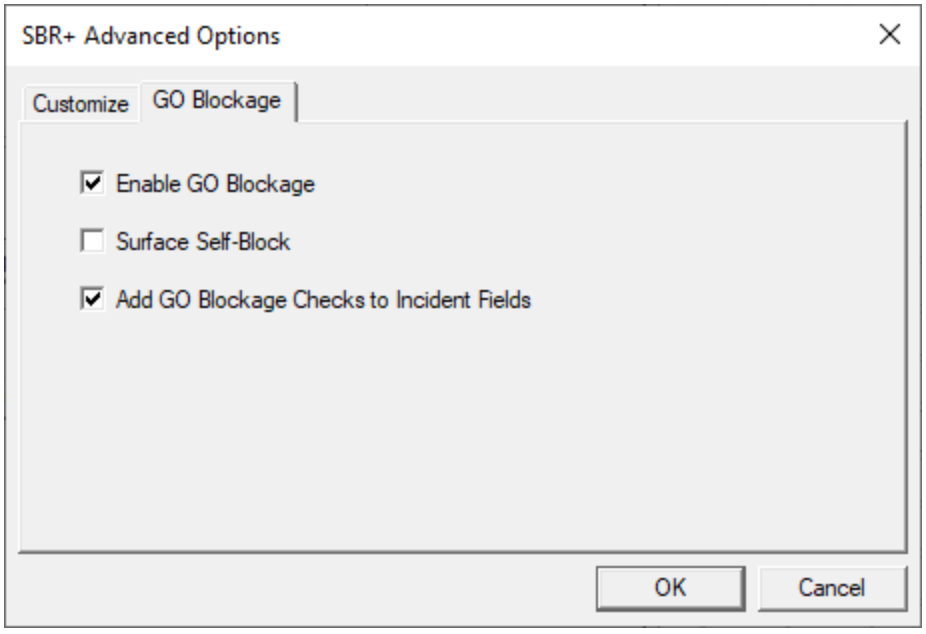
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Summary Workflow for GO Blockage:

Click the **Advanced SBR+ Options** button to open the **SBR+ Advanced Options** window. On the **Enable** tab, Check GO Blockage Options to enable the **GO Blockage** tab.



On the GO Blockage tab, you have check box options to enable or disable GO Blockage and enable or disable Surface Self-block.

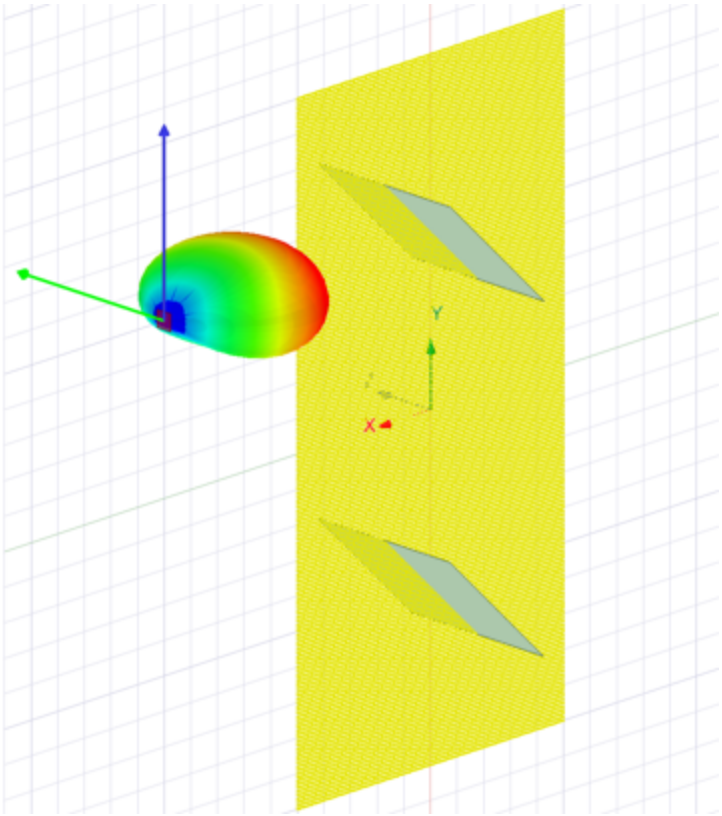


Surface Self-Block: if enabled, the surface where the footprint is radiated from is used for blockage check, that is, anything below the surface with respect to the incoming ray is not in line of sight. If the setting is disabled, the footprint surface is not used for blockage checks

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

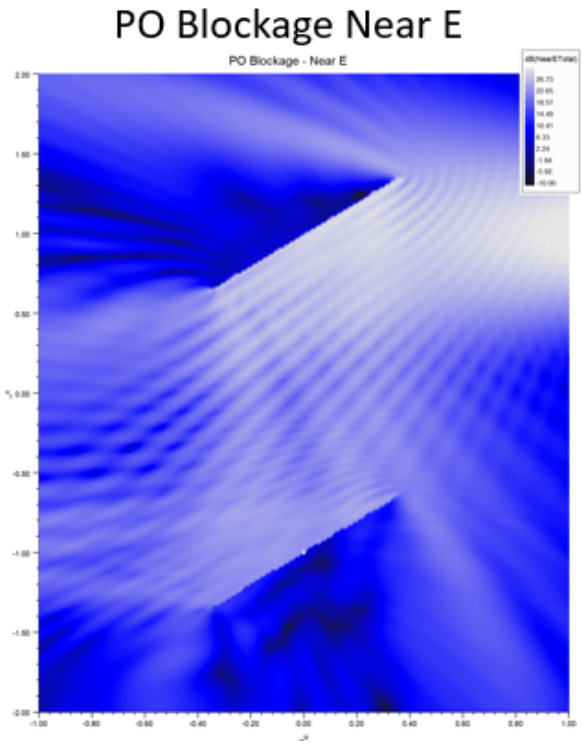
(anything below the surface with respect to the incoming ray is in line of sight), but other surfaces are used for blockage.

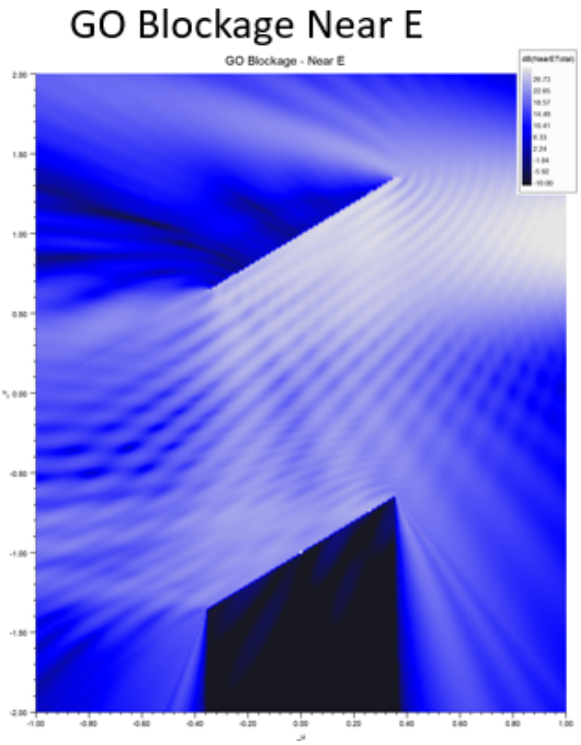
For example, consider a parametric beam antenna illuminating a series of plates.



In the plots, the shadows behind the plates are much more pronounced with GO Blockage enabled, but the fields have non-physical discontinuities.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





- **Enable SBR+ Self Coupling** computation capability for driven designs means that the diagonal S-param term takes into consideration the SBR+ scattering effect, and is more accurate. For a discussion of SBR coupling see, Select Tx/Rx Antennas for SBR+ Solutions.

MLFMM Usage Guidelines

The MLFMM solver option in HFSS enables the use of the Multi-Level Fast Multipole Algorithm to increase the efficiency of the iterative solution of IE problems in FE-BI and IE-Region problems. This document provides a brief description of the solver and guidelines for its use.

- The MLFMM Solver
- Further Details of the MLFMM Algorithm
- Guidelines for Choosing Between ACA and MLFMM IE Solver Options

The MLFMM Solver

The MLFMM solver is based on a hierarchical algorithm that relies on the underlying physics of electromagnetic interactions to provide a more efficient scheme to compute the matrix vector product that is at the heart of every iterative IE solver. The algorithm views a matrix vector

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

product as the computation of E and H fields and it computes these fields using a multi-level multi-pole expansion.

The algorithm can be conceptually broken down in to the following 6 steps.

1. Hierarchical decomposition of the geometry into a set of nested boxes (Figure 1) and identification of near and far interactions for each box (Figure 2).

   The reason for the separation of near and far interactions is to be able to efficiently compute far interactions using multi-poles. See additional explanations in Further Details of MLFMM Algorithm.

2. Computation of multi-poles at the center of each box at the lowest level due to the currents contained in the boxes (Basis functions projected on to multipoles)

   In HFSS, the multi-poles are plane waves sampled on the surface of a sphere of diameter equal to the diagonal of the box and centered at its center.

3. Hierarchical computations of multi-poles at the center of each "higher" level box due to multi-poles at the centers of its children (Up-tree traversal).

   Since the multi-poles are effectively plane-waves, this step is equivalent to resampling plane waves on a larger sphere using spherical interpolation.

4. Construction of fields at each box center at the "highest" level due to all multi-poles at this level (Translation).

   The fields at the receiving box can be again computed on a per-multi-pole (per-plane-wave) basis using the addition theorem for the Green's function. The fields at the receiving box center are, thus, again represented using plane-waves.

5. Hierarchical computation of fields at the center of each "child" box due to the "received" fields at its parents (Down-tree traversal)

   Due to the plane-wave nature of the multi-poles, this step is equivalent to a "down-sampling" from a sphere at the parent level to one at the child level.

6. Computation of fields at the observer locations inside each lowest level box (local fields projected to testing function)

   This is done using integration (summation) over all the plane-waves to compute the field, followed by testing using appropriate testing functions.

The first step, the computation of the absolute "near-field" and all RHS-independent pre-computations possible for steps 2 - 6 are performed during the "Matrix Assembly" stage and the remaining computations for steps 2 - 6 are performed during the "Iterative Solution" stage (Figure 3).

Due to the way the algorithm is constructed the MLFMM solver will take significantly less amount of time than ACA for "Matrix Assembly". It is, however, possible that the MLFMM solver may take

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a larger amount of time than ACA per iteration for "Matrix Solution". Further, due to its construction, MLFMM algorithm tends to use significantly less memory than the ACA solver for large problems.

Further Details of the MLFMM Algorithm

Near-field interactions for a given reference box are defined to be between itself and the boxes that it touches at the same level. Far-field interactions for a given reference box are computed explicitly for all boxes it does not touch which reside in its parent's near field. The hierarchical nature of the algorithm automatically provides the near-field interactions at level n+1 when computing the various interactions at level n. This can be inferred by studying Figure 1 and 2. An outcome of this fact is that only the lowest level contains explicit near-field interactions which are done using standard MoM. Further, note that the far-field interactions between a reference box and boxes which are further away than the near-field of its parent are accounted for by the far-field interactions of its parent (and higher level parents if they exist).

Figure 1: shows a cross-section of "hierarchically nested" boxes overlaid on the mesh of an arbitrarily shaped sheet.

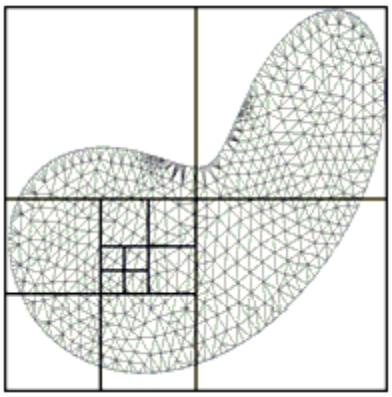


Figure 2: shows the decomposition of boxes into "near" and "far" fields at levels 1, 2, and 3. In each case, a reference box is shown in green and its immediate parent is shown with a red border. The orange color indicates boxes in the near-field of the reference box and the blue color indicate boxes in the far-field at each level. Note that the "near field" exists only at level 1.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

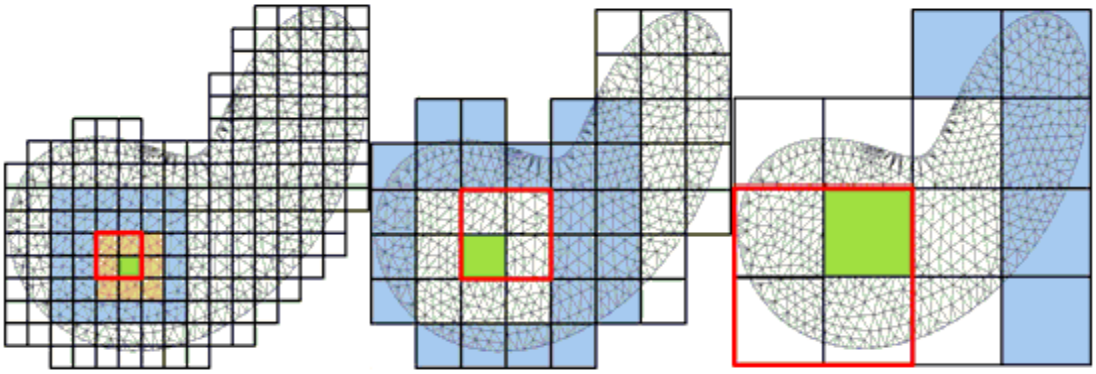
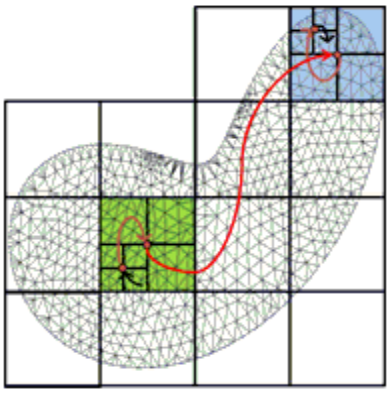
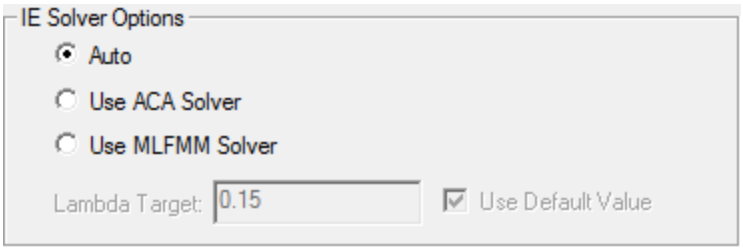


Figure 3: illustrates steps 2-6 in the MLFMM algorithm above. The black arrows indicate the multipole creation from basis functions and construction of the tested field from multipole expansions (steps 2 and 6). Orange arrows indicate the up-tree and down-tree traversals (steps 3, 5), and the red arrow indicates the translation of multipoles to its far field (step 4).



Guidelines for Choosing Between ACA and MLFMM IE Solver Options

The following guidelines have been arrived at by analyzing the features of the MLFMM algorithm and comparing its performance to ACA for several test problems.



1. Problem size

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In general, the **ACA solver will be more efficient** than the MLFMM solver in terms of both memory and solution time for problems whose scattering surface(s) has (have) **less than 100,000 surface unknowns (~100,000 triangles)**.

2. Aspect Ratio (largest dimension : smallest dimension):

In general, the **ACA solver will be more efficient** in memory and speed than MLFMM for problems whose geometry has a **bounding box with a highly skewed aspect ratio (on the order of 100:1 or higher)**.

3. Design Type

HFSS-IE

Number of Excitations:

For problems having **very few excitations (<10)**, the **MLFMM solver will be more efficient** than ACA in both speed and memory for problems that satisfy items 1 and 2 above. Since the time required for matrix multiplication is higher in MLFMM, **as the number of excitations increases** (e.g., mono-static RCS problems), the # of unknowns in the problem needed to realize the increase in speed provided by MLFMM (vs. ACA) will increase (with respect to the limits provided in 1, above). However, **the memory savings provided by MLFMM will remain consistent** even while the number of excitations increases.

HFSS - FE-BI

For FE-BI surfaces involving **few excitations the MLFMM solver will provide similar or slightly better performances in solution time** and **better performance in memory** for problems that satisfy items 1 and 2 above.

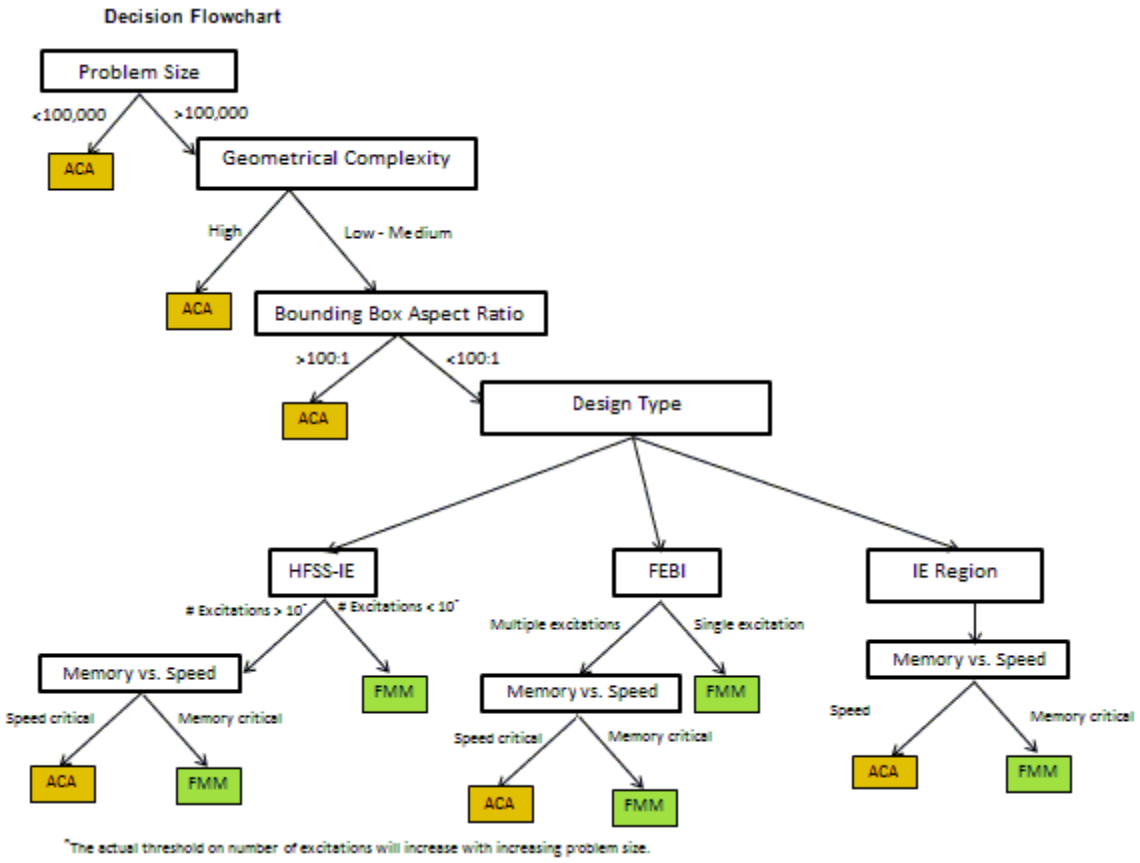
For FE-BI surfaces involving **a large number of excitations** (e.g., several ports, mono-static RCS problems), the **MLFMM solver's solution time may be larger than that for ACA but the memory savings** will remain.

IE Regions

For IE region problems the **ACA solver will nearly always be faster than the MLFMM solver**. However, the **MLFMM solver will provide memory savings** for problems satisfying criteria 1 & 2 above.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Decision Flowchart for Deciding on the IE Solver Options



Setting the Maximum Number of Passes

The **Maximum Number of Passes** value is the maximum number of mesh refinement cycles that you would like HFSS to perform. This value is a stopping criterion for the adaptive solution; if the maximum number of passes has been completed, the adaptive analysis stops. If the maximum number of passes has not been completed, the adaptive analysis will continue unless the convergence criteria are reached.

To set the maximum number of passes for an adaptive analysis:

- Under the **General** tab of the **Solution Setup** dialog box, enter a value for **Maximum Number of Passes**.

For driven problems and no frequency sweeps, HFSS requires at least one adaptive pass. Entering **1** will bypass adaptive analysis, generating a solution only at the solution frequency you specified.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The size of the finite element mesh — and the amount of memory required to generate a solution — increases with each adaptive refinement of the mesh. Setting the maximum number of passes too high can result in HFSS requesting more memory than is available or taking excessive time to compute solutions.

To shorten the solution time for solutions with a frequency sweep, you can set the **Maximum number of passes** to 0 to skip the adaptive pass phase during simulation.

Selecting 0 passes for a setup means that when you create a Report or a Field Overlay, in the Context field, the Solutions menu will not list ‘Adaptive Passes’ and ‘Last Adaptive’ for those setups. However, you can still plot the fields based on Sweep solutions if ‘Save field’ is turned on. Also, you can plot meshes using ‘Last Adaptive’ and this ‘Last Adaptive’ should be initial mesh or mesh link source mesh plus user defined mesh (if any).

If you change **Maximum number of passes** from non-zero to 0, then all plots or reports related to Last Adaptive or Adaptive pass are deleted once you click OK in the Driven Setup window. You can Undo this to recover if this was done in error. Messages in the Message window describe these actions.

If you choose 0 as the **Maximum number of passes**, the Solution Profile reports that analysis does not perform adaptive meshing.

Setting the Maximum Delta S Per Pass

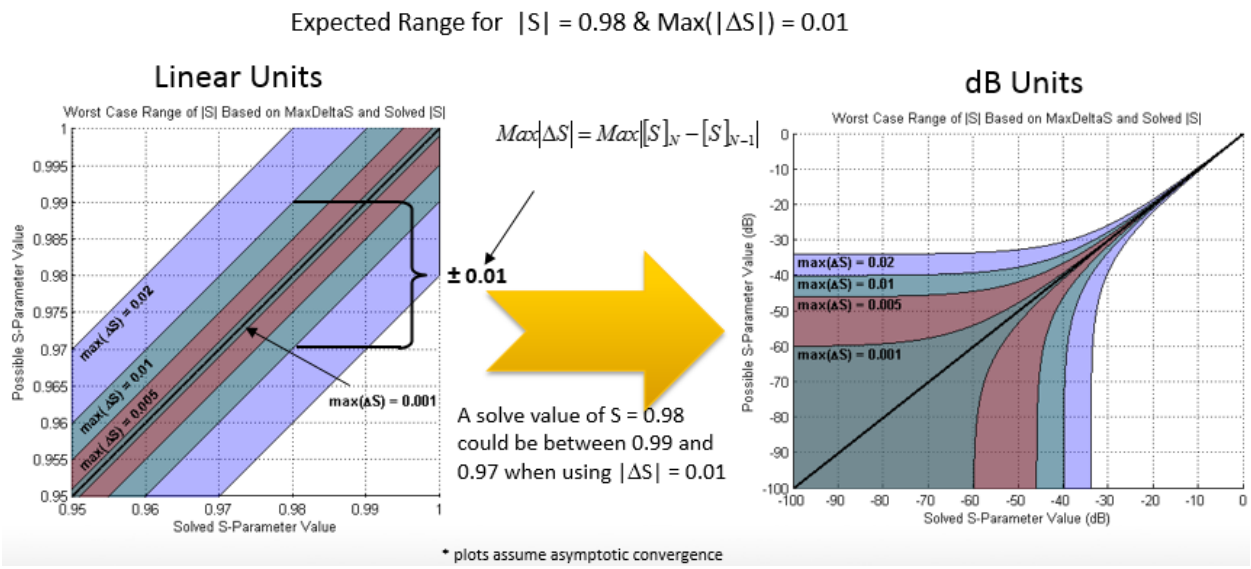
For designs with ports or Transient Solutions for Device Characterization.

The delta S is the magnitude of the change of the S-parameters between two consecutive passes. The solver reports the worst case violation. The value you set for **Maximum Delta S** is a stopping criterion for the adaptive solution. If the magnitude of the change of all S-parameters are less than this value from one iteration to the next, the adaptive analysis stops. Otherwise, it continues until the requested number of passes is completed. The default value is reasonable for most cases. Setting the Maximum Delta S too small wastes computer resources and time. Setting it too large jeopardizes accuracy. Always set the Delta S with your error tolerance in mind.

Consider the worst case scenarios when setting the Maximum Delta S.

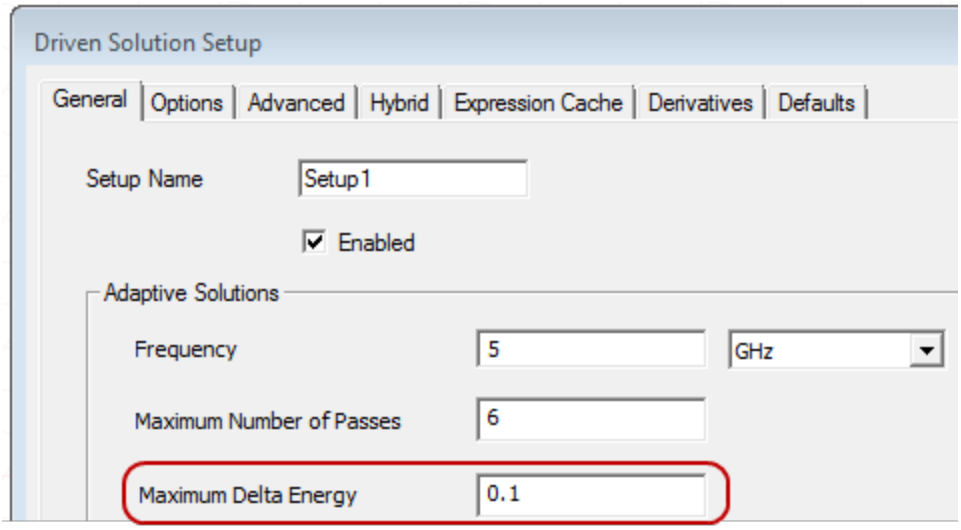
- Determine the worst case magnitude error by assuming the S-parameter magnitude is the only source of error (phase is perfectly accurate).
- Most likely the actual solution is much closer since the error will be split between the magnitude and the phase solve value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To set the maximum delta S per adaptive pass:

- Under the **General** tab of the **Solution Setup** dialog box, enter a value for **Maximum Delta S**.



Delta S data is available only after HFSS completes two iterations of the adaptive analysis process.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Delta S is computed on the appropriate S-parameters - modal or terminal - after the S-parameters have been de-embedded and renormalized.

Setting the Maximum Delta Energy Per Pass

For designs with voltage sources, current sources, incident waves or magnetic bias or Transient Solutions for Field Visualization.
Not applicable to designs with ports.

The delta Energy is the difference in the relative energy error from one adaptive solution to the next. The value you set for **Maximum Delta Energy** is a stopping criterion for the adaptive solution. If the delta Energy falls below this value, the adaptive analysis stops. Otherwise, it continues until the convergence criteria are reached.

To set the maximum delta Energy per adaptive pass:

- Under the **General** tab of the **Solution Setup** dialog box, enter a value for **Maximum Delta Energy**.

Delta Energy data is available only after HFSS completes two iterations of the adaptive analysis process.

Setting the Maximum Delta Frequency Per Pass

For Eigenmode solution types

The delta Frequency is the percentage difference between calculated eigenmode frequencies from one adaptive pass to the next. The value you set for **Maximum Delta Frequency Per Pass** is a stopping criterion for the adaptive solution. If the eigenmode frequencies change by a relative amount less than this value from one pass to the next, the adaptive analysis stops. Otherwise, it continues until the maximum number of passes is completed.

Maximum Frequency Delta Per Pass = max | F_current - F_previous|/|F_previous|

To set the **Maximum Delta Frequency Per Pass**:

- Under the **General** tab of the **Solution Setup** dialog box, enter a value for **Maximum Delta Frequency Per Pass**.

Delta Frequency data is available only after HFSS completes two iterations of the adaptive analysis.

Specifying Convergence on Real Frequency Only

For Eigenmode solution types.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting **Converge on Real Frequency Only** causes the percent difference calculation among a set of frequencies to be based only on the real parts of the frequencies; the imaginary parts of the frequencies are ignored. When the project contains only lossless material, the eigen frequency will be real, so checking this option does not make any difference. For lossy problems, selecting this option means the solver checks delta frequency on real part only as the eigen mode frequency in this case will be complex.

- Under the **General** tab of the **Solution Setup** dialog box, select **Converge on Real Frequency Only**.

Specifying Expressions for Adaptive Convergence

You can specify additional convergence criteria through the use of expressions and output variables. The **Max Delta** or the **Max Percent Delta** defined for expression convergence represents the difference in values of the expressions between consecutive adaptive passes. If the difference in the value of the expression between consecutive passes is less than the **Max Delta** or the **Max Percent Delta** value, this part of the convergence criteria is satisfied.

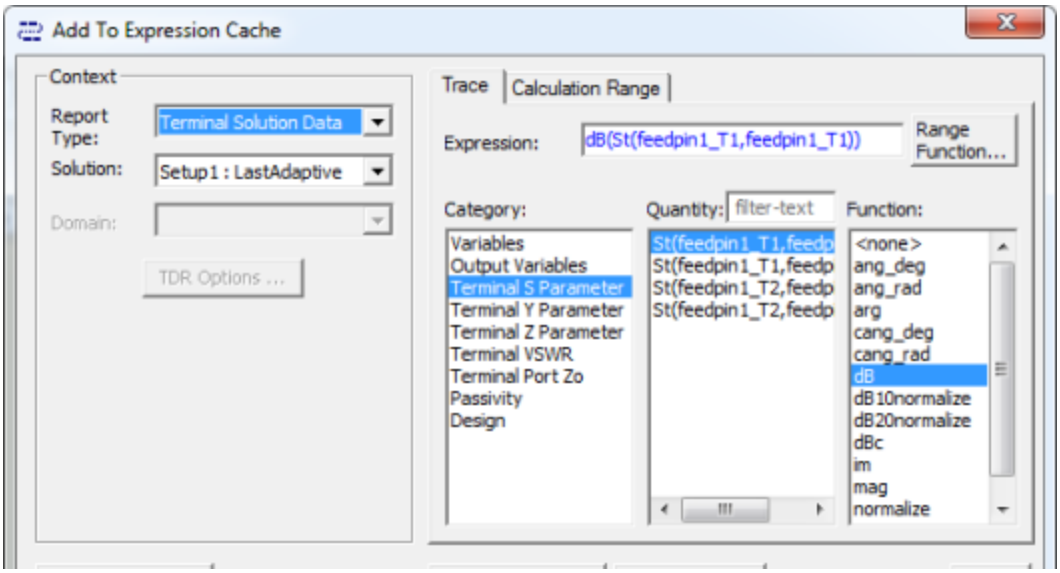
- For **driven solutions**, if the **Maximum Delta S**, **Maximum Delta E**, or alternate matrix convergence criteria are achieved in addition to any specified expression convergence criteria, the adaptive analysis stops. Otherwise, the solution continues until the requested number of passes is completed.
- For **eigenmode solutions,** if the **Maximum Delta Frequency Per Pass** criteria is achieved in addition to any specified expression convergence criteria, the adaptive analysis stops. Otherwise, the solution continues until the requested number of passes is completed.

To set expressions as **Convergence** criteria:

1. Double-click the Analysis setup icon in the Project tree to open the **Solution Setup** dialog.

2. Click the **Expression Cache** tab in the **Solution Setup** and click **Add...** to open the **Add to Expression Cache** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Specify the context for the expression you define. For Modal Solution Data, you do not need context other than the Solution. Other selections require more context:
   - Emission Test also requires a digital signal.
   - Far Field also requires a geometry such as an infinite sphere.
   - Fields or Near Fields also require a geometry such as a polyline.
4. Under the **Trace** tab, select from the Category, Quantity and Function lists to create expressions.

   Selecting a listed category lists the Quantities and Functions available for each category. If you have defined one or more output variables, you can see them listed as Quantities by selecting the **Output Variables** Category. The Output Variables button opens a dialog that lets you define additional output variables.

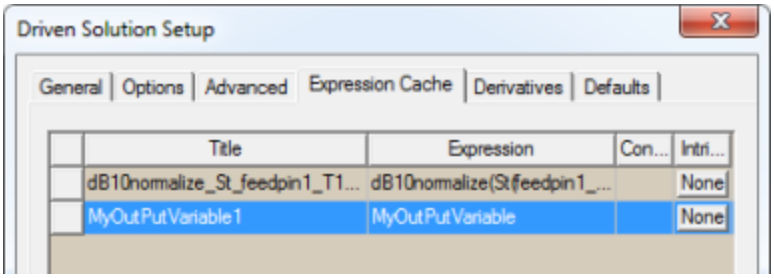
   When you have created an expression, it appears in the Expression field of the **Trace** tab. If desired, you can use the Range Function button to select range functions to apply to the expression.

   Under the **Calculation Range** tab, you can view the values of available sweep variables. Clicking the ellipsis [...] button in the Edit column opens a list of values.
5. When you have created an expression that you want to add to the cache, click **Add Calculation**.

   This adds the selected expression and the associated context to a table in the **Expression Cache** tab. You can select any additional expressions with contexts and add them in the same way.
6. When you have added the expressions you want, click **Done** to close the **Add to Expression Cache** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Expression Cache** tab of the **Solution Setup** lists the expressions you have added as a table.



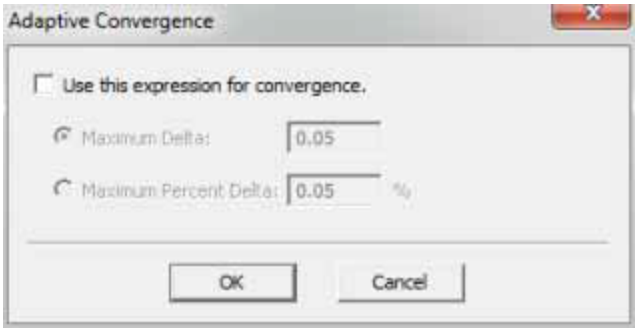
- The Title field is editable, by default showing the name as built from the expression, but removing underscores.
- The Expression field shows the full expression. If necessary, you can resize the **Solution Setup** dialog box. You can also resize each column in the table.
- The Context column shows None for Modal solutions, or the appropriate geometry for Fields calculations.
- The Intrinsics column shows a clickable button that opens an **Edit Calculation Range** dialog box. If the column button shows None you cannot edit the value. If the button shows variables, click the ellipsis [...] in the Edit column to display a list of the variable values that you can select. Click **OK** to close the **Edit Calculation Range** dialog box and apply your selections to the Expression Cache.
- The Convergence column contains a check box that lets you designate post processing variables, such as for renormalization or de-embedding. You can also use this feature to assign variables to non-model objects (for example, the properties of a solid, sheet or lines used for field calculations). You can also assign a PP Variable to a coordinate system (CS) that is not associated with model objects.

To excite particular sources for particular output variables, you can assign a post process variable to the magnitude of sources you might want to edit, and assign the expression value in the Expression Cache. With PP Variables, you can control convergence based on several combinations of active sources.

7. To designate one or more expressions for convergence, click the field for the Convergence column for each expression.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This opens an **Adaptive Convergence** dialog box.



Check **Use this expression for convergence** to enable the radio buttons. You can then specify the **Max Delta** between passes or the **Max Percent Delta** criteria. The **Max Delta** solves a potential issue if your expression is essentially zero and the numeric noise from pass to pass causes the maximum percentage delta to remain high. In that case adaptive refinement continues until you get to the maximum number of passes.

> **Note:** If you check **Use this expression for convergence**, the pass will be considered converged if both the energy error **and** the expression cache results have converged, not just expression cache.

8. Click **OK** to close the **Solution Setup** window.

Specifying a Source for the Initial Mesh

You may choose to specify a source for the initial mesh from either the current design or another design. The source mesh should represent a geometrically equivalent model.

> **Note:**
>
> For more information see Mesh Linking and Mesh Link Requirements.

To specify a source for the initial mesh:

1. Under the **Advanced** tab of the **Solution Setup** dialog box, click the check box for **Import Mesh**.

   This displays the *Setup Link* dialog box. By default, the **Source Project** check boxes for **Use This Project** and the **Source Design** check box for **Use This Design** are selected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> When you setup a link to an external source, the **Lamda refinement** option is deselected under the **Options** tab to avoid over-refinement of the linked mesh.

If you uncheck **Use This Project**, you enable fields for selecting a different project and radio buttons to specify whether to save the source path relative to **The project directory of the selected project** or **This project**.

2. To specify a Source Project file click the ellipsis [...] button to open a file browser window.

   When you selected the project, click the **Open** button to accept the project file for the setup. You can use the check box to **Open as read only**.

   Use the radio buttons to specify whether to save the source path relative to **The project directory of the selected project** or **This project**.

   When you select a Project File, the Design field and the Solution field are filled in with default values, and the drop down menus contain any available Projects and solutions.

   The "Default" solution is the product dependent solution of the first Setup. That is the setup listed first in the source design's project tree (alphanumerical order). A product specific solution of this setup becomes the default solution. In most products, it is LastAdaptive. In a Transient solution type, it is "Transient."

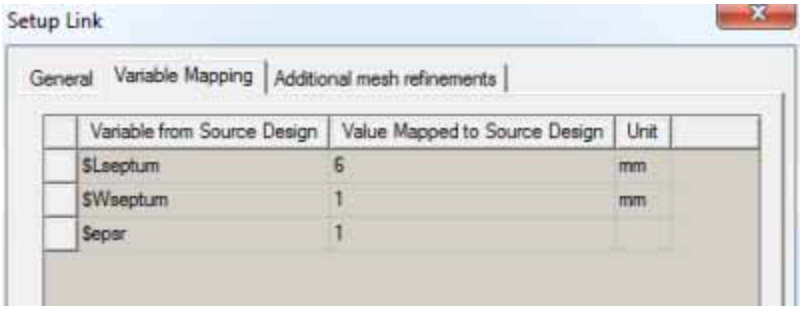
   > **Note:**
   >
   > The solution in the source design must provide data for the target design's adaptive frequency as well as its sweeps. That is, the adaptive frequency for the target design must be included in the sweep in the source design.
   >
   > If necessary, you can open the source design and add an appropriate frequency point to an existing sweep.

3. For the Source Design, if you leave **Use This Project** checked, you can uncheck **Use This Design** to enable the a drop-down menu for to select from other available designs.
4. For the **Source Solution** field, you can use the drop-down menu to select from other possible solutions.
5. Use the check box specify whether to **Simulate the source design as needed** to produce the mesh.
6. Use the check box to specify whether to **preserve the source design solution**. Note that in the **Extractor mode**, the source project will be saved upon exit. **Extractor mode** means that the software is opened during the link solely for the purpose of solving.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. The second tab, **Variable Mapping**, lets you view any variables contained in the Project you select.

   When there are variables in the source design, you can choose to "map" these variables to constant values, expressions or variables in the target designs. Variable mapping becomes more important when the datalink type requires source and target design to be geometrically identical and source design is geometrically parameterized.



   When a variable in the target design has the same name of a source design variable but the 2 variables are not mapped, the **Parameter** tab will become the active tab with the following message box popped.

   You can exit the **Setup Link** dialog directly by selecting "Accept Setup and Exit Dialog" or reexamine the parameter mapping by selecting "Return to Setup Dialog".

   In the **Setup Link** dialog box, for linked designs with variables of the same name, you can click **Map Variables by Name** to automatically map same named variables. A variable that does not have a same named counterpart, and retains its value in the source design.

   Click **OK** to proceed without the variable mapping.

8. You can select the **Additional mesh refinements** tab to specify the following:
   - For Mesh Operations, you can select radio buttons to either **Apply mesh operation in target design on the imported mesh**, or **Ignore mesh operations in target design** (default).
   - For *Port Adapt*, you can check **Perform port adapt in target solve setup**.
9. Click **OK** to accept the setup and close the **Setup** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clearing Linked Data

Linked data can be a mesh, field results, or some other post-processing data that the source design generated. The target design for the link caches the data internally to minimize the need to activate the source design.

If you have previously setup links to a design, the **HFSS**> **Analysis**> **Setup** shortcut menu contains an option to **Clear Linked Data**. This removes the linked data for all links in a design, therefore invalidating the solutions. You can also clear linked data through **HFSS**> **Results**> **Clean Up Solutions**, which displays a dialog box that includes options to let you selectively delete linked data only, or as part of other deletions.

Clearing linked data for some link types requires HFSS to revert to the initial mesh. Thus in some cases, this command removes the current mesh of the target design.

Mesh Linking

You can obtain a broadband mesh by linking between designs as follows:

- Link a mesh from an Eigenmode solution to a Driven Modal or a Driven Terminal design.
- Use a cascade of mesh links within the same design where each link adapts the mesh at a different frequency.

Mesh Link Requirements

If you want to reuse a mesh from an existing source design, make sure that the geometries in the target and source are identical. This is a requirement for a mesh link. There are two ways to generate geometrically equivalent models.

- Copy the entire source design and paste it in the **Project Manager** window to create the target design.
- Use **Import from Clipboard** as follows:

1. In the model window of the source design, press **Ctrl+A** to select all the objects.
2. Right-click the Modeler field and select **Edit>Copy** from the short-cut menu.
3. On the target design, go to **Modeler>Import from Clipboard** to paste in the design.

After you paste using one of the above methods, create a solution setup in the target, select **Import Mesh** on the **Options** tab, and link the mesh from the **Setup Link** dialog box.

> **Note:**
>
> For more information about linking the mesh, see Specifying a Source for the Initial Mesh.

Setting Lambda Refinement

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lambda refinement is the process of refining the initial mesh based on the material-dependent wavelength. It is recommended and selected by default.

HFSS performs solution-adaptive mesh refinement to produce accurate results. Lambda Refinement is necessary to avoid false convergence and achieve accuracy to the field data, especially for electrically large problems (approximately greater than 2*lambda). The Lambda refinement process helps simulation start with the optimal balance between element count and element size. For efficient convergence, the initial mesh should meet some element electrical size requirements determined by lambda refinement. Typically, a few elements per wavelength are needed for accurate results. Because wavelength depends upon the material, the element length target differs for each material. You can specify the size of the element target by which HFSS refines the mesh in the Lambda Target field or select the recommended Use Default Value. The default value in the Lambda Target field also depends upon the order of the basis functions. For example, if the lambda target is 0.3333, the initial mesh is refined such that on each solid the length of the elements are approximately smaller than 0.333*wavelength. Lambda refinement stops if the input mesh already reaches the Lambda refinement target.

If you use the **Advanced** tab to link to a mesh from another design or project, **Do Lambda Refinement** is unchanged.

To specify the size of the target value by which HFSS will refine the mesh:

1. From the **Solution Setup** window, select the **Options** tab and enable **Do Lambda Refinement**.

   This enables the **Target** field and the **Use free space lambda** check box. It also enables Lambda refinement from the background material for the IE solver. See: IE Solver Options area on the Advanced tab.

2. In the **Target** field, enter a value for the wavelength or accept the default.

   The **Target** defaults depend on the Order of Basis function selections. For example, for Driven solutions and a First Order basis function, the default target is 0.3333, which means that HFSS will refine the mesh until most element lengths are approximately one-third wavelength.

   For Eigenmode solutions and a First Order basis function, the default target is 0.2

   If you change the Order of Basis functions in the **Solution Setup** window, the default changes automatically. Setting the Order of Basis affects the default value of the lambda refinement in the solution setups as follows.

Zero order:	driven 0.1	eigenmode 0.1
First order:	driven 0.3333	eigenmode 0.2 (as is)
Second order:	driven 0.6667	eigenmode 0.4
Mixed order	driven 0.6667	eigenmode 0.6667

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. If you want the initial mesh to be refined based on the wavelength in free space, select **Use free space lambda**. Material-dependent lambda refinement will be deactivated.

> **Note:**
>
> Changing the Lambda refinement target invalidates any solutions that were performed with the previous lambda refinement.
>
> If an object is highly conductive, very little energy will penetrate into it, so it does not need a dense mesh. In such cases, free space mesh is applied.

Setting the Percent Maximum Refinement Per Pass

The value you set for percent **Maximum Refinement Per Pass** determines how many tetrahedra are added at each iteration of the adaptive refinement process. The tetrahedra with the highest error will be refined. The default value is 30%.

To set the percent refinement per adaptive pass:

- Under the **Options** tab of the **Solution Setup** dialog box, enter a value for percent **Maximum Refinement Per Pass**.

Setting the Maximum Refinement

This specifies the maximum number of tetrahedra that can be added during an adaptive pass. By default, this unchecked, to that there is no maximum. If you enable the **Maximum Refinement**, the initial value is 1000000.

To set a new value for the Maximum Refinement:

1. Under the **Options** tab of the **Solution Setup** dialog box, click the **Maximum Refinement** check box to enable the text field.
2. Enter the number of tetrahedra for **Maximum Refinement**.

You can also control these values in the docked properties window that displays when you select the setup with the **View>Properties** enabled. With the properties window displayed. you can click the check box for **Use Max Refinement** to apply the value in the Max Refinement text field.

Setting the Minimum Number of Passes

An adaptive analysis will not stop unless the minimum number of passes you specify has been completed, even if convergence criteria have been met.

- Under the **Options** tab of the **Solution Setup** dialog box, enter a value for **Minimum Number of Passes**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> For a solve setup with zero passes, no sweeps, and that is not ports only, validation produces a warning message.

Setting the Minimum Number of Converged Passes

An adaptive analysis will not stop unless the minimum number of converged passes you specify has been completed.

- Under the **Options** tab of the **Solution Setup** dialog box, enter a value for **Minimum Converged Passes**.

The convergence criteria must be met for at least this number of passes before the adaptive analysis will stop.

Setting Matrix Convergence Criteria

For designs with ports.

You can specify different stopping criteria for specific entries in the S-matrix. This is done in the **Matrix Convergence** dialog box. The adaptive analysis will continue until the magnitude and phase of the entries change by an amount less than the specified criteria from one pass to the next, or until the number of requested passes is completed.

To set the matrix convergence:

1. Under the **General** tab of the **Solution Setup** dialog box, select **Use Matrix Convergence**.
2. Click **Set Magnitude and Phase**.

   The **Matrix Convergence** dialog box appears.

3. Select one of the following from the **Entry Selections** drop-down menu:

All	Sets all of the matrix entries at once. (The default).
Diagonal/Off-Diagonal	Sets all of the diagonal matrix entries at once, all off-diagonal matrix entries at once, or both diagonal and off diagonal entries at once.
Selected Entries	Sets individual matrix entries that you will select.

   For the selection **All**, enter the convergence criteria for the **Maximum Delta (Mag S)** and the **Maximum Delta (Phase S)** in the fields to the right.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the selection **Diagonal**/**Off-Diagonal**, first check **Diagonal Entries**, **Off-Diagonal Entries**, or both, to enable the convergence criteria field or fields. Then enter the convergence criteria for the **Maximum Delta (Mag S)** and the **Maximum Delta (Phase S)** in the fields to the right.

For both of these **Entry Selections**, you can set the **Ignore Phase when Mag is less than** value. See the note on technical issues under step 4 c.

4. If you chose **Selected Entries**, the Matrix Convergence dialog displays some new fields:
   - a table showing columns for Matrix Entry 1, Matrix Entry 2, and the Delta Mag and Delta Phase.
   - **Entry 1** and **Entry 2** fields, which contain drop-down menus of ports and associated modes (or terminals).
   - an **Insert** button with which to move selections from the port list selections to the table

   To select the desired ports and mode (or terminal) pairs, do the following:

   a. Select Entry 1 and Entry 2 from their drop-down menus.
   b. In the **Magnitude** box, enter the maximum change in magnitude from pass to pass from the Entry 1 to Entry 2.
   c. In the **Phase** box, enter the maximum change in phase, in degrees, from pass to pass from Entry 1 to Entry 2.

      > **Note:** When the Mag S becomes small (near to zero) its phase becomes indefinite and insignificant due to mathematical issues. In HFSS 10 and 11 there was a 0.05 magnitude threshold that caused that Phase Margin to be discarded. However, some users may want to continue running additional passes to stabilize phase margins even when the S-parameter magnitude is below this threshold. For this reason, the magnitude threshold has been removed. This calls for caution if you want to see a good phase convergence for S_{12}, which is near to 1. If you set a small delta_phi with small delta_s, after a couple of passes, S_{12} will converge, but S_{11} never, since S_{11} ~0 and its phase changes with the mesh noise because the phase is indefinite. In other words, under these conditions the adaptive process never stops, so you should abort it.

   d. Click **Insert.**

      The entries appear in the table above. If you have selected multiple entries, all combinations of matrix entry1 and matrix entry2 populate the table.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting a Row in the table enables the Delete button, if you need to remove a row from the table.

Clicking in the Delta Mag and Delta Row fields of the selected row enables editing in those fields.

5. Click **OK** to apply the values and close the dialog.

Setting the Order of Basis Functions

You can change the basis functions HFSS uses to interpolate field values from nodal values.

- Under the **Options** tab of the **Solution Setup** dialog box, select **Order ofBasis**.

This can be First Order (the default), Zero Order, Second Order, or Mixed order.

Setting the Order of Basis functions affects the default value of the Lambda Refinement in the Solution setups as follows.

Zero order:	driven 0.1,	eigenmode 0.1
First order:	driven 0.3333,	eigenmode 0.2 (as is)
Second order:	driven 0.6667,	eigenmode 0.4
Mixed order	driven 0.6667	eigenmode 0.6667

The Zero order option is useful when a model requires a mesh that produces more than 100,000 tetrahedra, but the model size is small compared to wavelength. The higher order options solve progressively more unknowns for each tetrahedron. Mixed order uses higher order where more accuracy is required, and lower order where fields are weaker.

> **Warning:**
>
> If you select **Zero Order Solution Basis**, all tetrahedra in the model must have edge lengths less than $1/20^{th}$ wavelength. Thus, this option is usually selected in combination with a specific lambda refinement setting.

Enable Direct Solver

The direct solver provides a multi-frontal (MF) approach when a matrix is not well-conditioned for an iterative solution. The iterative solver significantly reduces memory usage, and it can also provide a savings in the solution time for large simulations.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you select the **Enable Direct Solver** option, HFSS automatically invokes the multi-frontal solver. If you select **Iterative Solver** and HFSS decides that the matrix is not conditioned well enough to take advantage of the iterative approach HFSS still uses the multi-frontal solver.

Enable the Direct Solver

From the **Solution Setup** window, select the **Options** tab. Then select **Direct Solver** from the *Solution Options* group box.

> **Note:** The Iterative Solver is not available for zero order basis solutions.

Enable Iterative Solver

The iterative solver provides an alternative to the multi-frontal solver when a matrix is well-conditioned for an iterative solution. The iterative solver significantly reduces memory usage, and it can also provide a savings in the solution time for large simulations.

When you select **Enable Iterative Solver**, HFSS automatically invokes the iterative solver when it decides that the matrix is conditioned well enough to take advantage of the iterative approach. HFSS uses the multi-frontal solver if the matrix does not meet this requirement.

For more detail, see the technical notes for Iterative Matrix Solver.

Enable the Iterative Solver

From the **Solution Setup** window, select the **Options** tab. Then select **Iterative Solver** from the *Solution Options* group box.

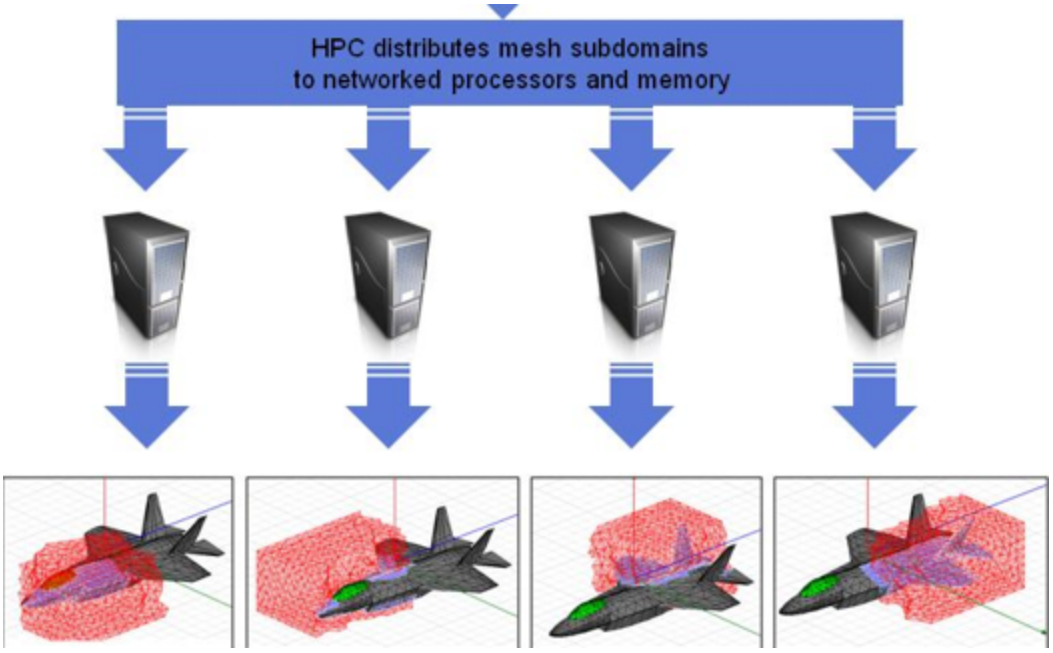
This enables the **Relative Residual** check box. The residual measures the convergence of the iterative solver. Check the box, if desired, and enter a **Relative Residual** value in the adjacent text field (default value is 1E-6).

> **Note:** The Iterative Solver is not available for zero order basis solutions.

Enable Domain Decomposition

If a problem is too large to solve on one machine, HFSS can automatically partition a design into domains that can be solved by separate processes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



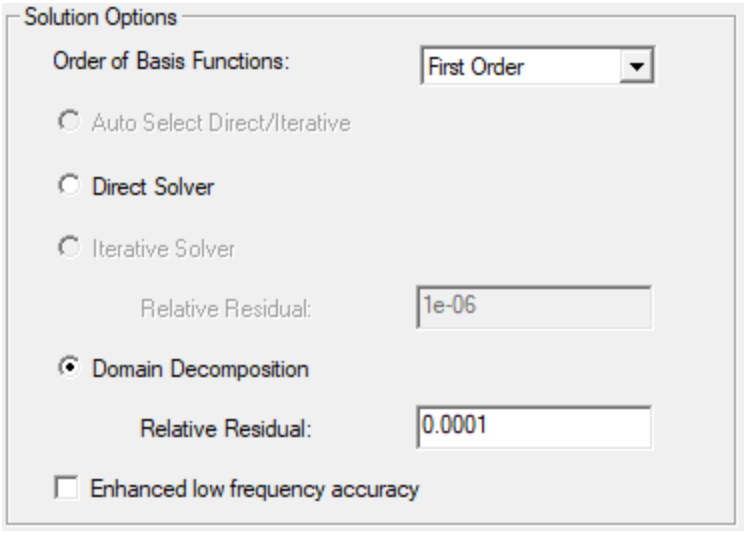
Before enabling solver domains, you must have the HPC license option, and you must have allocated at least three distributed machines to the solve workgroup. The number of domains that the solver creates will not exceed N-1, where N is the number of machines listed in the workgroup. (The first machine in the workgroup acts as the head node and is responsible for domain assembly, mesh refinement, and solution management). If more machines are present in the solve workgroup than are needed, HFSS creates the number of domains that leads to increased overall solver efficiency. Consequently, some machines remain idle if the problem size does not justify their use. See the Technical Note on domain decomposition for further details.

> **Note:**
>
> You can use the Domain Decomposition method and setup HPC to solve a large problem such as a helical antenna. HFSS automatically apportions the helical antenna design into domains and solves them by separate processes. See the discussion associated with the helical antenna example project that can be accessed with **File** > **Open Examples** > **Antenna**.

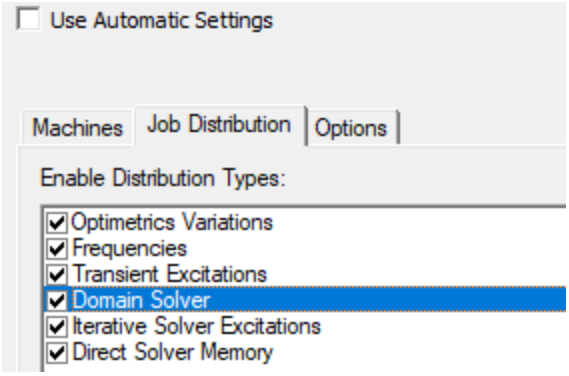
Domain use can be invoked for a solve when

- The **Domain Decomposition** radio button under the Solution Setup Options tab is checked.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- You have the HPC License.
- You have provided at least three distributed machines in the workgroup.
- Domain Solver is enabled in the Analysis configuration, either explicitly (as shown) or by selecting "Use automatic settings".



- The solver determines that the problem is large enough (the mesh has enough tets) to bother with domains.
- The design includes IE Regions and/or FE-BI Radiation Boundaries.

If you have configured your installations properly, the Domain solve can use Distributed Memory solutions. Memory used by the MPI-enabled HFSS solver is therefore limited by the set of machines that are available rather than the shared memory available on any single machine. This allows you to simulate larger structures than before and to optimally reconfigure the cluster of machines for the problem at hand. If an HFSS problem involves solver domains or a finite array, then frequency sweeps will not be done using DSO. Also, DSO for Optimetrics will not be allowed.

Restrictions on solver domains are that the design and analysis setup cannot include:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The design cannot contain primary and secondary (Lattice Pair) boundaries.
- Eigenmode solution type.
- Fast frequency sweeps.

If any condition is not met, the problem is solved with the non-domain solver defined under the solution options. When these conditions are met:

- The solver chooses the number of domains to use, based on the machines available. The number of domains chosen will likely be close to the maximum. The maximum is one less than the total because the first machine in the workgroup is used to control domain iterations.
- The solver creates the domain meshes of roughly the same size
- Domain meshes are created every time the global mesh changes (before each adaptive pass)

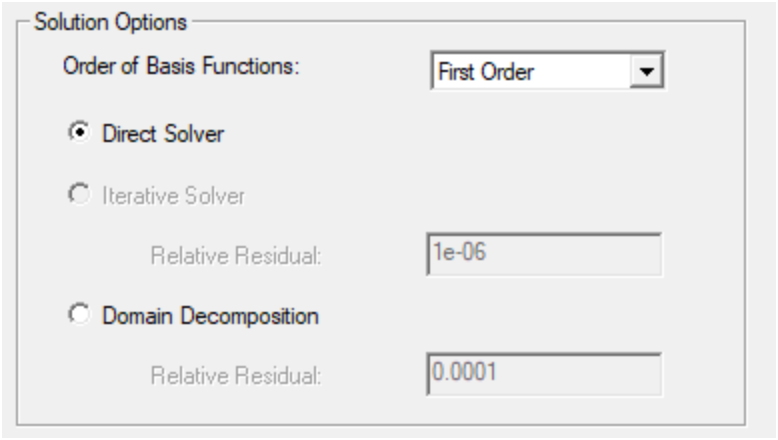
Note:

- You provide resources for the distributed solve by adding machines to the distributed machine workgroup. A machine can appear in the workgroup more than once. You should use this capability to maintain a balanced load.
- Because the domains should be roughly the same size, you should provide balanced resources.

Beta Option for Domain Decomposition Allowing Distributed Direct Solver

This release includes a Beta Option allowing to use the Direct Solver for regular arrays and 3D component arrays. When the option is enabled, and the design is appropriate, the Direct Solver radio under the under the Solution Setup Options tab can be selected. Both non-distributed and distributed direct are used the same way as traditional direct solver. This can be very beneficial in some cases such as non-radiating structures which may a long time for standard Domain Decomposition iterations to converge.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Distributed Right-Hand-Side Solver in Domain Decomposition

For versions prior to HFSS16 in the domain decomposition solver, domains are distributed across separate machines and each right hand side (RHS) or excitation is solved one at a time. For designs with several excitations, this process can take much time. In HFSS16 a distributed right hand side (RHS) solver can use extra machines (if available) and speed up simulation.

Balancing Resources for Domains

The Domain Decomposition option for large models requires that you provide resources for the distributed solve by adding machines to the distributed machine workgroup.

Because the domains should be roughly the same size, you should provide balanced resources. As an example of balancing the load, suppose that machine called Biggie has 256 GB and another called Smalls has 64 GB. If you want up to five domains, it makes sense to enter Biggie in the workgroup four times and Smalls once.

Under Analysis Configuration Options, you should set the **RAM Limit %** to the memory desired for each individual domain.

In the example above, a good choice for Mem_Limit_Soft would be 64 GB.

Remember:

- A machine can appear in the workgroup more than once. The user should use this capability to maintain a balanced load.
- The first machine in the workgroup is used to control domain iterations.
- Under the Analysis Configuration Machine tab, the **Number of Cores** is also set per domain.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the example above, a machine named Biggie has 16 processors and a machine named Smalls has 8 processors. Four domains will go on Biggie, so **Number of Processors** should be set to 4. Four processors on Smalls will be unused.

Recommendation: it is more important to use memory efficiently than to use all the processors.

Domains with FE-BI Radiation Boundaries or IE Regions

The solve uses domain solvers if there is a FE-BI radiation boundary and/or at least one IE Region.

- Solve sequentially using domains if solver domains are not enabled
- Solve distributed using domains if solver domains are enabled, even if mesh not big enough to split into solver domains

Number of domains

- One or more for FEM region (depending on whether using solver defined domains)
- One or more for FE-BI surfaces and dielectric cavity, depending on size and connectivity
- One or more for metallic IE Regions, depending on size and connectivity
- With solver defined domains, determines number of machines

Use Radiation Boundary on Ports

If the design includes wave ports, the **Use Radiation Boundary on Ports** option is enabled under the **Advanced** options tab of the **Solution Setup** dialog box.

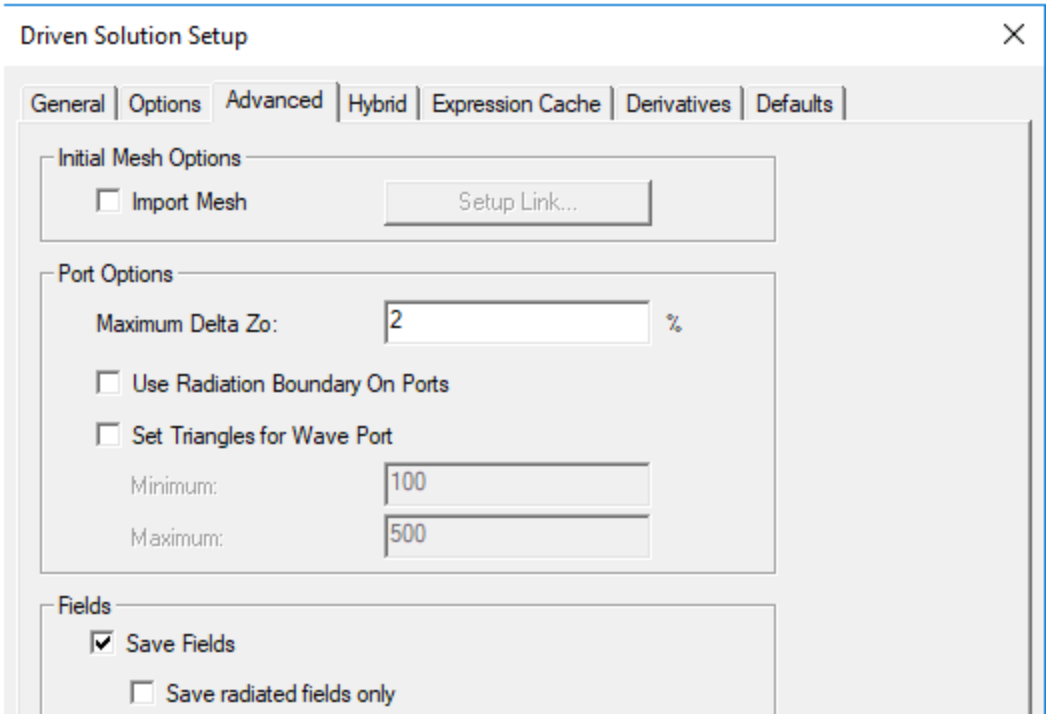
- If you select this setting, edges that touch a port and are assigned to an ABC have a radiation boundary condition applied during the port solution.
- If you do not select this setting, a perfect conducting boundary condition is used during the port calculations.

In most cases this setting has a limited effect on the overall fields or post processed quantities.

Port Options

If the design includes wave ports, the **Port Options** appear under the **Advanced** options tab of the **Solution Setup** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



These options include:

- **Maximum Delta Zo** - change to Zo specified as a target percentage. The default is 2%. The port solver computes mode patterns and propagation constants based on solutions of a finite element system of equations. The finite element mesh associated with each port is a 2D mesh of triangles corresponding to the faces of the tetrahedra that lie on the planar port surface. The port solver performs an iterative refinement of the triangular mesh. During the port solve a 2D adaptive process analogous to the well known 3D process is employed to determine the fields for the port. As we have a max delta S for the overall 3D solution, on the port the convergence criteria is tracking the port impedance, Zo, as a function of these rapid 2D passes to the port. For a technical explanation, see Mesh Refinement on Ports.
- Use Radiation Boundaries on Ports
- **Set Triangles for Wave Port -** unchecked by default.

If you check **Set Triangles for Wave Port**, the **Minimum** and **Maximum** fields are enabled. You can edit the default values of 100 for the minimum and 500 for the maximum.

For designs with lumped ports, this option is not active. Higher numbers of triangles would not benefit a solution setup in this case.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Beta Option: Enhanced Tau Port Meshing

The port solver requires an all planar mesh on ports. If the ports overlap with other objects or faces, occasionally the initial mesher may generate a non-planar mesh or an inaccurate mesh and then the port solver will fail or rejects the mesh. In such situation, you may need to use this Beta feature for accurate port mesh.

It uses a two stage approach to mesh generation. In stage-1, mesh on all the ports or user specified “high priority surface representation” faces will be generated, and strict mesh verification will be applied. Repair methods are involved if an inaccurate port mesh is found. This ensures that a satisfactory port solver driven mesh is created. Then in stage-2, all the remaining objects will be placed to the pool for the mesh. When an overlap or tiny gap is found, these faces will be treated as lower priority objects compared to ports.

Technical Notes: Port Solutions

Produce Derivatives for Selected Variables

The **Derivatives** tab of the **Solution Setup** window allows you to produce derivatives of S-parameters and related matrix quantities, such as Y or Z. The solver can also output partial derivatives of *E* and *H* over radiation surfaces. You can then create reports to calculate the partial derivative of the far fields using of *E* and *H* over the radiation surfaces based on the same formulation as far fields. Thus the tuned far fields are derived from the nominal fields and the partial derivative of the far fields.

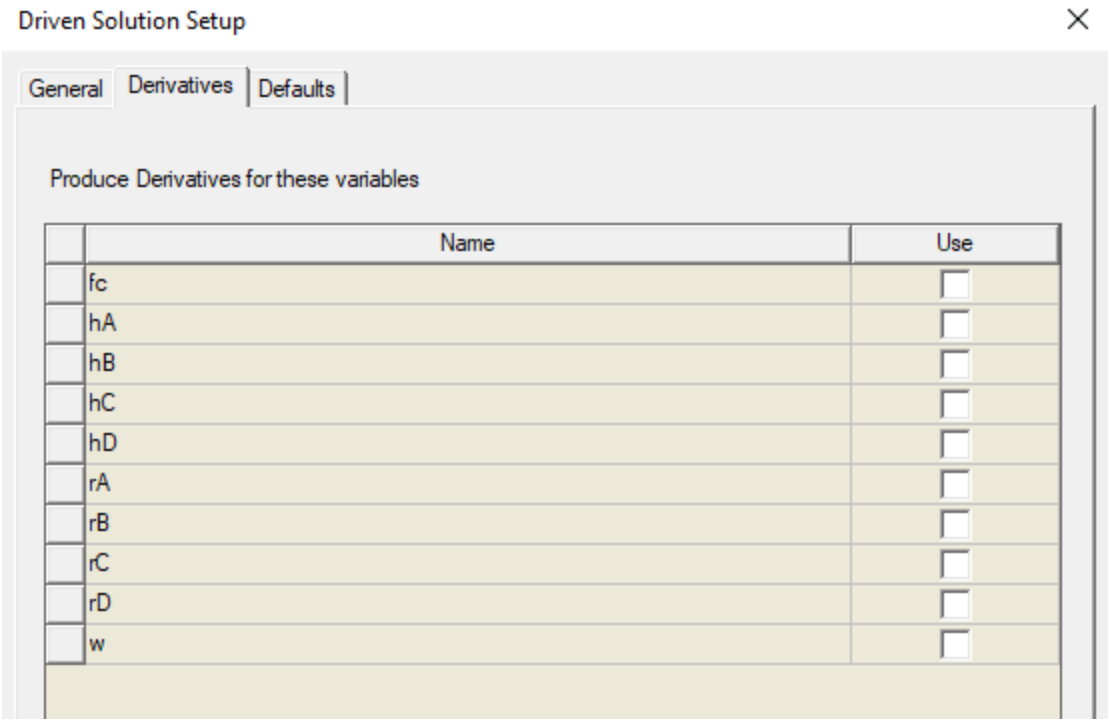
Far field tuning supports:

- Incident Wave
- Linked Field
- Voltage Source
- Current Source
- Magnetic Bias

> **Note:**
>
> Sensitivity analysis of a design variable associated with the magnetic bias region assumes that the magnetic bias field does not change. In some cases, this assumption may not be appropriate and requires user discretion to ensure its applicability, especially for non-uniform ferrite models.

You can select design properties and project variables as variables of differentiation. If your design has appropriate candidate variables, the **Derivatives** tab of the **Solution Setup** window lists them. Enable the derivative calculation(s) by selecting the appropriate **Use** check boxes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can view the computed derivatives in the Reporter. You can also use them through the Tune Reports command for interactive exploration of small variations in the design without the need to solve again.

Important:

This feature has the following limits:

- HFSS Transient projects do not support derivatives.
- HFSS Eigenmode projects do not support derivatives.
- Derivatives are not computed in a ports-only solution.
- Derivatives are supported in projects with non-port excitations, including incident waves, linked fields, voltage or current sources.
- Parameters of differentiation may not affect Floquet ports.
- Parameters of differentiation many not affect de-embedded Lumped ports.
- Derivatives are not supported for any solution employing Domain Decomposition Method (DDM), which includes FE-BI radiation boundaries, IE Regions, Finite Arrays.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Units for Calculated Derivatives

The derivatives calculated by HFSS are dependent on the units of the design parameters. For example, if the design parameter *p* is originally defined with units of millimeters, then the value of the derivative gives the (linearized) change in the output quantity when *p* is varied by one unit (1 mm). If the units of *p* are changed to an equivalent value in micrometers, and the derivative is recomputed, then it will appear to drop in magnitude by a factor of 1000 because it now represents how much the output changes when *p* is varied by one unit (now 1 micrometer.) Therefore, when reporting derivative values it is necessary to record the units of the output quantity as well as the units for the variable of differentiation.

Adding a Frequency Sweep (HFSS)

For Driven solution types.

To generate a solution across a range of frequencies, add a frequency sweep to the Advanced solution setup (an Auto Setup includes a discrete sweep by default). HFSS performs the sweep after the adaptive solution. If an adaptive solution is not requested, the sweep is the only solution generated. You can also disable a sweep, so that you can run only the adaptive solution (or a ports-only solution) without the sweep, then later reactivate the sweep definition.

To add a frequency sweep:

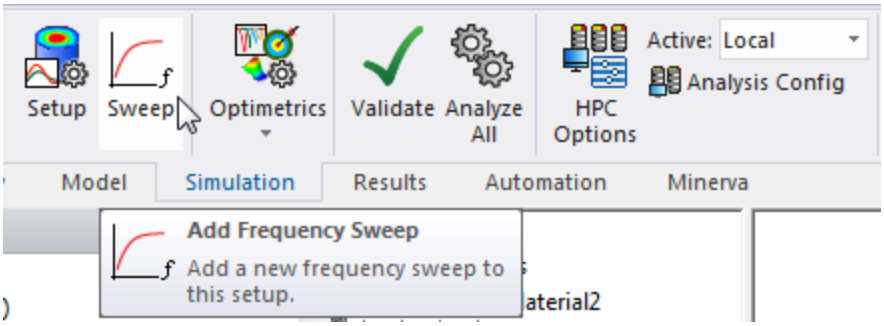
1. Click **HFSS**>**Analysis Setup**>**Add Frequency Sweep** .

   A dialog appears which lists the solution setups.

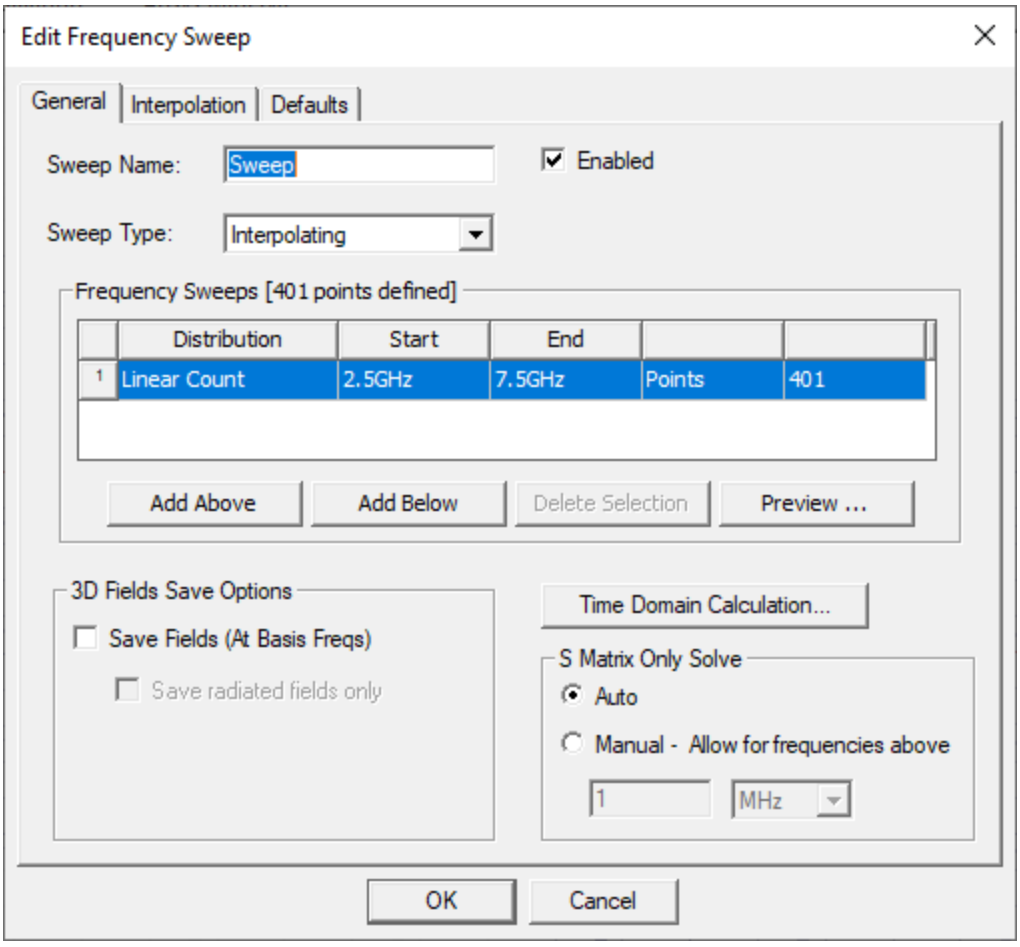
   Select the solution setup to which the sweep applies. To bypass this dialog right-click the desired setup in the Project tree and select **Add Sweep** from the shortcut menu, or select a

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup in the Project tree and on the Simulation tab, click the **Sweep** icon:



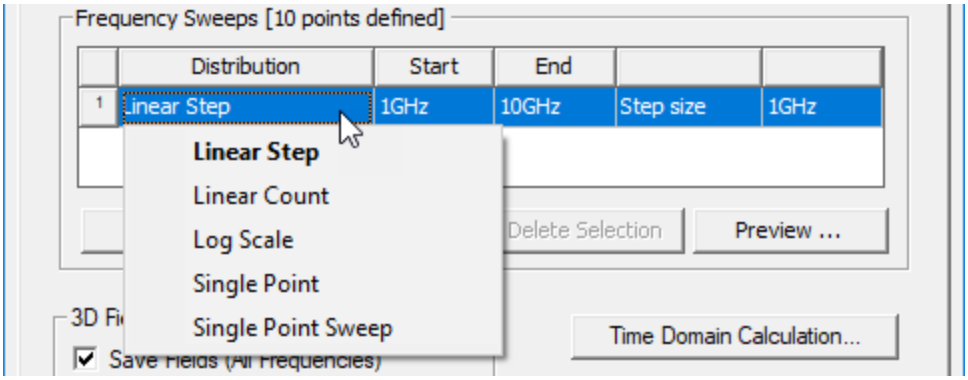
The **Edit Frequency Sweep** dialog box appears.



2. Specify the following sweep parameters:
   - Sweep Name.
   - Enabled check box.

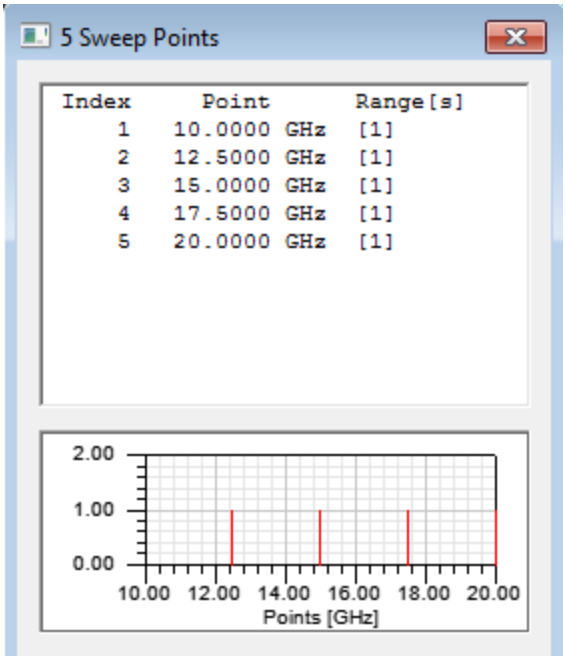
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Sweep type - Discrete, Fast, or Interpolating (the default). Each selection affects the tabs and options available. If a port is not assigned, Interpolating selection does not appear. The 3D Fields options are slightly different for each sweep type.
- Frequency Sweeps. You specify sweeps in terms of Distribution type, which can be Linear Step, Linear Count, Log Scale, Single Point, or Single Point Sweep, which adds a set of 10 Single Point Sweeps, defaulting from 1 GHz to 10 GHz in increments. The **Add Above**, and **Add Below** buttons permit you to add additional sweeps, including mixed sweep types. This feature provides flexibility. For example, you can define sweeps with log scale at lower frequencies, and linear step at higher frequencies. You can edit the Distribution, Start and End fields for any sweep.



If you have added more sweeps, you can **Delete** a selected distribution.

The **Preview** button displays the sweep(s) as currently defined.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If you plan to perform a Full-Wave SPICE analysis, click Time Domain Calculation tool to obtain assistance determining a suitable frequency sweep range for the solutions. Also see the Requirements for Full-Wave SPICE.
5. If you expect to use the same settings often, you can got to the Defaults tab and click the **Set Defaults** button. To use previously saved settings, click the **Use Defaults** button.

Once you have created a sweep, an icon for the sweep appears in the Project tree under the associated setup. Selecting the sweep icon causes the docked properties dialog to show the name, start, stop, step size, and sweep type. It also includes an **Enabled** check box.

Edit, Copy and Paste Existing Sweeps.

You can select an existing sweep, use the **Edit** commands to **Copy** it, and then and **Paste** the sweep into the Project tree. (By default, the copy is named Sweep*n*, where *n* increments with each new sweep.) You can edit the new copies of the sweep to make desired changes. For example, you can change a specific parameter, or for a distributed solve, you could assign different start and end points for each copy of the setup.

The **Paste** command for sweeps is design sensitive (that is, you cannot paste between Driven and Eigenmode designs) and context sensitive (for example, a sweeps can only be pasted in a setup.) Dependent setups are pasted along with the copied setup. You are warned if the dependent setup is already in the design and setup is not pasted again.

> **Note:**
>
> For a solve setup with zero passes, no sweeps, and that is not ports only, validation produces a warning message.

Selecting the Sweep Type

For Driven solution types.

Specify the type of sweep you want to perform in the **Edit Frequency Sweep** window. Choose one of the following sweep types.

Fast Sweep Options	Generates a unique full-field solution for each division within a frequency range. Best for models that will abruptly resonate or change operation in the frequency band. A Fast sweep will obtain an accurate representation of the behavior near the resonance. Fast sweeps are disabled if an anisotropic boundary condition is present.
Interpolating Sweep Options	Estimates a solution for an entire frequency range. Best when the frequency range is wide and the frequency response is smooth, or if the memory requirements of a Fast sweep exceed your resources.
(default)	All discrete basis solutions are solved prior to interpolating sweeps because it

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

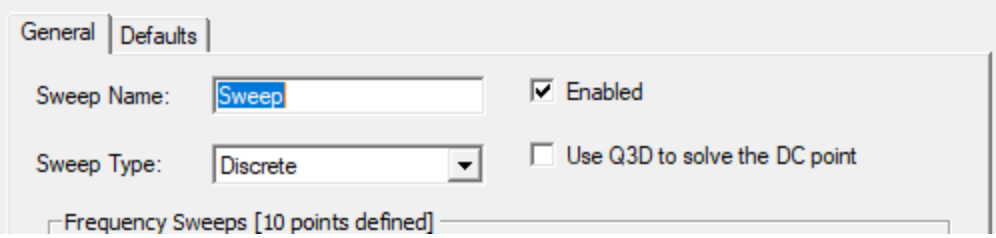
	is possible that an interpolating sweep can re-use already solved frequencies from a discrete sweep. For Time Domain Reflectometry plots (TDR), you must use an interpolating sweep. When you select Interpolating sweeps in the **Edit Sweep** dialog box, the **Interpolation** options tab is activated. This lets you specify a maximum number of solutions, and other interpolation values.
Discrete Sweep Options	Generates field solutions at specific frequency points in a frequency range. Best when only a few frequency points are necessary to accurately represent the results in a frequency range.

If you expect to use the same option settings often, you can click **Set Defaults**. To use previously saved settings, click **Use Defaults**.

Options for Discrete Sweeps

For Discrete sweeps, the **Edit Sweep** dialog options you can set include

- Sweep Name
- Enabled Checkbox
- The ability to Use Q3D for DC Simulation in HFSS means the dialog includes a Use Q3D to solve the DC point option.

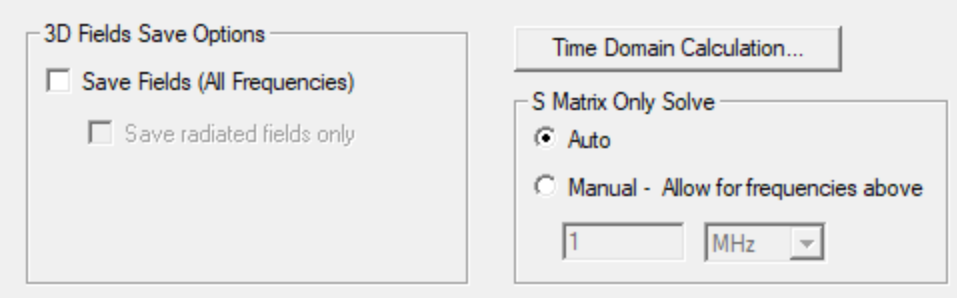


  If Q3D is turned on, the project will succeed only if classic can mesh the original model. Models where tau succeeds but classic fails fail if you turn on Q3D.

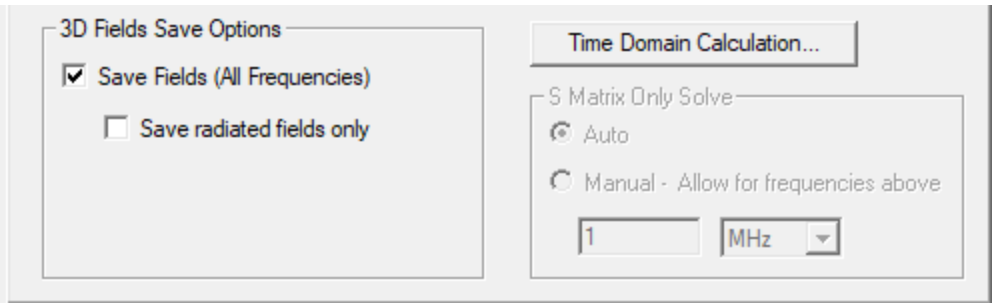
- Performance of HFSS with/without Q3D can be compared only with classic mesh. One can not compare it to tau.
- Frequency Setup

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 3D Fields Save Options.



  With **Save Fields** not selected, the default is for Auto **S Matrix Only Solve**. Here, the solver automatically determines threshold of S-matrix only solve. The **Manual** option is available for situations when:
  - The direct solver is being used.
  - If S-matrix only solve has a singularity matrix, you might want to increase the manual threshold.
  - If you want faster simulation, you might want to decrease the manual threshold.
- Whether to **Save Fields** (for all Frequencies). By default, all frequencies are saved. (This field is disabled under a Solve Ports Only setup. You can view port fields for the discrete frequencies under the port field display in the project tree.)
- For discrete sweeps, you can specify whether you want to save the 3D fields. Checking **Save Fields (All Frequencies)** enables the option to **Save radiated fields only**.



If you expect to use the same settings often, you can click **Set Defaults**. To use previously saved settings, click **Use Defaults**.

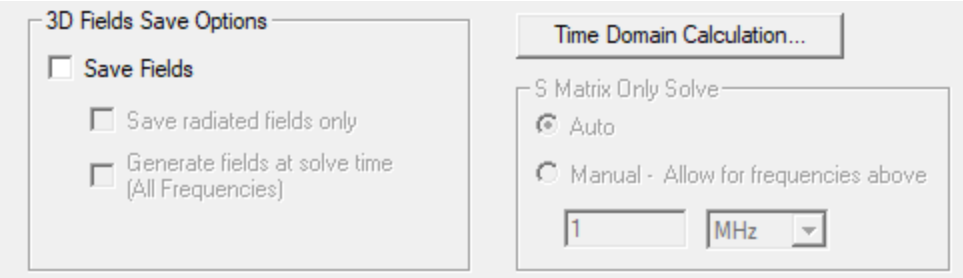
Options for Fast Sweeps

For Fast sweeps, the **Edit Sweep** dialog options you can set include:

- Sweep Name
- Frequency Setup
- For Fast Sweeps, you can select Save Fields, Save radiated Fields only, and whether to

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Generate fields at solve time (All frequencies).



By default, all fields are saved. (This field is disabled under a Solve Ports Only setup. You can view port fields for the discrete frequencies under the port field display in the project tree.)

Selecting **Save Fields** enables the check box for **Generate Fields for all frequencies while solving**. By default, fields are not generated. If you have more than 100 frequencies, checking the box generates a warning that disk space use may be excessive.

If you select this option, HFSS solves the fast sweep and then computes the fields at each freq in the sweep, and saves them. This has two effects: (a) the simulation takes longer due to saving the fields, (b) Post processing is much faster.

Since this option is exercised at solve time, it doesn't apply to existing solutions. Enabling this option for a previously solved sweep and re-solving will access the previously solved data and generate the requested fields.

- DC Extrapolation options (appearing depending on the Design Settings for DC Extrapolation)

If you expect to use the same settings often, you can click **Set Defaults**. To use previously saved settings, click **Use Defaults**.

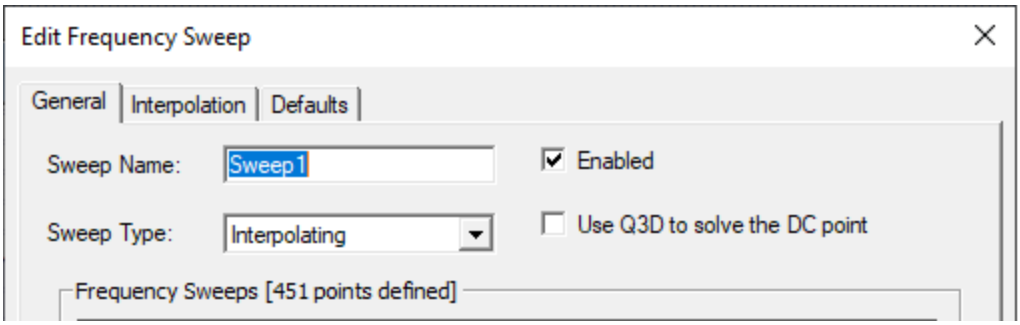
Options for Interpolating Sweeps

A wave port must be assigned for the Interpolating option to appear. For Interpolating sweeps, **Edit Frequency Sweep** options include:

General tab:

- Sweep Name
- Enabled check box
- Due to the ability to **Use Q3D for DC Simulation** in HFSS, the **General** tab includes a checkbox to **Use Q3D to solve the DC point**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



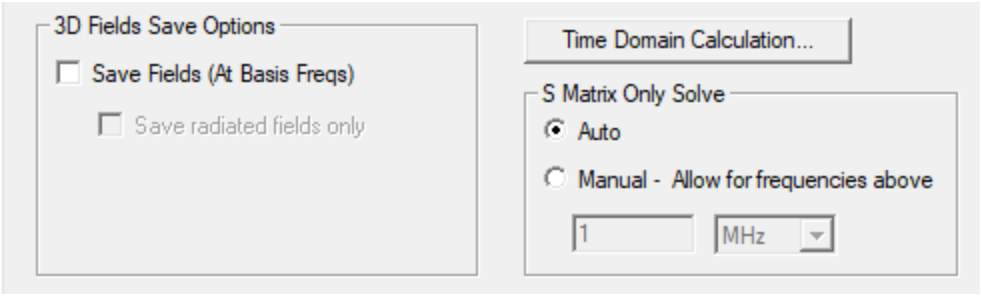
This feature accepts designs with gap type of lumped port, where the terminals touch planar faces. It is limited to RLC, finite conductivity and PEC for boundary. A Q3D DC solve will only use the classic/phi mesher. You can look at the S-parameter result for DC point, but no field will be available.

If Q3D is turned on, the project will succeed only if classic can mesh the original model. Models where tau succeeds but classic fails fail if you turn on Q3D.

- Performance of HFSS with/without Q3D can be compared only with classic mesh. One can not compare it to tau.

Note: Use Q3D to solve DC point provides network data only and cannot be used for fields overlays.

- Frequency Setup
- 3D Fields Save Options.

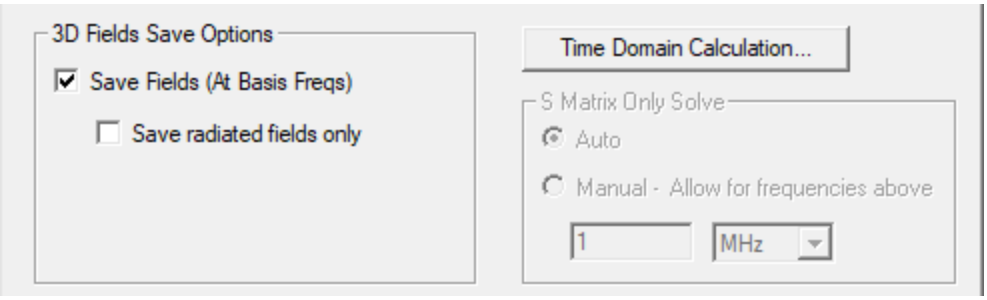


With **Save Fields (At Basis Freqs** not selected, the default is for Auto **S Matrix Only Solve**. Here, the solver automatically determines threshold of S-matrix only solve. The **Manual** option is available for situations when:

- If the direct solver is being used.
  - If S-matrix only solve has a singularity matrix, you might want to increase the manual threshold.
- If you want faster simulation, you might want to decrease the manual threshold.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Save Fields (at Basis Freqs)** selected:



  You can save fields or radiation only on the basis frequencies of the interpolating sweep. If you do so, the sweep will be available for selection by a target SBR+ design.This permits directivity and gain in the target SBR+ design.

Interpolation tab:

- Max Solutions
- Error Tolerance
- Click the **Advanced Options...** button to open the Interpolating Sweep Advanced Options dialog.

DC Extrapolation tab (appearing depending on the Design Settings for DC Extrapolation):

- DC Extrapolation option

If you expect to use the same settings often, you can click **Set Defaults** on the Defaults tab. To use previously saved settings, click **Use Defaults**.

DC Extrapolation options

If the Design Setting for DC Extrapolation is set to Advanced, the **DC Extrapolation** tab appears for Fast and Interpolating sweeps. Otherwise, an effective DC value is calculated.

1. Setting the Initial Frequency to 0 enables DC extrapolation for a sweep.
2. Enter a value for the **Minimum Solved Frequency**. This value represents the smallest frequency in the sweep for which a full solution is generated. The default is 100 Mhz.

> **Note:**
>
> The Minimum Solved Frequency that you enter is solved independently of the step size.

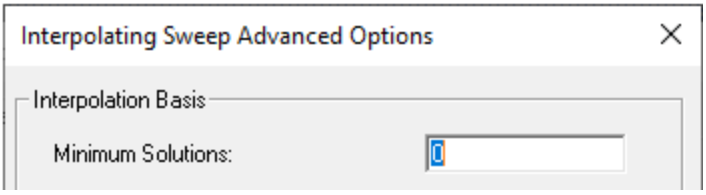
If you expect to use the same settings often, you can click **Set Defaults**. To use previously saved settings, click **Use Defaults**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

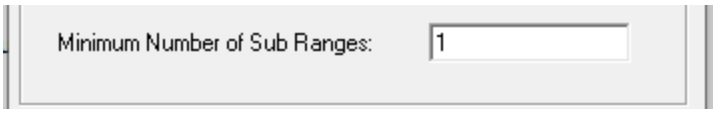
Set up Interpolating Sweep Advanced Options

For Interpolating sweeps, the **Setup Interpolating Sweep Advanced Options** dialog lets you specify the following settings for a sweep:

- The **Minimum Solutions** value is the minimum number of solutions that will be used for an interpolation basis of the desired frequency sweep. The finished sweep will satisfy both the desired error tolerance and the minimum number of bases. For example, if the minimum solutions is set to ten, then at least ten frequency points will be solved and used as a basis for the sweep even if the sweep converges before the ten points.



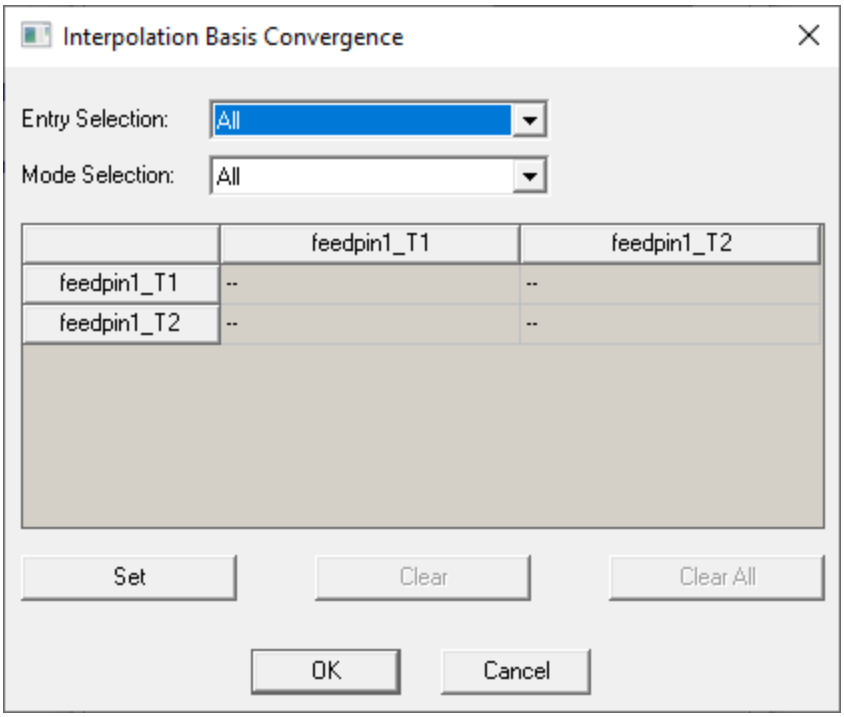
- Specify a **Minimum Number of Sub Ranges**. This number acts as an initial condition on the sweep to force initial even breakup of the null range into sub ranges. The end points and middle of each subrange will be solved. This controls the points at which the interpolating sweep is broken up and prevents redundant effort caused by neighboring interpolating sweeps solving the same point. For example, the 1GHz to 4GHz and the 4GHz to 9 GHz sweeps do not both solve the 4 GHz data point.



- Whether to use all or selected entries in the matrix of data types for the convergence. To choose, click the **Select Entries** button to display the Interpolation Basis Convergence

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dialog.



- The Data Types for Convergence. The defaults selections differ depending on the solution setup.

For Driven modal, Network Analysis 3D Solution Interpolating sweeps:

- S-Matrix - checked
- Port Impedance - unchecked.
- Propagation constants - checked.

For Driven terminal, Network Analysis 3D Solution Interpolating sweeps:

- S-Matrix - checked
- Port Impedance - checked.
- Propagation constants - checked.

For Driven modal, Network Analysis ports-only, interpolating

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- S-Matrix - unchecked
- Port impedance - checked
- Propagation constants - checked

For Driven terminal, Network Analysis ports-only, interpolating

- S-Matrix - unchecked
- Port impedance - checked
- Propagation constants - checked

If the design contains them, you can select **Derivatives**. If you select **Derivatives**, you can also set the **Error Tolerance**, overriding the tolerance specified in the Edit Sweep dialog..

> **Note:**
>
> If a driven setup's ports-only setup changes and then the problem type switches between driven modal and driven terminal, HFSS resets the interpolation basis data types for the interpolating sweep.

Enforce Passivity – This applies to driven modal and Terminal solutions.

Select this check box to enable passivity enforcement of an interpolating sweep at the requested sampling frequencies with the specified error tolerance. For instance, tolerance of .001 means simulation should attempt to reduce passivity to less than 1.001 at all sampling frequencies. You can view passivity using the reporter. Passivity is enforced on 50 ohm renormalized single ended terminals for terminal projects and 50 ohm renormalized modes for modal projects.

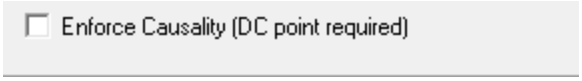
When using passivity, you may want to increase the Max Solutions value.

During an interpolating sweep, HFSS first adds basis points (solution frequencies) until the convergence goal is reached, and then tests the sampling frequencies using the converged sweep for whether they meet the specified passivity requirement. The sampling frequency having the worst passivity violation is selected for solving. After solving for a passivity violation frequency, the solver adds the newly solved data to the interpolation basis.

If necessary, the solver returns to solving for convergence, limited by the maximum number of solutions for the sweep and resumes passivity checking after reestablishing convergence. Messages in the Solution Profile indicate when the interpolated solver is working toward convergence or working to resolve passivity. If any of the bases violates the passivity, the current sweep terminates with a warning message.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enforce Causality(DC Point Required) -- This applies to Terminal solutions. Causality means that an output signal cannot start to change before the input signal changes. Causality can be enforced only when there is a DC point in the sweep. A design should follow "Best Practices for Accurate and Causal Broad Band Frequency Sweeps." In addition, you can select "Enforce Causality" so the sweep fits (rather than interpolates) the s-parameter results to a set of causal basis functions, thereby ensuring the extraction of a causal model..



Note:

You should follow recommended guidelines whenever setting up a numerical model in HFSS (provide correct material definitions, including information about frequency dependent behavior, and defining appropriate boundary conditions). Also when extracting s-parameter models for signal integrity designs, where providing passive and causal results is important, besides using this option, see "Best Practices for Accurate and Causal Broad Band Frequency Sweeps".

Note:

If you start and stop an interpolated sweep and change the convergence value and/or passivity tolerance, you may generate a non-repeatable sequence of frequencies in the basis. Both regular and passivity frequencies are used when re-evaluate convergence.

If you change sampling frequency delta subsequent passivity behavior may be totally different, since it is evaluated at new sampling frequencies.

Setting the Error Tolerance

For Interpolating sweeps.

The **Error tolerance** value is the maximum relative difference allowed between two successive interpolation solutions. Because interpolating sweeps converge non-uniformly, that is, the error will jump around, false convergence is possible. To prevent this, a secondary check based on the Loewner matrix must also succeed: the condition of the Loewner matrix must also be less than the error tolerance.

The default 0.5 percent for interpolating sweeps is usually satisfactory. However, if the Interpolated sweep Advanced Options include **Enforce Causality** and **Enforce Passivity**, you

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

may need to specify a lower tolerance (for example, 0.1). This is because of the sensitivity of Enforce Causality and Passivity.

To set the error tolerance for an Interpolating sweep:

1. Open the **Edit Sweep** dialog box (by either viewing the properties of an existing Sweep or by Adding a Frequency sweep to an existing Setup).
2. Type a value in the **Error Tolerance** box.

If you expect to use the same settings often, you can click **Set Defaults**. To use previously saved settings, click **Use Defaults**.

Setting the Maximum Number of Solutions

For Interpolating sweeps.

The **Max Solutions** value is the maximum number of solutions that will be solved for the frequency range. For fast sweeps and for interpolating sweeps the default is 250. The value must be greater than 3. To change the value:

1. Open the **Edit Sweep** dialog box (by either viewing the properties of an existing Sweep or by Adding a Frequency sweep to an existing Setup).
2. Type a value in the **Max Solutions** box and click **OK**.

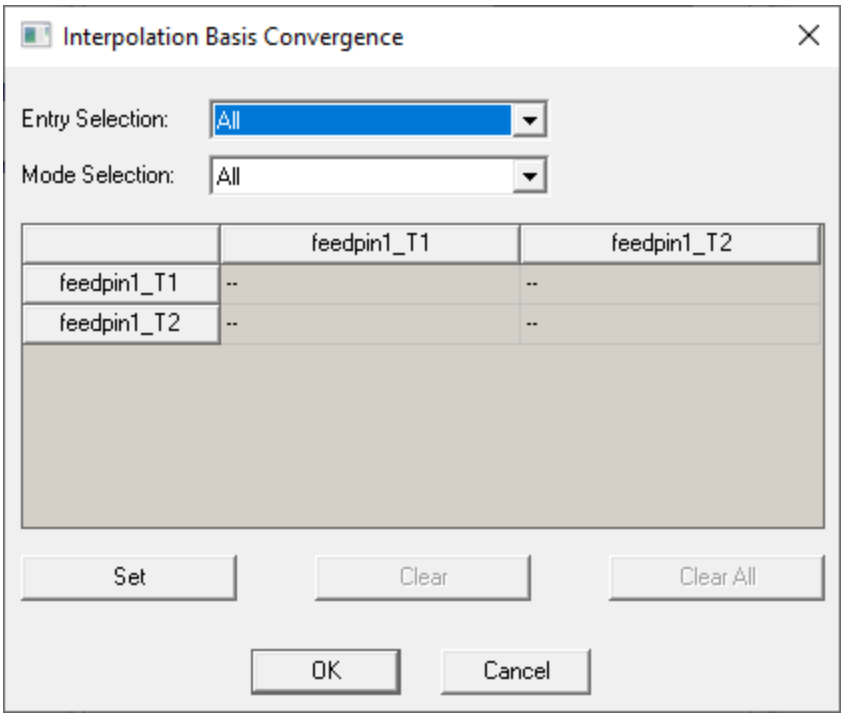
If you expect to use the same settings often, you can click **Set Defaults**. To use previously saved settings, click **Use Defaults**.

> **Note:**
>
> HFSS automatically subdivides the interpolating sweep range so that no single subrange gets too many basis elements. The effect is that you can now (if appropriate) request hundreds of basis elements in the Max Solutions box for interpolating sweep setup, without incurring any basis seeding performance penalty.

Interpolation Basis Convergence

From the Setup Interpolations Basis dialog box, select the **Use Selected Entries** radio button to enable the **Select Entries** button. Select this to display the **Interpolation Basis Convergence** dialog box. This dialog permits you to specify the convergence basis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



1. Select one of the following from the **Entry Selections** drop-down menu:

All	Selects all of the matrix entries at once. (The default).
Diagonal	Selects all of the diagonal matrix entries at once.
Off-Diagonal	Selects all of the off-diagonal matrix entries at once.

2. If you chose **All**, **Diagonal**, or **Off-Diagonal**, you may fine-tune the matrix entry selection process by selecting one of the following options from the **Mode Selection** drop-down menu:

All	Selects all of the mode matrix entries. Select in conjunction with **All**, **Diagonal**, or **Off-Diagonal** entry selections.
Dominant Only	Selects only the dominant mode matrix entries. Select in conjunction with **All**, **Diagonal**, or **Off-Diagonal** entry selections.
Higher Order Only	Selects only the higher-order mode matrix entries. Select in conjunction with **All**, **Diagonal**, or **Off-Diagonal** entry selections.

As you select the wave ports for convergence, you use the **Set**, **Clear**, and **Clear All** buttons in connection with the **Entry Selection** and **Mode Selection** settings. These buttons are enabled when the wave port matrix state and selection settings permit them do something.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, The **Clear** button is not enabled until there are entries in the wave port matrix to clear and those selections are permitted by the entry selection. The **Set** button is not enabled unless the available mode selections permit entries to be set.

You can also select individual entries in the wave port matrix by clicking on grid cells. This action displays a dropdown menu that lets you select ON or "-".

3. Specify **EntrySelections** and **Mode Selections** as desired and click **SET**, or click individual wave port cells and select ON.

In the table location corresponding to the selection, the dash in the display is replaced by ON. For example, selecting the first element in the row list and the fourth element in the column list, and then **Set Selection** places an ON in the first row, fourth column. You can select one entry at a time via the dropdown in the matrix cell, or clear the entire table with the **Clear All** button. You can also **Clear** only the entries specified by the **Entry** and **Mode** selection settings (such as off-diagonal, higher order).

4. Click **OK** to close apply the selections and close the dialog.

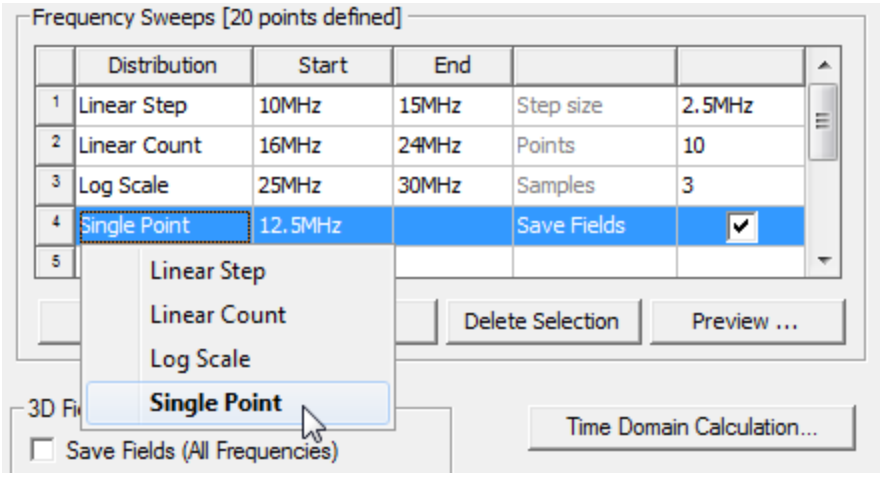
Specifying the Frequency Points to Solve

You can specify the following types of frequency points to solve within a frequency sweep:

Linear Step	A linear range of frequency points in which you specify a constant step size.
Linear Count	A linear range of frequency points in which you specify the number, or count, of points within the frequency range.
LogScale	A logarithmic range of frequency points in which you specify a frequency range and a samples number.
Single Points	Individual frequency points. *For Discrete sweeps*.

Select the type of frequency point entry from the **Type** dropdown menu.

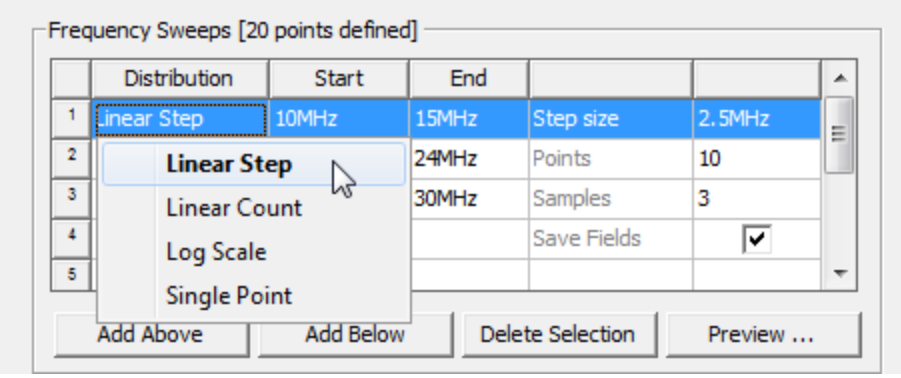
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Edit Frequency Sweeps window contains a Time Domain Calculation tool that you can use to help calculate frequency step sizes and maximum frequencies, particularly if you intend to perform Full-Wave Spice analysis. For Discrete sweeps, HFSS always solves the frequencies from highest to lowest.

Specifying Frequency Points with a Linear Step Size

1. In the **Edit Sweep** dialog box, click **Linear Step** in the **Type** pull-down list.



2. In the **Start** text box, type the starting frequency of the frequency sweep.

   For Fast and Interpolating sweeps, setting the initial value to 0 enables the DC Extrapolation option.

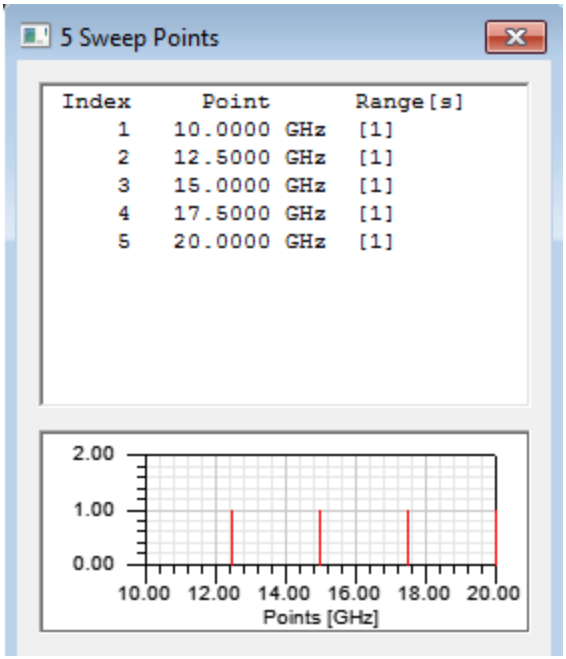
3. In the **End** text box, type the ending frequency of the frequency sweep.
4. In the **Step Size**box, type the difference between frequency points.

HFSS will solve the frequency point at each step in the specified frequency range, including the start and stop frequencies. For Discrete sweeps, HFSS always solves the frequencies from highest to lowest.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, specifying **10** for the start frequency, **20** for the stop frequency, and **2.5** for the step size**for a Discrete sweep** instructs HFSS to compute a solution for frequencies of 10, 12.5, 15, 17.5, and 20.

To view a table of the frequencies and count, click the **Preview** button.



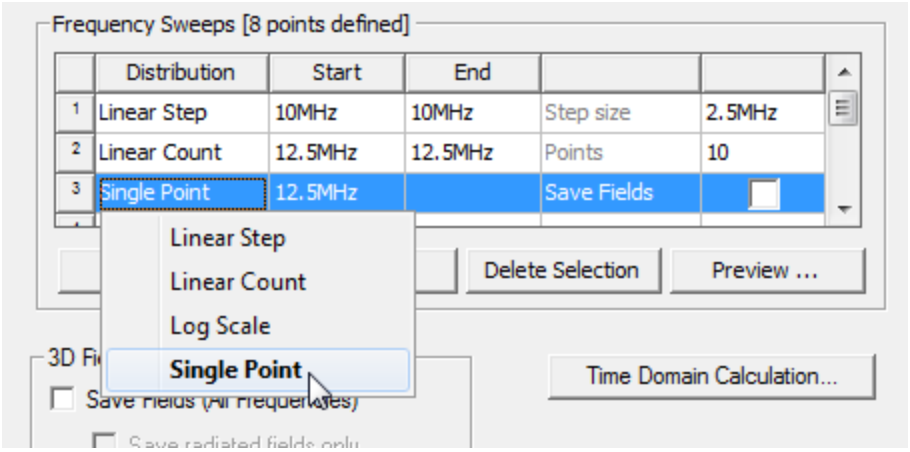
If the list of frequencies is longer than the visible display, you can use a scroll bar to view the list to the end of the count.

1. For Fast sweeps, select **Save Fields** if you want to save the calculated 3D field solutions associated with all port modes at the chosen frequencies.

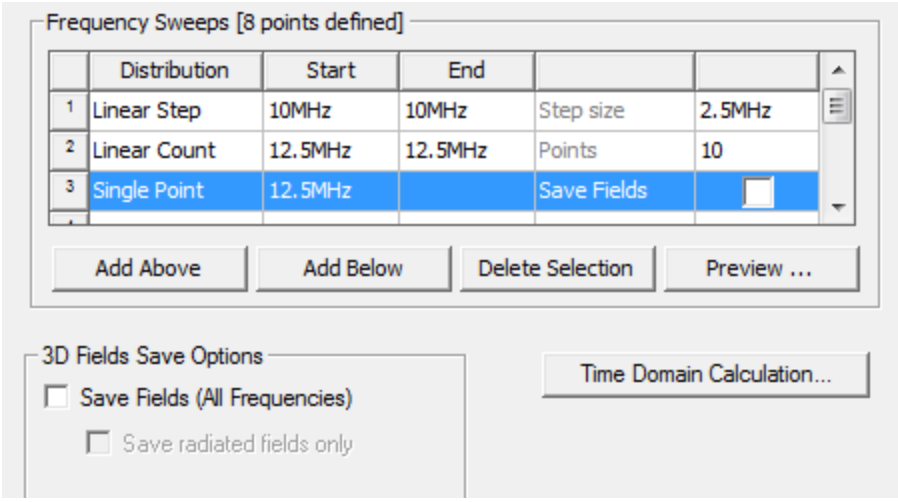
For Discrete sweeps, select **Save Fields (All Frequencies)** if you want to save the calculated 3D field solutions associated with all port modes at the chosen frequencies.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If want to save the fields for just one or a few Discrete sweep frequencies, create a single point sweep for that frequency.



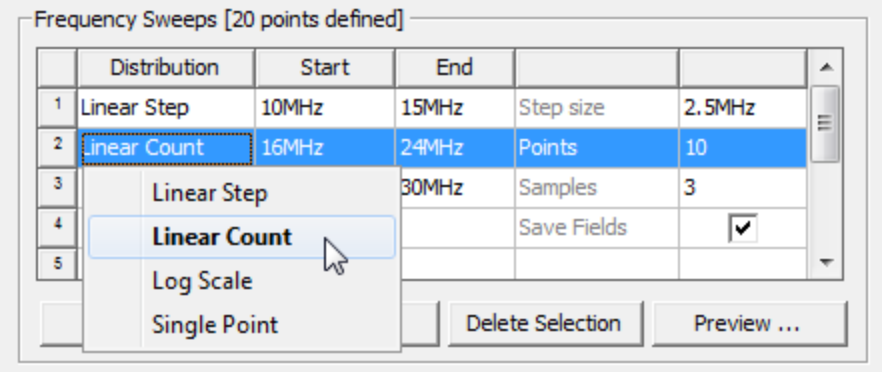
This single point sweep has a **Save Fields** column with a check box for each single point frequency you define. You can select the **Save Fields** check box for the desired frequency, or click the **Save Fields** (All Frequencies) check box to check or uncheck all frequencies.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specifying a Linear Count of Frequency Points

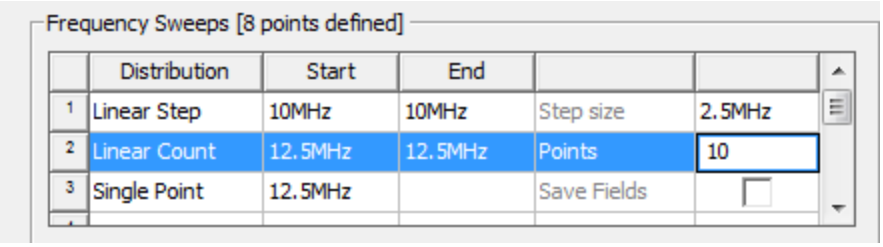
1. In the **Edit Sweep** dialog box, click **Linear Count** in the **Type** pull-down list.



2. In the **Start** text box, type the starting frequency of the frequency sweep.

For Fast and Interpolating sweeps, setting the initial value to 0 enables the DC Extrapolation option.

3. In the **End**text box, type the ending frequency of the frequency sweep.
4. In the Points text box, type the number of points in the sweep. The count value includes the Start and End values.



HFSS will divide the frequency range into the count you specify and solve each frequency point in the count.

To view a table of the frequencies and count, click the **Display** button. If the list of frequencies is longer than the visible display, you can use a scroll bar to view the list to the end of the count.

5. For Fast sweeps, select **Save Fields** if you want to save the calculated 3D field solutions associated with all port modes at the chosen frequencies.

For Discrete sweeps, select **Save Fields (All Frequencies)** if you want to save the calculated 3D field solutions associated with all port modes at the chosen frequencies.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

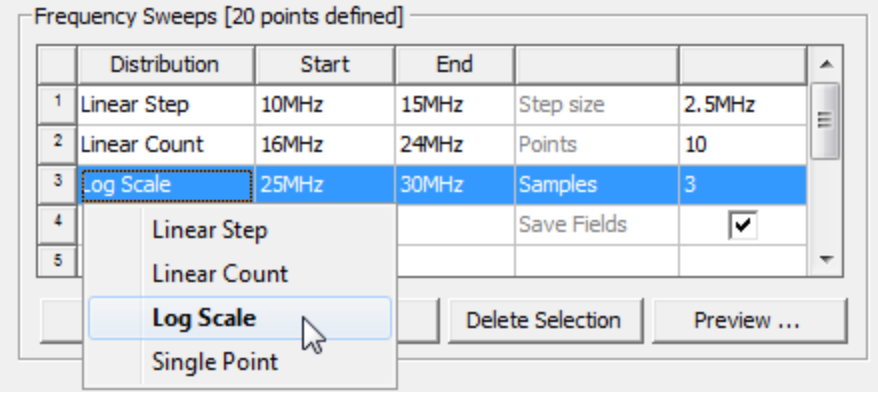
If want to save the fields for just one or a few Discrete sweep frequencies, define a Single Point sweep for those frequencies and check the Save Fields box for that point.

For Discrete sweeps, HFSS always solves the frequencies from highest to lowest.

Specifying a Logarithmic Spaced Frequency Sweep

For Discrete sweeps.

1. In the **Edit Sweep** dialog box, click **LogScale** in the **Type** drop-down menu.



2. In the **Start** text box, type the starting frequency of the frequency sweep.

   For Fast and Interpolating sweeps, setting the initial value to 0 enables the DC Extrapolation option.

3. In the **End** text box, type the ending frequency of the frequency sweep.
4. In the **Samples** text box, specify the number of frequency points to sample.

   HFSS assigns the sampled points using intervals based on a logarithmic scale.

   To view a table of the frequencies and count, click the **Display** button. If the list of frequencies is longer than the visible display, you can use a scroll bar to view the list to the end of the count.

5. Select **Save Fields (All Frequencies)** if you want to save the calculated 3D field solutions associated with all port modes at the chosen frequencies.

For Discrete sweeps, HFSS always solves the frequencies from highest to lowest.

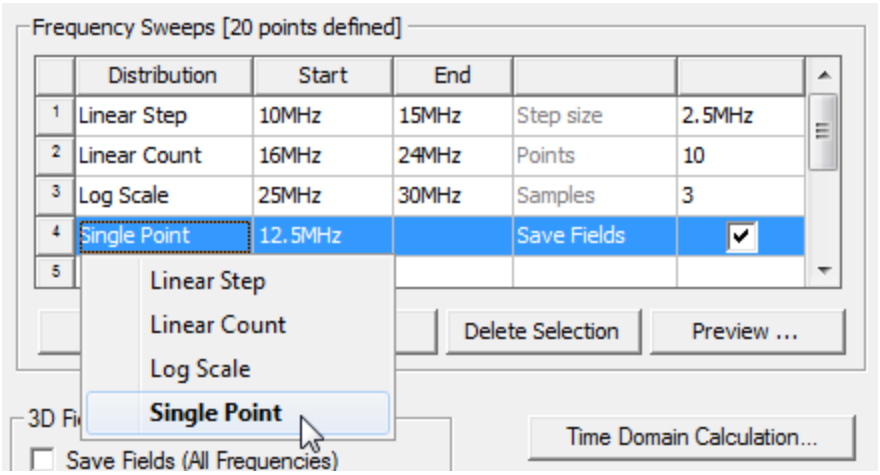
Specifying Single Frequency Points

For Discrete sweeps.

To specify Single Frequency points:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Edit Sweep** dialog box, click **Single Points** in the **Type** pull-down list.



2. In the **Single** text box, type a desired frequency point, and then select the frequency units.

A check mark in the **Save Fields** column indicates that the fields for the point will be saved. Optionally, click the check box in the **Save Fields** column.

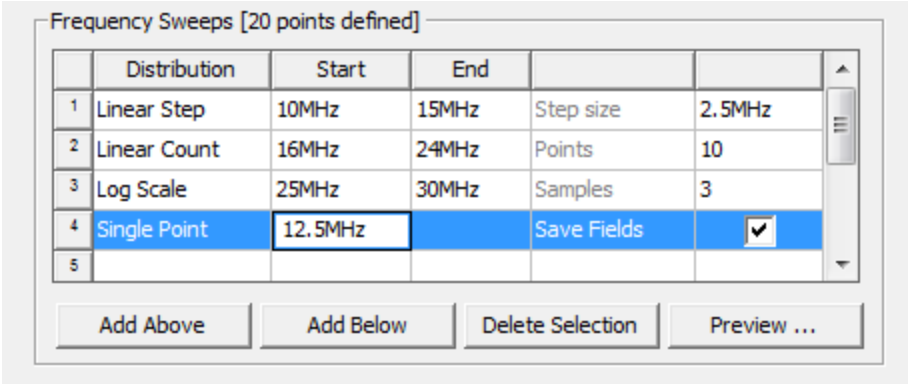
3. Create additional Single point sweeps for each frequency point you want to solve.

For Discrete sweeps, HFSS always solves the frequencies from highest to lowest.

Change the Value of an Existing Frequency Point

To change the value of an existing frequency point in a discrete sweep in the **Edit Sweep** dialog:

1. In the **Edit Sweep** dialog either select the text field in the Frequency column and edit an existing value field directly:



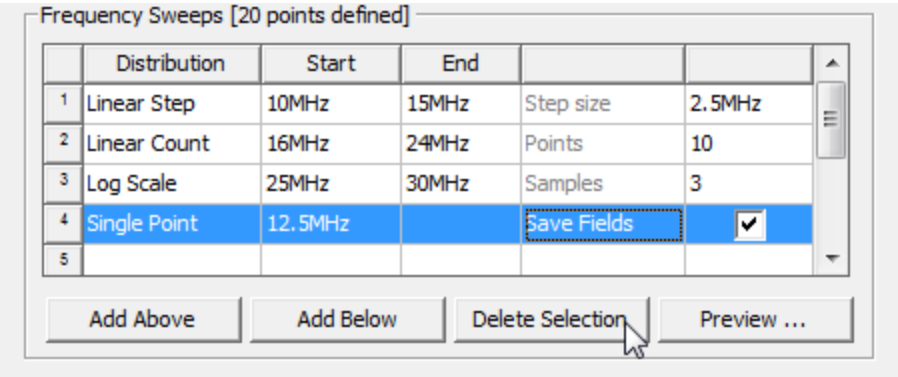
2. You could also create a new single point sweep.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select **Save Fields** if you want to save the calculated 3D field solutions associated with all port modes at that frequency.
4. Repeat for changing additional points.

Deleting Frequency Points

1. To delete a single frequency point from a sweep, select the row containing the frequency you do not want to solve.

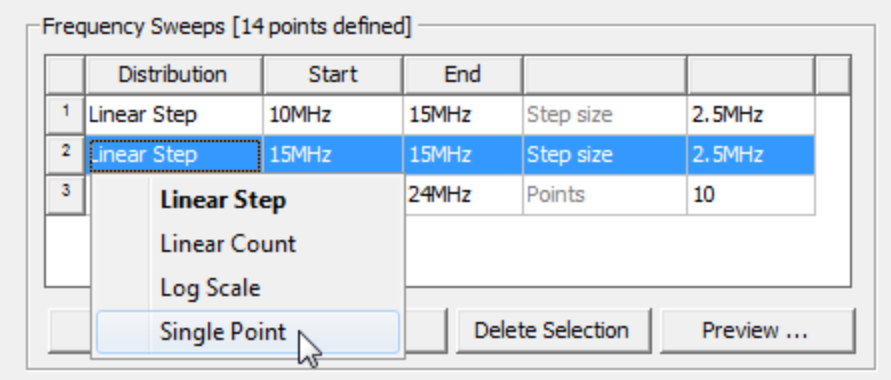


2. Click **Delete**.

Inserting Frequency Points

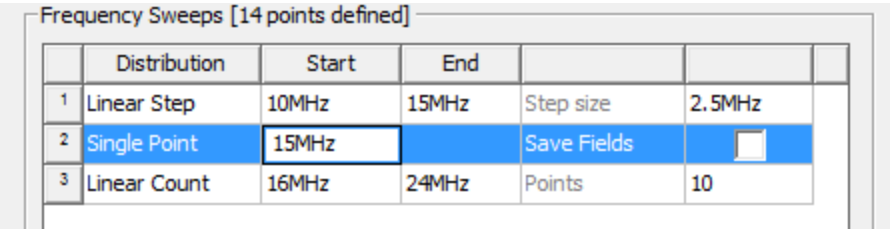
For Discrete sweeps, you can insert, change, or delete specific frequency points that you want to solve in the frequency range. They can be inserted individually to a new display, or to an existing display of uniform frequency points.

1. To pre-specify an initial uniform range of frequency points, use the Frequency setup of the **Edit Sweep** dialog to define the linear step, linear count, or log scale range you want.
2. Select an existing sweep and click Add Above or Add Below to insert the new sweep definition.
3. Select **Single Point** from the Distribution drop-down menu.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. In the **Start** text box, type a desired frequency point in the frequency units.



5. Select the **Save Fields** if you want to save the calculated 3D field solutions associated with all port modes at that frequency. You can also select, or deselect the check box to specify individual frequencies.
6. Add or edit additional Single Point frequency sweeps, and then click **OK** to save the current definitions.

For Discrete sweeps, HFSS always solves the frequencies from highest to lowest.

Choosing Frequencies for Full-Wave SPICE

If you plan to perform a full-wave SPICE analysis, use the **Time Domain Calculation** dialog box to help determine a suitable frequency sweep range for the solution.

To perform the calculation of suitable frequencies to solve:

1. In the **Edit Sweep** dialog box, click **Time DomainCalculation**.

   The **Time Domain Calculation** dialog box appears.

2. Type a minimum rise time value in the **Signal Rise Time** box.

   This value represents the time scale that will characterize the rate of change of the input time signal, which will be applied in the circuit simulator.

3. Type a value in the **Time Steps Per Rise Time** box.

   The time sampling increment for the entire signal is calculated using

$$\Delta t = \frac{\tau}{N_\tau} \quad (1)$$

   where

   - Δt is the time sampling increment.
   - τ is the signal rise time.
   - N_τ is the number of time steps per signal rise time.

4. Type a value in the **Number of Time Points** box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the input time signal duration is determined using $N \times \Delta t$, where *N* is the number of time points.

5. Click **Calculate**.

- HFSS now determines the **Maximum Frequency** using

$$F_{max} = \frac{0.5}{\Delta t} \quad (2)$$

- where F_{max} is the maximum frequency.

  HFSS determines the **Frequency Step Size** using $\frac{F_{max}}{N}$.

6. Click **OK** to transfer the data to the frequency sweep fields in the **Edit Sweep** dialog box.

Guidelines for Calculating Frequencies for Full-Wave SPICE

Keep the following guidelines in mind when you set up the calculation for the suggested frequency step size and maximum frequency:

- The maximum frequency should be at least five times the inverse of the rise and fall times. If the specified frequency band is too wide, an HFSS frequency sweep may have convergence problems. If this happens, try to decrease the maximum frequency until the solution converges.
- It is recommended, though not required, that the minimum frequency be less than the maximum frequency divided by the number of frequency steps. It is usually recommended to have at least 500 frequency steps. A higher number will slightly improve the full-wave SPICE solution accuracy, but will also increase CPU and memory requirements to solve the problem. For most cases, using 1000 frequency steps provides a good trade-off between the accuracy and computational requirements.

  > **Warning:**
  >
  > Occasionally, HFSS can fail to solve for the minimum frequency during a Discrete or Interpolating frequency sweep due to a failure of the port solver to converge. If this happens, try to increase the minimum frequency until the solution process completes successfully. However, the minimum frequency should be as low as possible because the low-frequency response determines the steady-state time response.

- The suggested frequency sweep ranges are estimates. You may have a pulse with a wider frequency content and HFSS's recommended frequency sweep range may miss some of the high frequencies.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Requirements for Full-Wave SPICE

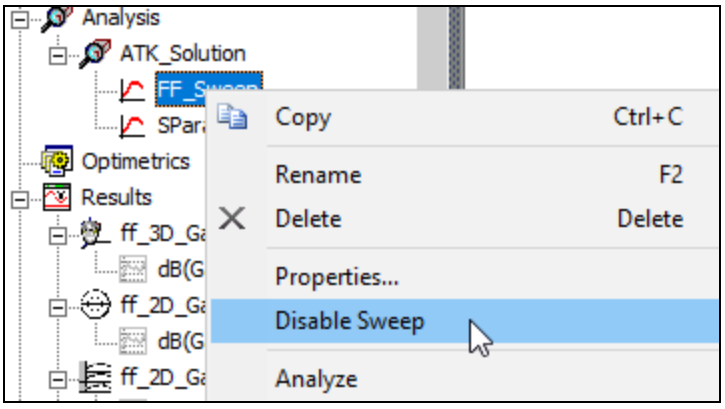
The Full-Wave Spice requirements are as follows:

1. The design problem type in which the solution data panel is opened must be driven terminal.
2. In the Matrix Data panel, for non-imported data, the view type for the solution data must be "Terminal Data" (not "Modal Data").
3. The data must be interpolating, or it must be discrete.
4. If the data is discrete:
    a. It must either be native terminal data, or it must be inferred as terminal. For instance, in a Touchstone file, the comment line "! Terminal data exported" will cause HFSS to interpret the data as terminal, while the comment line "! Modal data exported" will cause HFSS to interpret the data as modal. If HFSS finds neither comment line, it assumes that the data is terminal.
    b. At least 20 frequency points must be provided.
    c. HFSS must be able to generate an interpolation basis that converged with <= 0.5% error using no more than 100 basis elements.

Disabling or Enabling a Frequency Sweep

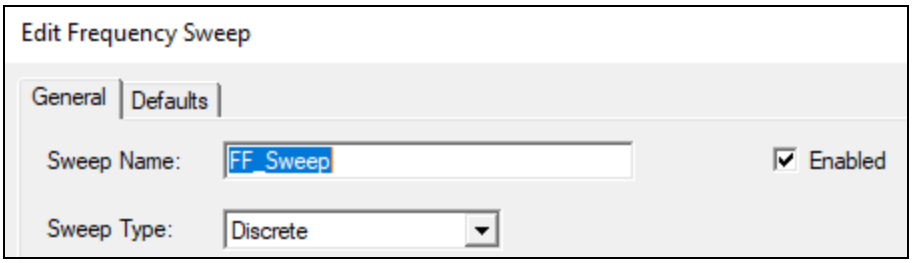
You can activate or deactivate a sweep definition. Disabled sweeps has an “empty graph” icon, and are not analyzed until enabled. To deactivate a sweep definition without deleting it:

1. Expand the tree hierarchy under the **Analysis** icon in the **Project Tree**.
2. Expand the tree hierarchy under the icon for the analysis setup that includes the sweep.
3. Right-click the icon for the sweep definition and select **Disable Sweep Analysis**. (To reactivate the sweep, right-click again and select **Enable Sweep Analysis**.)



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Also deactivate a sweep by right-clicking and selecting **Properties** to open the **Edit Properties** window. In the upper-right corner of the **Edit Properties** window, uncheck the **Enabled** area. Then click OK to apply the change. (To reactivate the sweep, re-open the **Edit Properties** window and check the **Enabled** area. Then click OK.)



Specifying the Number of Processors

If you want to use more than one processor, see HPC and Analysis Options,.

Specifying the RAM Limit Percentage

The **RAM Limit** (%) setting requests a restriction on the amount of memory the default solver used by the Ansys Electronics Desktop may allocate before it must stop solving on-core—solving processes entirely in memory—and start solving off-core. In off-core mode, the solver creates temporary solution files to which it spills, or shifts, data from memory, instead of relying on the operating system to start disk swapping. The default solver is finely tuned at handling its own memory, and can optimize loading only those blocks of memory required for its immediate needs.

Using this option may help to keep the entire solver from being swapped out in the normal course of process management on your computer. This kind of control may be especially important when multiple solvers are running on the same machine. Of course, if the total memory requirement of all processes grows large enough, the operating system will be forced into disk swapping.

> **Note:**
>
> The Iterative solver doesn't go off-core for efficiency reasons. Once the limit is exceeded, iterative solver will issue an "Out of memory" error message.

Regardless of this setting, processes are limited to 4TB of address space on 64 bit operating systems - no matter how much physical memory is installed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Regardless of the RAM Limit setting, if allocation fails, the solvers will automatically switch to off-core mode.

In case you receive an error message regarding insufficient memory on a 64-bit operating system, you may have reached a point where the sum of physical RAM plus available swap space exceeds the minimum amount of RAM needed by the off-core solver. Even for the off-core solver, the RAM usage cannot be made arbitrarily small. Off core does not mean it will use system swap space. Instead, the solver actively uses disk space to solve the problem. Users specify the % RAM allowed to be consumed by the solver on each node. The solver checks the total RAM on the node and applies the % RAM to arrive at the total memory that can be used. However, if you allow off core and it did not complete the solve the message estimates the minimum memory required to solve the off core solve.

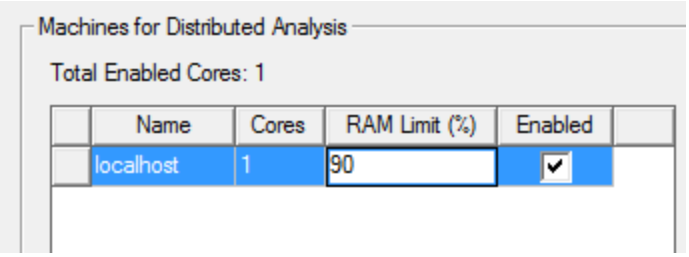
For Linux schedulers, where the environment may be non-uniform/heterogeneous, you have the option of using a batchoption to specify a RAMLimitPerCoreInGB. See the discussion under Distribution Command Line Options.

To specify the **RAM Limit** (%) of the machine on which Ansys Electronics Desktop s installed:

1. Click **Tools>Options>Setting HPC and Analysis Options**.
2. This opens the HPC and Analysis Options dialog.
3. Specify the Design Type for which you want to set a RAM limit. Make the Local machine Active.
4. In the **HPC and Analysis Options** dialog click the Edit button.

This opens the **Analysis Configuration** dialog.

5. Select the **Machines** tab. specify a **RAM Limit (%)** in the text field for the active machine list.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. This setting affects all projects for the current configuration for the user and machine..

> **Note:**
>
> Ansys recommends that you use the **RAM Limit**setting if you wish to limit the RAM that the MPS (default) solver may allocate. However, the Iterative solver doesn't go off-core for efficiency reasons. Once the limit is exceeded, iterative solver will issue an "Out of memory" error message.

> **Note:**
>
> Allocation of greater than 2 GB of RAM on 32-bit hardware platforms is only possible with the appropriate operating system and boot settings, even if more than 2 GB are physically installed.

Distributed Memory Solutions with HFSS

When properly configured, and with Domain Decomposition selected as a Setup option, the HFSS solver uses the industry standard Message Passing Interface ("MPI") and can perform solutions that distribute memory use across machines in a cluster or network. Memory used by the MPI-enabled HFSS solver is therefore limited by the set of machines that are available rather than the shared memory available on any single machine. This allows you to simulate larger structures than before and to optimally reconfigure the cluster of machines for the problem at hand.

To use the distributed memory solution in HFSS you will need to install HFSS and MPI software from one of the supported third party vendors on all the machines you intend to use. You may need to set passwords depending on the MPI vendor for authentication on the machines. Settings within HFSS are used to turn on distributed memory solutions and define the list of machines you intend to use. If an HFSS problem involves solver domains or a finite array, then frequency sweeps will not be done using DSO. Also, DSO for Optimetrics will not be allowed.

Detailed instructions about how to get distributed memory HFSS solutions up and running are outlined in the following sections.

- Installation Requirements for Distributed Memory Solutions with HFSS
- Setting up HFSS and Running Distributed Memory Solutions
- Select the MPI Version
- Running HFSS Distributed Memory Solutions from the Command line
- Discussion of HFSS Distributed Memory Solution
- Interconnects for HFSS Distributed Memory Simulation

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Authentication on Linux (RSH & SSH) for HFSS
- Troubleshooting for HFSS Distributed Memory Solutions

Installation Requirements for Distributed Memory Solutions with HFSS

You must install HFSS on all the machines you intend to use during the HFSS distributed memory solution process. The installation locations for HFSS must be identical and the machines must be uniform: all 64 bit, all Windows or all Linux. The machines must all use the same interconnect.

In addition, on Windows, you must install one of the supported versions of MPI from either Microsoft or Intel. Be sure to install the same version of MPI on all machines in your cluster. (Solving on a single Windows machine does not require MPI installation. Users running on Linux do not need to install MPI manually).

Platform	MPI Software
Linux Intel & AMD	Intel MPI 2018.3.210
Windows 10	Microsoft MPI 10.0
	Intel MPI 2018.3.210

You will need to set the password you want to use for your MPI runs on all the machines in the cluster. You can either use the batch command provided by Ansys to set your password or refer to the MPI vendor's documentation.

InfiniBand Support for Windows

By default, the MPI vendors use the fastest interconnect by default (typically InfiniBand is faster than Ethernet). If you want to override the default behavior and force the use of Ethernet, you can set the ANSOFT_MPI_INTERCONNECT environment variable to “eth” for the job.

Setting up HFSS and Running Distributed Memory Solutions

After setting up your HFSS project, you normally need to set the list of machines used for the distributed memory solution process and configure the distributed memory interface.

The distributed machine list is selected under Tools>Options>HPC and Analysis Options, and created under the Analysis Configuration dialog.

Distributed memory solutions are turned using the **Domain Decomposition** radio button under the Solution Setup Options tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

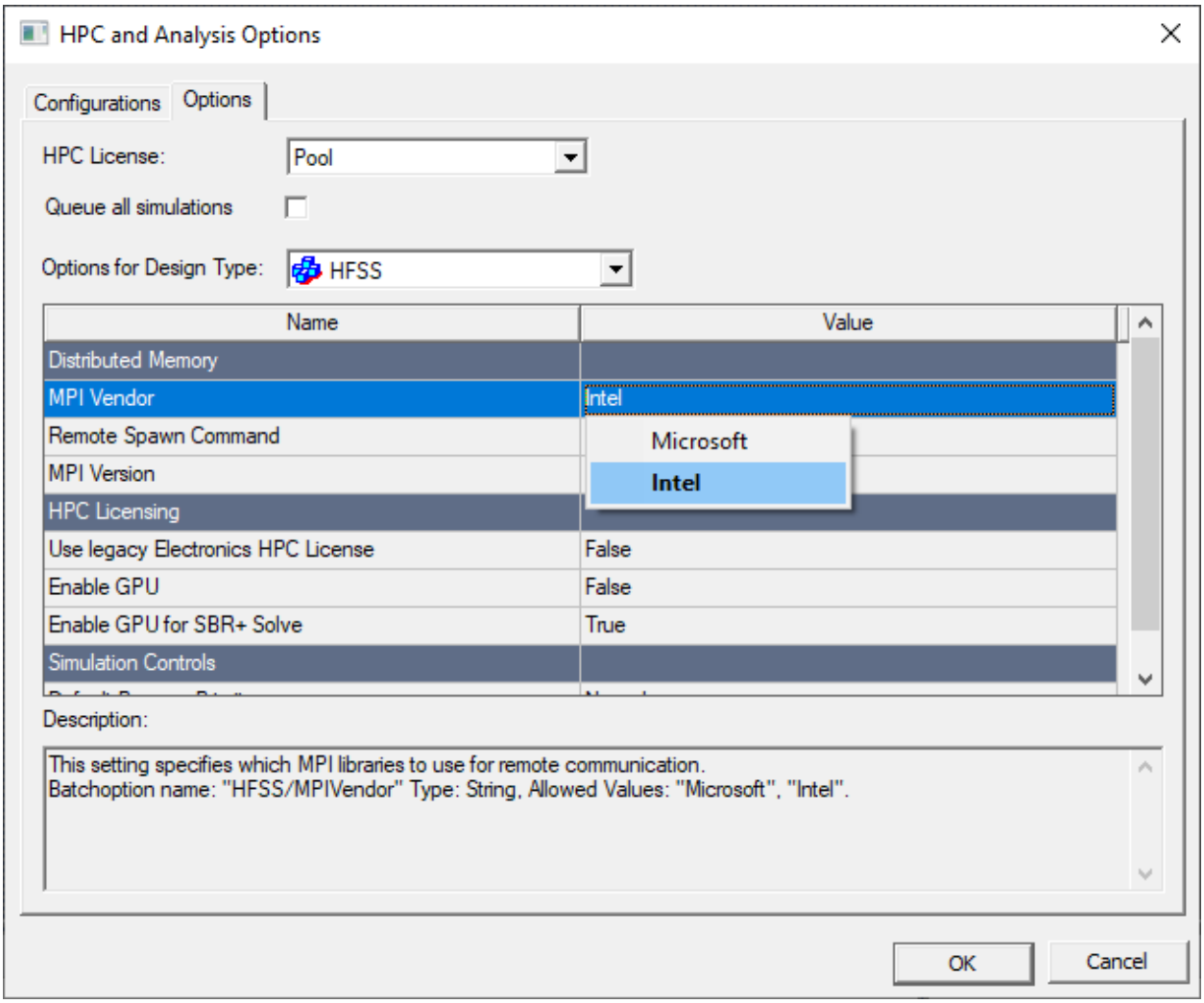
If an HFSS problem involves solver domains or a finite array, then frequency sweeps will not be done using DSO. Also, DSO for Optimetrics will not be allowed. Distributed frequency solutions are also known as "DSO" solutions.

MPI Vendor and MPI Version

For HFSS, HFSS 3D Layout, Q3D Extractor, Maxwell, and Icepak design types:

These settings may be used to specify the MPI Vendor and Version for distribution via MPI.

Go to the **Options** tab of the **HPC and Analysis Options** dialog box, and select the design type. Then you can select the MPI Vendor and MPI version.



MPI Vendor

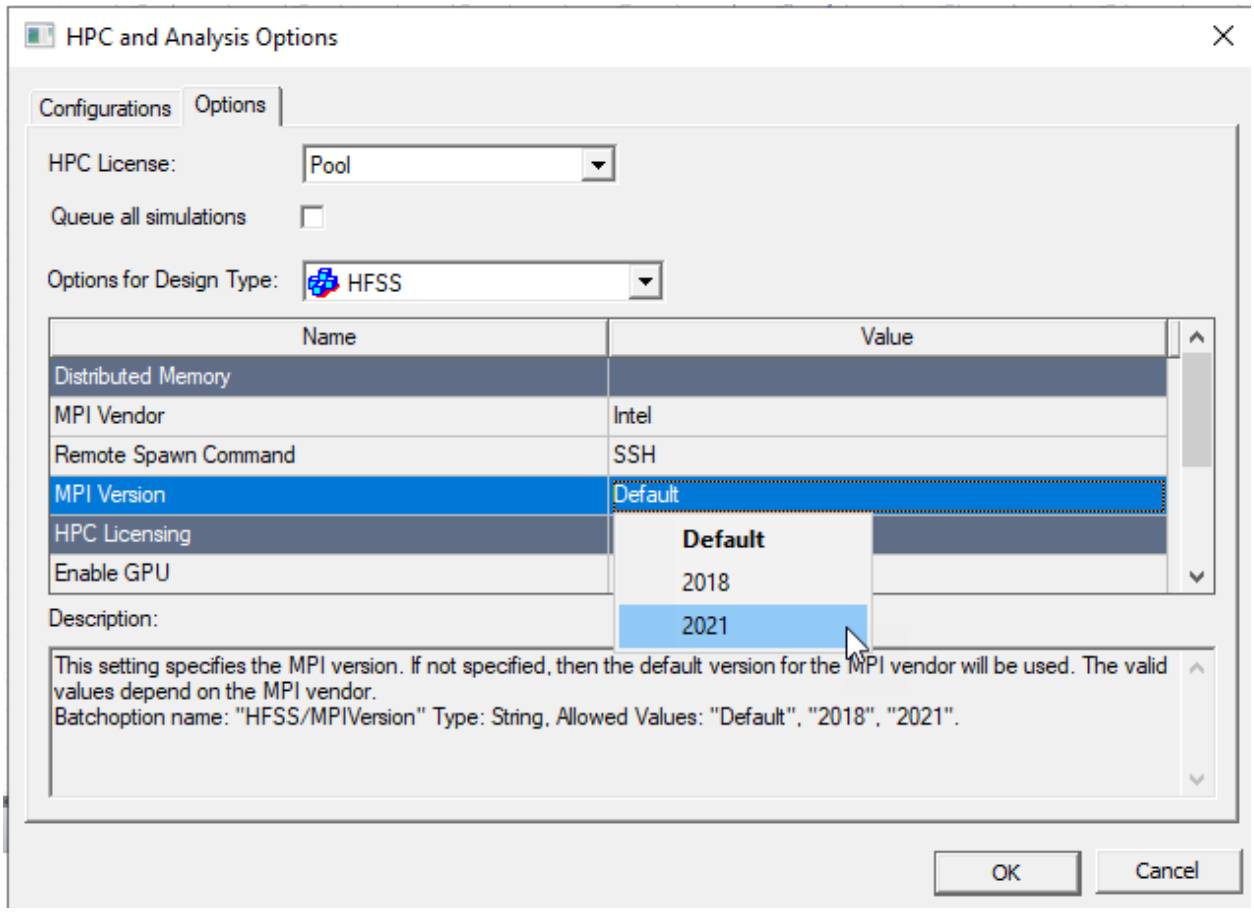
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The default MPI Vendor is "Intel". On Linux, "Intel" is the only supported MPI vendor. On Windows, both "Intel" and "Microsoft" are supported MPI vendor options. For jobs distributed to multiple hosts, the "Microsoft" MPI Vendor is only supported when running as a Windows HPC job.

MPI Version

If only a single MPI Version is supported for the selected MPI Vendor, then the MPI Version option is ignored. Multiple versions are only supported for Intel MPI, as follows:

- The value "Default" indicates that the default Intel MPI version should be used. This is Intel MPI 2021 in most cases.
- The value "2018" indicates that Intel MPI 2018 should be used.
- The value "2021" indicates that Intel MPI 2021 should be used.



Also see Setting Intel MPI Interconnect for more details.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Intel MPI Interconnect

To use a particular interconnect or interconnect variant, the user must set ANSOFT_MPI_INTERCONNECT and ANSOFT_MPI_INTERCONNECT_VARIANT environment values. For example, if you want to override the default behavior and force the use of Ethernet, you can set the ANSOFT_MPI_INTERCONNECT environment variable to appropriate Ethernet interface name for the job.

Supported options for MPI interconnect with Electronics Desktop based on Intel MPI Versions are described below.

With Intel MPI 2018

- ANSOFT_MPI_INTERCONNECT = mpi-auto-selected | shmem | myri | myrinet | ib | infiniband | default | eth | ethernet | (not set)
- For Linux ANSOFT_MPI_INTERCONNECT=infiniband | ib, you can set interconnect variant values to:
  ANSOFT_MPI_INTERCONNECT_VARIANT = ofed | infinipath | dapl | (not set)

With Intel MPI 2021

The Intel® MPI Library switched from the Open Fabrics Alliance* (OFA) framework to the Open Fabrics Interfaces* (OFI) framework and currently supports libfabric*. The DAPL, TMI, and OFA fabrics are deprecated.[1]

Intel MPI 2021 interconnect selection for Windows

- ANSOFT_MPI_INTERCONNECT = mpi-auto-selected | shmem | ib | infiniband | default | eth | ethernet | efa | (not set)

By default, MPI vendors use the fastest interconnect by default (typically InfiniBand is faster than Ethernet). To override the default behavior and force the use of specific interconnect, you can set the ANSOFT_MPI_INTERCONNECT.

Intel MPI 2021 interconnect selection for Linux

- ANSOFT_MPI_INTERCONNECT = mpi-auto-selected | shmem | ib | infiniband | default | eth | ethernet | efa | (not set)
- For ANSOFT_MPI_INTERCONNECT=infiniband | ib, user can set interconnect variant values to:
  ANSOFT_MPI_INTERCONNECT_VARIANT = verbs | mlx

For Linux environments, Intel MPI 2021 requires setting fabric provider variables, FI_PROVIDER, I_MPI_OFI_PROVIDER and FI_TCP_IFACE/FI_VERBS_IFACE (i.e. in case fabric provider is TCP/VERBS respectively) for a successful run. AEDT will set these values based on the available interconnects on Linux. AnsysEM will always prefer ethernet.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** This interconnect selection by AEDT does not guarantee the selection of faster interconnect. To bypass this interconnect selection, the user needs to set FI_PROVIDER and I_MPI_OFI_PROVIDER explicitly.

[1] https://www.intel.com/content/www/us/en/developer/articles/technical/mpi-library-2019-over-libfabric.html

Running Distributed Memory Solutions from the Command line

You can run distributed memory HFSS solutions from the command line using the -BatchSolve option. Set the distributed memory solve setup option before running the simulation and use the "BatchSolve" flag with the "Distributed" and MachineList" options. For example:

```
ansysedt -BatchSolve -Distributed -MachineList list="machine1,
machine2" TheProject.aedt
```

This simulates "TheProject.aedt" as a distributed memory solution on machines "machine1" and "machine2."

Note the distributed memory solve setup option can be turned on via scripting if desired.

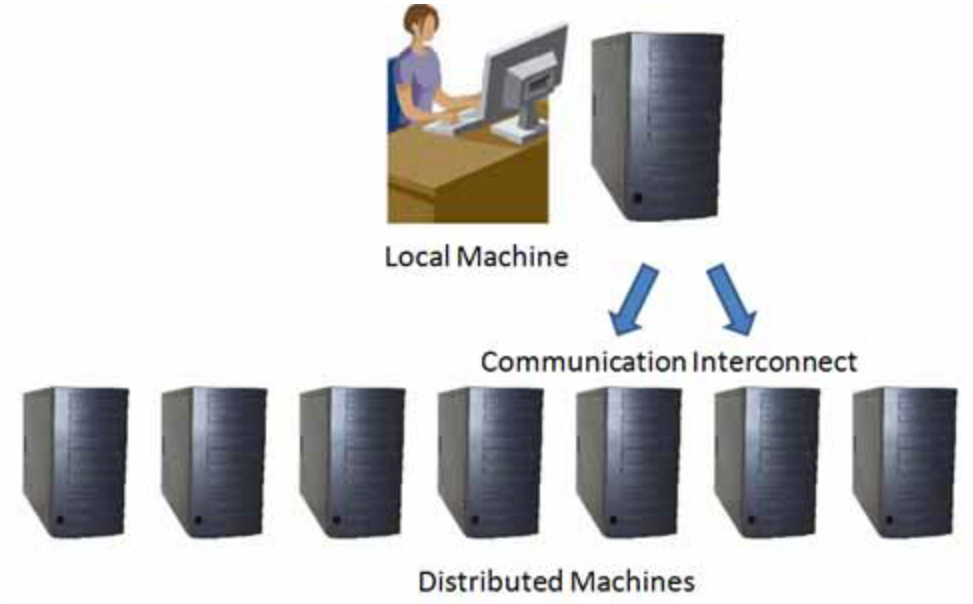
Discussion of HFSS Distributed Memory Solutions

Each machine or "compute node" is connected to the other nodes via a "communication interconnect" and relies on the message passing library (MPI) to exchange data and synchronize computational tasks. Ethernet, Myrinet and Infiniband are common communication interconnects. Each node is identified by a unique integer ID or rank number. The local machine is known as the "Rank-0 node" and is the master. The Rank-0 machine has many tasks including:

- Management of all communication with the HFSS user interface. None of the distributed machines communicate directly with the user interface but pass all information through the Rank-0 machine.
- Mesh generation. The mesh is generated only on the Rank-0 machine.
- Disk access. None of the distributed machines access their local discs. The simulation mesh, intermediate and solution data are passed to and from the distributed machines using MPI.
- Distribution and control of computational tasks on the distributed machines. The simulation process is dynamic and the Rank-0 machine will determine which of the distributed machines has memory available to distribute tasks accordingly.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Post-processing of the HFSS Distributed Memory Solution



The algorithms used in the distributed memory version of the HFSS solver engine resemble those used in the non-distributed memory version. The matrix solution algorithms in the distributed memory version of HFSS have been adjusted to use slightly more memory so that larger problems can be simulated in less time.

Most of the ANSYS solvers are multi-threaded. You can specify the hostname, # tasks, and # cores instead of repeating the hostname multiple times. The "auto" mode eliminates the need for end-users to specify tasks. They simply indicate the number of cores to use on the specified machine.

During the "Matrix Assembly" and "Matrix Solve" steps of the solution process the HFSS engine attempts to distribute memory use evenly. At various points in the matrix solution process the software will poll the machines in the cluster and determine which machine has the most memory available and then reserve a block of memory on that machine. If a particular machine does not have a large block of memory available the memory use on that machine will grow only slowly. If none of the machines in the cluster have sufficient memory the solution process will terminate and an error message will be posted to the HFSS message window.

Many factors affect solution time. In general, the solution time will decrease as the number of compute nodes increases. However, parallel efficiency decreases as the ratio of communication to computation increases so to some extent you need to match the size of the problem to the size of parallel machine. Simulating small structures on a large cluster will not be efficient and may take longer and use significantly more memory than if the structure was simulated on a single machine.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Network interconnect speed and topology can affect performance significantly in homogeneous clusters. Performance can degrade if machines are "doubled up" to the point of causing memory bus contention or if the cluster is significantly inhomogeneous and certain faster machines need to wait for slower machines to catch up to synchronization points in the solution process.

Interconnects for HFSS Distributed Memory Simulation

To obtain the best possible performance we recommend the use of a network interconnect that supports communication speeds greater than 1000MB/sec or higher. Some high performance interconnects plug into a PCI (Peripheral Component Interconnect), PCI-X (extended), or PCIe (PCI Express) slot on the system.

HFSS-IE 15 supports the following network interconnects:

Platform	Interconnects
Win32	Ethernet/GiGE
Win64	Ethernet/GiGE (default), Myrinet, Infiniband
Linux	Ethernet/GiGE (default), Myrinet, Infiniband

Ethernet/GiGE is the default interconnect on all platforms. You can choose one of the alternate interconnects by setting the ANSOFT_MPI_INTERCONNECT environment variable to "myri" for Myrinet and "ib" for Infiniband.

Interconnect variants are supported on Linux. Set the ANSOFT_MPI_INTERCONNECT_ VARIANT to the desired interconnect variant. For example, set "ANSOFT_MPI_ INTERCONNECT_VARIANT=silverstorm" to use the silverstorm variant.

Authentication on Linux (RSH & SSH) for HFSS

An important step in using a high performance cluster is setting up authentication across machines in such a way that the machines can be accessed without a password. By default HFSS uses SSH authentication on Linux to spawn commands on the remote machines but also supports RSH. The selection of which to use is made on the **Options** tab of the Tools/Options/HPC and Analysis Options dialog .

Troubleshooting for Distributed Memory Simulations

A number of things can prevent distributed memory solutions from completing successfully. This section provides suggestions to debug problems.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

It is often a good idea to set up and run a small simulation with two processes on a single machine before moving to a large cluster. Using a single machine will allow you to verify that the EM software and MPI are installed correctly while eliminating problems arising from remote installation, authentication and firewall settings.

Many problems occur because the MPI software cannot start and run due to authentication and firewall issues. Please check with your MPI vendor and their end user documentation for information about how to verify that authentication and firewall settings are correct.

Recall that HFSS must be installed in the same directory on all machines in the cluster and that the cluster must be uniform (e.g., all Linux machines, all 64 bit Windows machines).

Verify that the version of the third party MPI software is identical on all the machines and that it is a supported version.

Verify that the machine names are correct and that all the machines can be reached on the network.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17 - Assigning Boundaries in HFSS

Boundary conditions specify the field behavior at the edges of the problem region and object interfaces. Within the context of HFSS, boundaries exist for two main purposes:

1. to create either an open or a closed electromagnetic model or,
2. to simplify the electromagnetic or geometric complexity of the electromagnetic model.

In an open problem, an air volume encompassing the outer radiating surfaces is modeled by a surrounding object. Radiation boundary conditions (ABC, PML or FEBI) are assigned to the outer radiating surfaces (i.e., faces of the region) to absorb all outgoing waves. HFSS users can choose one of the three workflows to analyze an "open" design.

1. Auto-open region mode using the Solution type setting.

   This workflow eliminates as much as possible the required interactions from a user and allows a designer to get to a robust solution quickly. It is intended for users who do not want to be aware of the region and the exterior radiation boundaries.

   The open problem is enforced by the software automatically creates/maintains the region object and the ABC boundaries. HFSS applies the most appropriate out-of-the-box settings based on the geometries and the operating frequency. You do not need to (and cannot) edit these settings directly.

   This mode restricts the design to a single solve setup. Users cannot manually create the region object, radiation boundaries or PML in this mode. Infinite ground plane, metallic IE region, primary/secondary (Lattice pair) and symmetry boundaries are also not allowed.

2. Automation in setting up an open region problem using the **Create Open Region** command and **Update Open Region Padding** commands.

   This workflow provides automations that assist a user to quickly setup an open problem. As a single user-customizable command, the software creates the region and the radiation boundaries based on user's selections on the region padding type, radiation boundary type, etc.

   If necessary, you can further customize the design by editing these design objects directly.

3. Advance mode

   This is the original workflow where you manually set up the open design by assigning radiation boundary conditions.

You may assign the following types of boundaries to an HFSS design:

Anisotropic	Represents a boundary condition used to replace a surface a planar screen

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Impedance	or grid with periodic geometry.
Aperture	Represents a hole in a metallic sheet assigned as an IE Region.
Finite Conductivity	Represents an imperfect conductor.
Fresnel (SBR+)	Represents either a perfect absorbing boundary or a user-defined Fresnel table file that describes a good approximation of the reflection of an arbitrary wavefront off a surface. Supported for driven modal/terminal and SBR+ solution types.
Gradient Surface Roughness	Gradient Surface Roughness Boundaries address the situation where the conductors drawn in CAD models are perfectly smooth, but the manufactured surface is bumpy at microscopic scale.
Half Space	Represents a background comprising a dielectric half space. For antenna and scattering problems in HFSS in which all objects are assigned as Hybrid IE Region.
Impedance	Represents a resistive surface.
Lattice Pair	A Coupled pair, the same as one Primary Boundary and one Secondary Boundary together, but with an auto-set coordinate system. A red arrow points from the primary to the secondary face of the Lattice Pair.
Layered Impedance	Represents a structure with multiple layers as one impedance surface.
Linked Impedance	Represents a data link to an isotropic or anisotropic impedance boundary in another design. These can include infinite ground planes and shell elements.
Lumped RLC	Represents any combination of lumped resistor, inductor, and/or capacitor in parallel on a surface.
Multipaction SEE	For Multipaction Analysis, this represents multipaction Secondary Electron Emission (SEE). The Multipaction SEE boundaries should be added to vacuum-material interfaces where secondary electrons will be generated.
Perfect E	Represents a perfectly conducting surface.
Perfect H	Represents a surface on which the tangential component of the H-field is the same on both sides.
PML	Represents an open boundary condition using several layers of specialized materials that absorb outgoing waves.
Primary	Represents a surface on which the E-field at each point is matched to another surface (theSecondary boundary) to within a phase difference.
Radiation	Represents an open boundary by means of an absorbing boundary condition (ABC) that absorbs outgoing waves.
Secondary	Represents a surface on which the E-field at each point has been forced to match the E-field of another surface (the Primary boundary) to within a phase difference.
Symmetry	Represents a perfect E or perfect H plane of symmetry.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You may also choose to designate a perfect E, finite conductivity, or impedance boundary as an infinite ground plane if you want the surface to represent an electrically large ground plane when the radiated fields are calculated during post processing.

> **Note:**
>
> Hiding boundaries also turns off a check for boundary overlaps during boundary assignment. In the case of very large models with many boundaries, hiding boundaries can prevent delays during boundary assignment. Full model validation will subsequently check for boundary overlaps.

For convenience, you can access the **Edit Global Materials** command from the Boundaries menu.

> **Note:**
>
> By default, the history tree in the 3D modeler window groups sheet objects according to boundary assignment. To change this, select the **Sheets** icon and right-click to display the **Group Sheets by Assignment** check box.

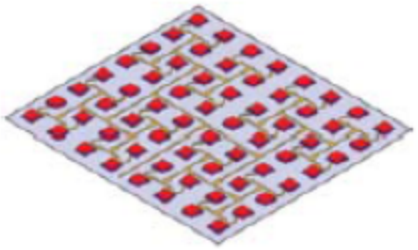
Boundaries for Open or Closed Models

In HFSS a closed model simply represents a structure, or a solution volume, where no energy can escape except through an applied port. For an Eigenmode simulation, this could be a cavity resonator. For a driven modal or terminal solution, this could be a waveguide or some other fully enclosed structure.

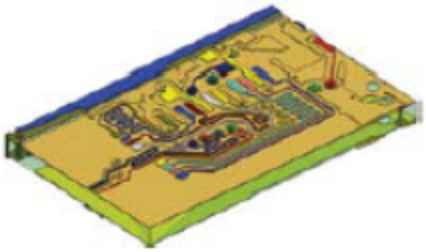
An open model represents an electromagnetic model that allows electromagnetic energy to emanate or radiate away. Common examples would be an antenna, a PCB, or any structure that is not enclosed within a closed cavity. While most HFSS simulations deal with models that are open, by default, HFSS initially assumes that any given model is closed. HFSS assumes all outer surfaces of the solution space are covered, or coated, by a perfect electric conductor boundary. In order to create an open model, you specify a boundary on the outer surfaces that will overwrite the default perfect electric conductor boundary.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Electromagnetically "Open" Structures



Antenna Array



Printed Circuit Board

Electromagnetically "Closed" Structures

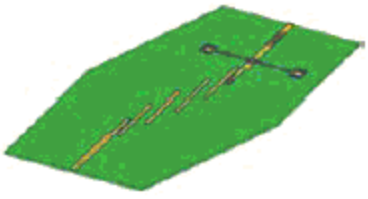
Waveguide

Cavity Resonator

Boundaries for Simplifying Models

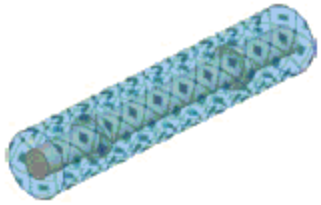
Boundaries can be used within HFSS to decrease the geometric/electromagnetic complexity of a given structure or model. These boundaries should only be used internally to a model or possibly on a symmetry plane (as in the Dielectric Resonator Antenna example or the Coax Bend Transient example). They should be applied to specifically created 2D sheet objects or to specific surfaces of 3D objects. While boundaries can be very useful, a user should exercise caution when using them as they can create unintended results if applied incorrectly.

Electromagnetically Simplified Structure



Filter where microstrip traces are modeled using 2D objects with boundary applied.

Geometrically Simplified Structure



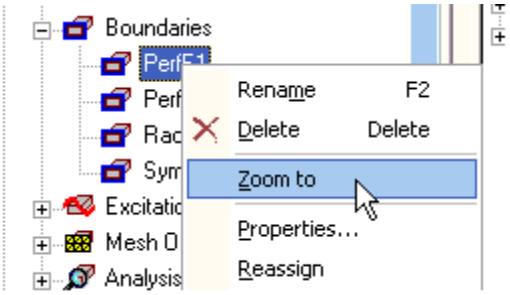
Coaxial structure where "shield" is replaced by appropriate boundary.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Not every HFSS model, however, will use simplifying boundaries. When using boundaries to create simpler models, users should take care to not create a model that has unreasonable or inappropriate boundaries applied.

Zoom to Selected Boundary

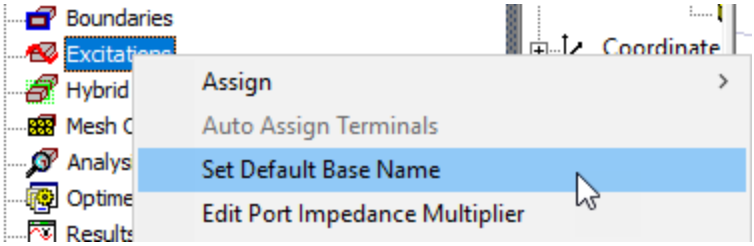
To zoom to a selected boundary, right-click a boundary name in the Project and select the **Zoom to** command on the pop-up menu. This zooms the view in the Modeler window in or out to show the selected boundary. The current orientation does not change. This can be very useful checking the assigned geometry.



Setting Default Boundary, Hybrid Region and Excitation Base Names

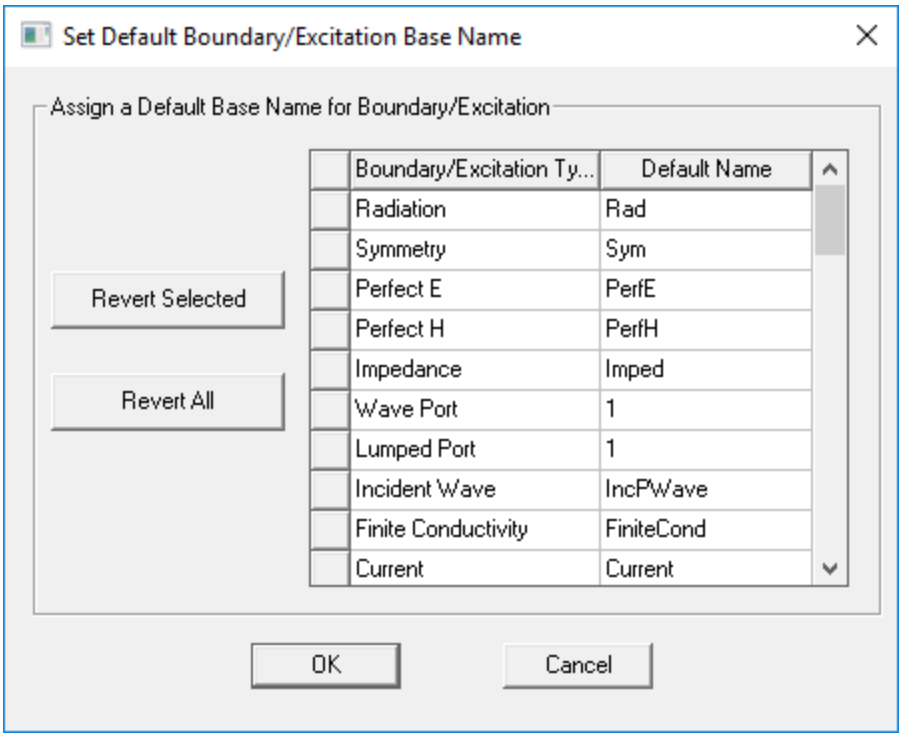
To change the default boundary, hybrid region, or excitation base names, so that subsequent names increment from the base of your choosing:

1. Click **Boundaries>Set Default Base Name** or **Excitations >Set Default Base Name** or **Hybrid Regions>Set Default Base Name**.



This displays the **Set Default Boundary/Excitation Base** dialog box. This contains a list of all boundary, hybrid region, and excitation types, and the base names for each. The base names for each type have editable text fields. The base names for boundaries and excitations are incremented from the base names here.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Edit the text fields to your preferred naming conventions. Names must be less than sixty characters and cannot include spaces. Letters, numbers, and special characters are permitted. Illegal names are not accepted and generate a warning message.
3. Click OK to accept the changes or Cancel to close the dialog without accepting changes.

If you want to revert all or selected names to defaults, use the **Revert All** or **Revert Selected buttons**.

Excitation Name Ordering Conventions

Ansys Electronics Desktop tries to sort excitations by name, using a sort of "improved" dictionary (lexicographic) ordering that takes into account trailing numerals as a special case.

So if you create some ports in the following chronological order, Port3, Port1A, Port10, Port1, Port3A, Port1A1, Port10A, you will find that they appear in the list as:

Port1, Port3, Port10, Port10A, Port1A, Port1A1, Port3A

Explanation:

- Port1, Port3, and Port10 start with the same string but end in trailing numerals, so they are sorted according to the numeric value of the trailing numerals: 1 < 3 < 10. This is different from the standard dictionary sorting order, which would order them as Port1, Port10, Port3.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Port10A and Port1A both start with 'Port1' but don't end with trailing numerals, so standard dictionary sorting applies. Numbers come before letters, thus Port10A comes before Port1A (0 is before A in position 6).
- Shorter string ABC comes before longer string ABCD (ABC is a "prefix" of ABCD), so Port10 is before Port10A, and Port1A is before Port1A1.

If the leading strings contain punctuation characters (. and _). The ordering is by the character's ASCII code, so period (ASCII 46) is before underscore (ASCII 95.)

Automation in Setting up an Open Problem

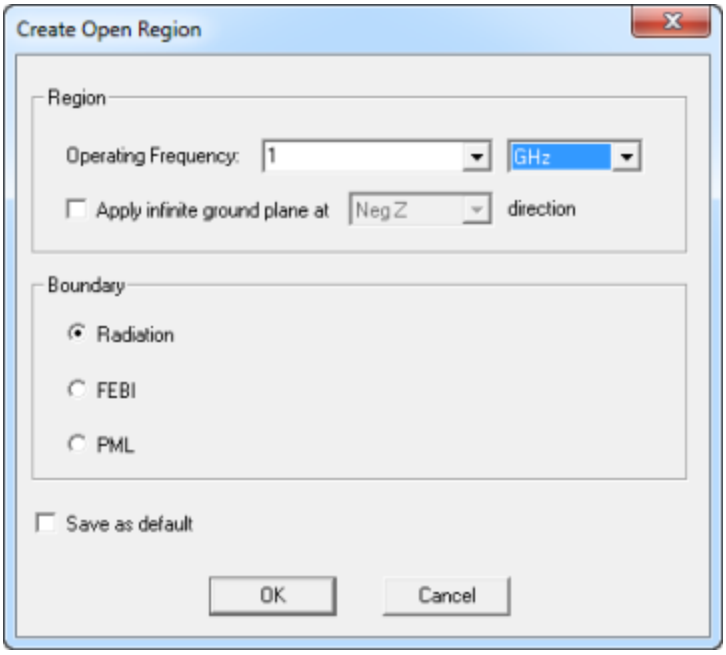
In an open problem, an air volume encompassing the outer radiating surfaces is modeled by a surrounding object. Radiation boundary conditions (ABC, PML or FEBI) are assigned to the outer radiating surfaces (i.e., faces of the region) to absorb all outgoing waves. HFSS users can use the **Create Open Region** command and **Update Open Region Padding** command to automate the setup of an open region problem. If you have specified the **Auto Open Region** check box in the Solution type, you will receive an error message. You must leave that check box unchecked to use this approach to automating an open problem.

You can access the **Create Open Region** command in three different ways:

- Right-click in the modeler window and select **Create Open Region...** from the context menu.
- Click **HFSS>Model>Create Open Region...**
- Right-click Model in the Project tree and select **Create Open Region...**

This displays the **Create Open Region** wizard.

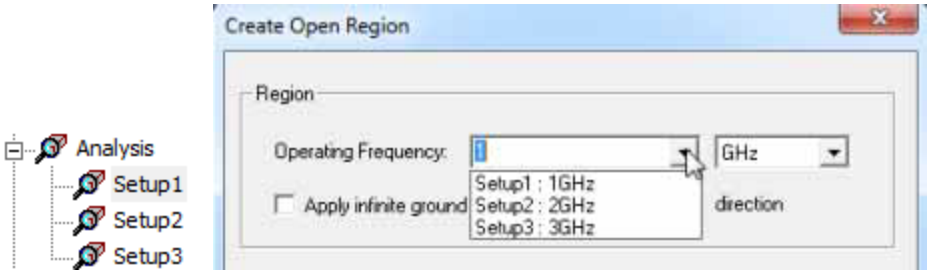
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Your selections here guide the software in setting up the region and boundaries that are required to model the device as an open problem.

- Operating Frequency: This value is used to determine the region padding and CANNOT be parameterized. The default factory value is 1 GHz when there is no solve setup in the design.

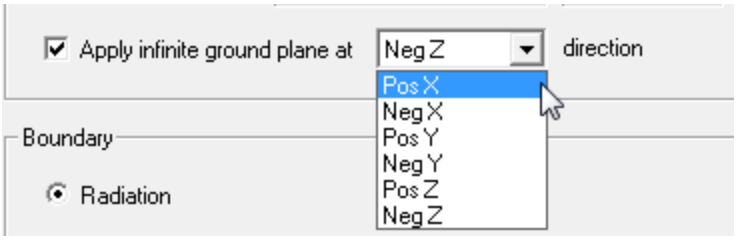
If you can created two or more solution setups, a pull-down provides you the ability to select the adaptive frequency. The default is to use the value from the first solve setup (in alphabetical order).



By option you can manually type in a value.

- Infinite ground plane can be specified at one of the 6 directions by checking Apply infinite ground and selecting from the drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Default is at the Neg Z direction.

- Boundary can be one of:

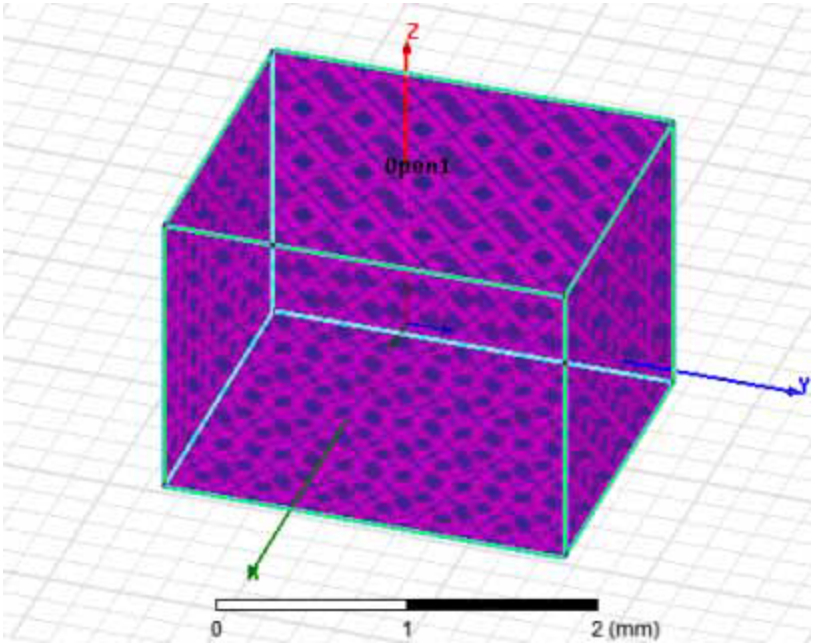
Radiation, an absorbing boundary condition (ABC) that absorbs outgoing waves..

FEBI, finite element-boundary integral (FEBI) method. Unlike ABC and PML, the IE hybrid region can be of arbitrary shape, both concave and convex thus in some cases allowing the size of the finite element solution domain to be significantly reduced.

PML, perfectly matched layer, uses several layers of specialized materials that absorb outgoing waves.

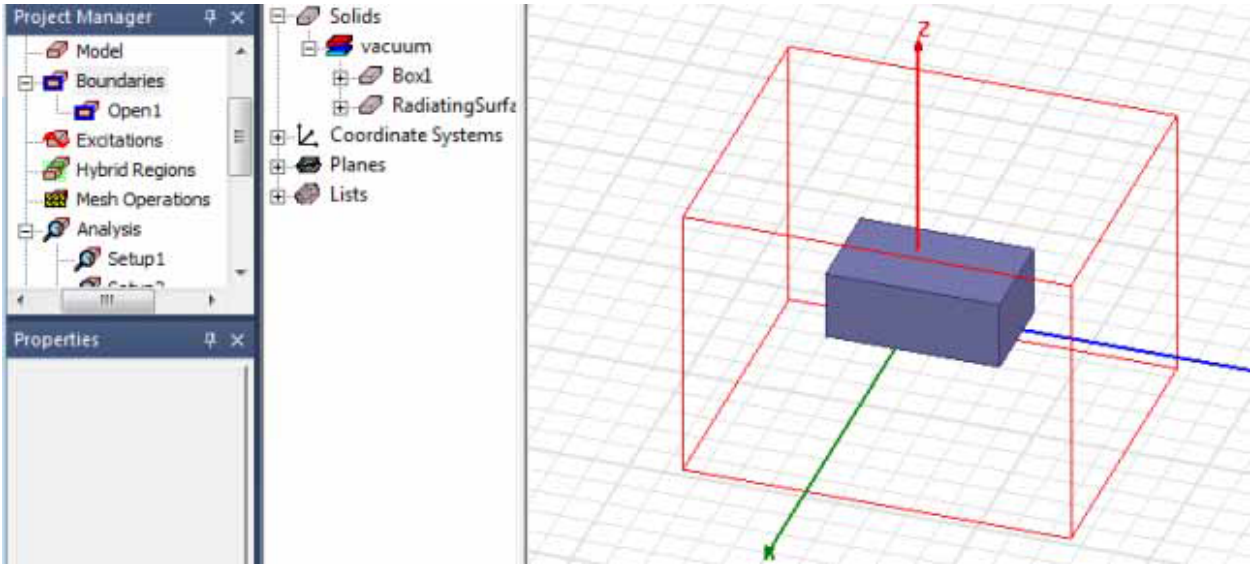
Default region padding = min(L/2, Lambda/<Boundary Type Specific>) where L is the diagonal of device's bounding box and Lambda is the wavelength of the operating frequency. This is the same formulation used in the "auto-open" design mode. The padding for ABC = min(L/2, Lambda/3). The padding for FEBI is min(l/2, Lambda/8). The padding for PML = min (L/2, Lambda/4).

When you have made your selections, and OK the dialog, region is created and displayed as a wireframe with the boundary highlighted as selected.

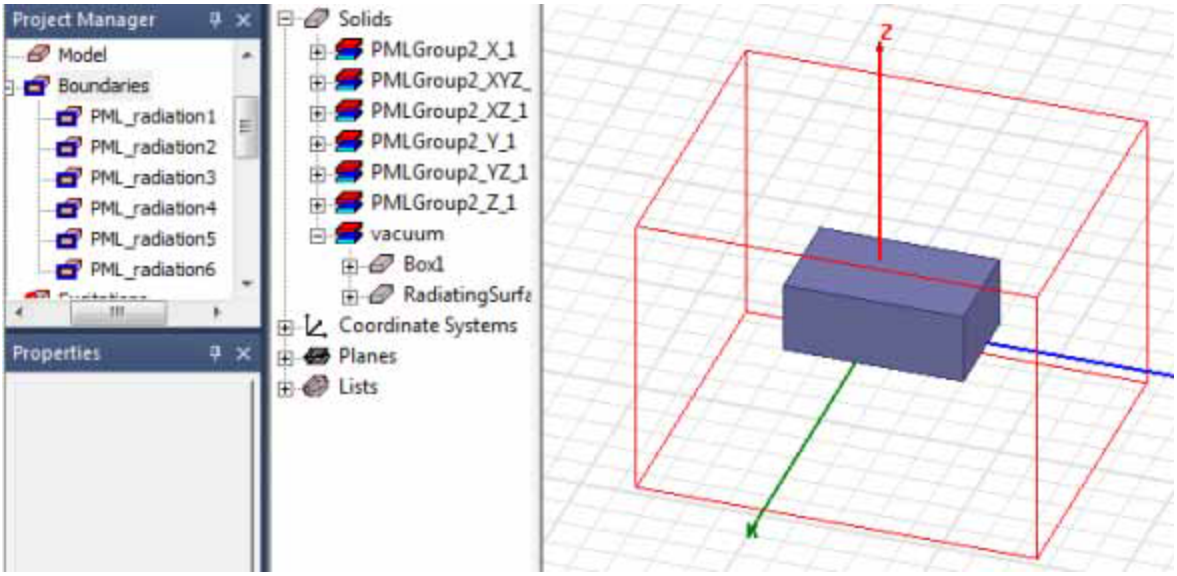


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clicking outside the boundary de-selects the boundary. The boundary appears in the Project tree under boundaries (for Radiation and PML) or Hybrid regions (for FEBI) and under radiating surface in the history tree.



In the case of a PML, the Project manager window and history tree show the PML objects. The visibility of these objects are defaulted to off.



You can edit the properties of the created region objects in the history tree. You can also edit the properties of radiation and FEBI boundaries in the Project tree. More direct control of PML settings can be had by creating the PML boundaries via the PML wizard.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The region's size is automatically updated if you change the size of your modeled device, but the region padding is not updated automatically.

Recalculate the Padding for Regions

The **Update Open Region Padding** command is enabled whenever there is a region object in a design. To Recalculate the Padding for Regions, select **Update Open Region Padding**.

- Right-click in the modeler window and select **Update Open Region Padding...** from the context menu.
- Click **HFSS>Model>Update Open Region Padding...**
- Right-click Model in the Project tree and select **Update Open Region Padding...**

This opens the **Update Open Region** dialog.

The function resembles that in the **Create Open Region** dialog box, where you can specify the operating frequency by selecting an already created setup or manually typed a value.

On **OK**, the region padding is updated based on the out-of-the box formulation min(L/2, Lambda/4) where L is the diagonal of the device's bounding box and Lambda is the wavelength of the adaptive frequency of the solve setup. The region is automatically resized when users edit the solve setup's adaptive frequency. Initial meshes and solutions are invalidated when the region is resized.

For example, suppose you began with a device diagonal size as 10 mm. You then launched the wizard and specified an operation frequency of 10 GHz. This ends with the region padding set to 5 mm. You then increase the device diagonal to 20 mm. The region is resized to encompass all geometries but padding remains at 5 mm. You then launch **Update Open Region Padding** and specify an operation frequency of 10 GHz On **OK**, the region padding is updated to 7.5 mm.

Assigning Perfect E Boundaries

A perfect E boundary is used to represent a perfectly conducting surface in a structure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To create a Perfect E boundary

1. Select a surface on which to assign the boundary and click **HFSS>Boundaries>Assign>Pefect E** to bring up the **Perfect E Boundary** dialog box.

2. For HFSS projects, you can select **Infinite Ground Plane** if you want the surface to represent an electrically large ground plane when the radiated fields are calculated during post processing. For PEC boundaries only, multiple infinite ground planes are supported.

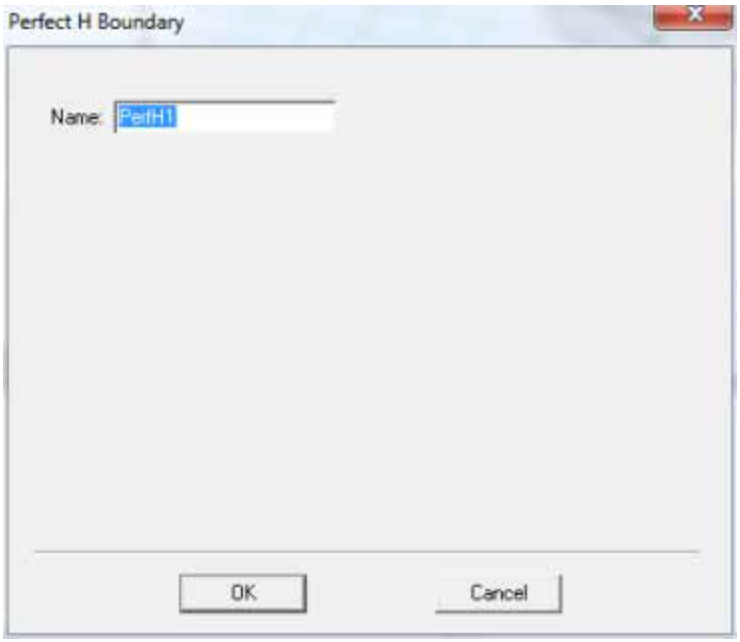
Assigning Perfect H Boundaries

A perfect H boundary represents a surface on which the tangential component of the H-field is the same on both sides. For internal surfaces, this results in a natural boundary through which the field propagates. For surfaces on the outer surface of the model, this results in a boundary that simulates a perfect magnetic conductor in which the tangential component of the H-field is zero.

To Assign a Perfect H Boundary:

1. Select a surface on which to assign the boundary and click **HFSS>Boundaries>Assign>Pefect H** to bring up the **Perfect H Boundary** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

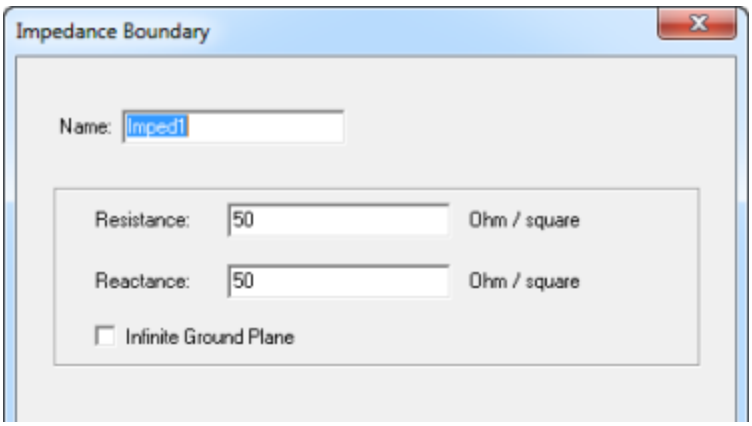


2. No parameters need be set..

Assign Impedance Boundaries

An impedance boundary (IB) represents a resistive surface. The behavior of the field at the surface and the losses generated by the currents flowing inside the resistor are computed using analytical formulae. Ansys Electronics Desktop does not simulate any fields inside the resistor.

1. Select a surface on which to assign the boundary, right-click **Assign Boundary>Impedance** to bring up the **Impedance Boundary** dialog box.



2. Enter the **Resistance** and **Reactance** for the boundary.

The default Resistance and Reactance are 50 Ohms per square.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you want to create a boundary that demonstrates **spatial dependence**, you must first assign that boundary to a sheet object. Then to define a spatial dependence for the boundary, you can create an expression for resistance or reactance using x, y, or z. For example, 50+y.

3. Select **Infinite Ground Plane** if you want the surface to represent an electrically large ground plane when the radiated fields are calculated during post processing. .

> **Note:**
>
> If you select **Infinite Ground Plane**, the effect of the impedance boundary will be incorporated into the field solution in the usual manner, but the radiated fields will be computed as if the lossy ground plane is perfectly conducting. Only one infinite ground plane is permitted in designs with impedance boundaries

> **Note:**
>
> You can assign a variable as the resistance and reactance values. Eigenmode designs cannot contain design parameters that depend on frequency, for example, a frequency-dependent impedance boundary condition.

Assigning Radiation Boundaries

> **Note:** For information on assigning radiation boundaries for Maxwell, see Assigning a Radiation Boundary for the Eddy Current Solver.

For Eigenmode, Driven Modal or Driven Terminal Designs

A radiation boundary is used to simulate an open problem that allows waves to radiate infinitely far into space, such as antenna designs. Ansys Electronics Desktop absorbs the wave at the radiation boundary, essentially ballooning the boundary infinitely far away from the structure. In HFSS, these are sometimes described as Absorbing Boundary Condition, or ABC.

A radiation surface does not have to be spherical, but it must be exposed to the background, convex with regard to the radiation source, and located at least a quarter wavelength from the radiating source. In some cases the radiation boundary may be located closer than one-quarter wavelength, such as portions of the radiation boundary where little radiated energy is expected.

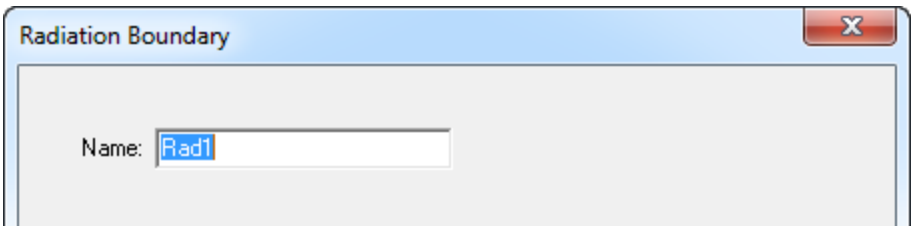
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Whenever additions/changes are made to radiation boundaries that affect fields, it invalidates those solutions that can possibly have fields. Meshes are not invalidated.

To assign a radiation boundary:

1. Select a surface on which to assign the boundary and click **HFSS>Boundaries>Assign>Radiation** to bring up the **Radiation Boundary** dialog box.



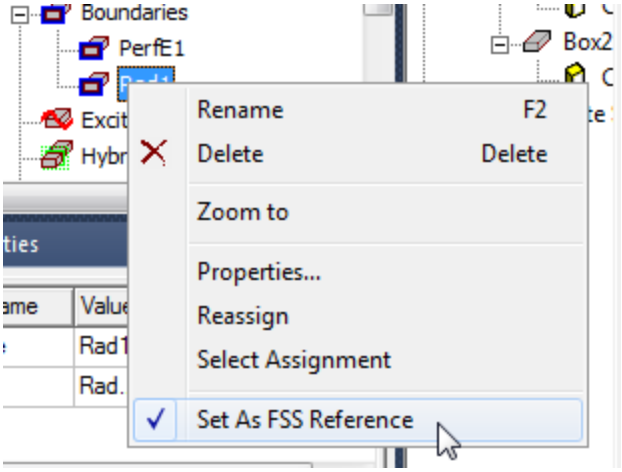
Note:

Do not define a surface that cuts through an object to be a radiation boundary.

In general, do not define the interface of two internal objects to be a radiation boundary. The only exception is when one object is a perfectly matched layer boundary (PML) and the other is the PML base object.

Set as Frequency Selective Surface Reference

You can set one reference of a Frequency Selective Surface. To setup Frequency Selective Surface, users right-click a radiation boundary and click the menu **Set As FSS Reference.** A check indicates that the boundary is selected as reference. This menu is always enabled. Inappropriate setups are handled by design validation checks.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This surface become the input surface for calculations of the reflection/transmission coefficients. The other radiating surface automatically becomes output. Only one FSS can be defined in a given model. Using the option is advantageous for highly reflective and resonant structures. Reflection/Transmission coefficients for FSS designs can be viewed in the solution data panel as S-parameters or you can create an S-parameter report.

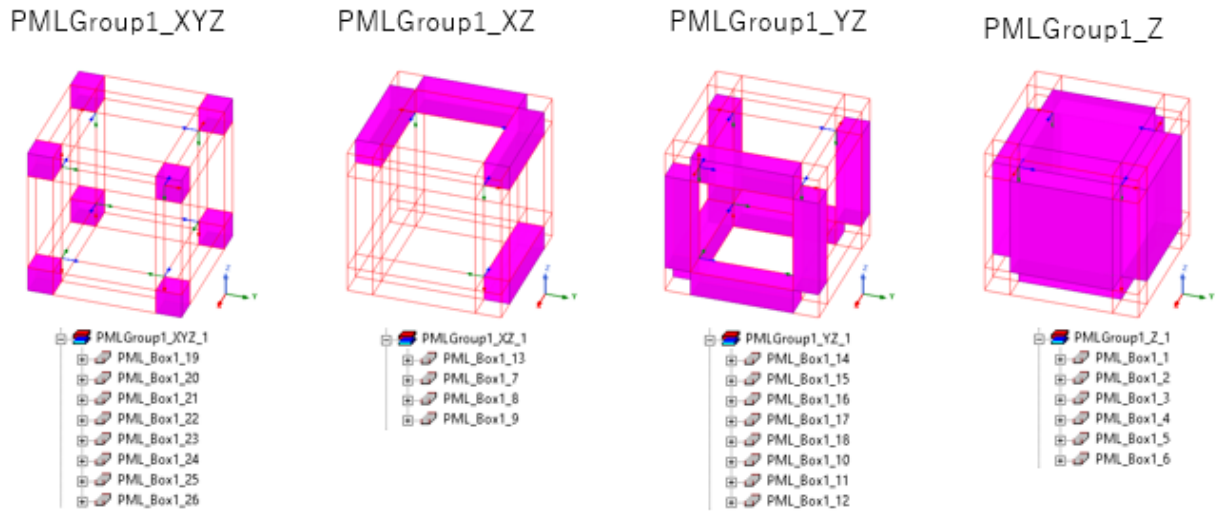
The requirements are:

- At most one reference is permitted
- Can only be set on a single face
- When a radiation boundary is selected as reference, there must be exactly one plane propagating incident wave that is without geometry assignment or assigned to the face of the radiation boundary
- Require at least one primary/secondary (Lattice pair) boundary
- Incompatible with ports (design cannot have ports)

Assigning PML Boundaries

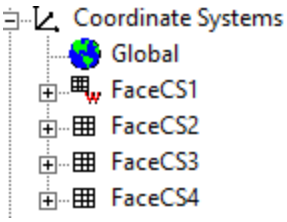
A perfectly matched layer (PML) boundary is used to simulate materials that absorb outgoing waves. Setting up a PML boundary is similar to setting up a radiation boundary. You start by selecting faces on the radiating object or by drawing a virtual object around the radiating structure. However, instead of placing a radiation boundary on its surfaces, you add PMLs to fully absorb the electromagnetic field. You can also assign PMLs to Region objects. Ansys Electronics Desktop can create PMLs automatically with the PML Setup Wizard, or using the Create Open Regions command, or you can create them manually. Create PMLs automatically if the base object touching the PML is planar and its material is homogeneous. HFSS creates a separate PML object for each covered face. The following figure shows PML objects automatically generated around a cubic region. The objects are grouped in the History tree as PML groups.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



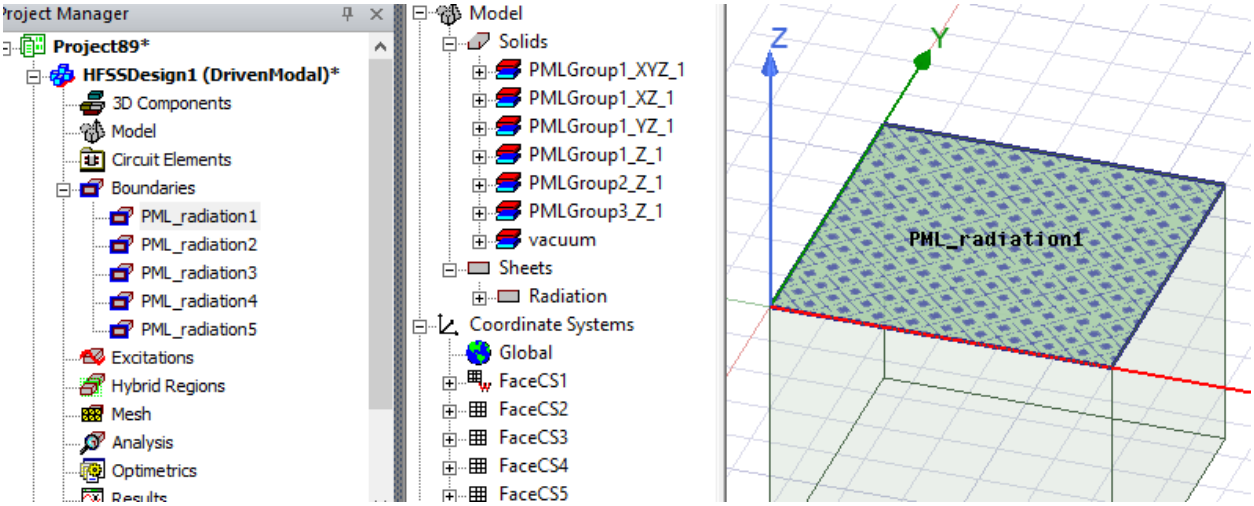
	Name	/	C...	Relative Permittivity	Relative Permeability	Bulk Co...	Dielectric Loss Tangent	Magnetic Loss Tangent
	PMLGroup1_XYZ_1	Pr...	N...	(1.596723046027...	(1.59672304602784,...	0	(1, 1, 1)	(1, 1, 1)
	PMLGroup1_XZ_1	Pr...	N...	(1, 0.01, 1)	(1, 0.01, 1)	0	(0, 509.904897143286, 0)	(0, 509.904897143286, 0)
	PMLGroup1_YZ_1	Pr...	N...	(0.01, 1, 1)	(0.01, 1, 1)	0	(509.904897143286, 0, 0)	(509.904897143286, 0, 0)
	PMLGroup1_Z_1	Pr...	N...	(1.596723046027...	(1.59672304602784,...	0	(1, 1, -1)	(1, 1, -1)
	vacuum	Pr...	N...	1	1	0	0	0

Each PML object is created on a Face coordinate system. You will see each FaceCS for PMLs listed under the Global CS.

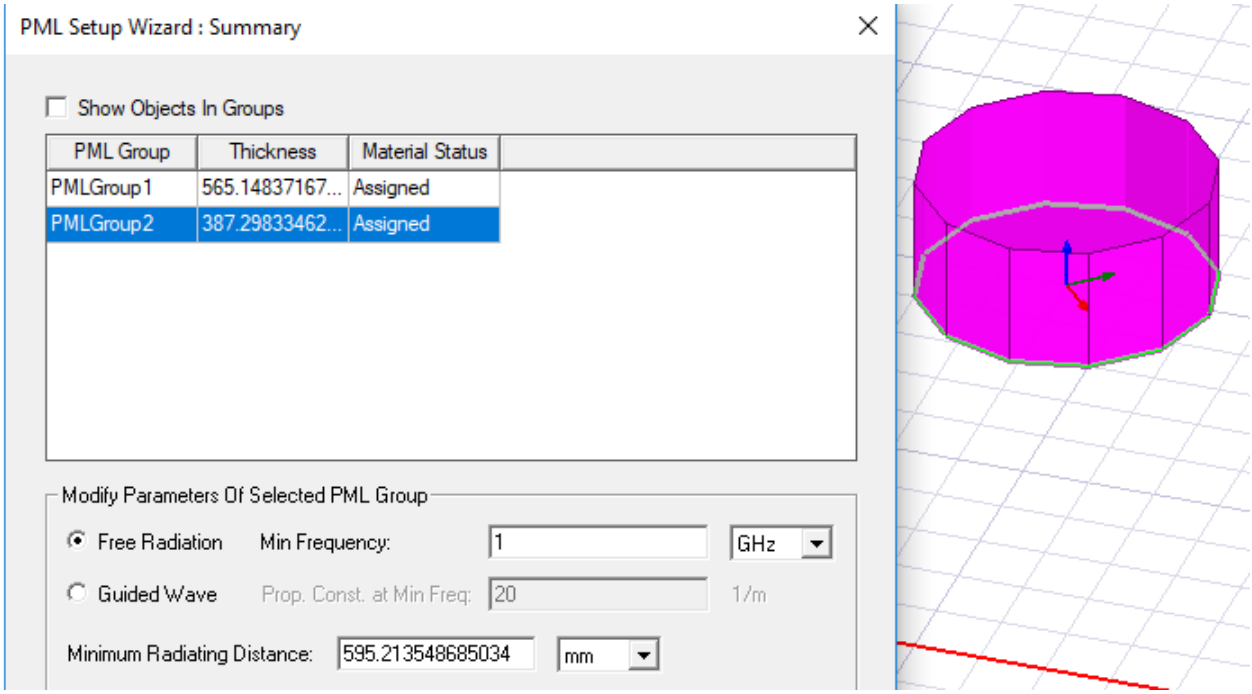


The PML boundaries are grouped in the **Project** tree under the **Boundaries** icon. Within these groupings, you can edit the radiation parameters (for example, as Incident Wave Port) in order to set up the right total field excitation based on the physical optics approach.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In creating PMLs, you can select non rectangular sheet objects as long as they do not touch any other selected face.



You can assign variables to the dimension properties of the base object. Changing the variable values then changes the associated PMLs. PML objects on a region are automatically updated when the region is updated by changing a region property or variable value.

Limitations

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you do assign PML objects to a region, you cannot perform geometry operations or copy and paste operations on that region.
- Region-dependent CS cannot be set as current CS.
- PML radiation boundaries are not generated in Eigenmode projects.
- The underlying object for assigning PML Boundaries does not have to be a box. However, any touching faces must be locally box-like.

PML Compared to Radiation Boundaries

Compared to Radiation boundaries, which create absorbing boundary conditions (ABC), PMLs in general make it more difficult for the iterative solver to reach convergence compared to the same model with using ABCs. PMLs also require significantly more RAM. The advantages for PMLs are that they absorb a much wider range of waves in terms of frequency and direction. As a result, you can put PMLs much closer to the discontinuities. This gives a smaller model. ABCs efficiently absorb normal incident waves. You have to put ABCs far away enough from the discontinuities.

What do you want to do?

Create PMLs automatically using the PML Wizard.

Create PMLs manually.

Guidelines for Assigning PML Boundaries

Creating PMLs Automatically

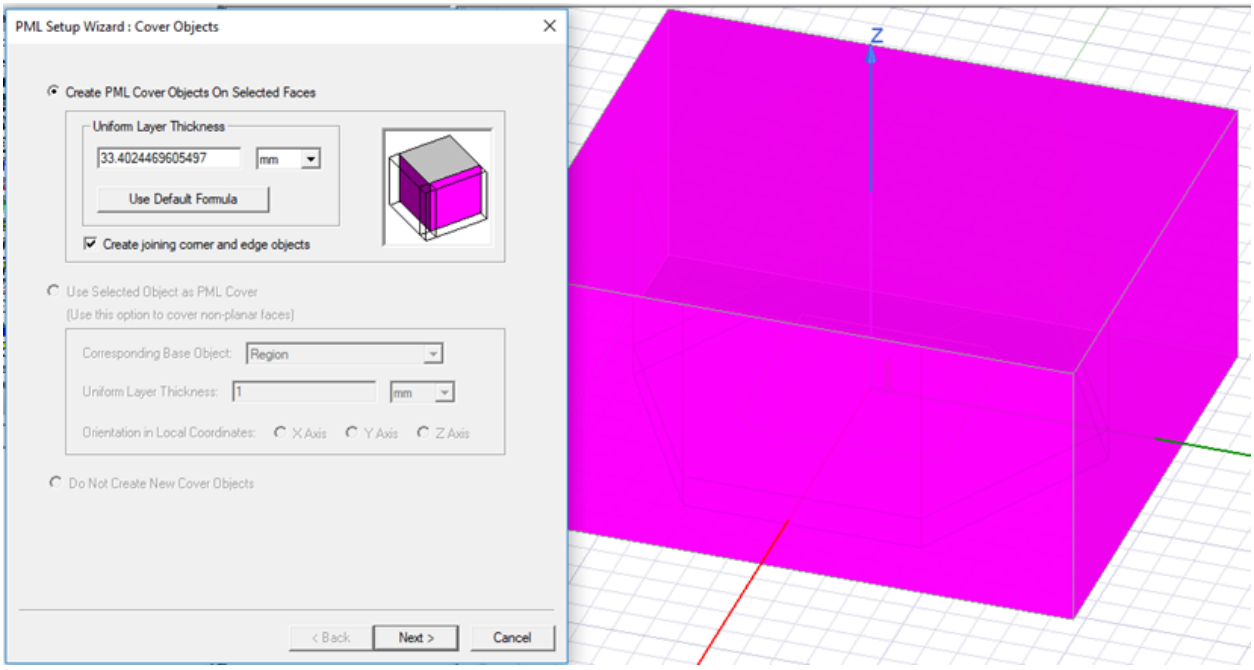
1. Draw a PML base object at the radiation surface.
2. Depending on the design you may select one or more faces of the object or all faces of object.

   When you create a PML that completely covers an object, for example, an air box or region around an antenna, you may find it convenient to select the object and have a boundary applied to all faces of the object.

   You can select the faces of the PML base object to turn into PMLs. In these cases, select only external, planar faces and exclude faces defined as symmetry boundaries.
3. Click **HFSS** > **Boundaries**>**PML Setup Wizard**.

   The **PML Setup** wizard appears.
4. If you have selected a non-planar object for a PML or one that completely covers another object, you can select **Use Selected Object** *objectName* **as PML Cover**. If you have selected object faces, select **Create PML Cover Objects on Selected Faces**.

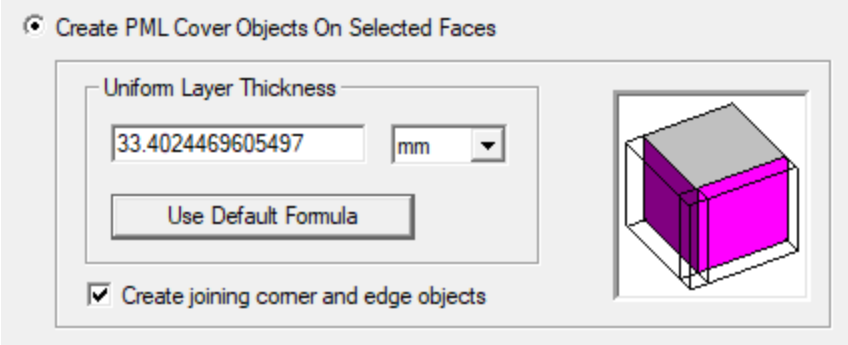
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Type the thickness of each layer in the **Uniform Layer Thickness** text box. You can assign a variable as the thickness value.

> **Note:**
>
> The layer thickness cannot be modified directly after PML objects have been created. If you want to be able to modify the thickness, assign a variable as the thickness value.

If you do not assign a value, you can select **Use Default Formula** to have HFSS calculate a value for you based on geometrical analysis.

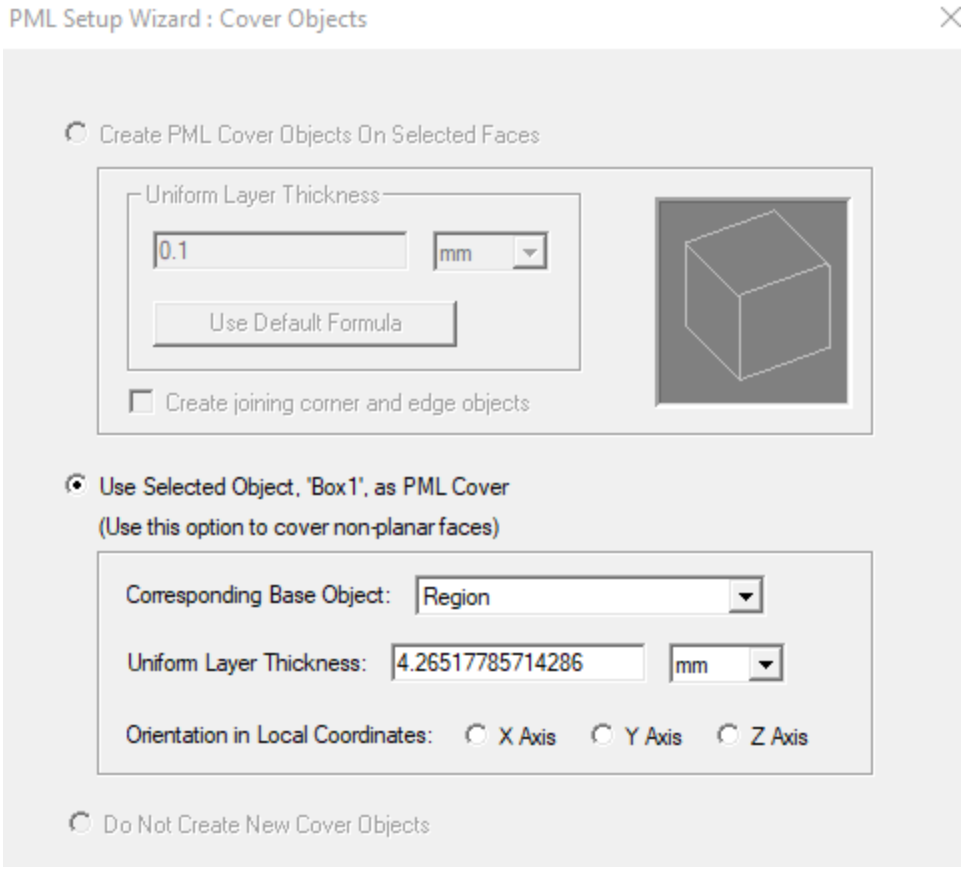


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. If the selected faces are on a box object. you can see the option to select **Create joining corner and edge objects**.

Edge and corner PML objects will be created to join adjacent PML surfaces together, ensuring complete coverage.

If the design contains another qualified object that you select, the **Use Selected Object** fields will be enabled.



If the selected object does not have a PML prefixed, you will see a dialog asking you to OK a name change. Clicking **Next** creates a PML cover object, but not a PML boundary.

If you select a face on a design that has a PML cover object defined, the dialog lets you chose to Create PML Cover Objects on Selected Faces or Do Not Create New Cover Objects. There is also a check box for **Reference for FSS** where you can also specify whether the surface is used as **Reference for FSS**, that is, as a Frequency Selective Surface - this surface becomes the input surface for calculations of the reflection/transmission coefficients. The other radiating surface automatically becomes output. Only one FSS can be defined in a given model. Using the **Incident Field** option together with **Reference for FSS** is advantageous for highly reflective and resonant structures. Reflection/Transmission

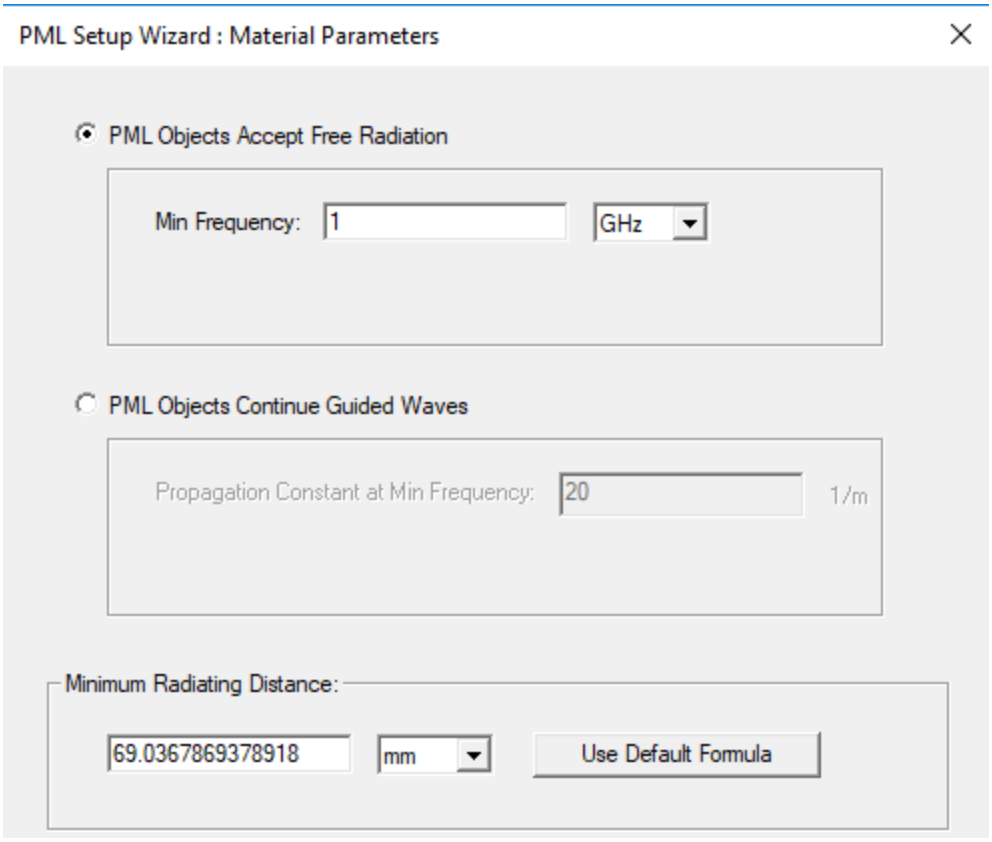
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

coefficients for FSS designs can be viewed in the solution data panel as S-parameters or you can create an S-parameter report.

If you check **Reference for FSS**, the PML objects will stay visible.

8. Click **Next**.

Ansys Electronics Desktop creates PMLs from the faces you selected. Names are automatically given to the layers. that start with *PML*, which is necessary for Ansys Electronics Desktop to recognize them as PMLs.



9. Specify how the PMLs terminate by selecting one of the following:
   a. **PML Objects Accept Free Radiation** if the PMLs terminate in free space.
      - Then enter the lowest frequency in the frequency range you are solving for in the **Min Frequency** text box.
   b. **PML Objects Continue Guided Waves** if the PMLs terminate in a transmission line.
      - Then specify the propagation constant at the minimum frequency.
10. Specify the minimum distance between the PMLs and any of the radiating bodies in the **Minimum Radiating Distance** text box. You may choose to have HFSS calculate the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

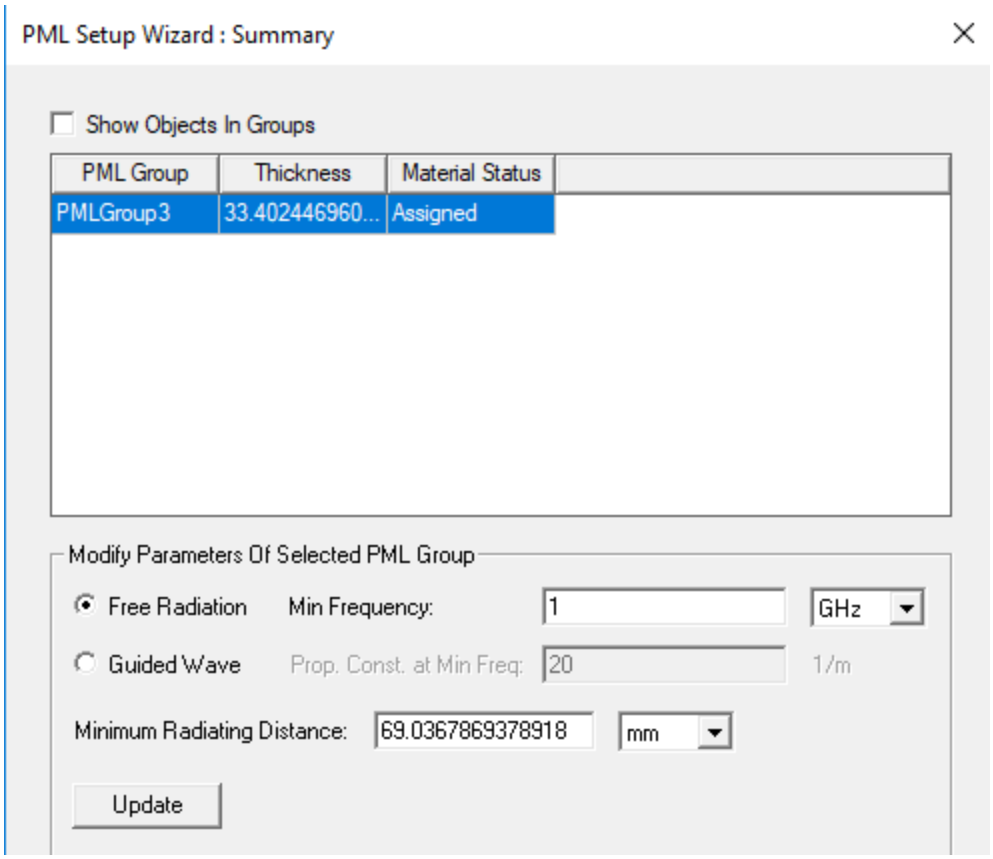
value by clicking **Use Default Formula**. The default distance is based on the extent of base object geometry.

The PML material characteristics depend on the cumulative effect of their near fields at the location of the PML surfaces.

11. Click **Next**.

Ansys Electronics Desktop calculates the appropriate PML materials based on the settings you specified and the material of the base object, and assigns these materials to the objects in the PML group.

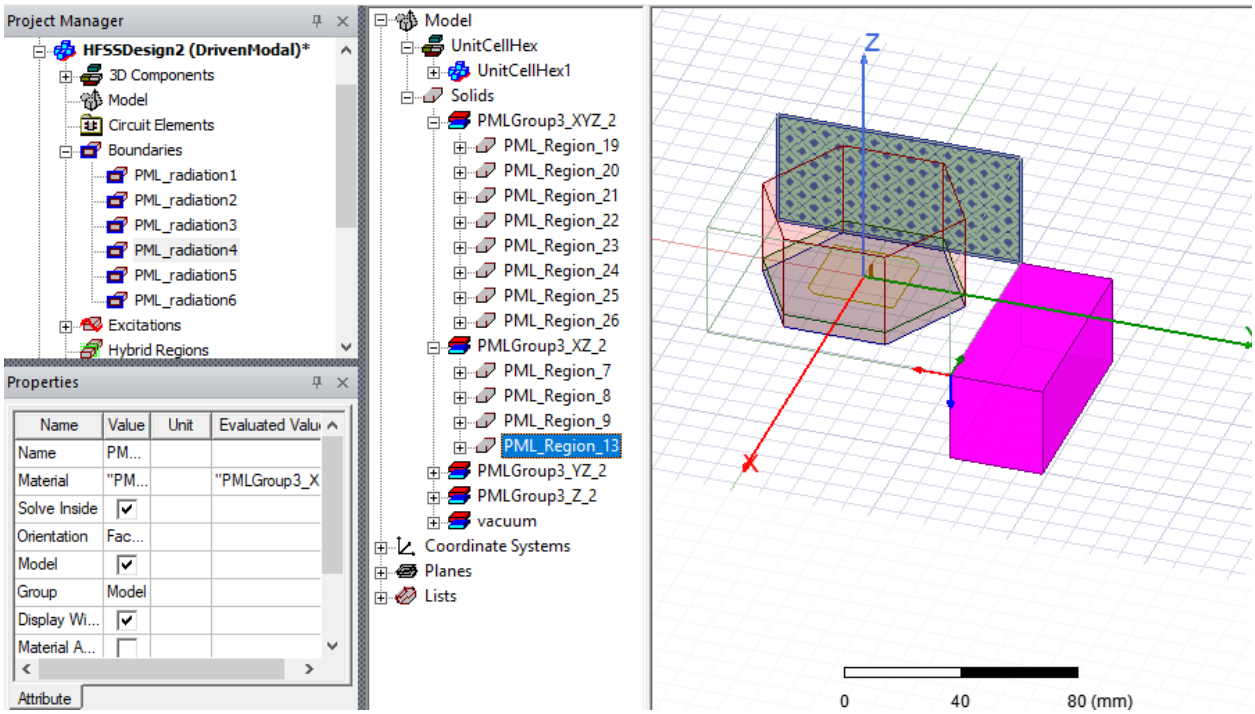
A summary dialog box appears, enabling you to modify the settings you specified.



12. Click **Finish**.

You can Fit the View to see the newly created PML boundaries in the Project tree and the PML groups and objects in the History tree. Selecting the boundaries or objects lets you view them in the Modeler window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Creating PML Boundaries Manually

See Guidelines for Assigning PML Boundaries.

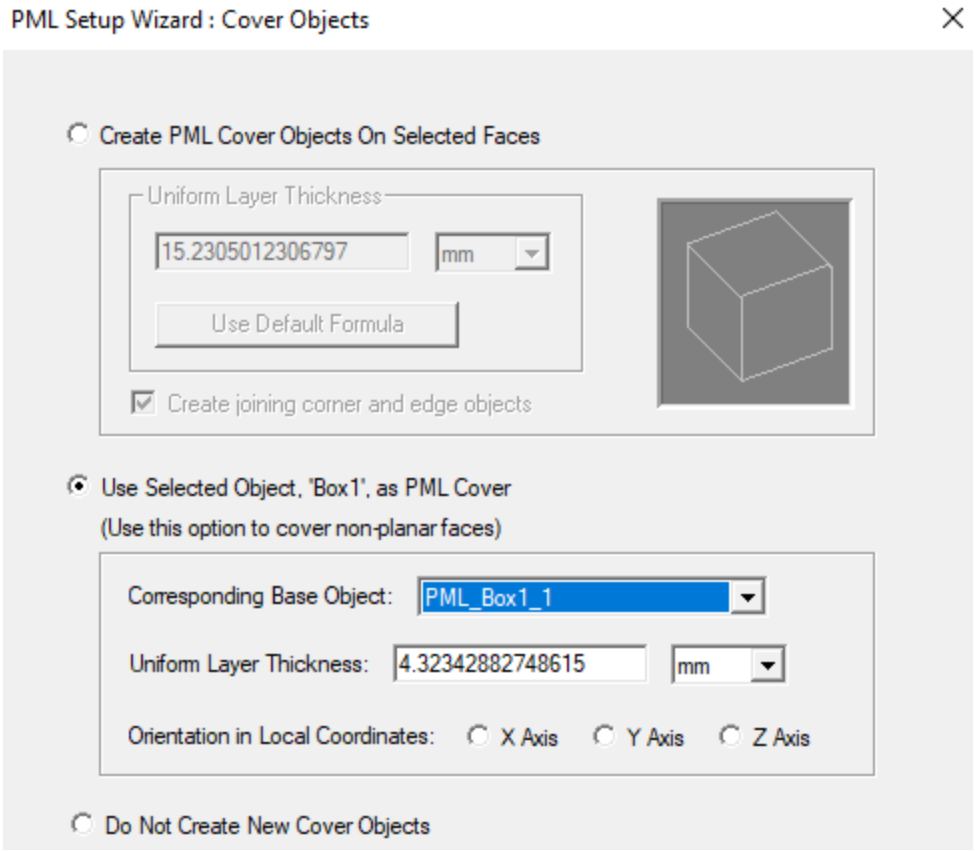
1. Draw the PML object at the radiation surface, and then select it.
2. In the **Properties** window, give the object a name with the prefix *PML*. If you so not change the name in advance, in a later step, you are prompted to allow an automatic name change.

Object names that start with *PML* are necessary for Ansys Electronics Desktop to recognize them as PMLs.

3. Click **HFSS>Boundaries>PML Setup Wizard**.

The **PML Setup** wizard appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



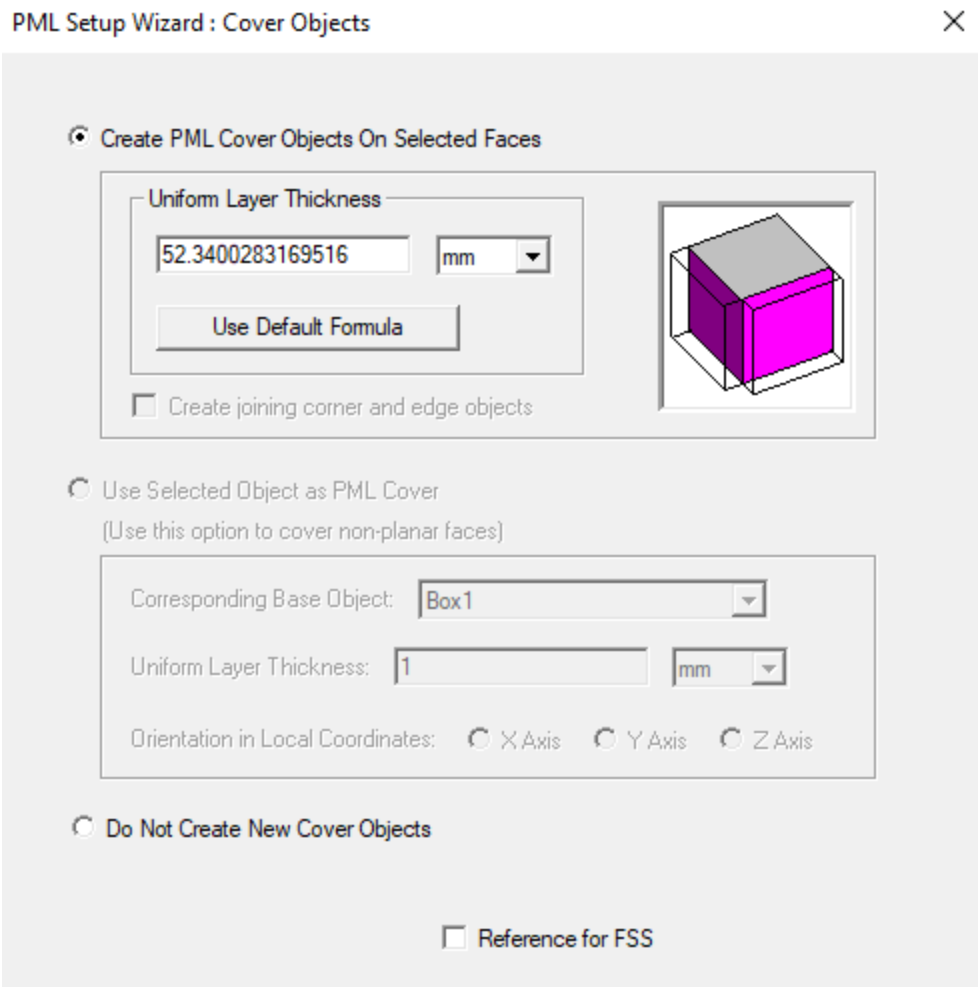
4. Select **Use Selected Object as PML Cover**.
5. Select the **Corresponding Base Object**, the object touching the PML, from the pull-down list.
6. Type the thickness of each layer in the **Uniform Layer Thickness** text box. You can assign a variable as the thickness value.
7. Select the **Orientation in Local Coordinates** of the PML object, X, Y or Z axis, the direction of outward propagation, in the relative, or local, coordinate system.
8. Under **Base Face Radiation Properties**, click a radio button to specify one of the following:

   If you selected a surface, the dialog will select **Create PML Cover Objects on Selected Faces** and disable **Use Selected Object as PML Cover**. You can also specify whether to **Do Not Create New Cover Objects** and designate whether a selected surface is used as **Reference for FSS**, that is, as a Frequency Selective Surface - this surface becomes the input surface for calculations of the reflection/transmission coefficients. The other radiating surface automatically becomes output. Only one FSS can be defined in a given model. Using the **Incident Field** option together with **Reference for FSS** is advantageous for highly reflective and resonant structures. Reflection/Transmission

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

coefficients for FSS designs can be viewed in the solution data panel as S-parameters or you can create an S-parameter report.

If you check **Reference for Frequency Selective Surface (FSS)**, the PML objects will stay visible.



9. Click **Next**.

If the selected object does not have a PML prefixed, you will see a dialog asking you to OK a name change. Clicking **Next** creates a PML cover object, but not a PML boundary.

10. Specify how the PML terminates by selecting one of the following:
    a. **PML Objects Accept Free Radiation** if the PML terminates in free space.
    - Enter the lowest frequency in the frequency range you are solving for in the **Min Frequency** text box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. **PML Objects Continue Guided Waves** if the PML terminates in a transmission line.

- Specify the propagation constant at the minimum frequency.

11. Specify the minimum distance between the PML and the radiating body in the **Minimum Radiating Distance** text box. You may choose to let HFSS calculate the value by clicking **Use Default Formula**.The default distance is based on the extent of base object geometry.

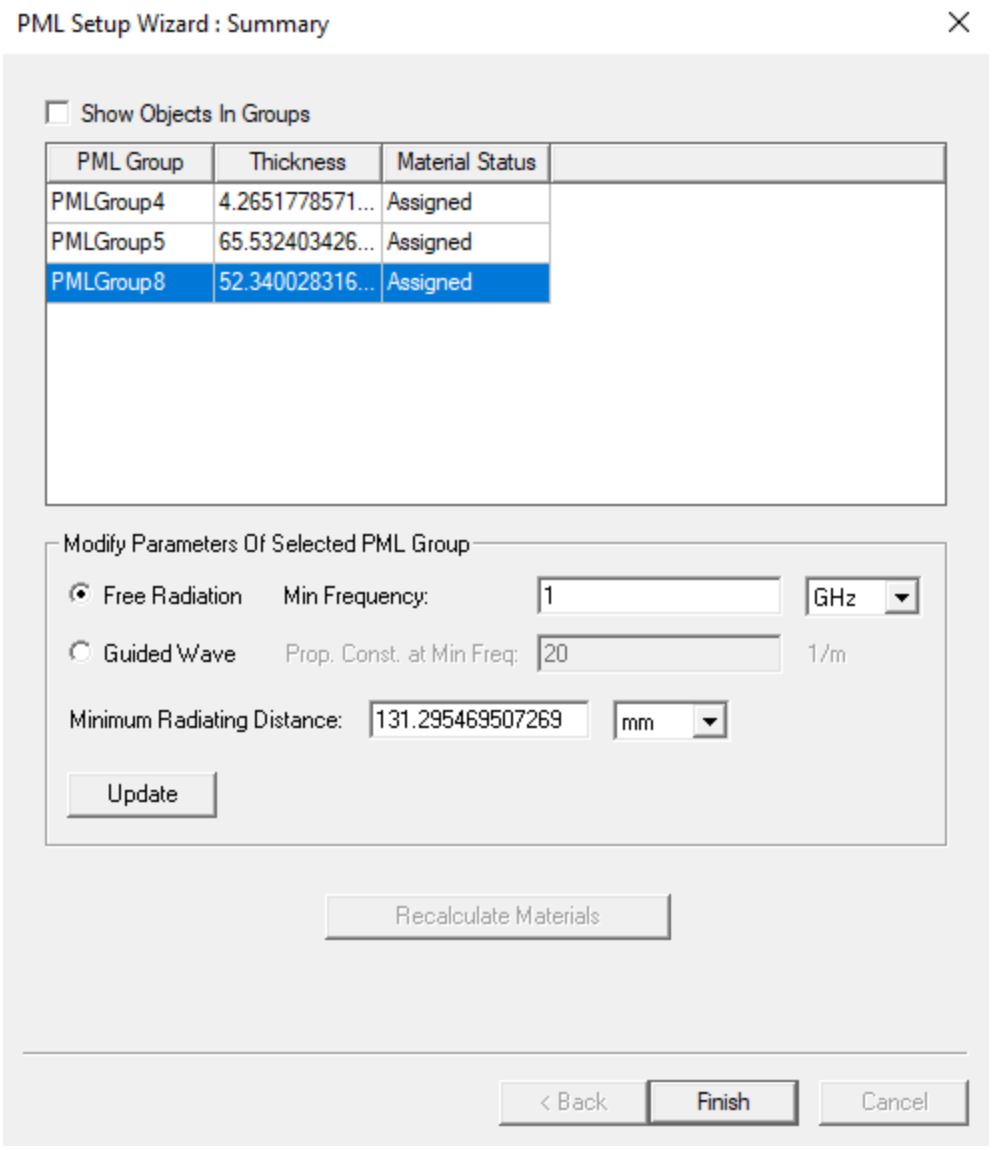
The PML material characteristics depend on the cumulative effect of their near fields at the location of the PML surfaces.

12. Click **Next**.

If Ansys Electronics Desktop calculates the appropriate PML material based on the settings you specified and the material of the base object, and assigns this material to the PML.

A summary dialog box appears, enabling you to modify the settings you specified.

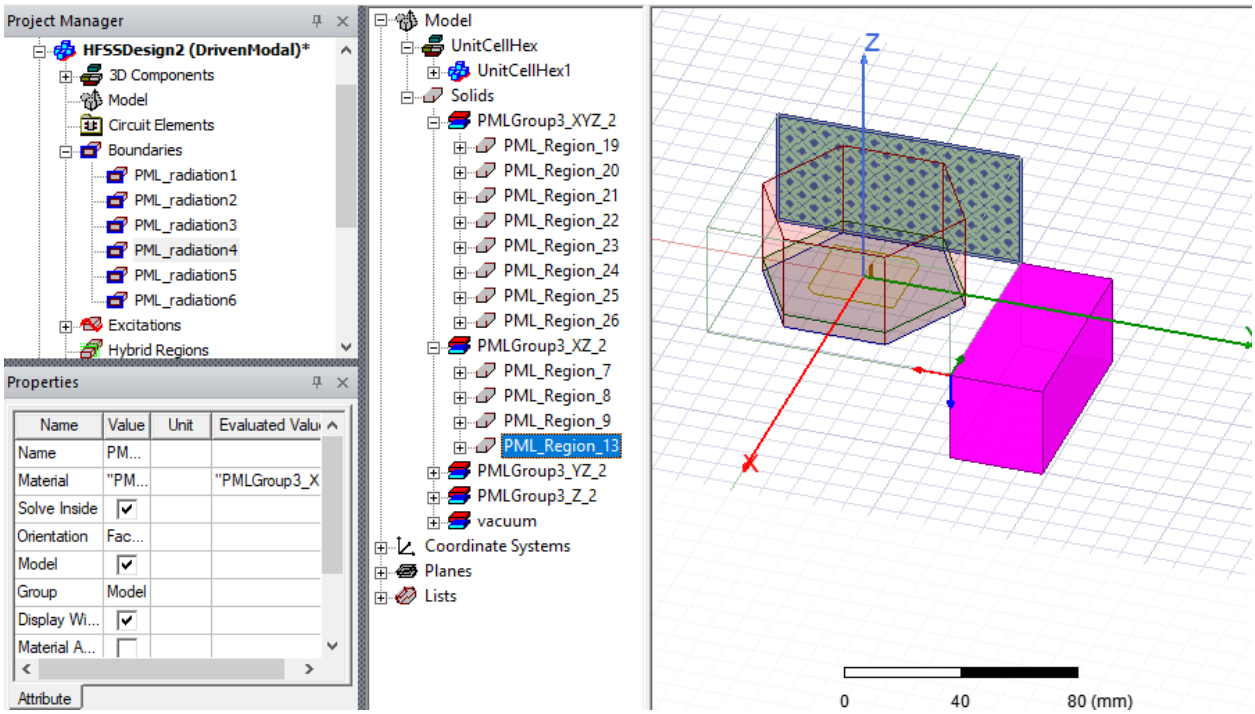
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



13. Click **Finish**.

You can Fit the View to see the newly created PML boundaries in the Project tree and the PML groups and objects in the History tree. Selecting the boundaries or objects lets you view them in the Modeler window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Guidelines for Assigning PML Boundaries

Keep the following guidelines in mind when assigning PML boundaries:

- When automatically creating PMLs, Ansys Electronics Desktop creates a new relative coordinate system for each PML object. This results in the z direction of the PML object coinciding with the normal direction of the base object's face.
- Ansys Electronics Desktop treats PMLs uniformly with regard to thickness. If the PMLs in your design vary in thickness, create a separate PML group for each thickness.

You should manually create a PML in the following situations:

- The base object is curved.

  Ansys Electronics Desktop calculates the PML material properties using the normal vector at the center of the base object's face. If the face is curved, the normal vector changes with position. The PML materials will only be good approximations if the normal vector at each point on the face is close to the normal vector at the face center.

  It is a good idea to segment the curved surface of the base object for greater accuracy. Create separate PMLs for each segment. Note that each segment's thickness is treated uniformly. The view angle of the segments should be no wider than 45 degrees. The smaller the angle of each segment, the greater the accuracy of the corresponding PML.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The material of the corresponding base object touching the PML is not homogenous. An example is a metal-shielded microstrip line with a substrate. One PML could be drawn to terminate the microstrip and another could correspond to the substrate.

Create as many PML objects as there are subsections of material properties in the base object.

Modifying PML Boundaries

You can modify parameters of PML boundaries through the **PML Setup Summary** dialog box. You can also modify the dimensions of PML boundaries by editing the History tree properties for the original object for which you assigned the PML boundaries.

To modify dimension properties for the original PML object:

1. Go to the History tree and open the hierarchy under the original PML base object.
2. Select the CreateBox command for each part of the geometry you want to modify.
3. In the Properties dialog for that geometry command, edit the properties for XSize, YSize, or ZSize as required.

The changes to the CreateBox parameters apply to the associated PML objects. Note that you can create variables to parameterize these properties.

To modify PML parameters:

1. Make sure that nothing is selected in the **3D Modeler** window.
2. Click **HFSS**>**Boundaries**>**PML Setup Wizard**.

The **Summary** dialog box of the **PML Setup** wizard appears. By default the Show Objects in groups box is not checked. A table shows each PML Group, its thickness, and material status. Checking the box causes the table list the objects under each group.

3. If more than one group of PMLs were defined, select the PML group you want to modify from the table.
4. Modify the PML parameters.

A Radio button lets you select either the **Free Radiation** minimum frequency and units, or the **Guided Wave** propagation constant at a minimum frequency.

You can also specify a Minimum Radiating distance and units.

5. If you make changes, click **Update**.

This performs the update and enables the **Recalculate Materials** button.

6. Click **RecalculateMaterials** to apply the updates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electronics Desktop automatically recalculates and assigns the appropriate PML materials to the objects in the PML group.

7. Click **Finish**.

> **Note:**
>
> If objects are modified after PMLs are created, the PML materials will be invalid and must be recalculated in the **PML Setup Wizard**. For example, if the material of the PML base object is modified, the associated PML materials must be recalculated in the **PML Setup Wizard**.

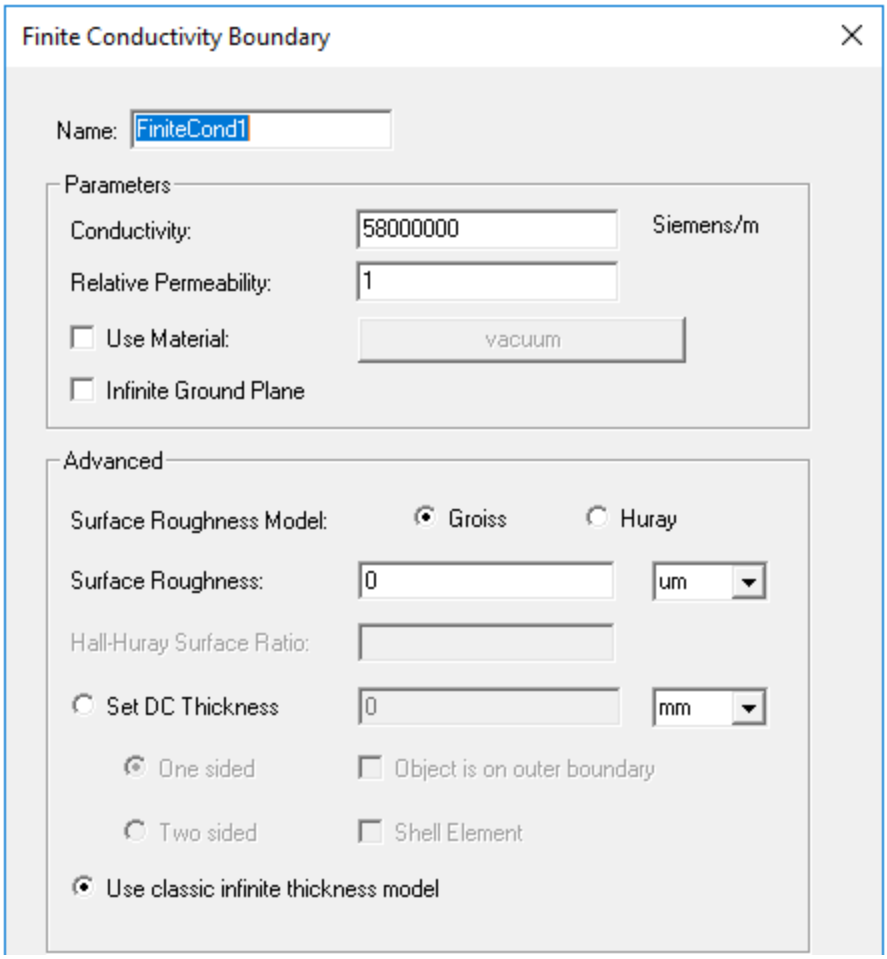
Assigning Finite Conductivity Boundaries

A finite conductivity boundary the behavior of the field at the object surface. The finite conductivity boundary is valid only if the conductor being modeled is a good conductor, that is, if the conductor's thickness is much larger than the skin depth in the given frequency range. If the conductor's thickness is in the range or larger than the skin depth in the given frequency range, HFSS takes the thickness into account, if it has been defined.

To assign a Finite Conductivity boundary:

1. Select a surface on which to assign the boundary and click **HFSS>Boundaries>Assign>Finite Conductivity** or **HFSS>Boundaries>Assign>Surface Roughness>Groiss (Finite Conductivity) ...** to bring up the **Finite Conductivity Boundary** dialog box with Gross selected as the Surface Roughness Model. If you select, **HFSS>Boundaries>Assign>Surface Roughness>Huray (Finite Conductivity) ...**, the Huray model is selected. For a description of the Huray model, see step 4 below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Do one of the following:

- Enter the conductivity in inverse ohm-meters, and then enter the permeability.

> **Note:**
>
> You can assign a variable as the conductivity or permeability values or roughness model parameters.

To assign Finite Conductivity so that the boundary is **spatially dependent** (that is, in which the material properties change over the length), use the method described via this link.

- Select **Use Material**, click the default material name button , and then choose a material from the material editor. The conductivity and permeability values of the material you select will be used for the boundary. Note that selecting a perfectly conducting material for a finite conductivity boundary triggers a validation error.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select **Infinite Ground Plane** if you want the surface to represent an electrically large ground plane when the radiated fields are calculated during post processing.

Note that if you select **Infinite Ground Plane**, the effect of the finite conductivity boundary will be incorporated into the field solution in the usual manner, but the radiated fields will be computed as if the lossy ground plane is perfectly conducting.

For designs with port excitations but no FE-BI boundary and metallic IE region defined, selecting Infinite Ground Plane affects only the calculation of near field and far field radiation during post processing; otherwise, the design has to be re-simulated to model the effects of infinite ground plane. The boundary condition infinite ground plane divides the problem region into two halves. The entire model resides in the half above the boundary, and the radiated fields are set to zero in the half below the boundary. Antenna parameters involving radiated power are consistent with these properties.

Lossy ground planes can be approximated by selecting the Infinite Ground Plane check box when defining Finite Conductivity Boundary or Impedance Boundary. The effects of these boundary conditions are incorporated in the field solution and in the radiated fields accordingly.

Remember the following requirements when defining an infinite ground plane:

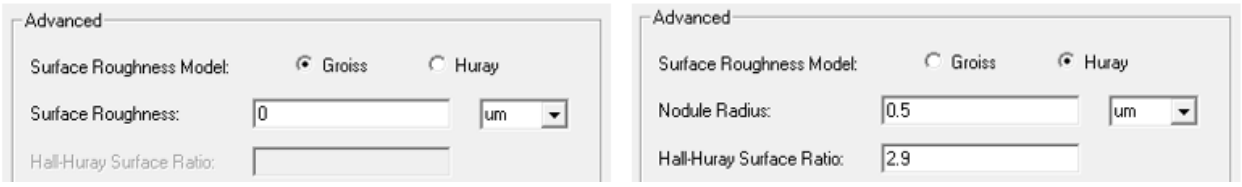
- It must be defined on a planar surface.
- All infinite ground planes and symmetry planes must be mutually orthogonal.
- For impedance, layered impedance, or finite conductivity boundaries, only one infinite ground plane can exist in a design. For perfect E boundary conditions, multiple infinite ground planes are supported and they must be co-planar.

4. Because surface roughness can increase conductor power losses more than two times, HFSS models surface roughness model based on the visible features of copper conductors used in circuit fabrication.To select the **Surface Roughness Model** used for surfaces such as the interface between the conductor and the substrate for a microstrip line, select either Groiss or Huray.

For the Groiss model, you specify a Surface Roughness parameter (traditional case) as a value (or variable) and units. The default is 0 um. Legacy projects use the Groiss model by default.

For the Huray Model, you specify the Nodule radius value (or a variable), which describes the radius of copper spheres that model the surface roughness. The default is 0.5 um. Also for the Huray model, you specify the Hall-Huray Surface Ratio, a unitless quantity. The default is 2.9. HFSS implements an enhanced version of the Huray surface roughness model. In addition to the benefits of the original Huray model (accurate broad-band modeling of the losses in very rough copper foils typically used in printed circuit board manufacturing), the enhanced model is causal, making it more suitable for time-domain signal integrity studies, and it accurately predicts increased phase delay due to the surface roughness.

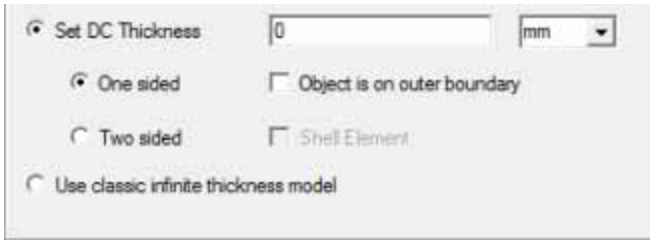
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



(Using surface roughness with the Finite Conductivity boundary may be more intuitive than using a layered impedance boundary to model the effects.)

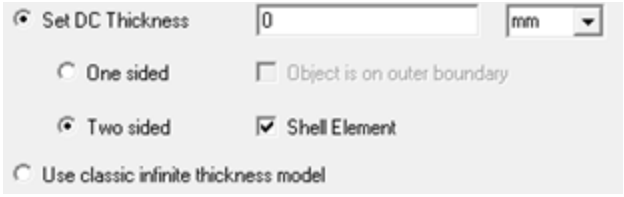
5. You can select either Set DC Thickness or Use classic infinite thickness model. You can specify a DC thickness to more accurately compute DC resistance of a thin conducting object for which Solve Inside is not selected. Skin impedance of the object will be calculated using the defined finite thickness.

To Set DC thickness, click the radio button to enable the **Layer Thickness** field, and enter a value and select units. You can also specify whether a One-sided object is on outer boundary of the model.



.

If you select **Two sided**, you can also specify whether to treat the boundary as a **Shell Element**. Shell elements maintain two sets of unknown coefficients for the top and bottom surface



Meshing thin layers such as signal traces or thin substrates of PCBs can be difficult and cause very inefficient simulation performance. Instead of meshing these thin layers, they can be replaced with sheets along with appropriately assigned boundary conditions. The appropriate boundaries are 2-sided layered impedance or finite conductivity boundaries where the latter is only applicable for a single layer of metallic material such as a signal trace made of copper.

When to Use Shell Elements

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

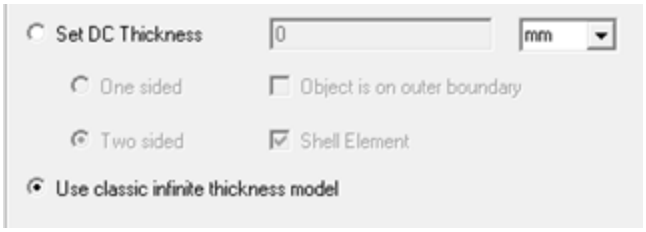
You can consider using **shell elements** on the sheet for enhanced accuracy at high frequencies where the currents on the two sides of the sheet will be allowed to be different and therefore better model the non-uniform current distribution of a thin structure at high frequencies. Some examples for this application would be studying shielding effects of a car chassis by modeling the chassis with sheets or replacing an antenna radome model with a thin sheet. Note that the shell element option is applicable for all frequencies and can therefore be used for broadband simulations without the need to modify boundary options depending on the simulation frequency. Further, a sheet in the above discussion can also be a face of a solve inside object.

For technical details, see Shell Elements Theory.

The following features are not currently supported when using shell elements:

- Fast sweep
- Scattered field formulation
- Eigen solver
- IE solver

Selecting **Use classic infinite model** disables the fields for Set DC Thickness and the outer boundary check box. Calculations for DC then assume infinite thickness.

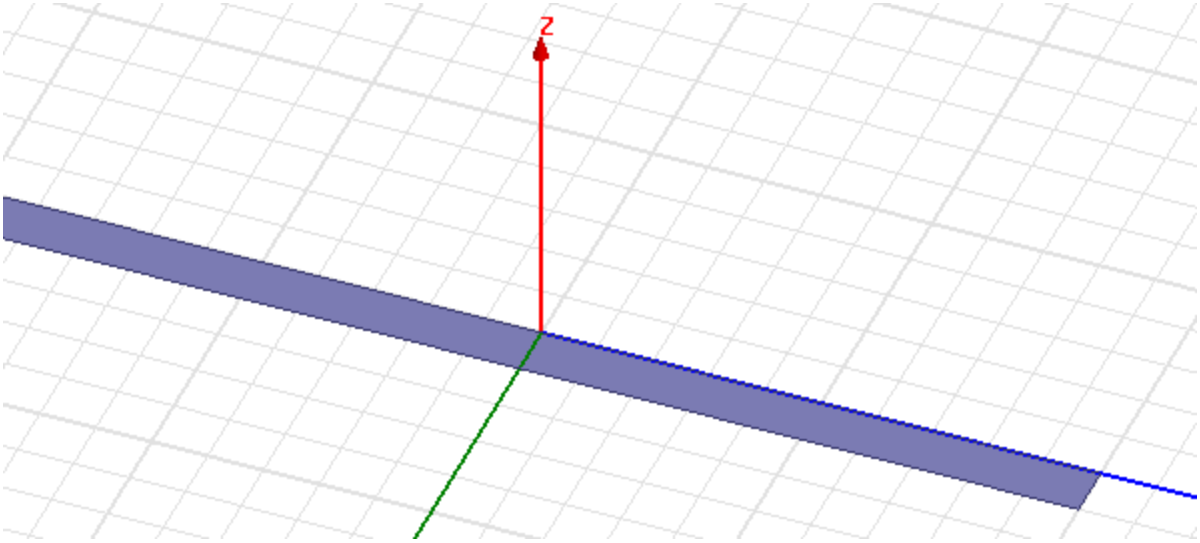


Also see Assigning DC Thickness under Materials for a discussion of design defaults and the how DC thickness is used and calculated.

Spatially Dependent Boundaries

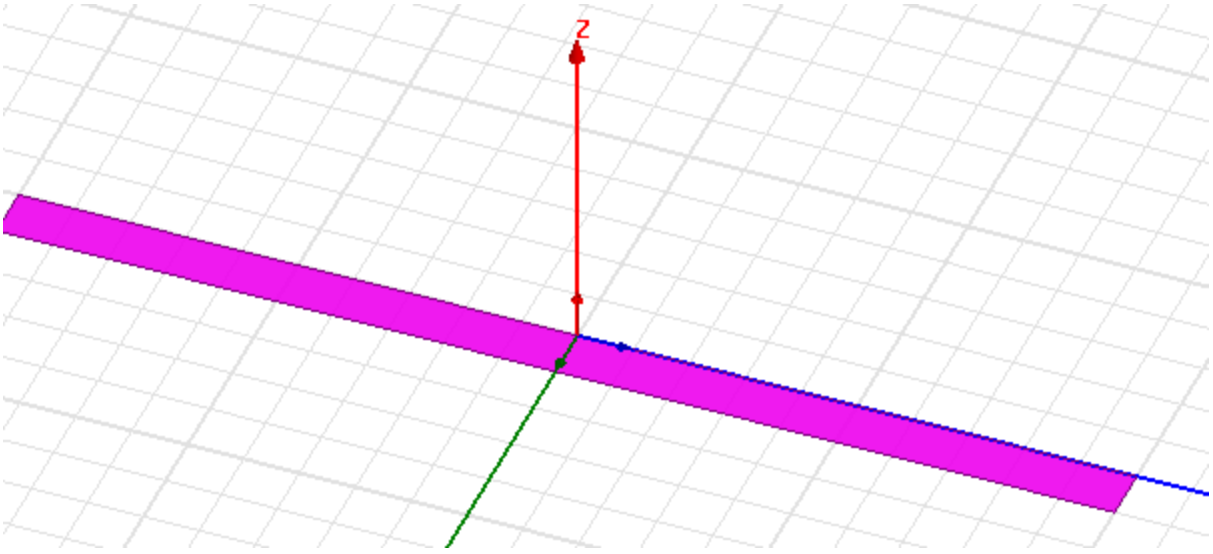
HFSS supports spatially dependent materials, that is, materials whose properties change over their length. Consider a long rectangular sheet object that lies on the XY plane along the Y axis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



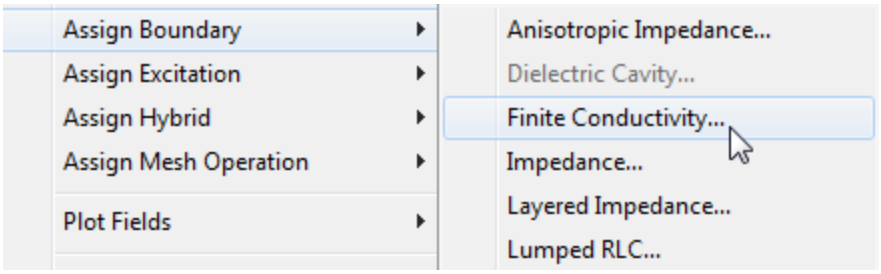
To assign **Finite Conductivity** such that the boundaries are spatially dependent follow these steps:

1. Select the line as shown in the figure below.



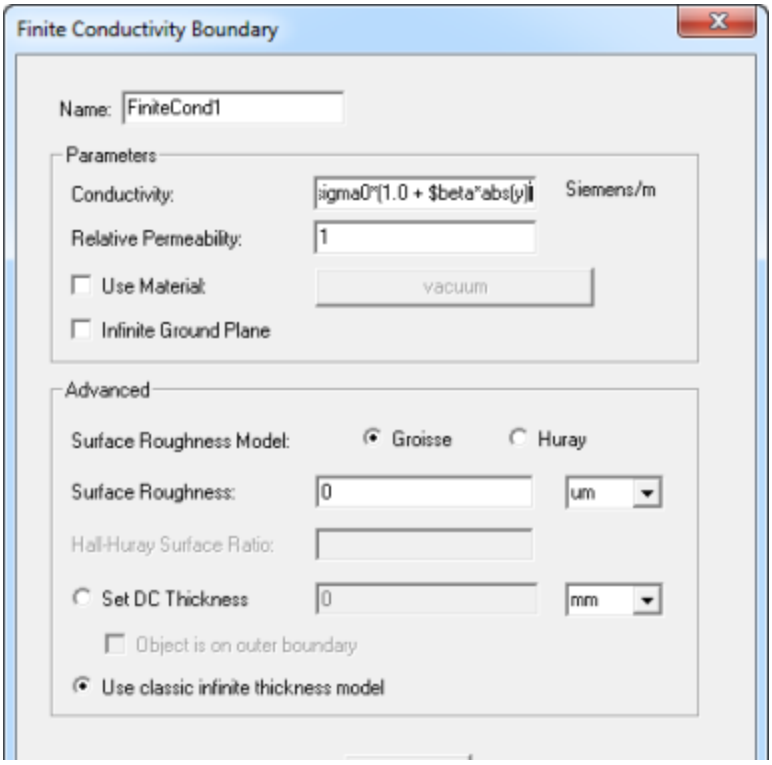
2. Right-click anywhere in the modeler and select **Assign Boundary** > **Finite Conductivity** as shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This causes the **Finite Conductivity Boundary** dialog box to appear as shown below.

3. Type the expression in the **Conductivity** field to include the spatially dependent material properties of the sheets. For example, in this design, the **Conductivity** field has the following expression: $sigma0*(1.0 + $beta*abs(y)), where $sigma and $beta are the predefined project variables and y specifies the local y co-ordinate associated with the sheet.



> **Note:** Whatever model units you define, specify the expression in the **Conductivity** field in SI units. To verify the **Coordinate System** associated with the sheet, double-click the **Create Rectangle** command associated with the object from the History Tree to open the **Attribute** window as shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command

	Name	Value	Unit	Evaluated Value
	Command	CreateRectangle		
	Coordinate System	Global		
	Position	0.1 ,-1 ,0	mm	0.1mm , -1mm , 0mm
	Axis	Z		
	XSize	-0.1	mm	-0.1mm
	YSize	7	mm	7mm

Note: The spatial dependency can only be expressed in Cartesian coordinates.

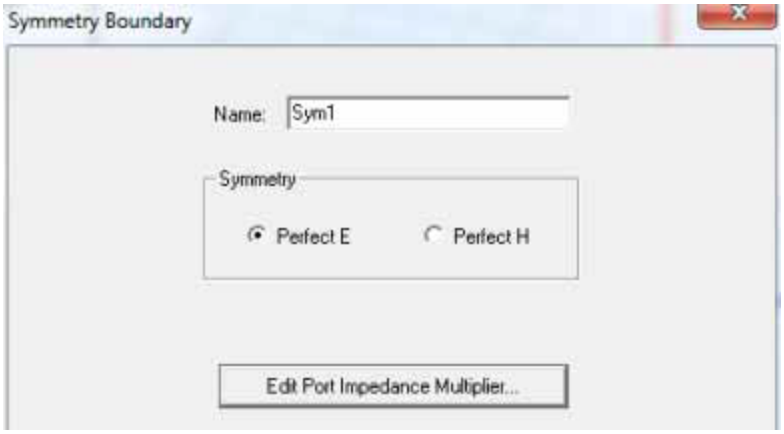
Assigning Symmetry Boundaries

For Driven Modal or Eigenmode Designs

A symmetry boundary represents a perfect E or perfect H plane of symmetry. Symmetry boundaries enable you to model only part of a structure, which reduces the size or complexity of your design.

To assign a Finite Conductivity boundary:

1. Select a surface on which to assign the boundary and click **HFSS> Boundaries> Assign> Symmetry** to bring up the **Symmetry Boundary** dialog box.



2. Select the type of symmetry plane the boundary represents: **Perfect E** or **Perfect H**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Impedance Multiplier**.

   If the design includes a port, you must adjust the impedance multiplier or the computed impedances will not be for the full structure.

   The **Port Impedance Multiplier** dialog box appears.

4. Type a value in the **Impedance Multiplier** box, and then click **OK**.

Assigning Coupled Lattice Pair Boundaries

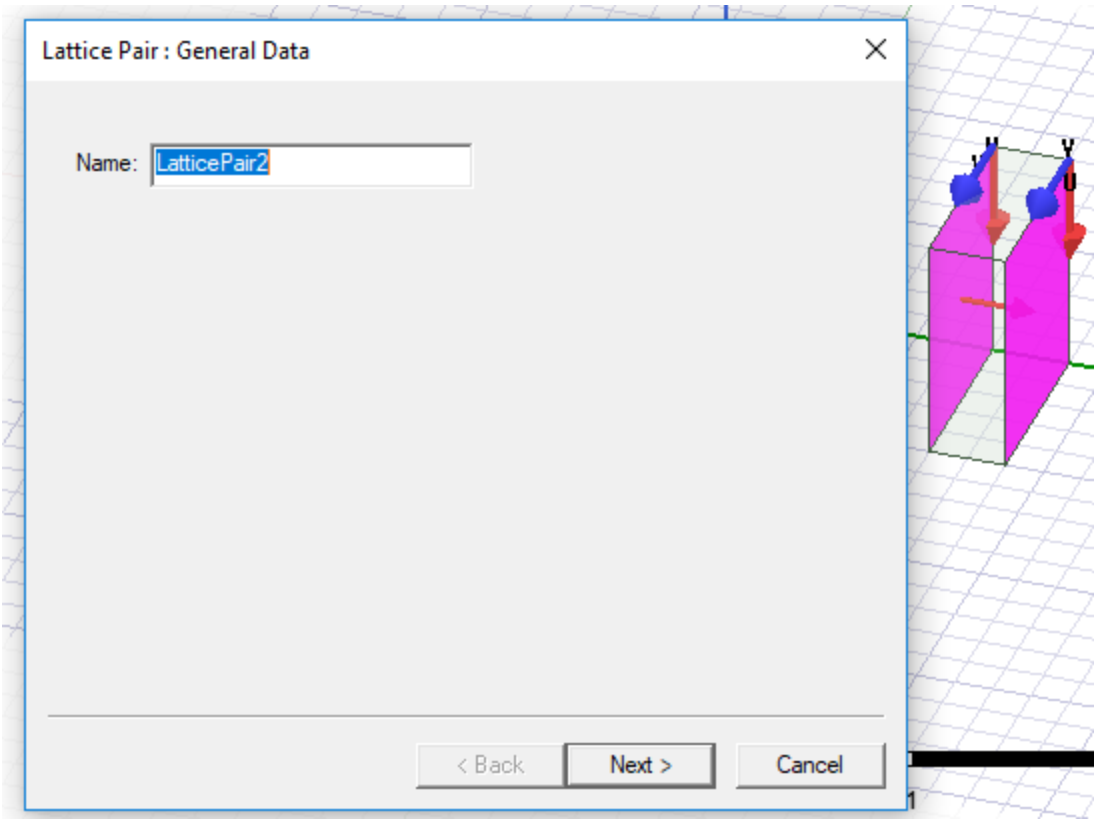
Coupled Lattice Pair boundaries enable you to model planes of periodicity where the E-field at every point on the Secondary boundary surface is forced to match the E-field of every corresponding point on the Primary boundary surface to within a phase difference. The transformation used to map the E-field from the Primary to the Secondary is determined by an auto-set coordinate system for the coupled lattice pair. The boundaries are useful in creating unit cells and arrays. (You can also assign Primary and Secondary boundaries for a coupled lattice pair separately.)

To assign a Coupled Lattice Pair:

1. Select two surfaces on which to assign the coupled lattice pair and click **HFSS>Boundaries>Assign>Couple...>Lattice Pair...**to bring up the **Lattice Pair** dialog

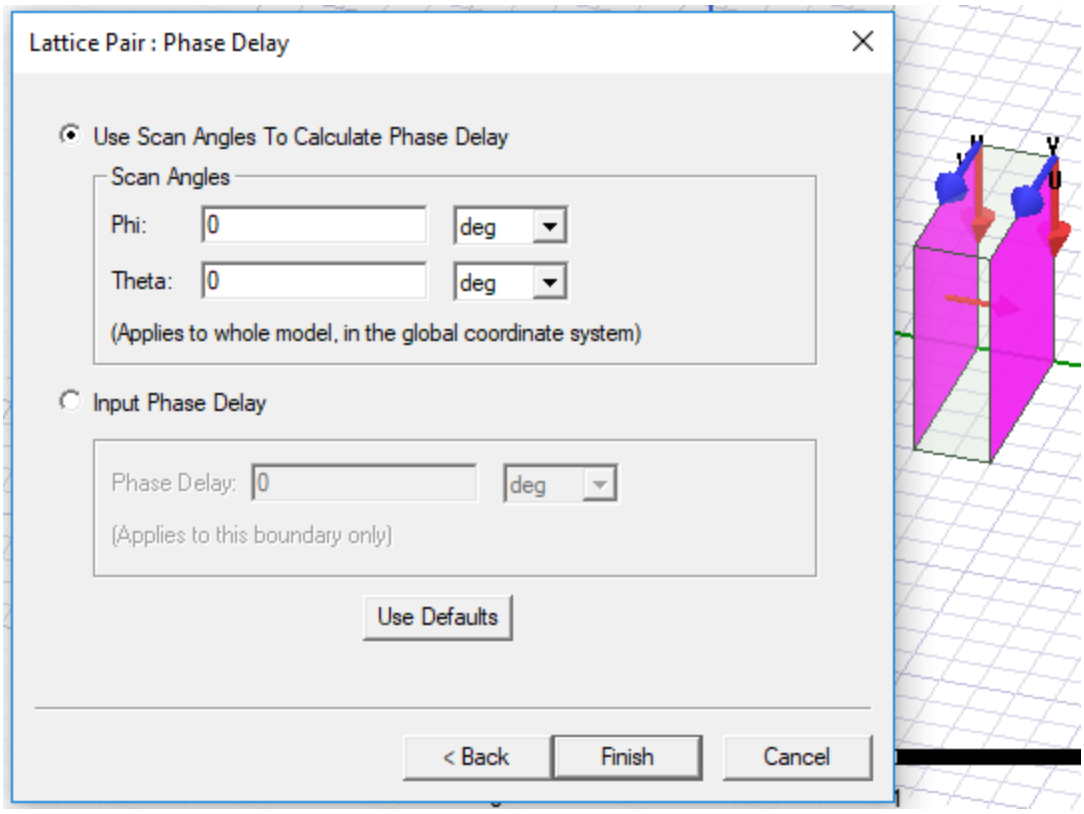
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

box.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Next**.



You have the option to relate the Secondary boundary's E-fields to the Primary boundary's E-fields in one of the following ways:

For driven designs, select **Use Scan Angles to Calculate Phase Delay** to enable the **Scan Angle fields**. Then enter **Phi** and **Theta** scan angles. These apply to whole model, in the global coordinate system. The phase delay is calculated from the scan angles; however, if you know the phase delay, you may enter it directly in the **Phase Delay** box.

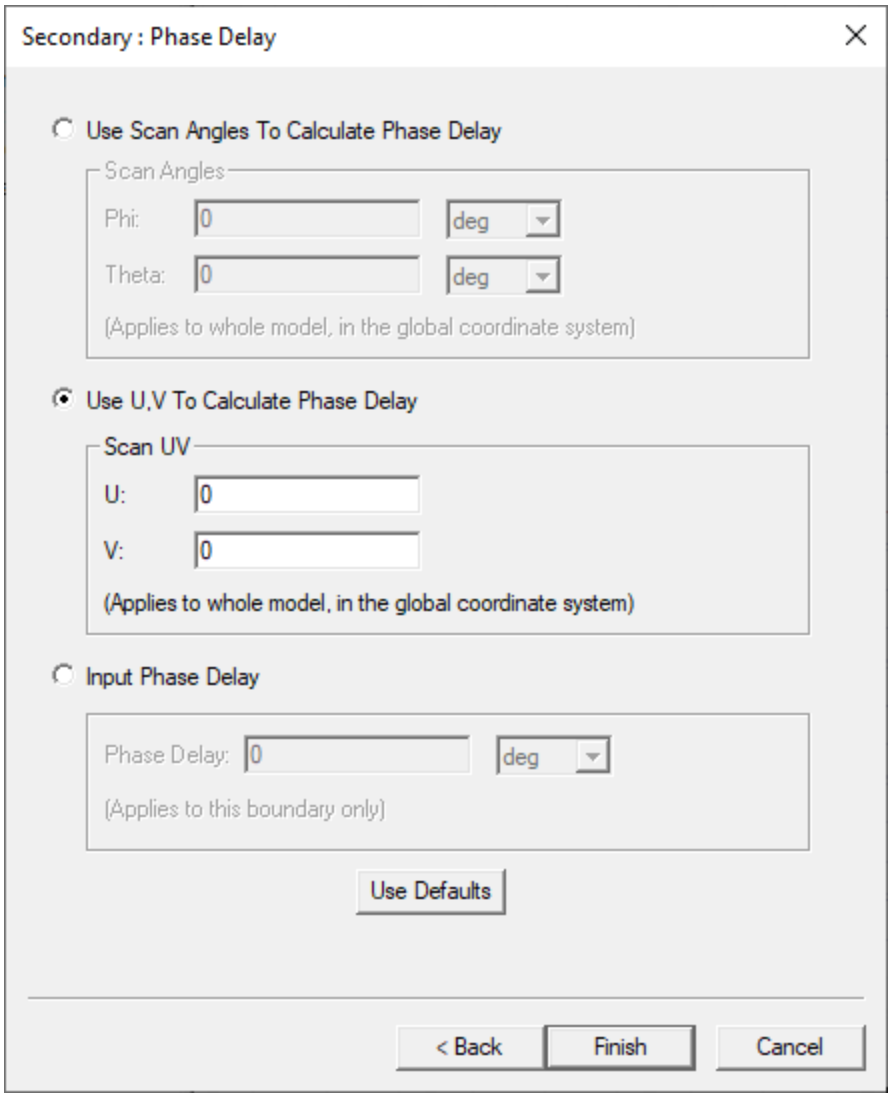
> **Note: Note**: For Eigenmode problems, the **Use Scan Angles to Calculate Phase Delay** fields are disabled.

Select **Input Phase Delay**, and then enter the phase difference, or phase delay, between the boundaries' E-fields in the **Phase Delay** box. The phase delay applies only to this boundary.

> **Note:** : You can assign a variable as the phi, theta, or phase delay values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A Beta Option allows you to **Use U,V to Calculate Phase Delay**.



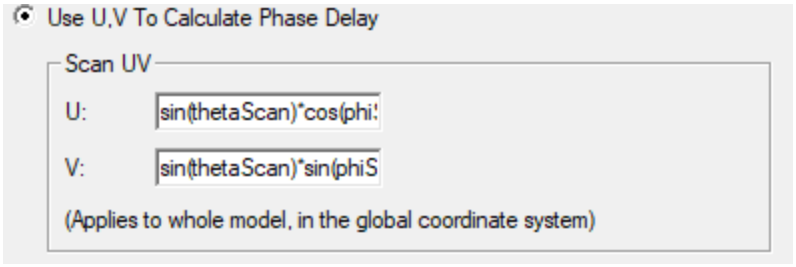
This permits you to scan to scan in the invisible range which is conveniently specified by u,v coordinates. U or V can span anything between -infinity to + infinity or practically some large negative value to large positive value. When $U^2+V^2 > 1$, it is the invisible range. You can specify U and V as equations in this form:

$u = \sin\theta \cdot \cos\varphi$

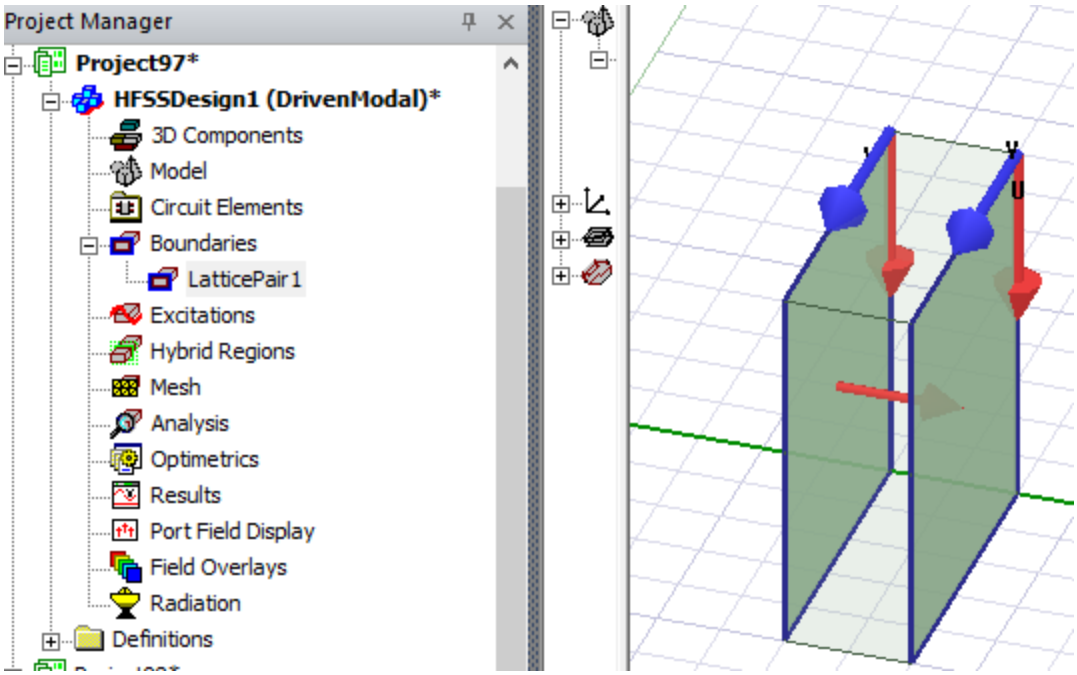
$v = \sin\theta \cdot \sin\varphi$

Note that you can employ variables in the equations:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electronics Desktop will compute the E-field on this boundary and map it to the Secondary boundary using the transformation defined by the Primary and Secondary coordinate systems.



Assigning Primary Boundaries for a Coupled Lattice Pair

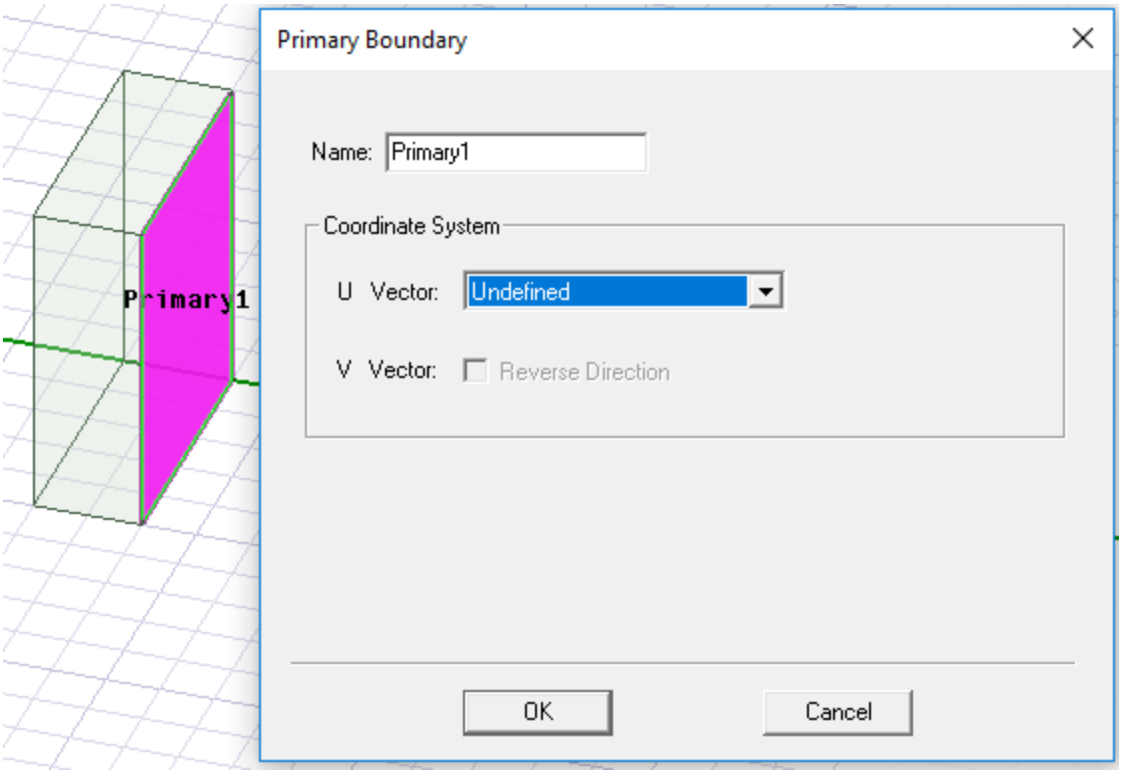
Coupled Lattice Pair boundaries enable you to model planes of periodicity where the E-field at every point on the Secondary boundary surface is forced to match the E-field of every corresponding point on the Primary boundary surface to within a phase difference. The transformation used to map the E-field from the Primary to the Secondary is determined by specifying a coordinate system on both the Primary and Secondary boundaries. You can use the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coupled...>Lattice Pair... command to auto-set the coordinate system, for both Primary and Secondary boundaries together, you have the option of assigning the Primary boundary directly.

To assign a Primary boundary:

1. Select a surface on which to assign the boundary and click **HFSS>Boundaries>Assign>Primary** to bring up the **Primary Boundary** dialog box.



2. You must specify the coordinate system in the plane on which the boundary exists. First draw the U vector of the coordinate system. HFSS uses the U vector that you draw and the normal vector of the boundary face to calculate the Vvector. If necessary, you can reverse the direction of the V vector.
   a. Select **New Vector** from the **U Vector** drop-down menu.

      The **Primary Boundary** dialog box disappears while you draw the U vector.

   b. Select the U vector's origin, which must be on the boundary's surface, either by:
      - Clicking the point for the vector origin.
      - Typing the point's coordinates in the **X**, **Y**, and **Z** boxes.

   c. Select a point on the u-axis to indicate the U vector direction.

      The **Primary Boundary** dialog box reappears and the model display shows the U vector and V vector as red and blue arrows respectively.

   d. If you need to reverse the direction of the V vector, select **Reverse Direction**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

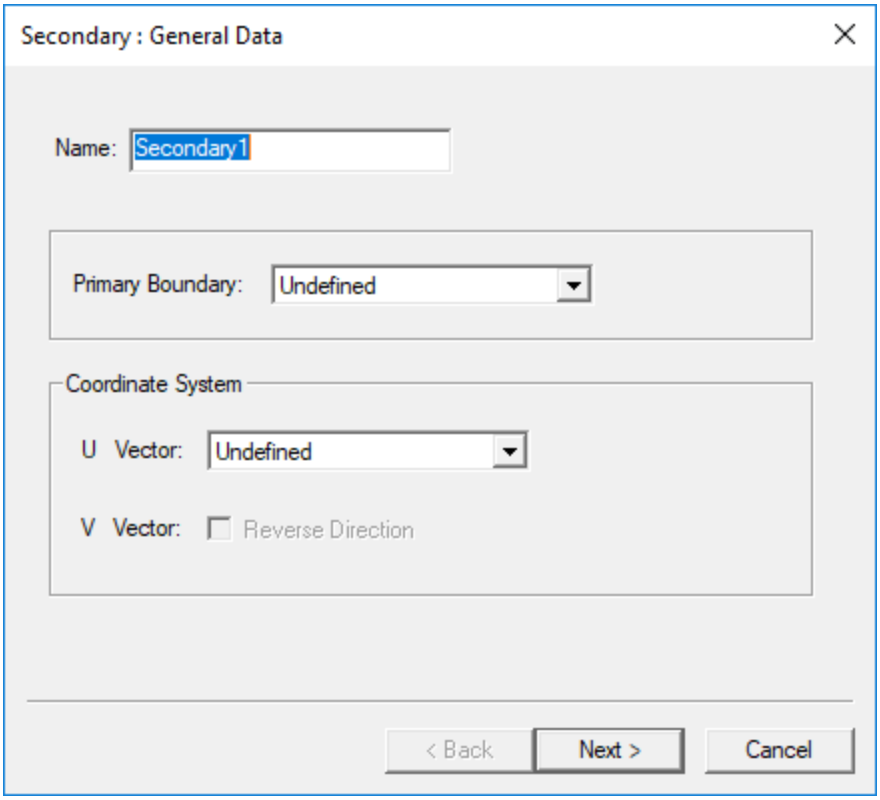
Ansys Electronics Desktop will compute the E-field on this boundary and map it to the secondary boundary using the transformation defined by the Coupled Lattice Pair coordinate systems.

Assigning Secondary Boundaries for Lattice Pair

Coupled Lattice Pair boundaries enable you to model planes of periodicity where the E-field at every point on the Secondary boundary surface is forced to match the E-field at every corresponding point on the Primary boundary surface to within a phase difference. The transformation used to map the E-field from the Primary to the Secondary is determined by specifying a coordinate system on both the Primary and Secondary boundaries. You can use the **Coupled...> Lattice Pair...** command to auto-set the coordinate system, for both Primary and Secondary boundaries together, you have the option of assigning the Secondary boundary directly.

To assign a Secondary boundary:

1. Select a surface on which to assign the boundary and click **HFSS> Boundaries> Assign> Coupled...> Secondary** to bring up the **Secondary Boundary** dialog box.



2. Select the corresponding Primary boundary from the **Primary Boundary** drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

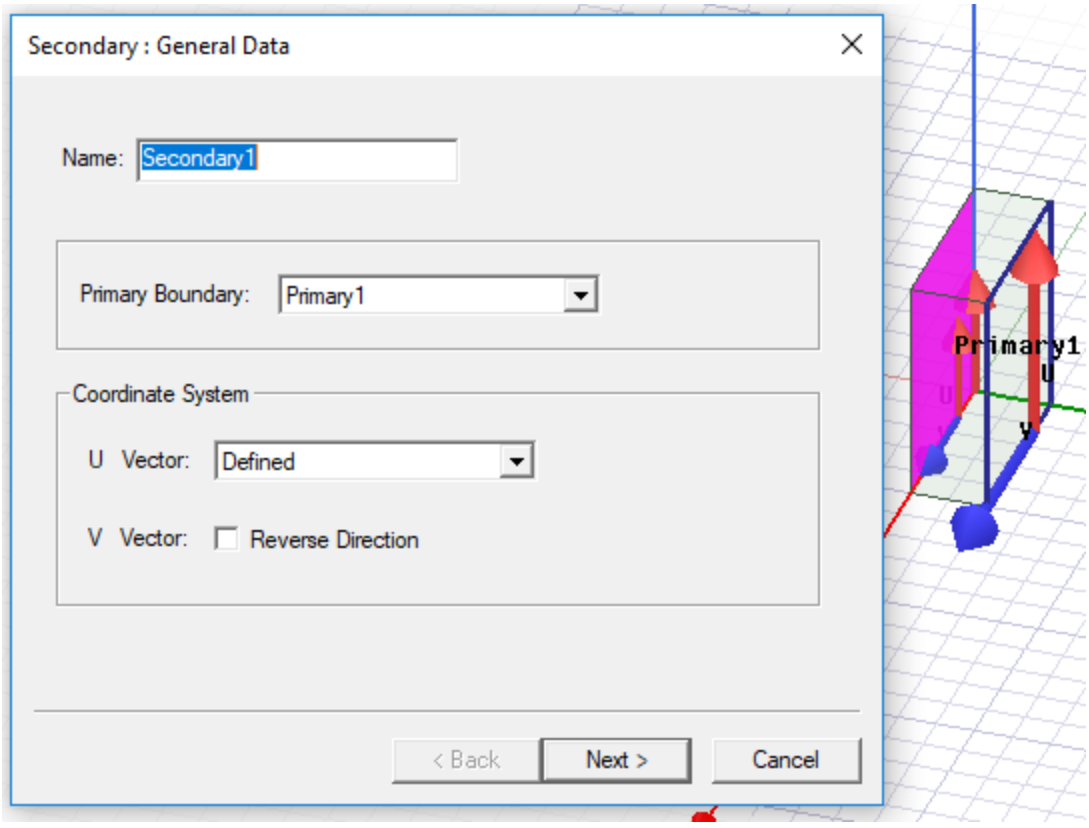
If a primary boundary has not yet been defined, return to make this selection when it has been defined.

3. You must specify the coordinate system in the plane on which the boundary exists. First draw the U vector of the coordinate system. HFSS uses the U vector that you draw and the normal vector of the boundary face to calculate the V vector. If necessary, you can reverse the direction of the V vector.
   a. Select **New Vector** from the **U Vector** drop-down menu.

      The **Secondary** dialog box disappears while you draw the U vector.

   b. Select the U vector's origin, which must be on the boundary's surface, in one of the following ways:
      - Click the point for the vector origin.
      - Type the point's coordinates in the **X**, **Y**, and **Z** boxes.
   c. Select a point on the u-axis to indicate the U vector direction.

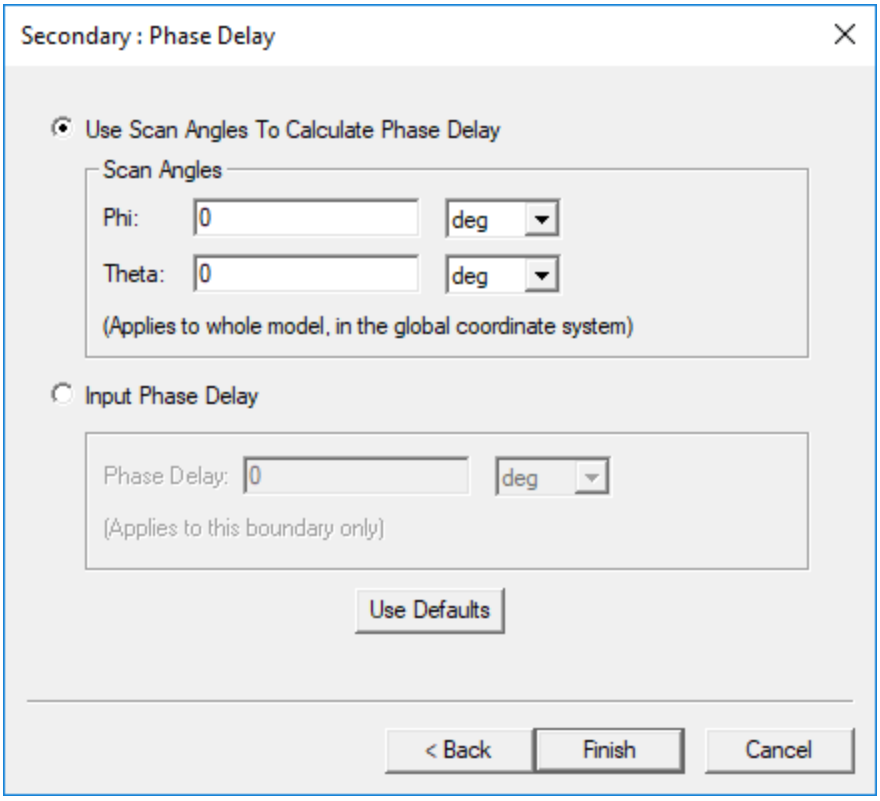
      The **Secondary Boundary** dialog box reappears and the model display shows the U vector and V vector as red and blue arrows respectively.



   d. If you need to reverse the direction of the V vector, select **Reverse Direction**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **Next**.



5. You have the option to relate the secondary boundary's E-fields to the primary boundary's E-fields in one of the following ways:
   - For driven designs, select **Use Scan Angles to Calculate Phase Delay** to enable the **Scan Angle** fields. Then enter **Phi** and **Theta** scan angles. These apply to whole model, in the global coordinate system. The phase delay is calculated from the scan angles; however, if you know the phase delay, you may enter it directly in the **Phase Delay** box.

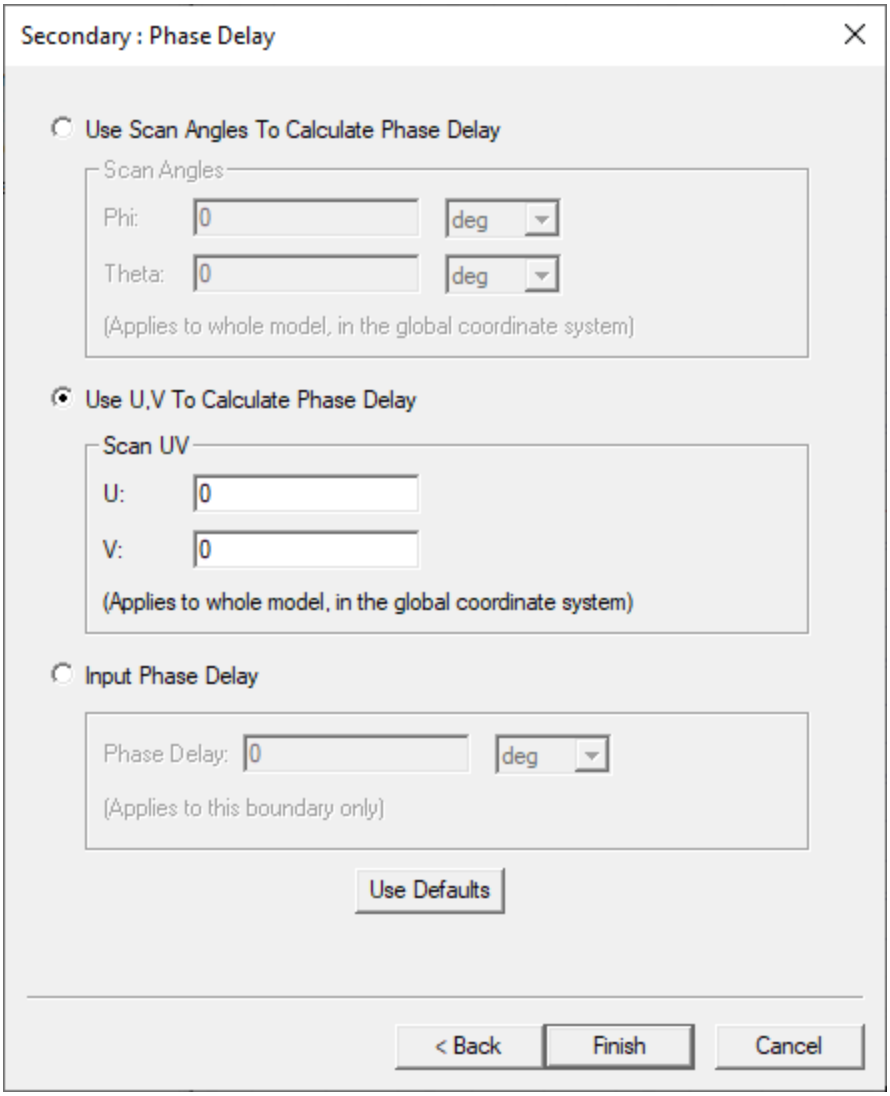
     > **Note:**
     >
     > For Eigenmode problems, the **Use Scan Angles to Calculate Phase Delay** fields are disabled.

   - Select **Input Phase Delay**, and then enter the phase difference, or phase delay, between the boundaries' E-fields in the **Phase Delay** box. The phase delay applies only to this boundary.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> You can assign a variable as the phi, theta, or phase delay values.

A Beta Option allows you to **Use U,V to Calculate Phase Delay**.



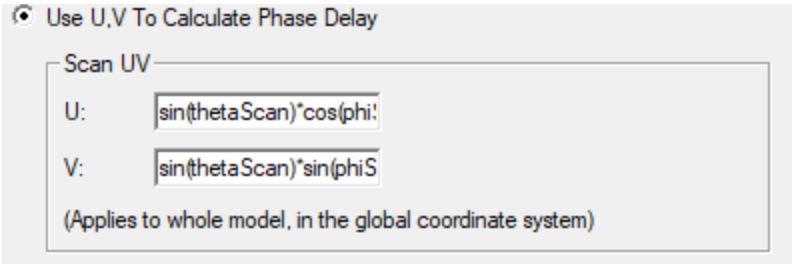
This permits you to scan to scan in the invisible range which is conveniently specified by u,v coordinates. U or V can span anything between -infinity to + infinity or practically some large negative value to large positive value. When $U^2+V^2 > 1$, it is the invisible range. You can specify U and V as equations in this form:

$u = \sin\theta \cdot \cos\varphi$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

v = sinθ·sinφ

Note that you can employ variables in the equations:



Ansys Electronics Desktop will compute the E-field on the primary boundary and map it to this boundary using the transformation defined by the Primary and Secondary Boundary coordinate systems.

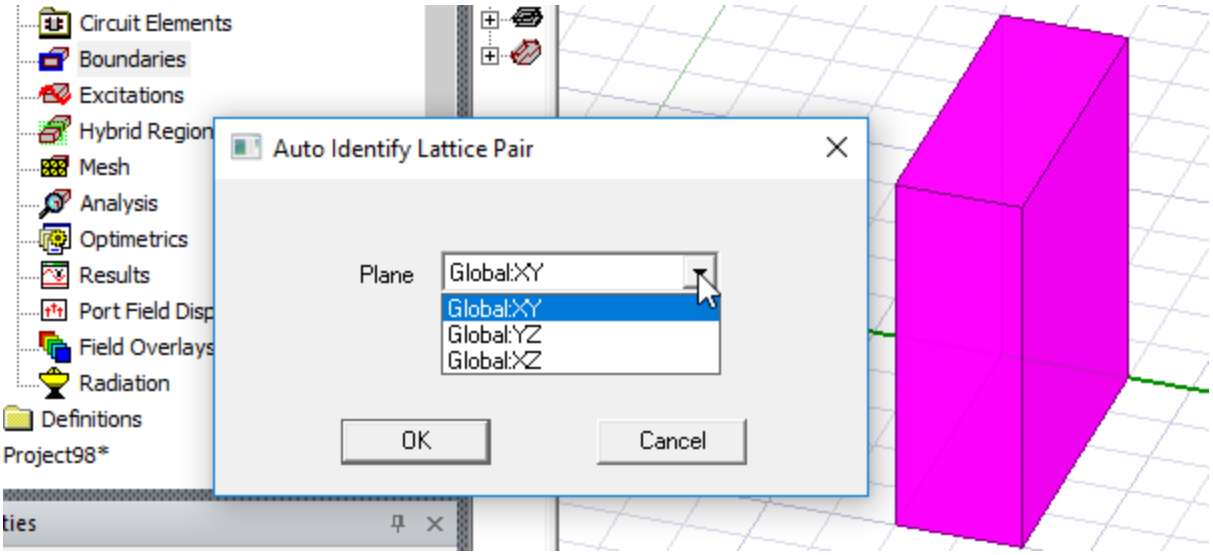
Auto Identify Lattice Pair for Coupled Boundaries

Coupled Lattice Pair boundaries enable you to model planes of periodicity where the E-field at every point on the Secondary boundary surface is forced to match the E-field of every corresponding point on the Primary boundary surface to within a phase difference. The transformation used to map the E-field from the Primary to the Secondary is determined by specifying a coordinate system on both the Primary and Secondary boundaries. You can use the **Auto Identify Lattice Pair** command to auto-set the coordinate system for all surfaces and assign lattice pairs to all faces parallel to the selected plane.

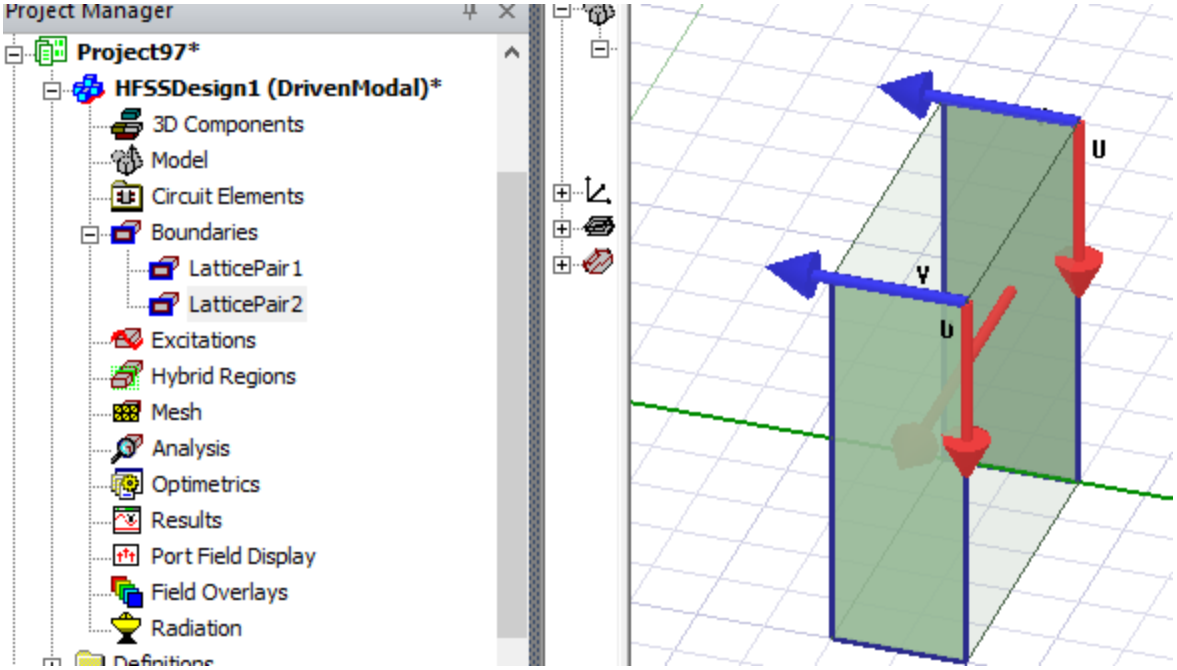
To automatically identify and assign lattice pairs on a selected object:

1. Select an object on which to assign lattice pairs and click **HFSS>Boundaries>Assign>Coupled>Auto Assign Lattice Pair**to bring up the **Auto Identify Lattice Pair** dialog box. Select a plane from the menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **OK**.
   Lattice Pair boundaries will be assigned to all the pairs of parallel faces which are perpendicular to the selected plane. Each pair is formed by exactly two parallel faces. Notice that in this example two Lattice Pairs appear in the history tree, with the second highlighted in the modeler.



Ansys Electronics Desktop will compute the E-field on this boundary and map it to the secondary boundary using the transformation defined by the Coupled Lattice Pair coordinate systems.

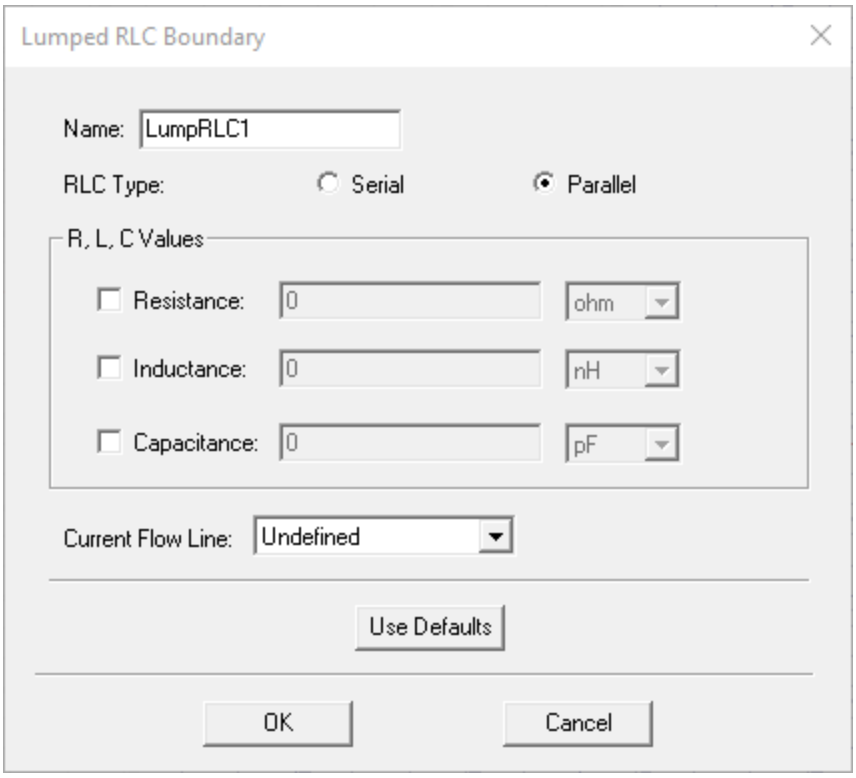
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning Lumped RLC Boundaries

A lumped RLC boundary represents a parallel or serial combination of lumped resistor, inductor, and/or capacitor applied to a surface. Multiple RLC boundaries can be used to model other circuit configurations. For example, a lumped RLC serial circuit connection can be modeled with three connected RLC surfaces: one with only resistance, one with only inductance, and one with only capacitance.

To create a lumped RLC boundary:

1. Select a surface on which to assign the boundary and click **HFSS >Boundaries>Assign>Lumped RLC** to bring up the **Lumped RLC Boundary** dialog box.



2. Select **Resistance**, **Inductance**, and **Capacitance** as needed and specify values and units for each selected element. Optionally, you can assign a variable to any of these values.
3. To specify where on the surface the current and voltage will be controlled, define a **Current Flow Line**. The selection field initially appears as **Undefined**. Select **New Line** to define a vector line on the boundary surface.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> HFSS assumes the lumped RLC is assigned to a rectangular face. If you assign a non-rectangular face, HFSS issues a warning, but proceed with the solution. Using a non-rectangular face can result in less accurate representation of the lumped RLC. See the technical notes on RLC boundaries for more information.

Vector Line

To draw a vector line to indicate the current flow:

1. Select **New Line** from the Lumped RLC Boundary dialog's drop-down menu.

   The dialog box disappears while you draw the vector line.

2. Select the start point in one of the following ways:
   - Click the point.
   - Type the point's coordinates in the in the **X**, **Y**, and **Z** boxes on the status bar.
3. Select the endpoint using the mouse or the keyboard.
4. Once the line has been defined, you can edit it as follows:

   Select **Swap End Points** from the dialog drop-down menu to switch the start and endpoints of the line, reversing the line's direction.

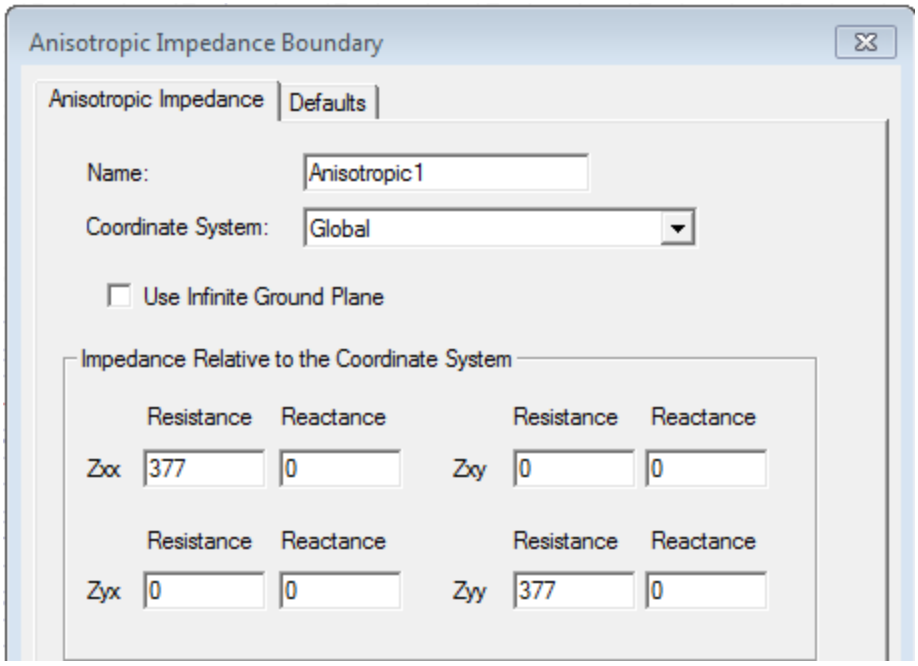
Assigning Anisotropic Impedance Boundaries

Planar screens or grids of large extent with periodic geometry can be replaced by an Anisotropic impedance boundary. The boundary applies a homogeneous characteristic impedance to the surface in an effort to create an equivalent electrical representation of the geometric grid pattern. For legacy designs, this boundary replaces the Screening Impedance boundary.

To assign an Anisotropic Impedance boundary

1. Select a surface on which to assign the boundary and click **HFSS>Boundaries>Assign>Anisotropic Impedance** to bring up the **Anisotropic Impedance Boundary** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. The Coordinate System drop-down menu lists the Global Coordinate system and any relative coordinate systems if you have defined them in the design. Select the Coordinate System that defines the anisotropic characteristic of the impedance boundary. (See Creating a Relative Coordinate System.)

> **Note:**
>
> An anisotropic boundary must not touch ports in a design.

3. Select **Infinite Ground Plane** if you want the surface to represent an electrically large ground plane when the radiated fields are calculated during post processing.
4. In the Impedance Relative to the Coordinate System area, you set the **Resistance** and **Reactance**. If Anisotropic Impedance is checked, the wizard shows Resistance and Reactance fields for X Axis alignment and Y axis alignment. These values can be numeric or can reference variables, including spatial variables x, y, and z. Isotropy will be inferred if Zxy and Zyx are left as zero.

Assigning Layered Impedance Boundaries

A layered impedance boundary is used to model multiple thin layers in a structure as one impedance surface. The effect is the same as an impedance boundary condition, except that Ansys Electronics Desktop calculates the impedance of the surface based on data you enter for the layered structure, including surface roughness.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Materials with thermal modifiers can have layered impedance boundaries assigned to them. The layered impedance boundary is supported for single-frequency solutions and for Discrete and Interpolating frequency sweeps. The order of the layers does not matter to the implementation.

Layered impedance boundaries can be one sided or two sided. One-sided boundaries terminate their model in an infinite space, such as air or vacuum. Two-sided boundaries do not. Two-sided boundaries can be used in 3D models. By default, Ansys Electronics Desktop assumes the layered structure is one sided, external to the design.

If the layered structure is within a 3D model, meshing thin layers and solving inside is inefficient. You can replace the highly conductive thin layer by a shell element. Shell elements are 3D layers that have been replaced by a sheet. They cannot represent a one-sided impedance boundary. The **Shell Element** option only applies to HFSS Driven solution types and does not apply to HFSS Eigenmode and Transient solution types or to HFSS-IE designs. Shell elements maintain two sets of unknown coefficients for the top and bottom surface. If the angle of incident deviates significantly from the boundary, then the accuracy of the shell elements degrades and you should use a layered impedance boundary instead.

Practical uses of shell elements in simulations include:

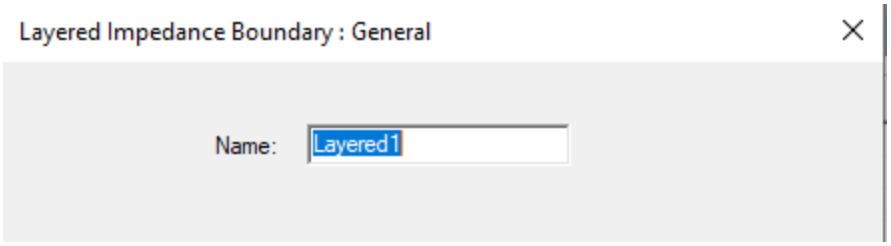
- an antenna inside a car chassis when the chassis is modeled using sheets
- an antenna inside a dielectric radome when the radome is modeled using sheets

If you specify that a two-sided sheet is to be modeled as a shell element, the following HSS features are disabled:

- Scattered field formulation: HFSS disables this when all boundaries are radiating boundaries AND when the design has an Incident Wave excitation.
- IE solver

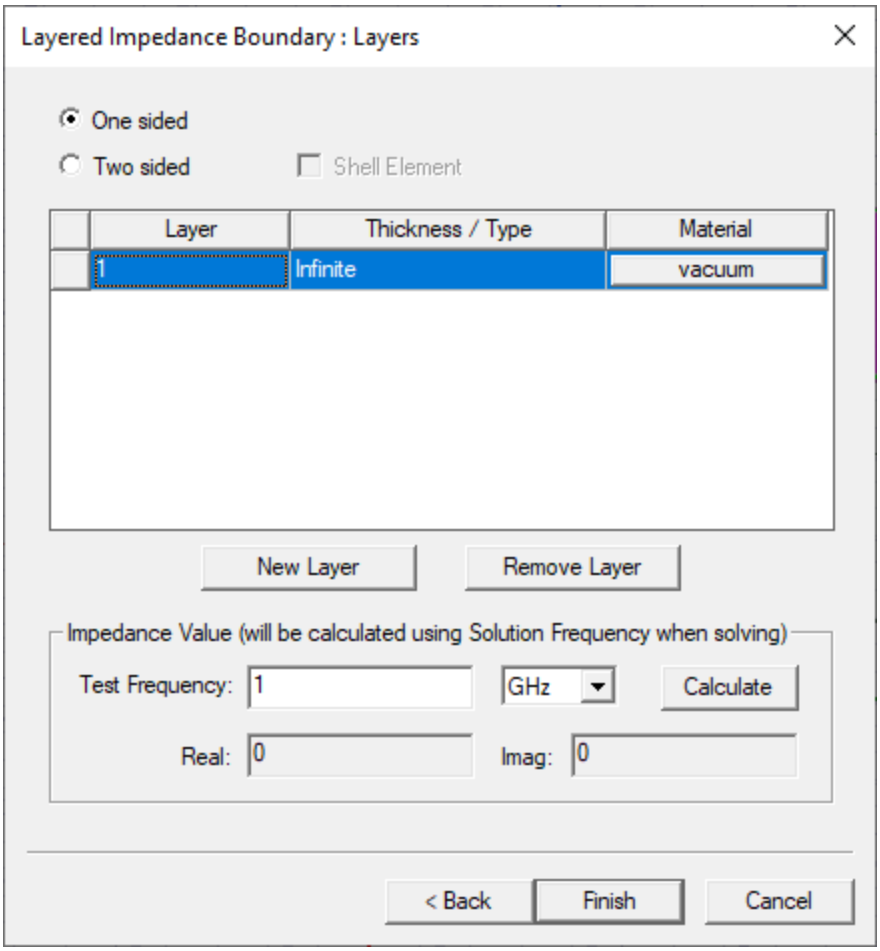
To assign a layered impedance boundary:

1. Select a surface on which to assign the boundary and click **HFSS**> **Boundaries**>**Assign**>**Layered Impedance** or **HFSS**>**Boundaries**>**Assign**>**Surface Roughness**> **Groiss (Layered Impedance) ...**to bring up the **Layered Impedance Boundary** window.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Next** or the **Layers** tab, depending on the general option setting.



3. If the layered structure is external to the design, do the following:

   By default, Ansys Electronics Desktop assumes the layered structure is **One sided**, external to the design; the outermost layer of the structure is listed.

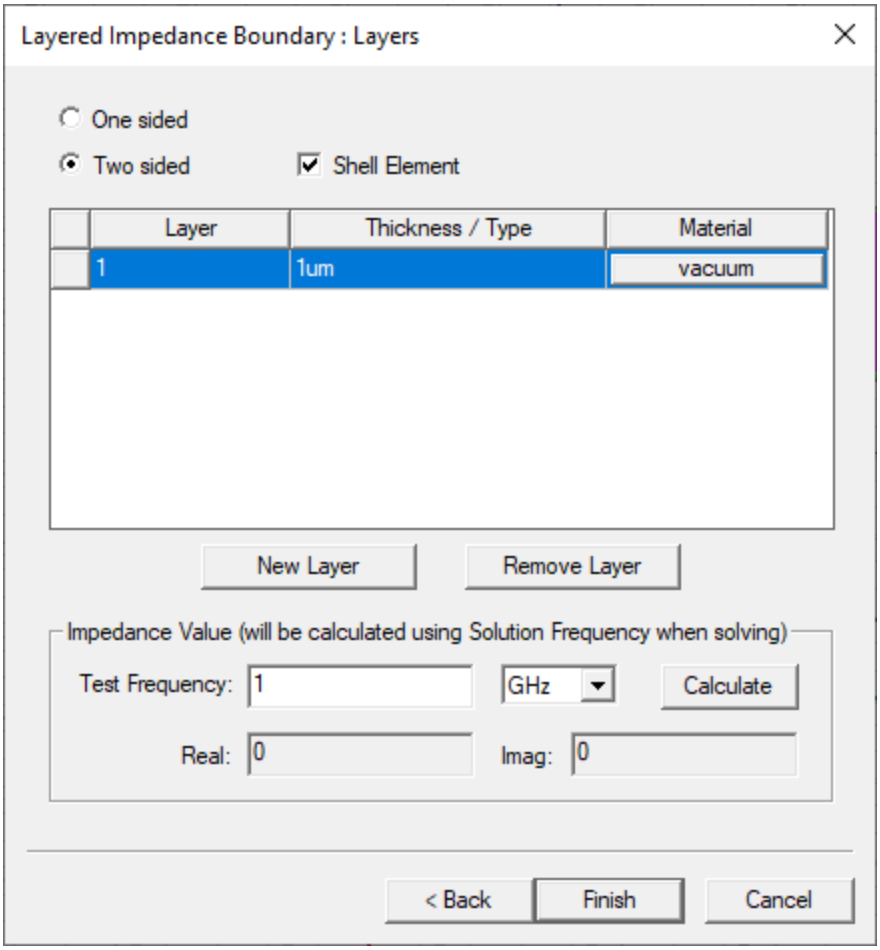
   a. Select whether this layer is an **Infinite**, **Perfect E**, or **Perfect H** layer from the **Thickness/Type** column.
   b. If the layered structure is within a 3D model, do the following:
      1. Select the **Two sided** option.
      2. Enter a thickness for the first layer in the **Thickness/Type** column. You can assign a variable as this value.

      If it is a two-sided BC and not assigned to an SBR+ region, layer thickness should not exceed lambda/10 for for dielectric objects. For thin conductors, if the skin depth is much smaller than the layer thickness, you cannot replace it by an internal sheet.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Neither of these restrictions apply if the layered structure is part of an SBR+ region. For an SBR+ objects, there is no limit to the electrical thickness (i.e., measured in wavelengths) of the layers for result quality. However, it is not recommended that the absolute thickness of the layers exceed 20% of the lateral extent of the surface to which the BC is applied.

4. To change the first layer's material, click **vacuum** and follow the procedure for assigning a material. The material properties that will apply to the layers can be functions of complex numbers.
5. The **Shell Element** check box is for HFSS Driven situations where the layered structure is within the 3D model and you want to use sheets to model thin but finite thickness objects.



6. To add a new layer to the structure:
   a. Click **New Layer**. The new layer is added at the end of the list.
   b. Enter a thickness for the layer in the **Thickness/Type** column. You can assign a variable as this value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

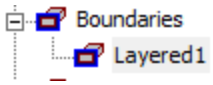
c. To change the layer's material, click **vacuum** and follow the procedure for assigning a material.

7. Optionally, to reorder layers, click the first row square and drag the row to the desired position.
8. Optionally, to view the impedance values that will be calculated based on the data provided, do the following:
   a. Enter the frequency at which the solution is being solved in the **Test Frequency** text box.
   b. Click **Calculate**.

      The real and imaginary components of the Ansys Electronics Desktop-calculated layered impedance value appear.

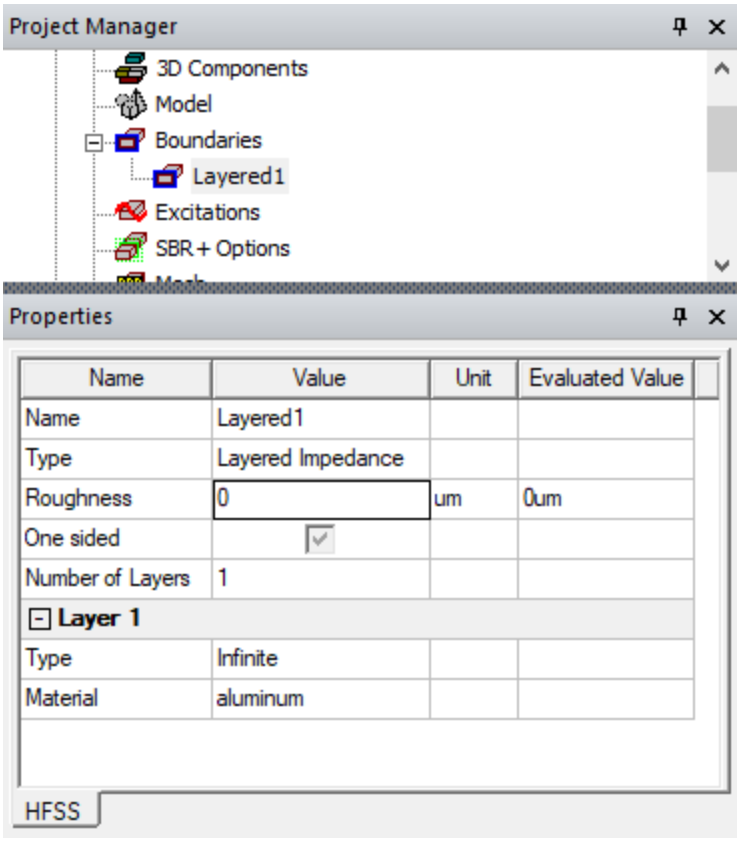
      Enter the **Surface Roughness** for the layered structure.
      If the layered structure is internal to the design, enter the average surface roughness of the two outermost sides. You can assign a variable as this value. Ansys Electronics Desktop uses the Groiss model for surface roughness. When surface roughness is activated, it is applied to the first conducting layer (not necessarily the first layer) for an external boundary, and to both the first and the last conducting layer for an internal boundary.

9. Click **Finish** to assign the boundary. The boundary appears under the **Boundaries** icon in the **Project** tree.



Optionally, you can use the **Properties** dialog to enter the **Surface Roughness** for a layered structure. Selecting the boundary displays the Properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the layered structure is internal to the design, enter the average surface roughness of the two outermost sides into the **Roughness** field. You can specify Units by clicking the drop-down menu in the Units column. You can assign a variable as this value. Ansys Electronics Desktop uses RMS deviation of the conductor surface from a plane. When surface roughness is activated, it is applied to the first conducting layer (not necessarily the first layer) for an external boundary, and to both the first and the last conducting layer for an internal boundary.

> **Note:**
>
> If a fast sweep is defined in a design that contains a layered impedance boundary, the impedance may only be accurate for the center frequency.

Assigning Linked Impedance Boundaries

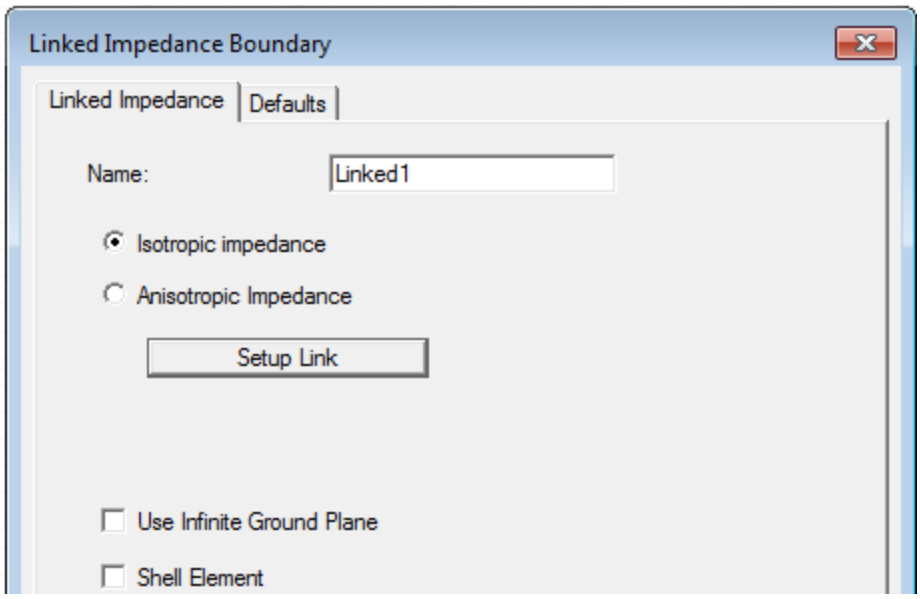
A linked Impedance boundary can be an isotropic or anisotropic impedance boundary in another design. Planar screens or grids of large extent with periodic geometry can be replaced by an Anisotropic impedance boundary. The boundary applies a homogeneous characteristic

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

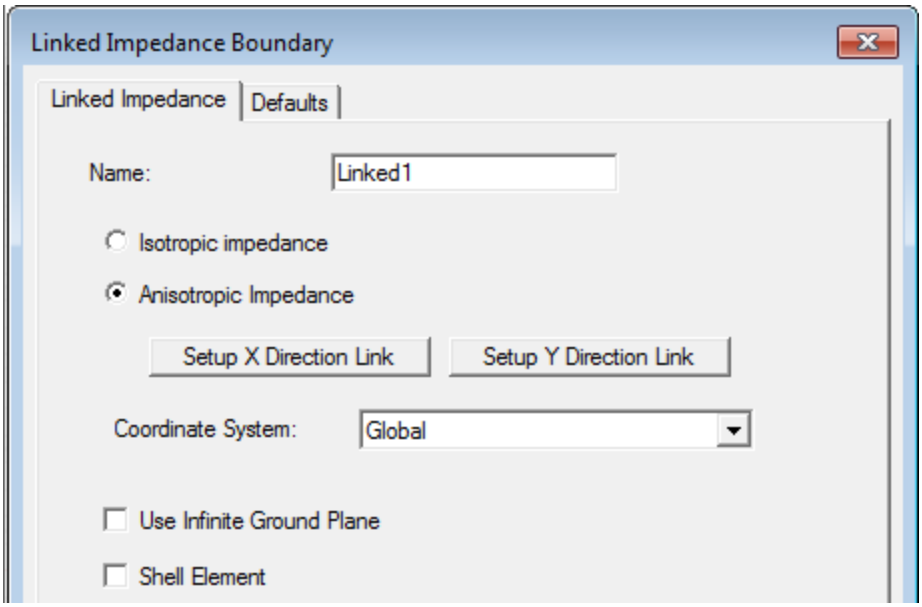
impedance to the surface in an effort to create an equivalent electrical representation of the geometric grid pattern. The link boundary can use an infinite ground plane and shell elements.

To assign a Linked Impedance boundary

1. Select a surface on which to assign the boundary and click **HFSS>Boundaries>Assign>Linked Impedance** to bring up the **Linked Impedance Boundary** dialog.



2. If you select Anisotropic Impedance, the dialog includes buttons to Setup X Direction Link, Setup Y Direction Link and a selection list for the Coordinate System.



If the linked projects have wave ports, x and y directed projects are different and have to

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

have one mode per port.
If the linked projects have Floquet ports, x and y directed projects should be identical and have to have 2 modes per port (TE and TM modes). In that case, the field of mode 1 serves as x directed field, while the field of mode 2 serves as y directed field. It is also a valid setup to assign two different projects with Floquet ports, but with one mode per port.

3. The Coordinate System drop-down menu lists the Global Coordinate system and any relative coordinate systems if you have defined them in the design. Select the Coordinate System that defines the anisotropic characteristic of the impedance boundary. (See Creating a Relative Coordinate System.)

> **Note:**
>
> An anisotropic boundary must not touch ports in a design.

4. Select **Infinite Ground Plane** if you want the surface to represent an electrically large ground plane when the radiated fields are calculated during post processing.
5. Select Shell Elements if you want to the boundary modeled as a shell element.

With shell elements: Make sure two ports are defined, and for isotropic 1 mode on each port and for anisotropic 1 mode one each port when two designs are linked and 2 modes on each port when one design is linked.

Without shell elements: Allow either one or two ports but only 1 mode each for isotropic, 1 mode each for anisotropic with two linked designs, and 2 modes each for anisotropic with one linked design. An info or warning message is issued when two ports are defined in the design to let you know we only extract the link from one of the ports and specifies which one.

Get Impedance from External Design

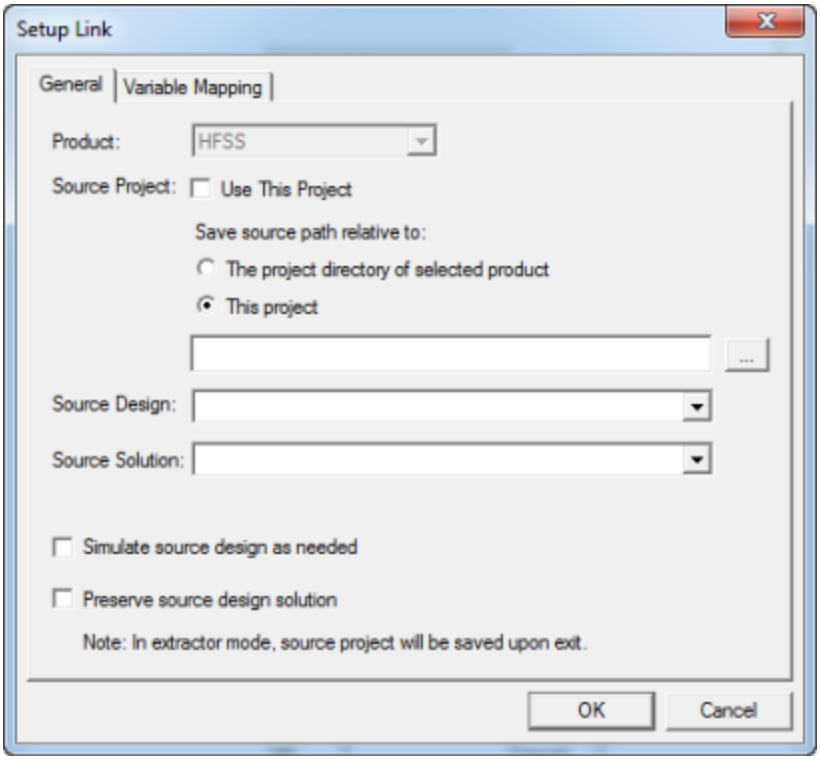
To get impedance from an external design for a Linked Impedance Boundary for an isotropic design:

1. Click the **Setup Link** button to display the **Setup Link** dialog.

For Linked anisotropic impedance cases, select **Setup X Direction Link** button to chose a design which will define the impedance in the X direction. Then select **Setup Y Direction Link** button to chose a design which will define the impedance in the Y direction.

The **Setup Link** dialog has three fields under the **General** tab: Project File, Design, and Solution.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Specify the Project file for the design that is the source. A browse button [...] lets you look through your file system. If you do not specify a project file, but select the current model, the current Project File is automatically filled in.
3. Specify the Design for the source. If the source is in the current design, you can select this from a drop down menu. If you select the current model, the Project File is automatically filled in.
4. Use the radio button to specify whether to save the source path relative to **The project directory of the source project** or **This project**.
5. Specify the Solution to use. A drop down list lets you select from the available solutions.

The "Default" solution is the product dependent solution of the first Setup.

That is the setup listed first in the source design's project tree (alphanumerical order). A product specific solution of this setup becomes the default solution. In most products, it is LastAdaptive. In a Transient solution type, it is "Transient."

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The solution in the source design must provide data for the target design's adaptive frequency as well as its sweeps. That is, the adaptive frequency for the target design must be included in the sweep in the source design.

If necessary, you can open the source design and add an appropriate frequency point to an existing sweep.

6. Use the check box specify whether to **Simulate source design as needed**.
7. Use the check box to specify whether to **preserve the source design solution**. Note that in **Extractor mode**, the source project will be saved upon exit. **Extractor mode** means that the software is opened during the link solely for the purpose of solving.
8. Under the **Variable Mapping** tab, you can set the desired variable values in the source design. If the source and target designs contain same named variables, you can choose to Map Variable By Name. In this case, same named variables are mapped automatically.
9. Click OK to close the **Setup Link** dialog and return to the Linked Impedance Boundary dialog.

Assign Gradient Surface Roughness Boundaries

Gradient Surface Roughness Boundaries address the situation where the conductors drawn in the model are perfectly smooth, but the manufactured surface is bumpy at microscopic scale. It generates a frequency dependent dataset for resistance and reactance values behind the scene for solve.

Parameters of the gradient model in HFSS are:

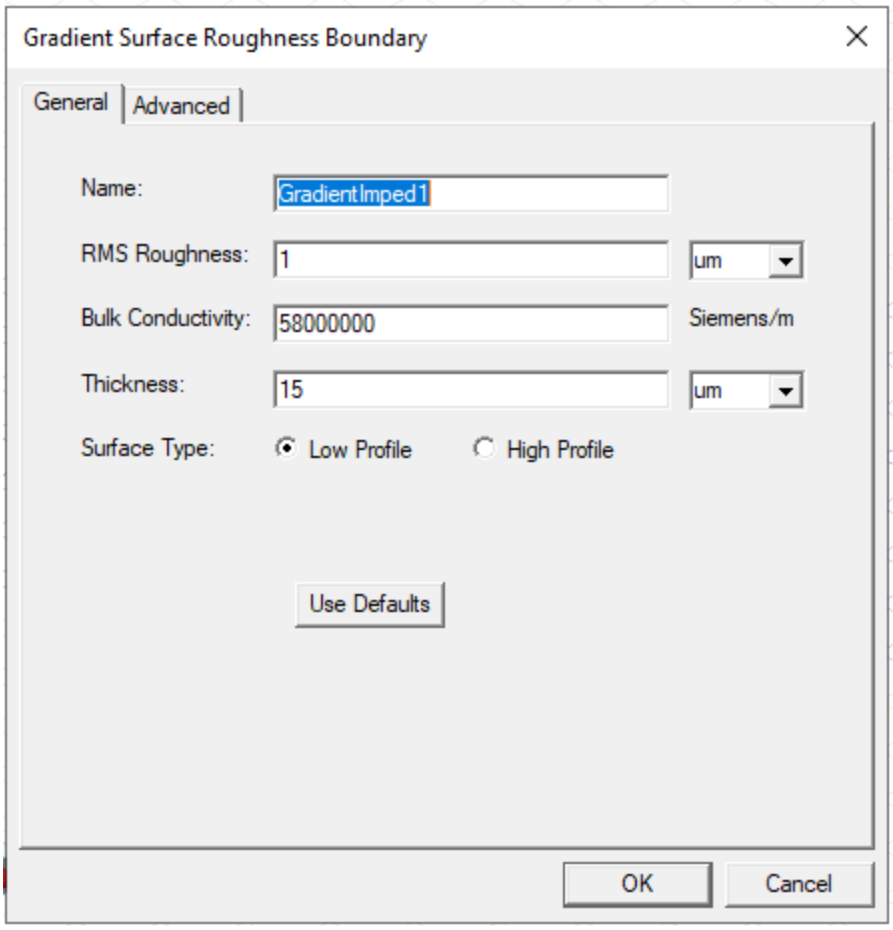
- The RMS Roughness parameter describes the root mean square (RMS) roughness of the surface (Rq). It is routinely measured by manufacturers, and can be entered as a value with units.
- The metal’s bulk conductivity.
- The conductor thickness.
- The surface profile. Choose Low-Profile for low-profile foils and polished surfaces, and High-Profile for high-profile foils and rough conductors.
- On the **Advanced** tab, the maximum frequency of interest in the simulation.

The next section first describes how to assign a Gradient Suface Roughness Boundary and the subsequent section provides a technical overview.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning Gradient Surface Roughness Boundary

1. Select a surface on which to assign the boundary, right-click **Assign Boundary** > **Gradient Surface Roughness** or **Assign Boundary**> **Surface Roughness**>**Gradient...** to bring up the *Gradient Surface Roughness Boundary* dialog box.



2. Set the Parameters. and click **OK**. The boundary is listed in the Project Manager and displayed in the Modeler window.

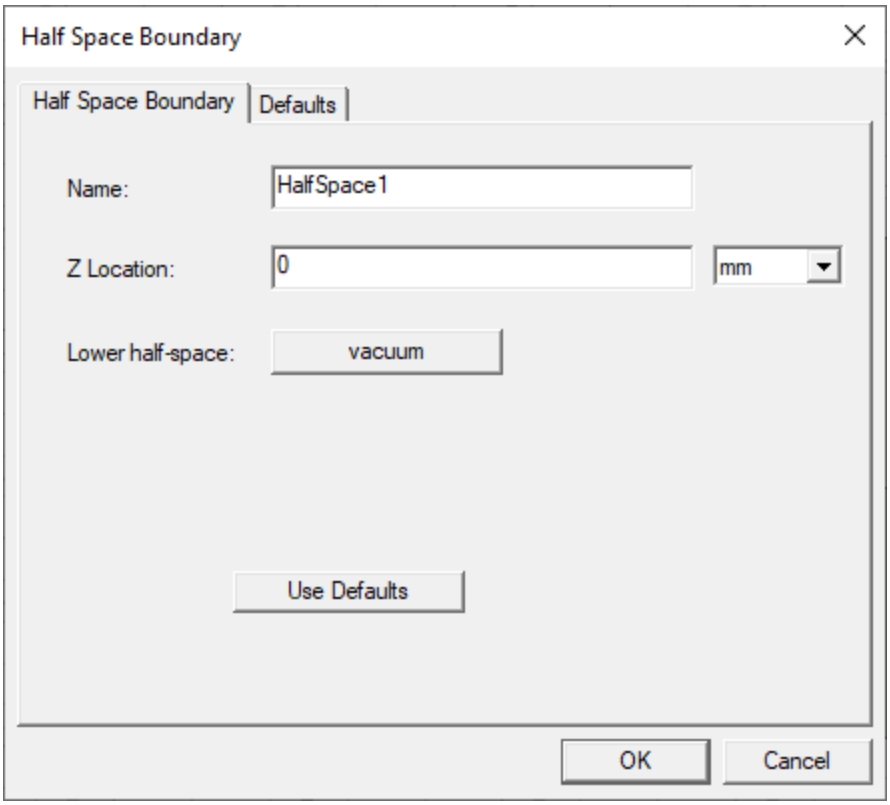
Assigning Half Space Boundaries in HFSS

The Half Space boundary is designed for solving antenna and scattering problems with a background comprising a dielectric half space. The Half Space boundary splits the background material along a z=constant plane. The antenna or scatterer can be located in the top medium (for example, air) or the bottom medium (for example, sea water) or embedded in both. Half space boundary is only available to an HFSS Hybrid and Arrays design but can include both

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FEM areas and IE regions. An HFSS Hybrid and Arrays design can only have one half space boundary. You do not select an object to assign a Half Space boundary. For such projects, compared to solving with a single domain, using a Half Space boundary the results will be similar though not identical but will use less time and fewer resources.

Right-click **Boundaries>Assign>Half Space** to display the Half Space Boundary dialog.



- Specify a Name in the text field.
- Specify a Z Location, Z and select units from the menu.
- Specify the lower-half space material. The "lower half-space" material defines the material in the lower half-space, and the upper half-space is defined using the global background material.

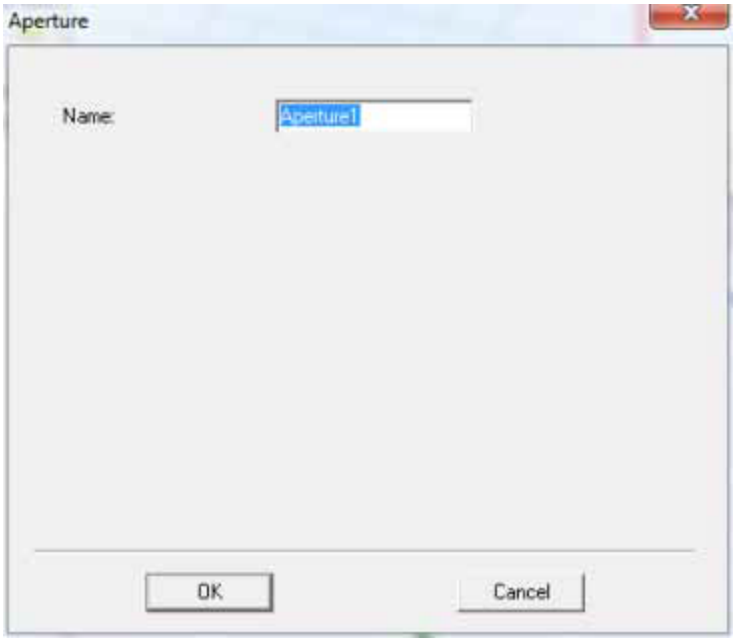
The Boundary appears in the Project tree. If you select the boundary in the Project tree, a rectangle representing the boundary shows in the Modeler window, and the docked properties shows the properties.

An aperture boundary can only be assigned on a 2D sheet object assigned as IE Region. These objects will be meshed as part of the solution process.

To create an aperture boundary in HFSS:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

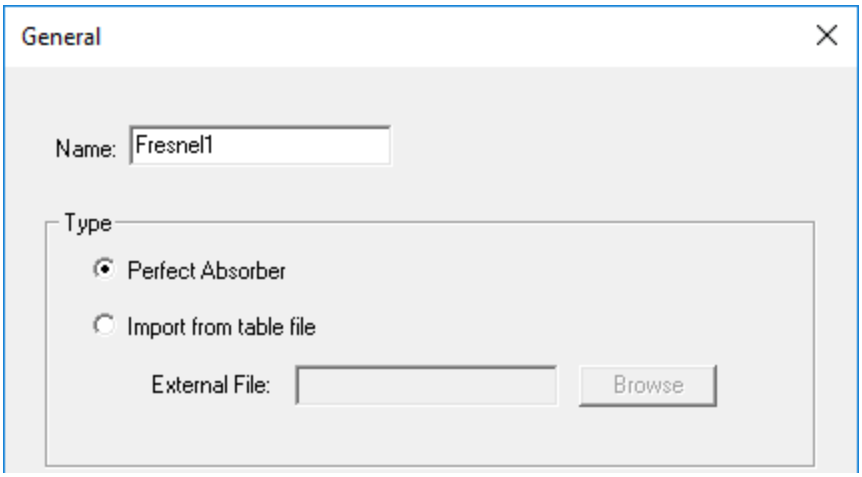
1. Create a 2D sheet object on the XY plane at the elevation of ground plane.
2. Select the object and right-click **Assign Hybrid>IE Region**.
2. Select the object and right-click **Boundaries>Assign>Aperture** to display the **Aperture** dialog.



3. Type the boundary's name in the **Name** text box or accept the default name. (To change the default base name to one of your choosing, see Setting Default Boundary/Excitation Base Names.)

Assigning (SBR+) Perfect Absorber Boundaries

The **Assign Boundary>Fresnel (SBR+)** command allows you to specify a perfect absorber.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

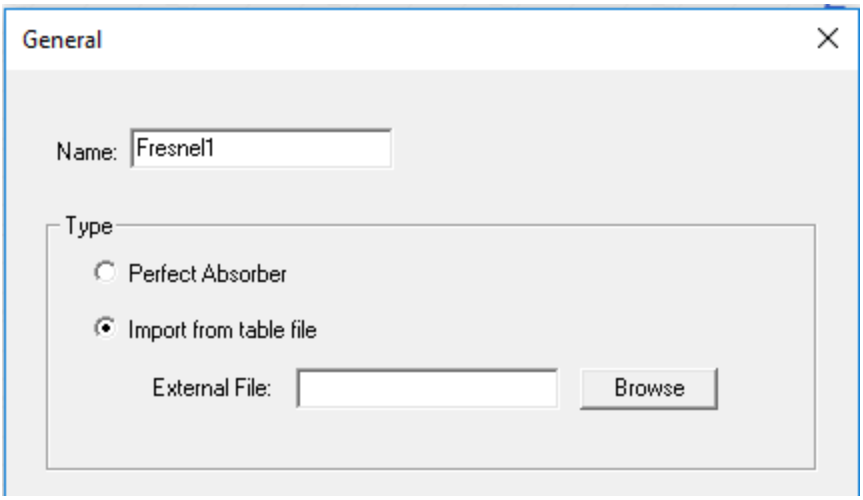
Perfect Absorber: Incident SBR+ rays are absorbed by surfaces with this boundary, and no reflection or transmission rays are generated.

This does not imply that there is no radiation from surface currents on perfectly absorbing boundaries. The perfect absorber still generates very strong forward scattering. This scattering is associated with the incident GO ray fields at the surface. These ray fields are converted into equivalent currents at the surface that are radiated to observation angles and receiving antennas in any direction (including the backward direction); it tends to be strongest in the forward direction. At the same time, since the reflected ray has zero field strength (and therefore no reflected ray is traced), the equivalent surface currents associated with the reflected ray are also zero, so scattering in the backward half-space is weak.

The Fresnel (SBR+) **Import from table file** option lets you directly load the Fresnel tables from an external file.

Assigning (SBR+) Fresnel Table Boundaries

The **Assign Boundary>Fresnel (SBR+)** command allows you to specify either a perfect absorber or to import a Fresnel table file that describes reflection and transmission coefficients.



Fresnel coefficients are based on the assumption of plane-wave incidence and reflection and represent the ratio of the reflected field to the incident field. Ray bundles can represent complex wavefronts, and each ray represents a sample of that wavefront that is assumed to be locally planar. Provided that the wavefront is not too irregular, and the coatings are not too thick compared to their lateral extent, the Fresnel coefficients applied in conjunction with individual rays provide a good approximation of the reflection of an arbitrary wavefront off a surface

When an SBR+ boundary is configured as an Impedance Boundary or a Layered Impedance boundary, a table of Fresnel coefficients is internally generated for use during the SBR+ ray-tracing simulation. The Fresnel (SBR+) **Import from table file** option lets you directly load the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

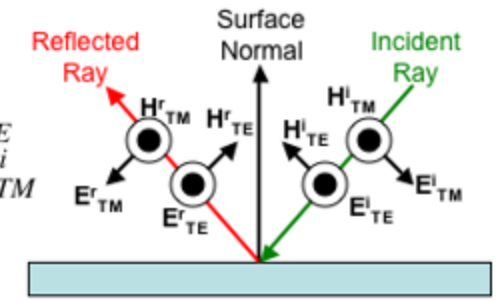
Fresnel tables from an external file. You can develop these from a number of sources, including analytic formulas, specialized codes for simulating the boundary (e.g., HFSS for simulating a frequency selective surface), or measurements. Two types of boundary tables are supported:

- reflection coefficients
- reflection + transmission coefficients

Tables of reflection + transmission coefficients are explained later.

Fresnel coefficients depend on the polarization of the incident plane wave. There are two unique, orthogonal polarizations: transverse electric (TE) and transverse magnetic (TM). HFSS-SBR+ adopts the convention given in the diagram below. In this convention, PEC metal has a TE reflection coefficient of -1 and a TM reflection coefficient of +1. In general, reflection coefficients are complex quantities that depend on incident angle and frequency

$$E^r_{TE} = R_{TE} E^i_{TE}$$
$$E^r_{TM} = R_{TM} E^i_{TM}$$



Reflective Coating Described by Fresnel Reflection Coefficients

When a ray hits a surface, the solver determines its incidence angle relative to the surface normal. The incident ray tube fields are decomposed into TE and TM components. The solver then looks up the TE and TM reflection coefficients for that angle, whether internally generated or provided by an external table as described here. These are applied to the respective incident ray field components to determine the TE and TM components of the reflected ray field. This, in turn, influences the surface currents and scattered fields generated at the current bounce and subsequent bounces.

Fresnel Reflection Table File Format

The Fresnel reflection table file format is documented in the header comments in the example below. This first example is for a table defined at a single frequency, which means it will be used at all frequencies during a simulation. A multi-frequency example is provided later for a reflection/transmission table, but since their formats are nearly identical, the same encoding can be used for a reflection-only table

```
# Reflection table file.
# Blank lines and lines beginning with are ignored.
```

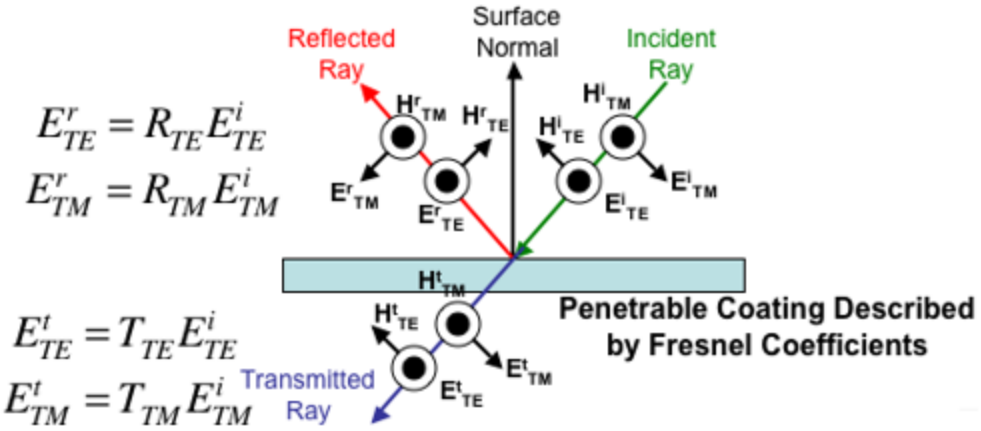

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
# The following key is critical in distinguishing between a
reflection table and a reflection/transmission table.
ReflTab1e
# Incident angle, theta, is measured from the normal to the surface .
# In the table, theta is assumed to vary from C (head-on) to 90 deg
(grazing) .
#The density cf angular sampling is specified by the number cf theta
steps.
# In this example, there is a sample every 1 deg.
# <num theta step> number of theta points - 1
90
# Frequency domain
# Mono freq. table for 1.000000 GHz data type
# <freq_domain_type> = { MonoFreq, Multi Freq}
MonoFreq
# Data section follows. Frequency loops within theta.
# <rte_rl> <rte_im> <rtm_rl> <rtm_im>
-3.81966e-01 0.00000e+00 3.81966e-01 0.00000e+00
-3.82C18e-01 0.00000e+00 3.81914e-01 0.00000e+00
-3.82174e-01 0.00000e+00 3.81758e-01 0.00000e+00
-3.82435e-01 0.00000e+00 3.81497e-01 0.00000e+00
-3.82799e-01 0.00000e+00 3.81132e-01 0.00000e+00
-3.83269e-01 0.00000e+00 3.80662e-01 0.00000e+00
-3.83843e-01 0.00000e+00 3.80086e-01 0.00000e+00
-3.84523e-01 0.00000e+OC 3.79403e-01 0.00000e+00
-3.8S3C9e-01 0.00000e+0C 3.78613e-01 0.00000e+00
-3.86201e-01 0.00000e+0C 3.77715e-01 0.00000oe+00
-3.87200e-01 0.00000e+0C 3.76708e-01 0.00000e+00
:
:
-9.16560e-01 0.00000e+00 -6.42464e-01 0.00000e+00
-9.32634e-01 0.00000e+00 -7.03164e-01 0.00000e+00
-9.49016e-01 0.00000e+00 -7.68667e-01 0.00000e+00
-9.65704e-01 0.00000e+00 -8.39528e-01 0.00000e+00
-9.82699e-01 0.00000e+00 -9.16389e-01 0.00000e+00
-9.99900e-10 0.00000e+O0 -9.99500e-01 0.00000e+00
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Penetrable boundaries, as are encountered in the case of canopies and radomes, should be described by both reflection and transmission coefficients. Likewise, arbitrary boundaries that are penetrable can be entered as a table.

When a ray hits a penetrable coatings, it generates both a reflected ray and a transmitted ray, as shown in the diagram below, which also extends the polarization convention of the previous diagram. In the diagram below, it is intentional that the transmitted ray is shown as starting from the top of the surface, rather than the bottom. In the modeler, CAD models can represent infinitely thin surfaces. The effect of boundary thickness is entirely captured in the reflection and transmission coefficients. Hence, in creating a reflection + transmission table input, it is critical to do so such that the phase reference of the transmission coefficient is the surface nearest the incident ray. That way, the transmitted ray will include the correct phase delay given that it is launched from exactly the same point as where the incident ray hits the surface



The following example reflection + transmission table is very similar in structure to the reflection-only table. The only difference is that each line in the table has four extra numbers to represent the complex TE and TM transmission coefficients. This particular example also shows how to define a frequency dependent table of coefficients

```
# EMA3D native format reflection/transmission table file
# The following key is critical in distinguishing between a
reflection table
# and a reflection/ transmission table.
RTTable
# <num theta step> = number of points-1
90
# Frequency domain
# In this example, the frequency samples are from 0.5 to 4 GHz in
20MHz steps
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
# MultiFreq <freq_start_ghz> <freq_stop_ghz> <num_freq_steps>
MultiFreq 0.5000000 4.000000 175
# Data section follows. Frequency loops within theta
# <rte_rl> <rte_im> <rtm_rl> <rtm_im> <tte_rl> <tte_im> <ttm_rl>
<ttm_im>
-6.23449e-01 1.00978e-04 6.23449e-01 -1.00978e-04 -5.88718e-01 -
3.66763e-01 -5.88018e-01 -3.66763e-01
-6.12741e-01 7.99650e-02 6.12741e-01 -7,99650e-02 -6.13162e-01 -
3.29510e-01 -6.13162e-01 -3.29510e-01
-5.78215e-01 1.54566e-01 5.78215e-01 -1.54566e-01 -6.44759e-01 -
2.91524e-01 -6.44759e-01 -2.91524e-01
:
:
-9.99927e-01 2.21130e-05 -9.99629e-01 9.58350e-05 -1.85062e-05 -
7.37198e-05 -1.07270e-04 -3.64781e-04
-9.99920e-01 2.99358e-05 -9.99596e-01 1.33667e-04 -2.89748e-05 -
7.33898e-05 -1.59538e-04 -3.61020e-04
```

Designating Infinite Ground Planes

To simulate the effects of an infinite ground plane in an HFSS design:

- Select the **Infinite ground plane** check box when setting up a perfect E, finite conductivity, or impedance boundary condition. For Impedance, Layered Impedance, and Finite Conductivity Boundary conditions, HFSS supports only one infinite boundary condition per design. For PEC, multiple antenna ground planes are supported.

This selection only affects the calculation of near- and far-field radiation during post processing. Ansys Electronics Desktop models the boundary as a finite portion of an infinite, perfectly conducting plane. If the infinite ground plane does not touch a radiation boundary, you will receive a warning. A finite sheet, which does not touch the radiation boundary condition cannot be an infinite ground plane.

Modifying Boundaries

To change the properties of a boundary, do one of the following:

- Double-click the boundary's icon in the project tree.

The boundary's dialog box appears, in which you can edit its properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Right-click the boundary in the project tree, and then click **Properties** on the shortcut menu.

The boundary's dialog box appears, in which you can edit its properties.

- Click **HFSS**>**List**.

The **Design List** dialog box appears, in which you can modify the properties of one or more boundaries.

Deleting Boundaries

To delete *one boundary*:

1. Select the boundary you want to delete by selecting its icon in the project tree.
2. On the **Edit** menu, click **Delete** .

To delete *all boundaries*:

- Click **HFSS**>**Boundaries**>**Delete All**.

You can also *delete one or more boundaries* in the **Design List** dialog box:

1. Click **HFSS** menu, click **List**.

   The **Design List** dialog box appears.

2. Under the **Boundaries** tab, click the row of the boundary you want to delete.
3. Click **Delete**.

Reassigning Boundaries

You can reassign a boundary to another surface. This is useful when you have modified objects with assigned boundaries, invalidating the boundaries. For example, if you unite two objects with assigned boundaries, the second object's boundary will become invalid because united objects maintain the characteristics of the first object selected. In this case, you would need to reassign the boundary or delete it. You can use a **Reassign Boundary** dialog box accessed through the [*solver*] > **Boundary** > **Reassign** menu, or right-click the Boundary of interest in the **Project** tree, and select from the shortcut menu to **Reassign**, **Add to Assignment**, or **Remove from Assignment**. You can also determine current assignments by right-clicking on the boundary on the Project tree and using the **Select Assignment** command on the short cut menu.

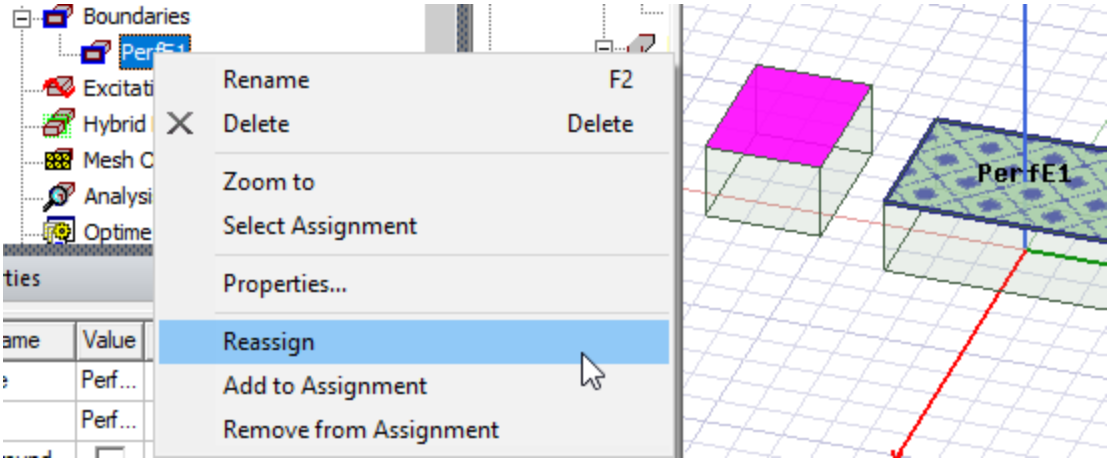
Prerequisites:

- You must first select the surface or object that you want to reassign, or add to an assignment, or remove from an assignment.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To use the short-cut menu to reassign, add assignment, or remove from assignment:

1. With the target surface selected, right-click the Boundary of interest to invoke the menu.



2. Select from the menu to make the change you want.

The boundary assignments change.

> **Note:**
>
> When reassigning a boundary that includes vectors in its definition, Ansys Electronics Desktop attempts to preserve the vectors with the new assignment, but this is not always possible.

Reprioritizing Boundaries

Each boundary you assign overwrites any existing boundary which it overlaps. You can change the priority of a previously assigned boundary to be greater than a more recently assigned boundary.

The order of boundaries and excitations is important because, for any given triangle of the mesh, only one boundary or excitation can be visible to the solvers. The highest priority boundary condition applied on a certain surface overwrites the lower priority one(s) applied on the same surface. If any surfaces with boundary conditions have a partial overlap, then in the overlap region, the precedence rule applies. The highest priority boundary condition takes precedence over the lower one(s) such that for all mesh nodes on any surface carrying a boundary condition, only one boundary condition is actually enforced and visible to the solver.

Excitations cannot be reprioritized. Only boundaries can be reprioritized.

1. Click **HFSS** > **Boundaries** > **Reprioritize** to reprioritize boundaries.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Reprioritize Boundaries** window appears. The order the boundaries and excitations appear in the list indicates the order in which they were defined. The lowest priority assignment appears at the top of the list.

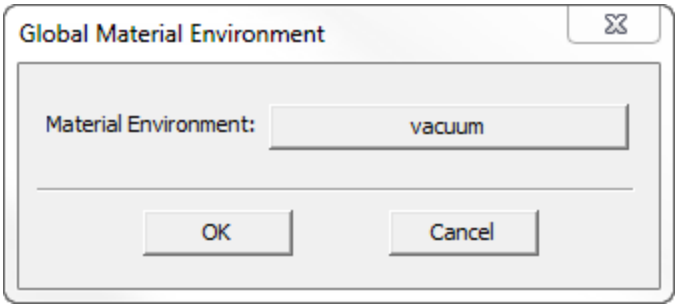
Ports are automatically placed at the bottom (highest priority) of the list; you cannot move a boundary to a higher priority than a port. Magnetic Bias Excitations (if any) have the lowest priority. Other boundaries and excitations appear between these two extremes.

2. Drag the boundary you want to change to the desired order of priority.

> **Note:**
>
> The order of boundaries and excitations in the project tree is alphabetical. The order does not correspond to the order of boundaries and excitations visible to the solvers.

Editing The Global Material Environment

For HFSS, the **Boundaries>Edit Global Material Environment** command displays the **Global Material Environment** dialog box. You can access the command via both the **Tools** menu and via right-click **Boundaries** in the Project window for the shortcut menu.



By clicking the **Material** button, you can access the Select Definition dialog box. This lets you work with the materials library.

This setting tells HFSS what material properties to use when calculating far fields. The default setting is vacuum. If you simulate an antenna underwater, for example, you should set the **GlobalMaterial Environment** to water.

Selecting anisotropic material is disabled because the solver doesn't support that.

See additional discussion of the effect this setting has on calculations in Global Material Environment.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Duplicating Boundaries and Excitations with Geometry

To duplicate a boundary or excitation when its geometry is pasted or duplicated:

1. Click **Tools**>**Options>GeneralOptions** to open the **Options** dialog.
2. Under **HFSS>Boundaries**, select **Duplicate boundaries with geometry**.

All boundaries and excitations will be duplicated with their associated geometries until you choose to clear this option.

> **Tip:**
>
> Use this option to copy and paste boundaries. For example:
>
> 1. Select the face to which you want to assign the boundary.
> 2. Click **Modeler**>**Surface**>**Create Object From Face**.
> 3. Assign the boundary to the new face object.
> 4. Copy and paste the new face object to copy and paste the boundary.

Showing and Hiding Boundaries and Excitations

You can choose to show or hide a boundary or excitation's geometry, name, or vectors, in the active view window or in all view windows.

What do you want to do?

Show or hide a boundary or excitation in the active view window.

Show or hide a boundary or excitation in every view window.

Showing and Hiding Boundaries and Excitations in the Active View Window

1. On the **View** menu, click **Visibility>** **Active View Visibility**.

   The *Active View Visibility* dialog box appears.

2. Select the tab for the objects you want to show or hide. The dialog box contains tabs for 3D Modeler objects, Color Key objects, Boundaries, Excitations, and Fields Reporter objects.
3. Under the tab you need, select the **Visibility** option for the objects you want to show in the active view window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click the **Boundaries** tab if you want to show or hide boundaries.

   > **Note:**
   >
   > Hiding boundaries also turns off a check for boundary overlaps during boundary assignment. In the case of very large models with many boundaries, hiding boundaries can prevent delays during boundary assignment. Full model validation will subsequently check for boundary overlaps.

   Click the **Excitations** tab if you want to show or hide excitations.

   - For designs with large numbers of objects, you can resize the dialog for easier selection.
   - By default, objects are listed in alphabetical order. You can invert the order by clicking the Name bar above the Name fields. A triangle in the bar indicates the direction of the listing.
   - You can also use the **Name** field to type in an object name and apply the visibility via the **Show** and **Hide** buttons.

   The objects you select and designate as Visible (by selecting the property or using Show) appear.

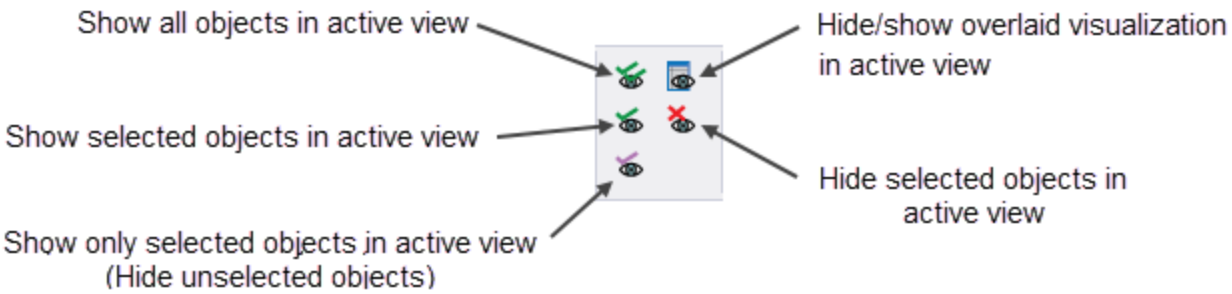
5. Clear the **Visibility** selection of boundaries or excitations that you want to hide from view.

   The boundary or excitation will only be visible in the active view window if it is selected.

6. Select the **Visibility** option for boundaries or excitations that you want to show in the active view window.

   The boundary or excitation will be visible in the active view window when it is selected or when it is not selected.

   You can also use the show/hide icons on the **View, Draw,** and **Model** ribbon tabs:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Showing and Hiding Boundaries and Excitations in Every View Window

1. Click **HFSS> Boundaries> Visualization** if you want to show or hide boundaries. Or:

   Click **HFSS> Excitations> Visualization** if you want to show or hide excitations.

2. Clear the **View Geometry**, **View Name**, or **View Vector** option for boundaries and excitations that you want to hide from view. Select the options you want to show.

   The options affect all view windows.

> **Note:**
>
> Hiding boundaries also turns off a check for boundary overlaps during boundary assignment. In the case of very large models with many boundaries, hiding boundaries can prevent delays during boundary assignment. Full model validation will subsequently check for boundary overlaps.

Reviewing Boundaries and Excitations in the Solver View

After you have assigned all the necessary boundaries and excitations to a model, you should review their order of priority according to the Ansys Electronics Desktop solver. Reviewing the solver's view of the model's boundaries and excitations enables you to verify that their order during the solution process will be as you intended.

To check the solver's view of boundaries and excitations:

1. On the **HFSS menu, click Boundary Display (Solver View)**.

   Ansys Electronics Desktop generates an initial mesh and determines the locations of the boundaries and excitations on the model.

   The **Solver View of Boundaries** window appears, which lists all the boundaries and excitations for the active model in the order specified in the **Reprioritize Boundaries and Excitations** dialog box.

2. Select the **Visibility** option for the boundary or excitation you want to review.

   The selected boundary or excitation will appear in the **3D Modeler** window in the color it has been assigned.

- **Visible to Solver** will appear in the **Solver Visibility** column for each boundary or excitation that is valid.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Overridden** will appear in the **Solver Visibility** column for each boundary or excitation that will be ignored by the solver as a result of it overlapping an existing boundary or excitation with a higher priority.

3. Verify that the boundaries or excitations you assigned to the model are being displayed as you intended for solving purposes.
4. If the order of priority is not as you intended, reprioritize the boundaries and excitations.

Setting Default Values for Boundaries and Excitations

When assigning a boundary or excitation, many of the fields in the boundary and excitation dialog boxes have default values associated with them. These default values are initially set by Ansys Electronics Desktop, but can be overridden.

To modify the default values associated with a specific boundary or excitation type:

1. Assign a boundary or excitation.
2. Modify any default values.
3. Close the boundary or excitation's dialog box.
4. Re-open the new boundary or excitation's dialog box. It now includes a **Defaults** tab.
5. Under the **Defaults** tab, click **Save Defaults**.

   The values assigned to this boundary are saved as the default values and will be assigned when new boundaries of this type are created.

6. Optionally, click **Revert to Standard Defaults**.

   The default values you set for this boundary type will be cleared and will revert to the default values set by Ansys Electronics Desktop.

> **Note:**
>
> For PML boundaries, the defaults are set via a formula, rather than a value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

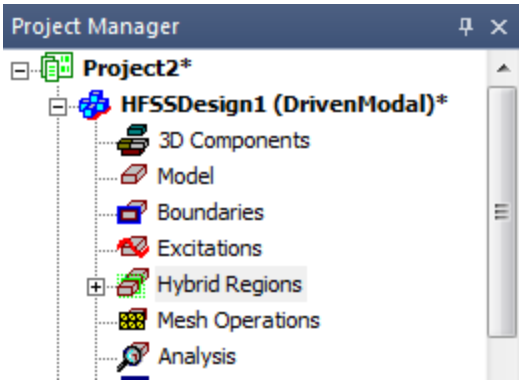
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

18 - Assigning Hybrid Regions in HFSS

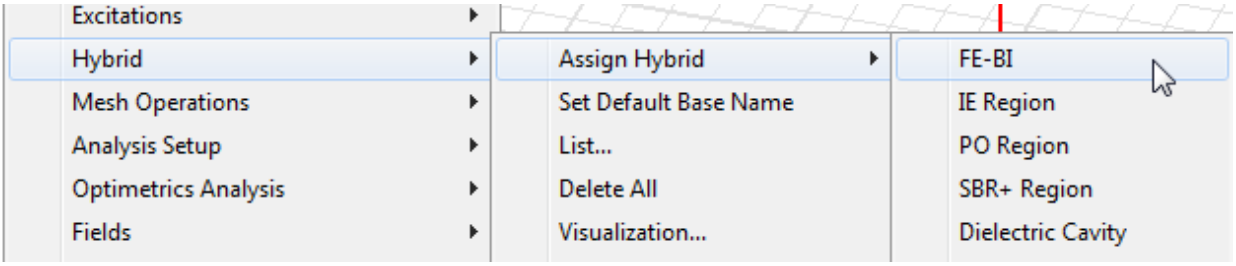
To use Hybrid Regions, the solution type must be HFSS with Hybrid and Arrays. Hybrid regions can include:

- **FE-BI** – only Radiating and Incident Field types are supported.
- **IE Region** – using the Integral Equation solver.
- **PO Region** – using a Physical Optics solver, which can provide a fast approximation for a class of problems.
- **SBR+ Region** – using shooting and bouncing ray (SBR) technology to compute scattered field contributions of defined platform geometry to antenna coupling (S-parameters) and far-field radiation results.
- **Dielectric Cavity**

The Project tree contains a folder for Hybrid regions.

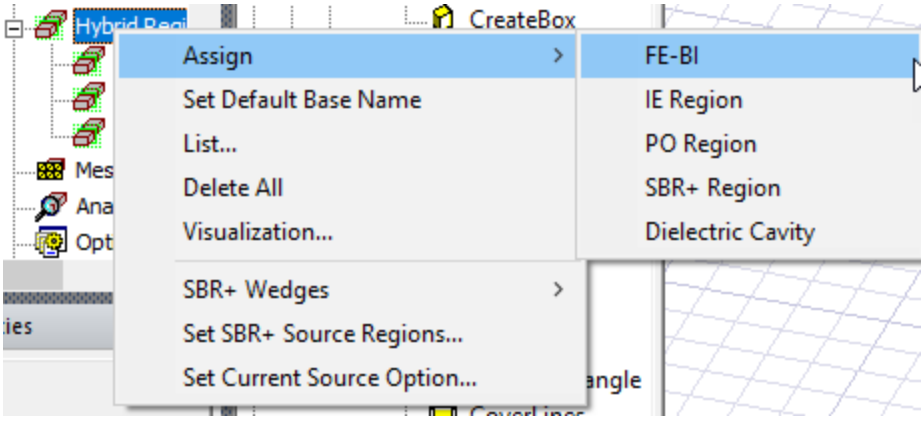


Assign Hybrid regions by selecting an object in the 3D Modeler window, or an object in the History tree, right-click for the shortcut menu, click **Assign Hybrid** and selecting from the enabled options. You can use **HFSS > Hybrid > Assign Hybrid** for the options to be enabled. The selection must meet the conditions for that assignment.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also right-click the **Hybrid Regions** icon in the Project tree to access a shortcut menu there:



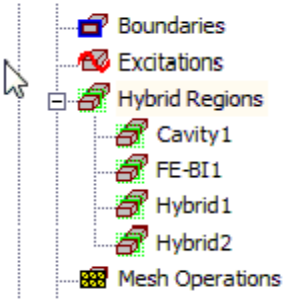
The Toolbar menu and Project tree right-click menus for Hybrid regions display commands for managing Hybrid regions.

- **Set Default Base Name**: opens a dialog for setting default names for new boundaries, excitations, and hybrid regions.
- **List ...**: opens the Design list dialog to the **Hybrid Regions** tab.
- **Delete All**: deletes all hybrid regions.
- **Visualization...**: opens a Visualization Options dialog that lets you show or hide regions and other design objects.
- **SBR+ Wedges**: for designs with SBR+ regions assigned, this command includes further menu for Settings, for Wedge visibility, and for Set Current Source option (see below).
- **Set Coupling**: for designs without SBR+ regions assigned, this command appears and opens a menu that lets you specify whether regions are all one-way coupled or two-way coupled. In one-way coupling, one design acts as as the field source, and the other as a target. This feature improves the efficiency of the solver. If an SBR+ region is assigned, the design becomes one-way coupled and the **Set SBR+ Source Regions** command replaces **Set Coupling** on the short-cut menu. The **Advanced** command on the **Set Coupling** menu opens a dialog that lets you group region objects, other than SBR+ Region objects. Grouped objects become two-way coupled when the design is otherwise one-way linked. See Set Coupling for Hybrid Regions for details.
- **Set SBR+ Source Regions**: for designs with SBR+ assigned, the **Hybrid Regions** right-click menu provides this command for you to group source regions for SBR+ regions in the design. If a source region contains multiple FEBI or IE regions, you can decide if the regions are 1-way or 2-way coupled. All source regions interact as one-way links to SBR+ and coupling between the source regions are handled by SBR+ using reciprocity theory. The dialog excludes all SBR+ regions as those are targets of the sources defined in the interface.

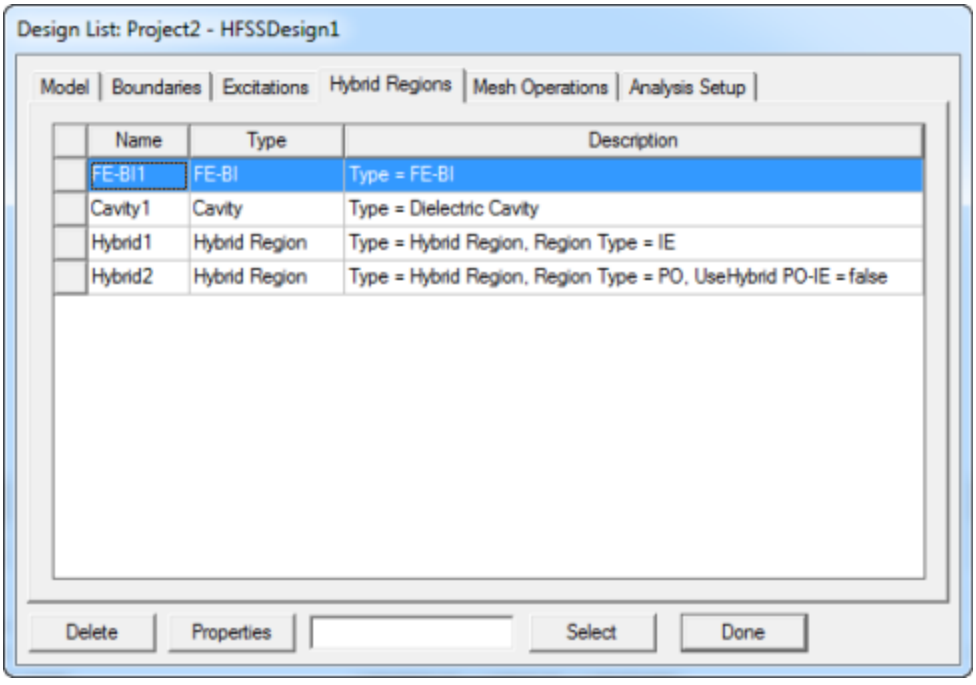
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Set Current Source Options...**: this allows you to specify customized power fraction for designs with SBR+ Regions assigned. Use Global is the default. Enabling Thin Sources lets you specify a Power fraction should be greater than 0.1 and less than 1. Solutions are invalidated if you change current source conformance, use Thin Sources, Power Fraction, or change the Global option.

To view existing Hybrid Regions assignments, you can expand the list under the Hybrid Regions icon in the Project tree. Selecting an assigned region in the list also highlights that region in the Modeler window.



You can also select the **Hybrid Regions** icon in the Project Tree and right-click for menu and select **List.** This shows the **Design List** dialog with the **Hybrid Regions** tab selected.



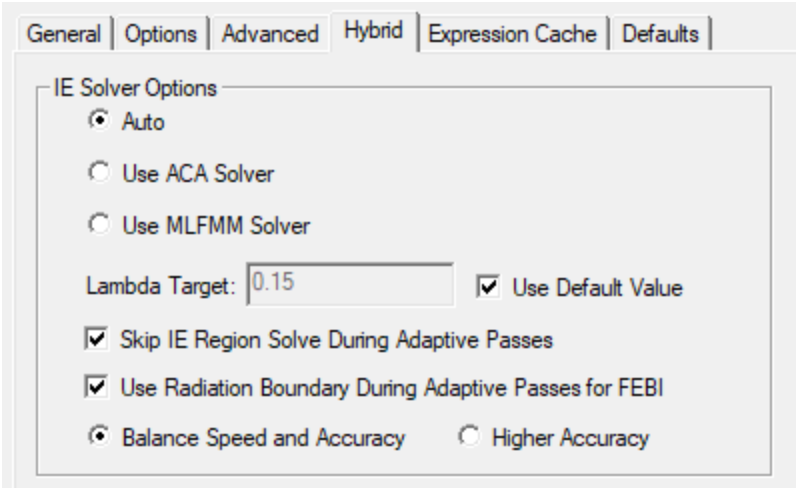
Click Delete to remove the assignment from the selected object(s).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning FE-BI Hybrid Regions

FE-BI- this uses integral equation formulation, which is an exact transparent condition. By taking advantage of conformal radiation volumes to reduce the overall finite element solution domain, using the FE-BI hybrid region will result in more efficient simulation for electrically large open boundary problems. It can be close to or on the structures, but for performance, 0.05 wavelength is recommended. If it is on a surface, you must turn off curvilinear elements. The FE-BI hybrid region should enclose the entire structure by itself, or with an Infinite ground plane. (Compare Assigning IE Regions). If you model a radiation boundary as a FE-BI hybrid region, the solution setup cannot include calculations of derivatives on those regions. Contact with dielectric IE Region or PO Region is allowed.

In the Solution Setup, when the **Hybrid** tab option, **Use Radiation Boundary During Adaptive Passes for FEBI** is enabled, solver uses a radiation boundary instead of FEBI during adaptive passes. When it converges, solver enables the FEBI for the last adaptive pass. If the FEBI surface is sufficiently large, this option can lead to a faster adaptive mesh run.



To assign a FE-BI Hybrid Region:

1. Select the object you want to assign as FE-BE, and use one of the following menus:

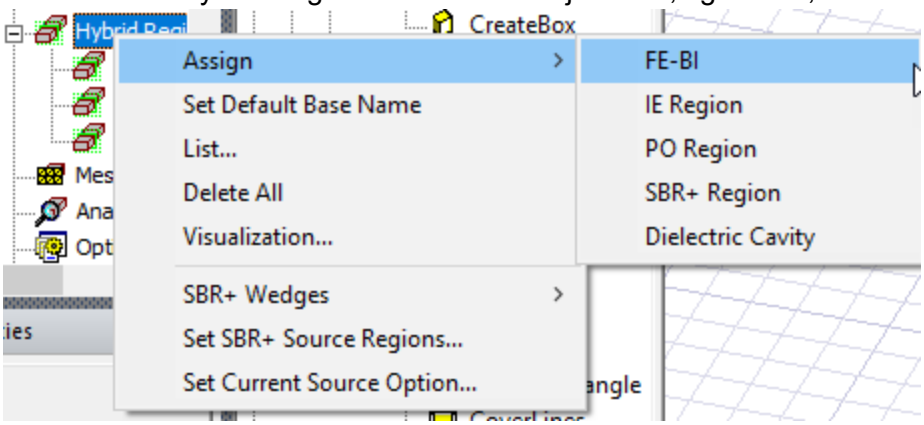
   HFSS>Assign Hybrid>Assign>FE-BI

   Right-click in the Modeler window and select **Assign Hybrid>FE-BI**.

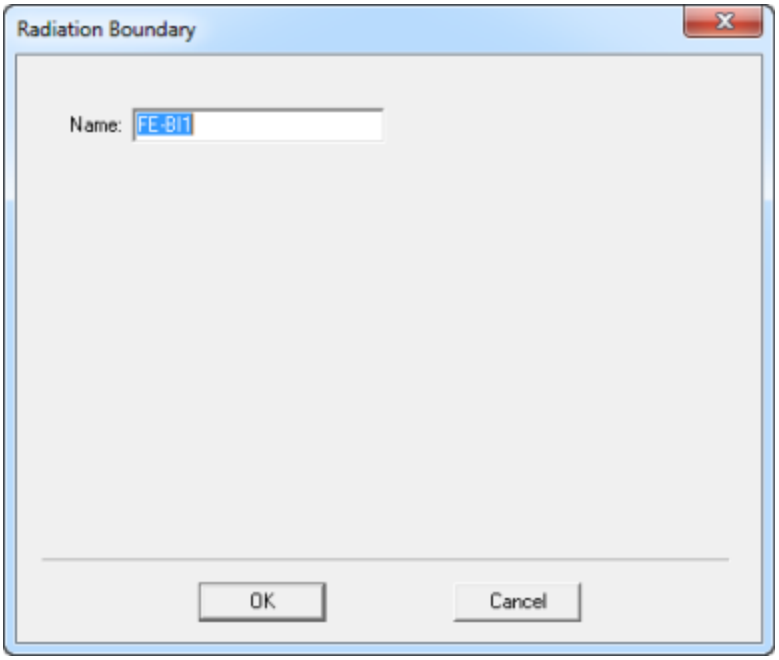
   Right-click in the Modeler window and select **Assign Hybrid>FE-BI**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select the Hybrid Regions icon in the Project tree, right-click, and click **Assign>FE-BI**.



The **Radiation Boundary** dialog opens, showing the default name and FE-BI assignment.



2. When you OK the dialog box, the assignment appears under the Hybrid Region icon in the Project tree.

Assigning IE Regions

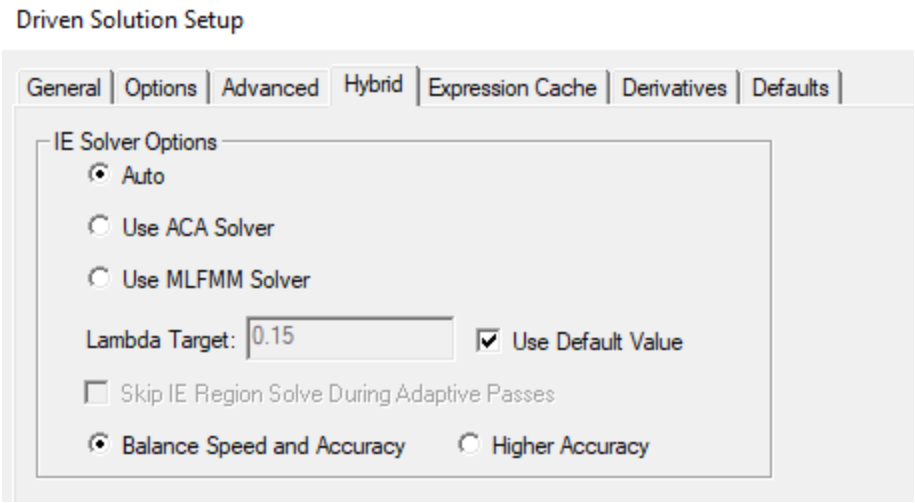
For driven modal and driven terminal solutions you can assign dielectric or conducting objects or sheets as IE Regions to be solved with the IE Solver as part of a hybrid simulation approach, using the advantages of the FEM and Hybrid Region solvers. IE Regions use an integral equation solver (sometimes called method of moments = MoM) and solves for the currents on *surfaces of objects.* It creates a triangular surface mesh on all objects - it solves for the currents

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or equivalent currents on conducting and dielectric objects. In general, HFSS uses the finite element method (FEM) to solve for the electromagnetic fields in the solution region. It meshes over the *entire solution volume* (other than areas assigned as one of the Hybrid Regions) and solves for the electric field throughout that volume.

To use Hybrid IE Regions, the solution type must be HFSS with Hybrid and Arrays.

If you assign Hybrid regions, you can use the Hybrid tab in the Advanced **Solution** setup to specify the IE Solver type as ACA (the traditional method) or as MLFMM, which is superior than ACA for models with large FE-BI surfaces, and also works for IE regions. The default choice is Auto, in which the choice is made based on the characteristics of the design. See Guidelines for Choosing Between ACA and MFFMM IE Solver Options.



To Assign an IE Region

To use Hybrid IE Regions, the solution type must be HFSS with Hybrid and Arrays.

You must select an appropriate object or face in order to enable the menu.

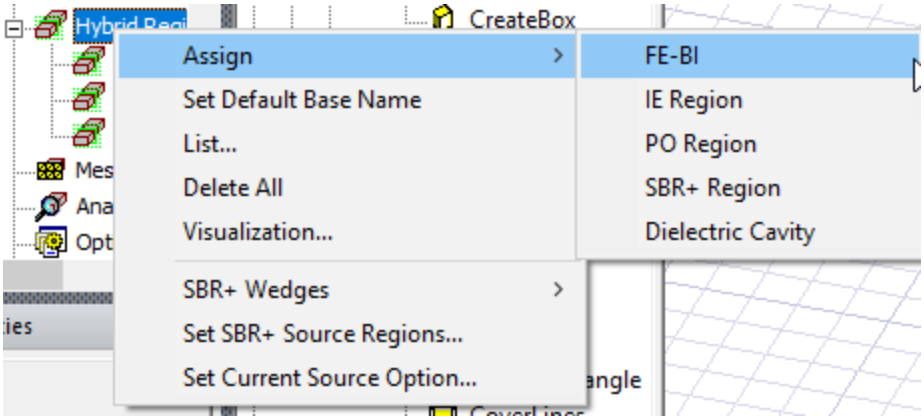
1. Select the object you want to assign as an IE Region, and use one of the following menus:

   HFSS>Hybrid>Assign Hybrid>IE Region

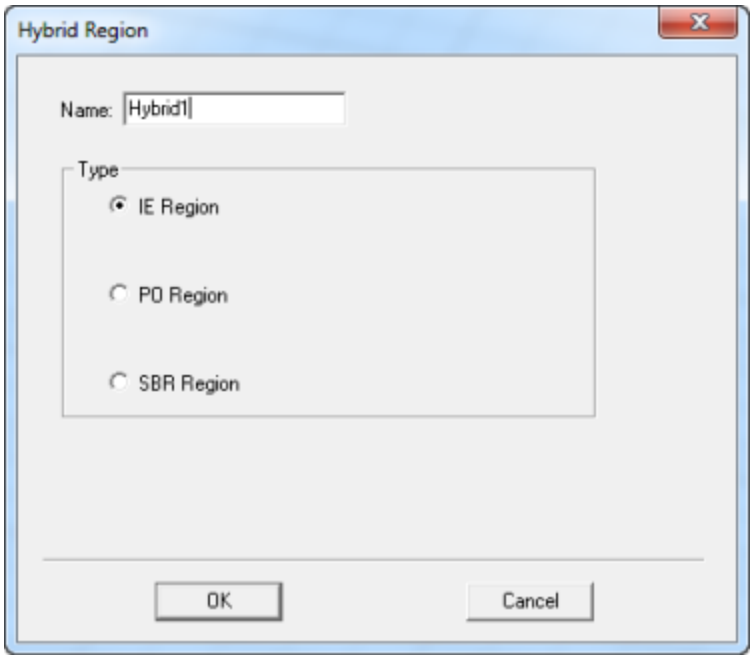
   Right-click in the Modeler window and select **Assign Hybrid>IE Region**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select the Hybrid Regions icon in the Project tree, right-click, and click **Assign>IE Region**.

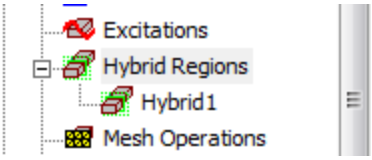


The dialog for **Hybrid Region** opens, showing the default name and type selection as IE Region. Because you may decide to change an assignment for IE Regions, and PO Regions, the dialog shows these types as radio button selections.



2. When you OK the dialog box, the assignment appears in the **Project** tree under **Hybrid Regions**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Applications and Technical Considerations for Using IE Regions

Application areas for IE Regions include:

- Radar cross section (RCS)
- Antenna placement (for example, antenna on a vehicle)
- Stand alone antennas
- Coupling. EMI/EMC
- When a layered impedance boundary touches a dielectric to model paint on fascia which is important for ADAS radar simulations

In assigning IE Regions:

- 3D objects must be dielectric or conducting based on bulk material conductivity. Some boundary conditions can also be assigned on the surfaces of 3D objects. These boundaries are: PEC, Finite Conductivity, Impedance, Layered Impedance, Lumped RLC, and Anisotropic Impedance.
- Sheet objects can also be allowed only if they have supported boundaries assigned. These boundaries are: PEC, Finite Conductivity, Impedance, Layered Impedance, Lumped RLC, or Anisotropic Impedance boundaries. They cannot be dielectric.
- An infinite ground is allowed with dielectric IE region.
- Dielectric Cavity must either:
  - be contained within a FEM solve inside object.
  - have its surface covered by FEM solve inside objects.
- A free standing dielectric IE/PO region must not:
  - touch other free standing IE/PO regions.
  - have a conducting boundary assignment on free standing dielectric IE/PO region
  - have an IE/PO region to be assigned to both dielectric objects and metallic objects. One IE/PO region can only be metallic or dielectric
- Metallic IE Region must either
  - (Interior) Be contained within a Dielectric Cavity.
  - (Exterior) Be outside entire FEM region.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Faces of two dielectric IE Regions can touch. We allow metal IE region to touch (or to be contained in) a DIE. We also allow DIE regions to touch a DIE. The IE region can be a solid conductor, or a conducting/impedance sheet. You can also assign a boundary condition on the surfaces of a dielectric IE region.
-
  - Allow IE regions (solid metal or metal/impedance sheet) to touch dielectric IE region (DIE).
  - Allow boundary assignment on the face of dielectric IE region.
  - Allow metallic IE (both solid and sheet objects) contained inside dielectric IE.
  - Allow metallic sheet placed at the interface of two dielectric IE regions.
  - Allow mixed IE region with infinite ground.
- Two dielectric IE components can touch.
- Dielectric Cavity may contain implicitly subtracted FEM objects (and imply solver handling like FEBI on their interface)
- Metallic IE Region may touch FEBI surface.
- Models with IE Regions for curved surfaces (for example, a large antenna dish) benefit from improvements to the Tau mesher.

Note:

For designs like antennas mounted on a platform such as an aircraft or battleship, it is beneficial both in terms of memory and solution time to model antennas using the finite element method, while modeling remaining metallic structures as metallic IE regions. In these cases, a metallic IE region is actually in contact with a FEBI hybrid region where appropriate boundary conditions are enforced at the interface between the two solvers.

In cases where an IE region is in contact with a FEBI hybrid region, it is recommended that the FEBI hybrid region be placed at least a third wavelength away so that the FEM domain can have sufficient space to perform its adaptive mesh refinement more accurately.

The near field calculation can be inaccurate inside a cavity made from any metal boundary applied to a box made of background material. For PEC boundary or other metal boundaries at high enough frequencies, the total fields inside this cavity should be zero but that might not be the case due to solver formulation employed on surfaces with metal boundaries. However, the exterior to the cavity will be accurate.

There are two cases we do not handle:

- Mixed IE region with high loss impedance assigned to the surface of a closed IE region;
- Mixed IE region with a sheet (IE region) larger than the touching surface of a dielectric IE Region.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Solver issues an error for these cases.

The accuracy of this approach is the same as enclosing the structure inside a FEBI box. Performance is faster than the FEBI apporach.

IE Region Ports

You can assign ports on an IE Region. This is helpful, for example, when an antenna is attached to a large metal body. One-way coupling between regions allows for an easier setup in HFSS design, where you can switch between one way link and full solution in a single setup. The following cases are allowed:

- IE region ports in the presence of one or more non-port FEBI, IE, PO and SBR+ regions.
- One-way coupled for the above case.
- Can allow an IE component with port but skips port refinement for an IE component with port.

The following restrictions will still apply:

- IE region port cannot mix FEM ports with IE ports.
- IE region port is not allowed to contact both PO region and SBR+ region.
- IE region port cannot coexist with incident waves and FEM ports
- IE region port does not support composite excitation.

Assigning PO Region Hybrid Region

While IE Region provides an efficient and accurate alternative to FEM solver for metallic and finite conducting structures, for very electrically large structures it is time consuming and memory demanding. The physical Optics (PO) solver provides first-order scattering information and proves to be efficient approximation for metallic, smooth, locally flat structures. For such geometries for example, a dish reflector, physical optics (PO) can provide an accurate approximation with a significantly reduced computational resource requirement. PO Hybrid Regions are dielectric objects and perfectly conducting objects or sheets solved with physical optics formulation. In light of this, PO Region provides a fast alternative to IE region.

To use PO Region, the solution type must be HFSS with Hybrid and Arrays.

However, because PO is not a full wave solver, we stress that inappropriate use of PO Region may lead to an incorrect solution. PO Region is optimal when the following conditions are satisfied:

1. Structure is flat or locally flat, such as a dish reflector;
2. Frequency is sufficiently large such that the structure is more than 10 wavelengths in dimension and the structure is sufficiently far away (more than 10 wavelengths) from the radiating source.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Only a single incident wave excites the PO Region.
4. A free standing dielectric IE/PO region cannot touch other free standing IE/PO regions.

When these conditions are met, a large portion of the structure is visible to the radiating source. This region is called a lit region. The remaining region is termed a shadow region. Figure 1 shows visualization of such a concept.

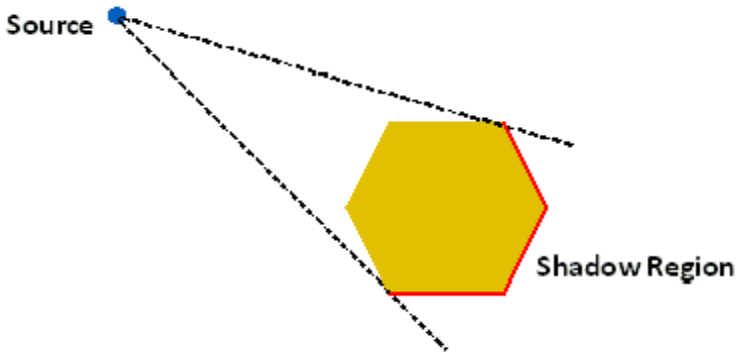


Figure 1 Shadow region of PO solver

In PO approximation, the currents on shadow regions are identically zero. This approximation is acceptable only when frequency is large. In the situation where condition 1) is not met but 2) is satisfied, PO region could still provide reasonable results.

While the accuracy of PO depends on an accurate identification of shadow region, this region is determined based on source location as can be seen from Figure 1. Therefore for the situation where there are multiple sources, shadow regions due to each source could be vastly different. In this situation PO region solver does not provide reliable results. Consequently PO region will only allow a single incident wave direction. Furthermore if there are multiple disjoint FEBI boundaries, the solver will report error message when it detects that a shadow region due to each FEBI box does not mostly overlap. Contact between dielectric PO region and a FE-BI region is allowed.

Note that PO regions may be used together with other useful features like 3D components. For parametric sweeps that vary the distance between assembled 3D components, you can preserve and reuse the component-level meshes between these variations. For example, if a horn and a reflector are defined as 3D components with enabled Do Mesh Assembly option, you can do a placement study running the parametric sweep over the distance between the horn and the antenna, recycling the mesh to speed up the parametric sweep simulations.

The following guidelines summarize the use of PO region.

- Use PO Region when you want a faster solution with sacrifice of accuracy.
- Use PO Region for smooth flat and locally flat structures at the operating frequency.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note several limitations of the PO region option. First, every conducting material of the metallic PO region will be treated as perfect conductor, and every surface will be treated as PerfE boundary condition disregarding actually assigned materials and boundary conditions. The warning message will be issued if the PO region is assigned on a metal object or a boundary condition that is not a perfect electric conductor. Assigning the PO region to a dielectric object automatically turns this hybrid region to a dielectric PO region. Second, if there are multiple sources modeled with multiple disjoint FE-BI boundaries that makes it hard to accurately define the shadow region, the PO region solver does not provide reliable results. Furthermore, the solver will report an error message when it detects that a shadow region due to each FE-BI region does not mostly overlap.

To Assign a PO Region:

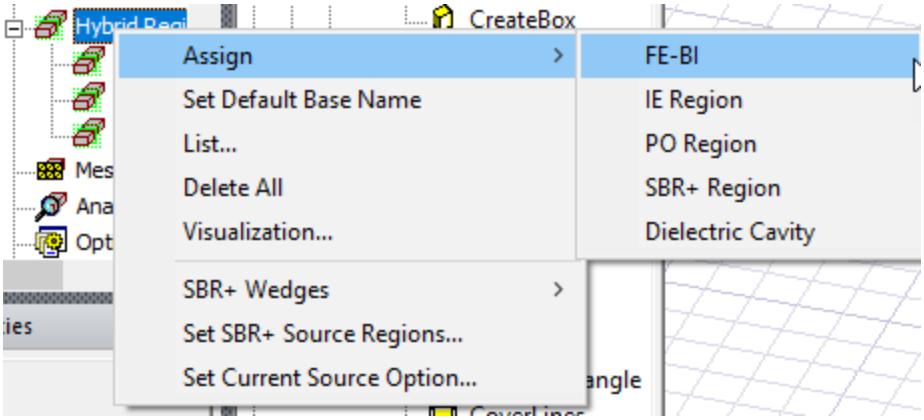
To use PO Region, the solution type must be HFSS with Hybrid and Arrays.

1. Select the object you want to assign as an PO Region, and use one of the following menus:

   HFSS>Hybrid>Assign Hybrid>PO Region

   Right-click in the Modeler window and select **Assign Hybrid>PO Region**.

   Select the Hybrid Regions icon in the Project tree, right-click, and click **Assign>PO Region**.



   The dialog for **Hybrid Region** opens, showing the default name and type selection as PO Region. Because you may decide to change an assignment for IE Regions, and PO Regions,

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the dialog shows these types as radio button selections.

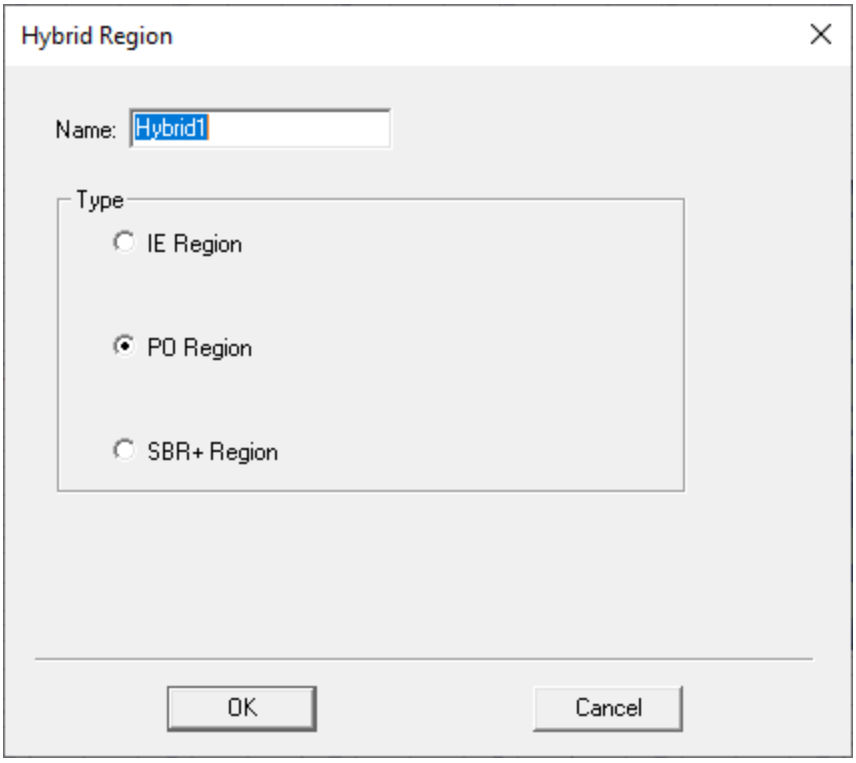


Fig. 2 Hybrid Region dialog for PO region

2. When you OK the dialog box, the assignment appears under the Hybrid Regions icon in the Project tree.

Examples of PO Hybrid Simulations

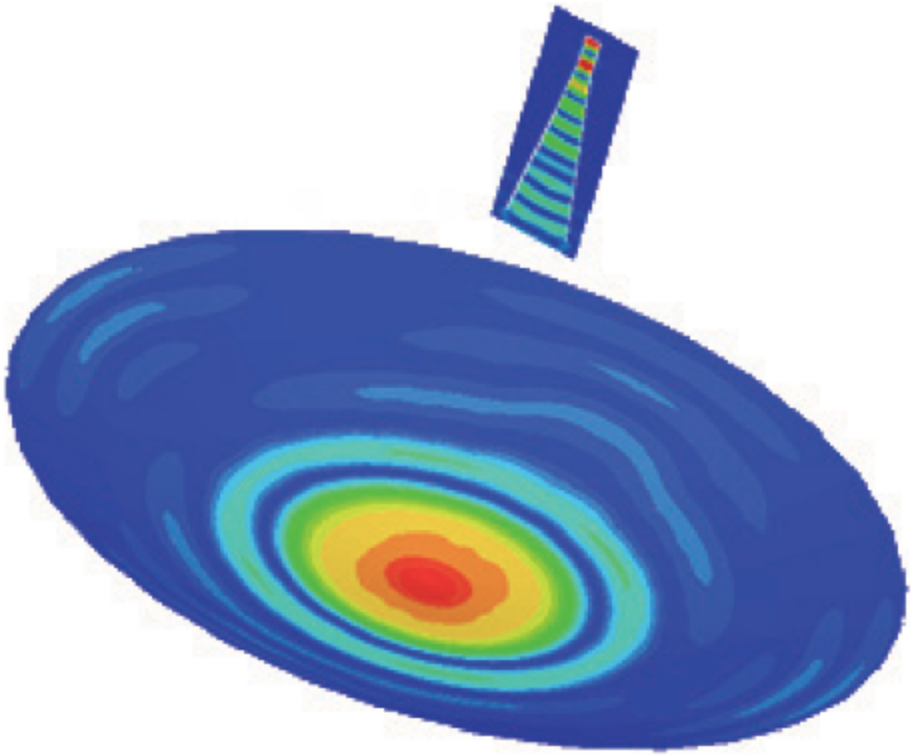
This section looks at three different projects that use PO Hybrid Regions.

Horn Fed Reflector Antenna with PO

An electrically large reflector of 22 wavelengths in diameter is fed with a rectangular horn. To model the system using a hybrid solution with the reflector solved as a PO region, all geometry objects are placed in an HFSS design. The horn geometry is surrounded by an air box named RadiatingSurface. The faces of this object are assigned as hybrid FEBI region, the assignment corresponds to radiation FE-BI boundary in previous versions of Ansys HFSS. To assign, select the air volume surrounding FEM part of a design, and use menu item **HFSS > HYBRID> ASSIGN HYBRID > FE-BI**. The reflector is modeled as a 2D sheet object with PerfE boundary condition assigned. This sheet object is modeled as a hybrid metallic PO region. To assign, select the reflector and use menu item **HFSS > HYBRID > ASSIGN HYBRID > PO REGION**. All hybrid regions are listed under the hybrid regions entry in the project manager.

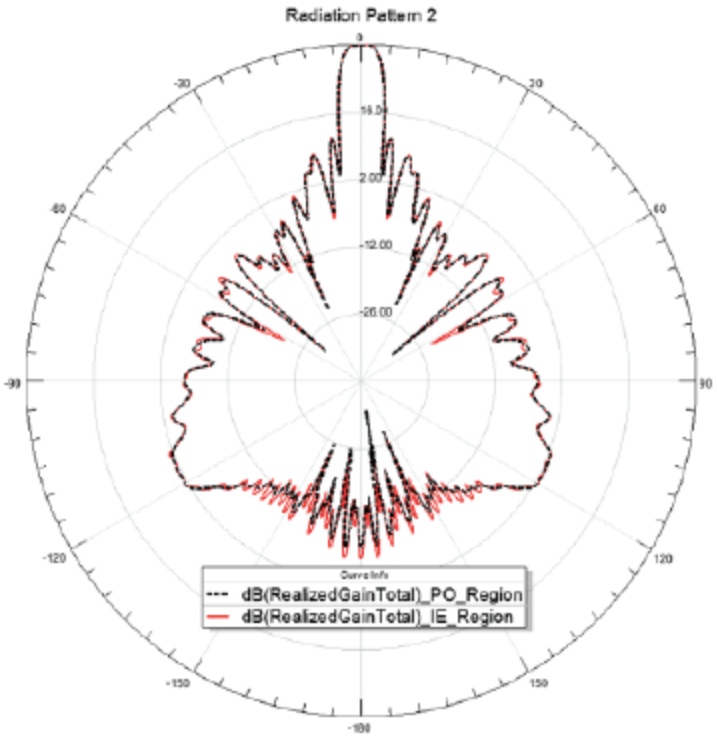
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After the solution has completed, standard result reports like field plots, S-parameters and far field characteristics are available. The following figure shows an E-field plot in the FE-BI region assigned to the horn and a J-field plot on the PO region assigned to the reflector dish.



The same geometry was solved with the reflector assigned as an IE region. We can see that the radiation patterns are almost identical and differ only for back lobes, while the PO region option significantly saves computational resources.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

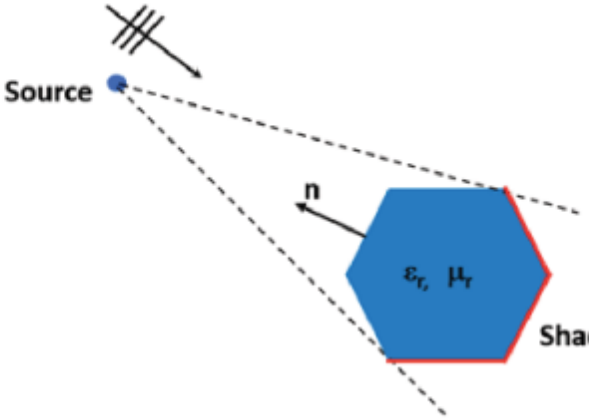


Reflector assigned as	RAM 1st pass	Time (1 core used) Mesh + 1st pass
IE Region	2.49GB	10 min 31sec
PO Region	1GB	1 min 27 sec

Antenna with Raydome using Dielectric PO Hybrid Region

The dielectric PO region option is well suited for simulating antennas covered with radomes. To model the system using the dielectric PO region, all geometry objects are placed in a single HFSS design. Horn geometry is surrounded by the air box that is assigned as a hybrid FE-BI region. The radome geometry should be a standalone 3D object assigned with dielectric material. This object is also assigned as a hybrid dielectric PO region.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



$$\mathbf{j} = (1-R)\mathbf{n}\times\mathbf{H}^{inc}$$
$$\mathbf{e} = (1+R)\mathbf{n}\times\mathbf{E}^{inc}\times\mathbf{n}$$

Lit Region

$$R = \frac{k_z^{\,2} - k_{zs_\mu}^{\,2} - (k_z^{\,2} - k_{zs_\mu}^{\,2})\,e^{-2jk_{zs}dd}}{(k_z + k_{zs_\mu})^2 - (k_z - k_{zs_\mu})^2\,e^{-2jk_{zs}dd}}$$

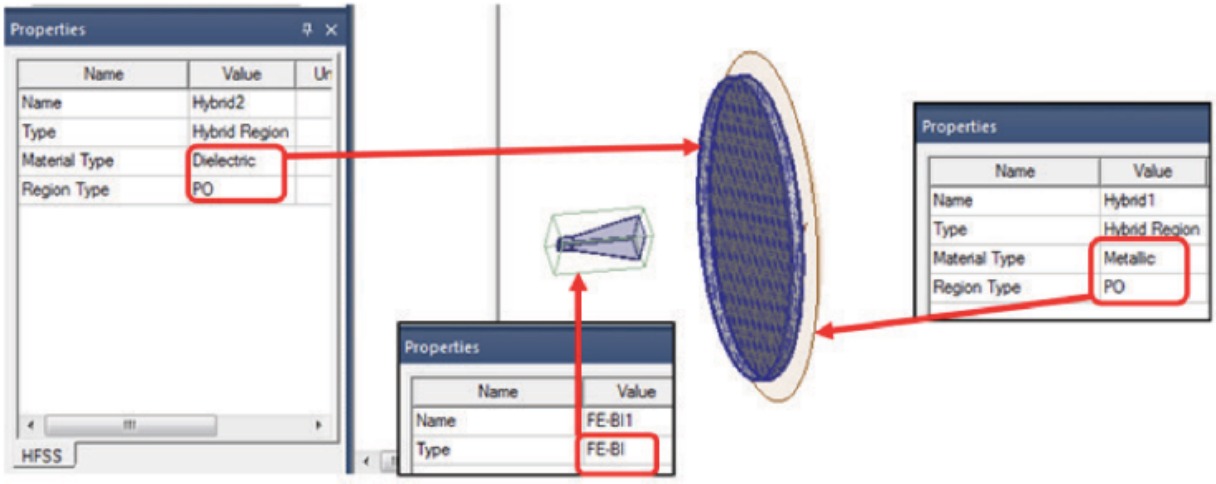
$$T = \frac{4(k_z k_{zs_\mu})e^{-jk_{zs}dd}}{(k_z + k_{zs_\mu})^2 - (k_z - k_{zs_\mu})^2\,e^{-2jk_{zs}dd}}$$

$$\mathbf{j} = -T\mathbf{n}\times\mathbf{H}^{inc}$$
$$\mathbf{e} = -T\mathbf{n}\times\mathbf{E}^{inc}\times\mathbf{n}$$

Shadowed Region

$$k_z = k_0,\quad k_{zs} = k_z\sqrt{\varepsilon_r\mu_r},\quad k_{zs_\mu} = k_{zs}/\mu_r$$

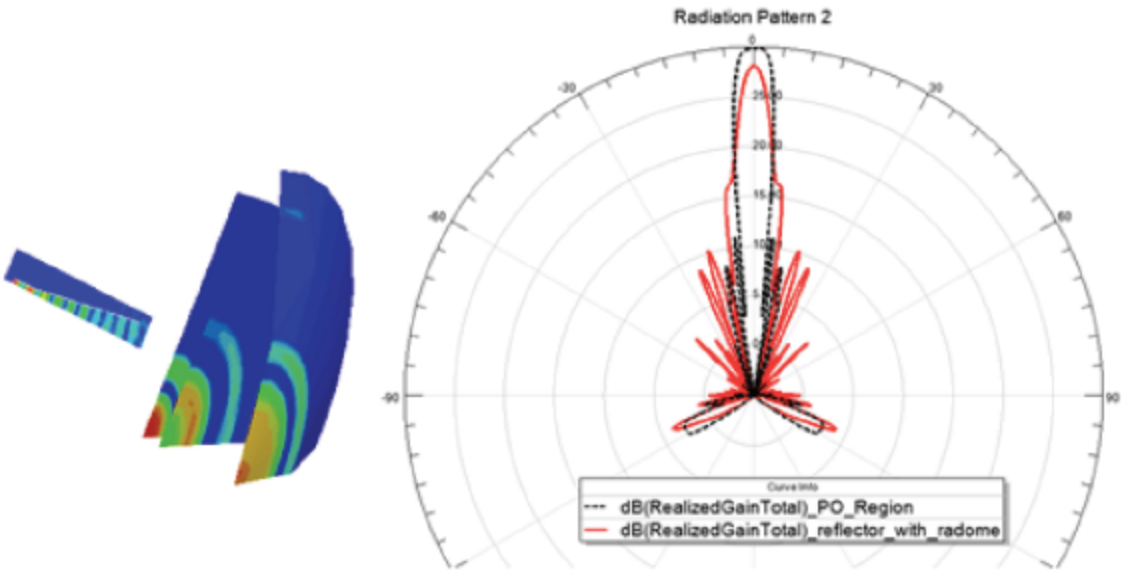
To assign, select the radome and use the menu item **HFSS > HYBRID > ASSIGN HYBRID > PO REGION**. The dielectric PO region should not touch any other model geometry. This option may be used in the same design together with other hybrid regions. Below is the example of reflector with the flat FR4 radome between the reflector and the horn. Three hybrid regions are assigned: FE-BI for the air box around horn, metallic PO for the PerfE reflector and Dielectric PO for FR4 radome.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note some limitations for a dielectric PO region: no boundary condition may be assigned on the surface of a dielectric PO region; dielectric PO region cannot touch any other metallic IE or PO regions. However, several touching dielectrics like layers of the radome may be assigned as one hybrid dielectric PO region.

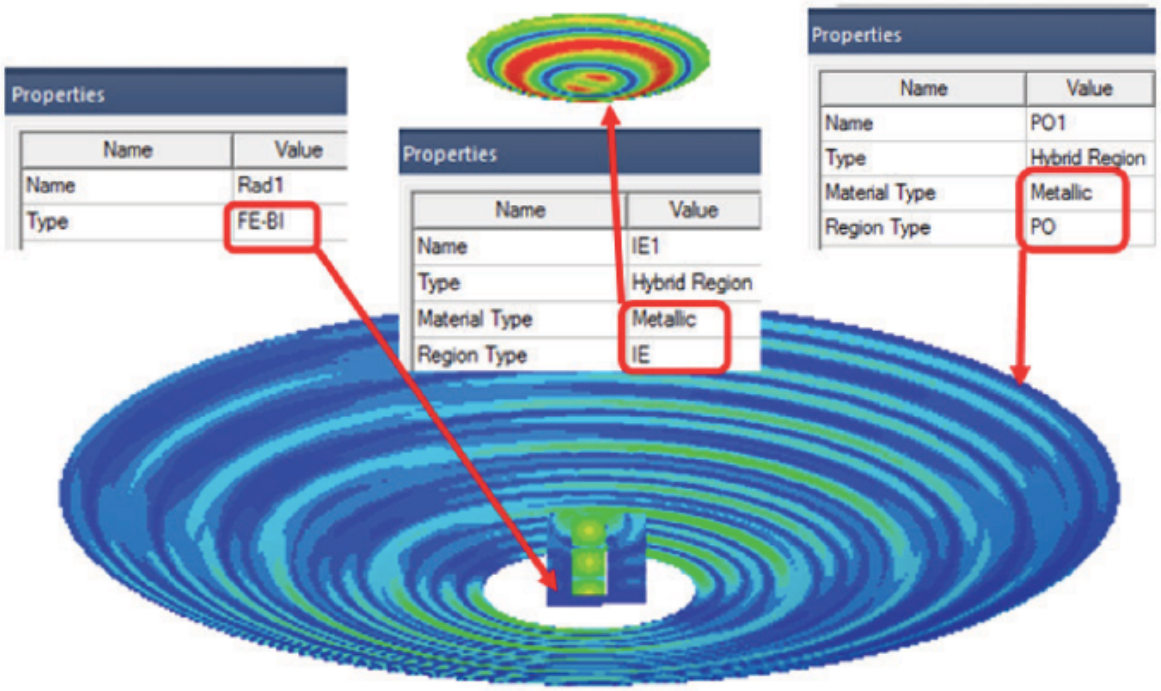
The following figure shows solved fields for a design with three hybrid regions, including, dielectric PO region for FR4 plate radome. Only a quarter of geometry is displayed for a field plot. In the Radiation Pattern plot, the black dotted line represents the far field of the reflector without radome, and the red line refers to the design with FR4 plate between the horn and the reflector.



Cassegrain Reflector: PO and IE Hybrid Regions in One Design

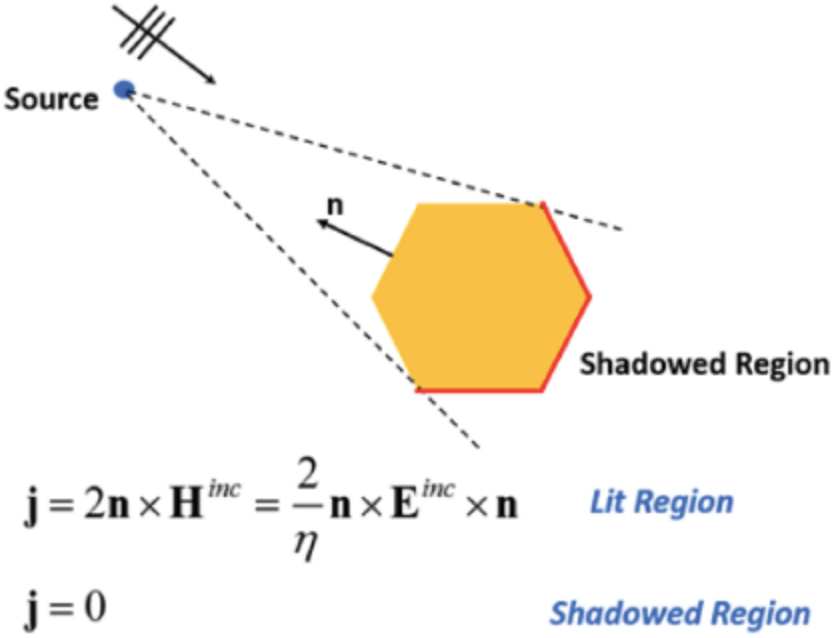
Hybrid PO regions may be combined with IE hybrid regions if desired. For applications like the Cassegrain reflector, there are 2 regions suitable for IE solver: a sub-reflector and a main reflector. A sub-reflector is usually not very large electrically but accuracy of the field solution on the sub-reflector is very important. A main reflector may be electrically very large and smooth making PO approximation justified for the main reflector. Below is the solved field for the Cassegrain system with a relatively small main reflector: The air box around the horn is assigned as a FE-BI region, the sub-reflector is assigned as a metallic IE region and the main reflector is assigned as a metallic PO region.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Technical Notes for Hybrid Regions and PO Solver

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Assigning SBR+ Hybrid Regions

HFSS can use shooting and bouncing ray (SBR) technology to compute the scattered field contribution of defined platform geometry via one-way coupling. This allows you to efficiently calculate installed near field patterns, far-field patterns and coupling (S-parameters) between antennas in the presence of an electrically large scattering geometry.

Note that the same SBR+ mesh used for both SBR+ simulation and VRT is also used in extracting wedges according to the user-specified criteria. This is done so that the wedge locations extracted are very near the physical edges of the SBR+ geometry model in the simulation, preserving accuracy of the line-of-sight visibility check for wedges.

- An SBR+ Region can only be assigned to no-solve inside object (metal).
- When an object is assigned as an SBR+ region, Boundary conditions (BCs) may be assigned to the geometry inside the SBR+ region(s), but there are some limitations in what boundary conditions are supported. HFSS currently supports assignment of Perfect E, Perfect H, Impedance, Finite Conductivity, and Layered Impedance boundaries to geometry in the SBR+ region. Other boundaries are not yet supported for SBR+, and produce an error. Infinite Ground Planes are not supported with SBR+. Note also that surface roughness must be set to “0” for all BCs assigned to SBR+ region geometry.
- An SBR+ Region cannot be contained by a dielectric cavity.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Incident plane waves are allowed. If you include an incident plane wave in the design, the Hybrid tab for the Driven Solution Setup dialog includes options for RCS type as Monostatic or Bistatic.
- If an SBR+ Region is assigned, the design becomes one way coupled. (See Assigning Hybrid Regions.) You can use the **Set SBR+ Source Regions** command to group source regions for SBR+ for two-way couplings between the grouped regions.
- Terminal ports with multiple terminals are supported if a an SBR+ region is present.
- Designs with SBR+ regions provide options to view total, scattered, or incident fields.
- SBR+ Region simulations can benefit from GPU Acceleration. See Setting HPC and Analysis Options.
- If your design contains an SBR+ Hybrid Region, you have an option on the Initial Mesh Settings dialog to **Allow tolerant meshing for SBR+ regions**, which can speed up the solve.
- If you want to consider far fields or near fields, the field sample points for SBR+ Hybrid regions must be defined as part of the simulation setup, rather than during post processing as is done for HFSS near and far fields. See the discussion of the **Setup** dialog box, **Hybrid** tab in step 3 below.
- HFSS with Hybrid and Arrays designs that include blockages have an enhanced work flow described.
- When a project contains SBR+ region and the following error appears: "[error] Solving adaptive frequency ... (Domain solver), process hf3d error: Error in using SBR+ region: Each source region should contain at least one excitation."

  "Each source region should contain at least one excitation" means there is at least one source region without an excitation. To work around the issue, please go to Set SBR+ Source Regions and, select all regions and click "**Group**".

You must select an appropriate object in order to enable the menu.

To assign an SBR+ Hybrid Region:

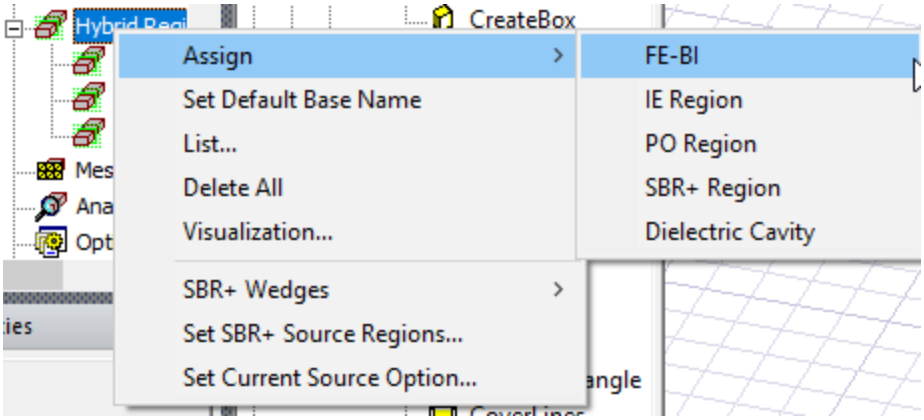
1. Select the object you want to assign as an SBR+ Region, and use one of the following menus:

   HFSS>Hybrid>Assign Hybrid>SBR+ Region

   Right-click in the Modeler window and select **Assign Hybrid>SBR+ Region**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

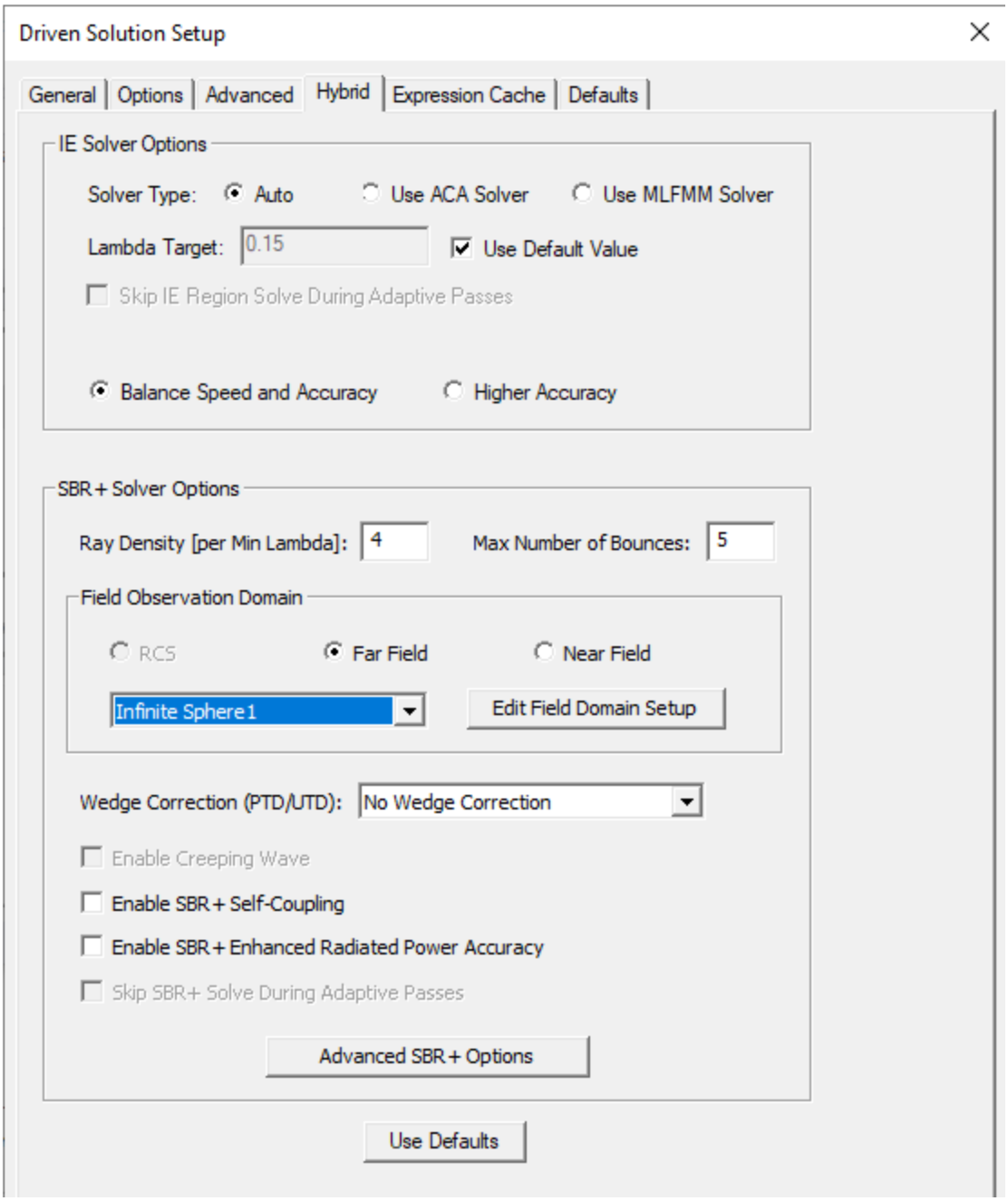
Select the Hybrid Regions icon in the Project tree, right-click, and click **Assign>SBR+ Region**.



The dialog for **Hybrid Region** opens, showing the default name and type selection as IE Region. Because you may decide to change an assignment for IE Regions, and PO Regions, the dialog shows these types as radio button selections.

2. When you **OK** the dialog box, the assignment appears in the Project Tree under Hybrid Regions.
3. SBR+ Solver Options are enabled in Solve setups for HFSS when an SBR+ Region is present.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

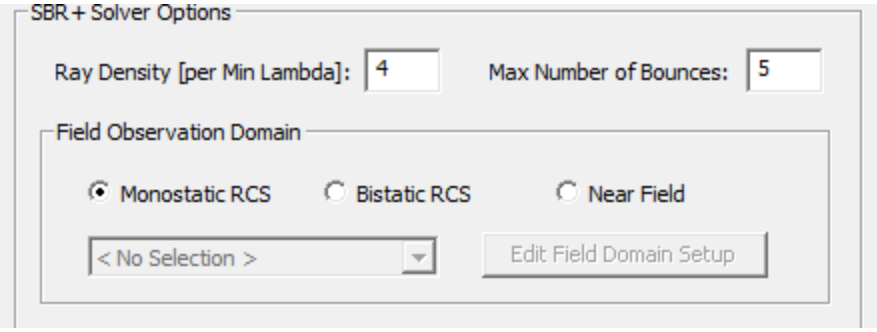


You can set Ray Density Per Wavelength and Maximum Number of Bounces, and if you have not done so, create or edit an infinite sphere setup for Far Field Observation. See Setting Hybrid Region Parameters for HFSS. You can also choose to Skip SBR+ Solve During Adaptive Passes. SBR+ regions are not being mesh adapted and SBR+ solutions have no impact on field solutions on FEM or IE regions. However, SBR+ does impact stopping criteria in some cases such as coupling between two separate source antennas. Therefore, to speed up mesh adaption, you can choose to not solve SBR+ regions until

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

source regions have converged in isolation. However, SBR+ does impact stopping criteria in some cases in the form of cache expression such as coupling between two separate source antennas or far field pattern. In such cases, SBR+ solve could not be skipped. Moreover, SBR+ solve is always launched when the maximum number of passes is reached regardless of source region convergence.

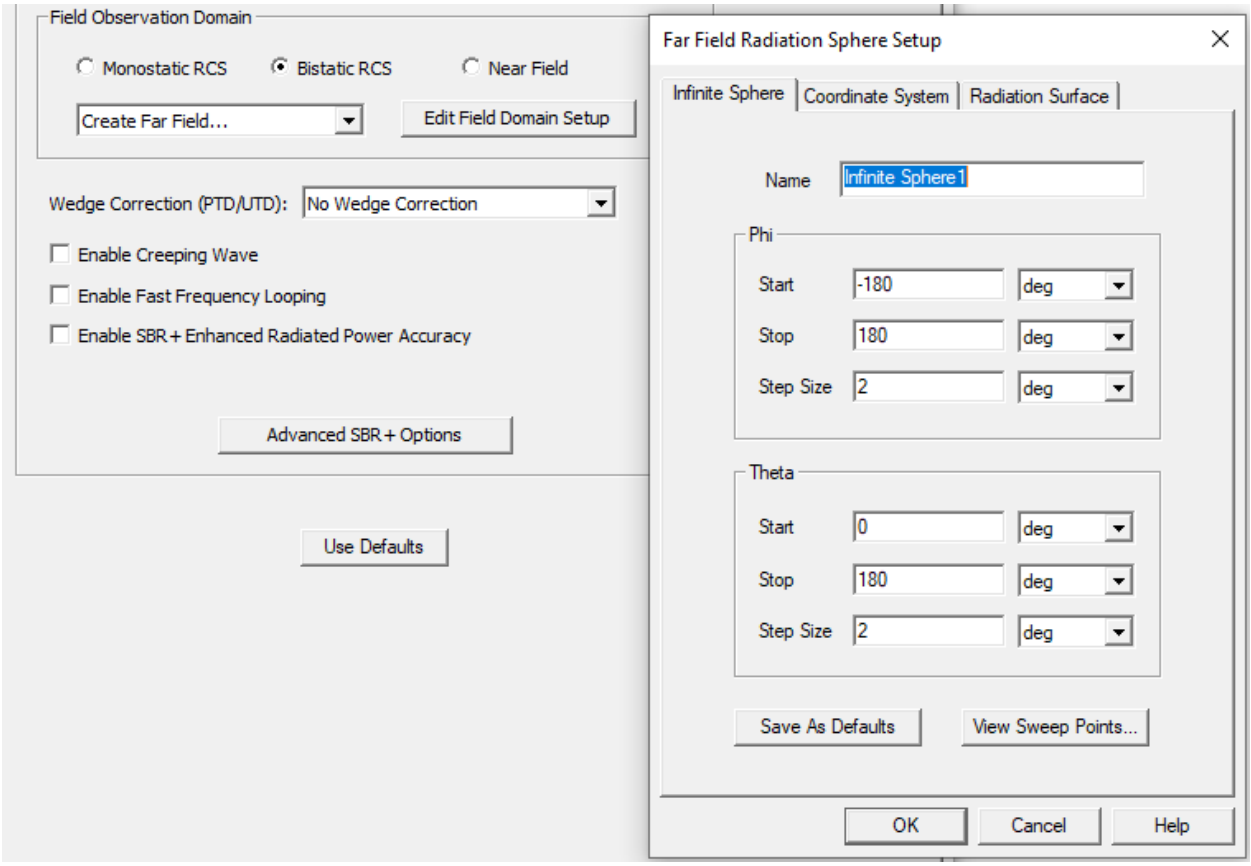
If you created an Incident Plane Wave for the design, the **Hybrid** tab lets you make an RCS Type selection for Monostatic or Bistatic.



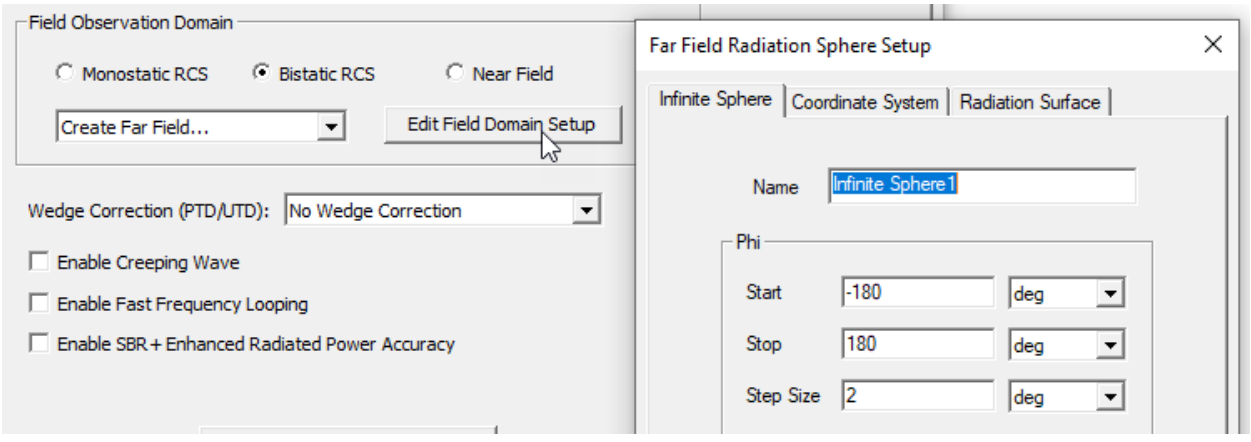
If you select Monostatic, the infinite sphere setup is disabled. In this case, the solver computes the scattered field in the direction of the plane wave. The Reporter will not require a geometry selection, and offers a range of Monostatic quantities for Monostatic RCS traces. See Generating Reports for Monostatic RCS.

If you select have assigned both an SBR+ region and an Incident Plane wave, and select Bistatic as the RCS Type, the Near Field and Far Field Observation domain fields are enabled, and the **Far Field Radiation Sphere Setup** dialog appears as follows, allowing you to create an infinite sphere for a Far Field or a rectangle for examining near fields. You can edit an existing Far Field Infinite Sphere or Near Field Rectangle setup by selecting from the drop down menu and clicking **Edit Field Domain Setup**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you have created one or more field observation domains, these are listed on the dropdown. You can edit an existing Far Field Infinite Sphere or Near Field Rectangle by clicking **Edit Field Domain Setup**.

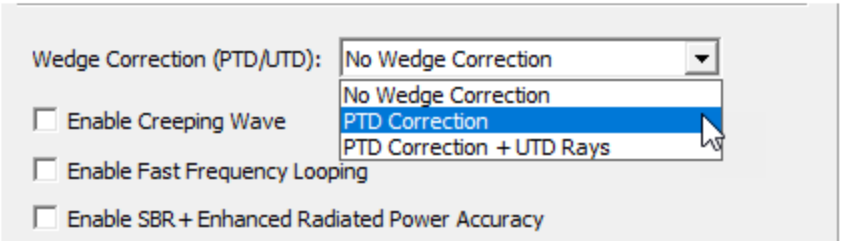


For Near Field rectangle setups, you can use the feature described in Overlay Contour Plot of Near Field on Rectangle.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Enable SBR+ Self Coupling computation capability for driven designs means that the diagonal S-param term takes into consideration the SBR+ scattering effect, and is more accurate. For a discussion of SBR coupling see, Select Tx/Rx Antennas for SBR+ Solutions.

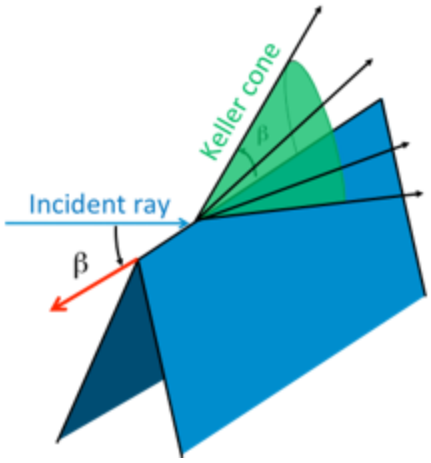
5. The PTD/UTD Simulation Settings allow for the inclusion of additional wedge diffraction phenomenology that can improve the accuracy of SBR+ simulations. You can opt out of using the PTD/UTD settings, or select PTD Correction or PTD Correction + UTD Rays. Selecting one of the PTD Correction options enables a field for specifying PTD Edge Density. This controls how densely wedges are segmented when establishing line of sight from the field source (for example, Tx antenna) to each wedge segment.



For SBR+ simulations, the Physical Theory of Diffraction (PTD) and Uniform Theory of Diffraction (UTD) features can account for additional phenomenology not well predicted by plain SBR due to truncation of uniform Physical Optics (PO) currents at sharp angular discontinuities (“wedges”) on metallic surfaces and blockage of SBR’s Geometrical Optics (GO) rays. PTD is a numeric correction to the scattered fields radiated by PO currents near wedges. UTD launches bundles of edge-diffraction rays from directly illuminated portions of each wedge along the Keller cone. Once launched, the UTD rays behave exactly like regular SBR rays, propagating according to GO and painting PO currents at each bounce that contribute to the scattered field. The UTD rays often illuminate portions of the SBR scattering geometry that are never reached by SBR GO rays launched directly

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

from the field source.



The PTD and UTD wedge features are only deployed for metallic wedges with line-of-sight visibility from the source (Tx) location. If either adjacent surface of the wedge is non-PEC and not within the tolerance for PEC-like, or if the entire edge segment is not visible to the source, the wedge will be ignored in the SBR+ simulation and for visual ray tracing (VRT).

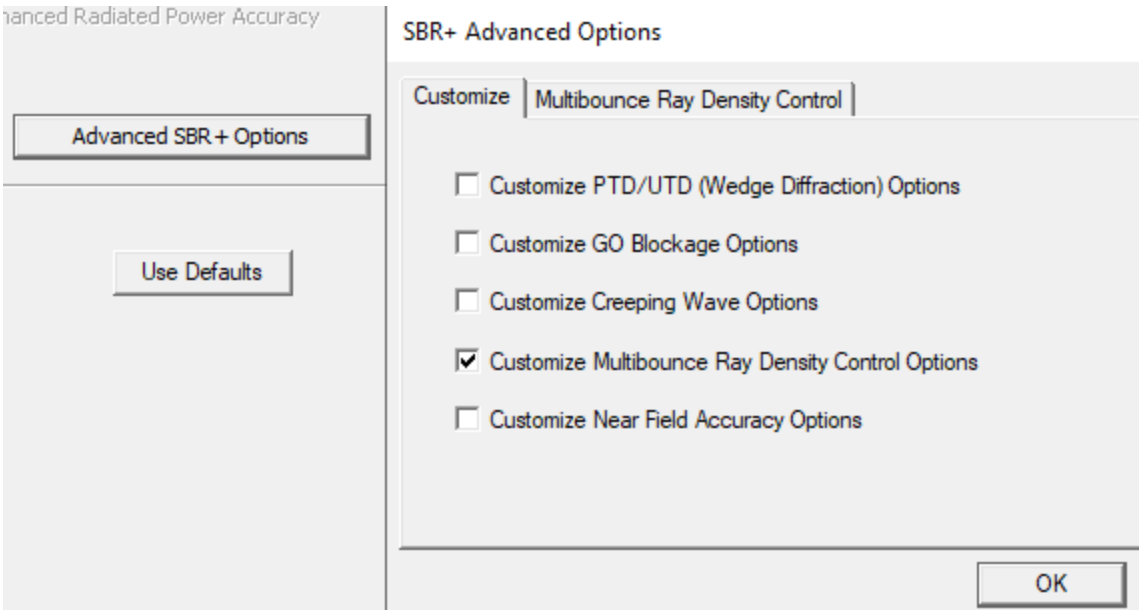
6. Advanced Option for NF Accuracy Settings for SBR+

The NF Accuracy settings (Beta feature, NF Accuracy Controls for SBR+) provide access to methodology settings relating to cases where Tx antennas, Rx antennas, and near-field observation points are in proximity to the scattering geometry (i.e., the platform CAD model). When this feature is switched off, HFSS SBR+ uses default near-field accuracy settings that should be adequate for most situations, including close-proximity Tx/observer conditions that this feature is designed to tune.

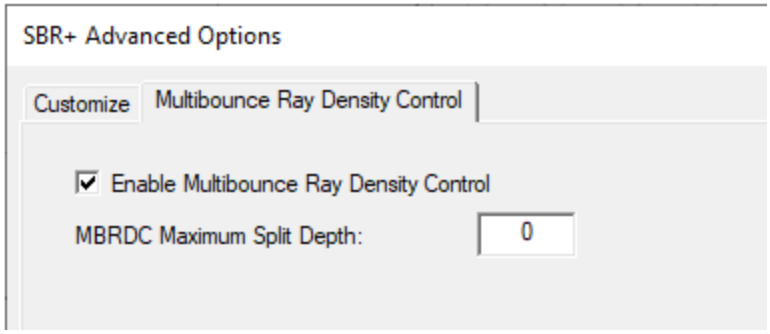
7. Advanced Option for Multibounce Ray Density

By option you can select the **Advanced SBR+ Options** and enable Customize Multibounce Ray Density Control Options.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can then select the **Multibounce Ray Density Control** tab.



Use the checkbox to **Enable Multibounce ray density** and enter the maximum number of subdivisions to be used in the solve. Enabling the feature may increase the solve time.

The **MBRDC Maximum Split Depth** field value increases the number of spawned ray tracks on each ray bounce. The MBRDC algorithm is designed to achieve the ray density per wavelength criterion at any bounce depth. If a footprint at depth *N* in a ray track does not satisfy the ray-density criterion, the MBRDC algorithm attempts a refined ray shoot from an earlier bounce, just for the ray in question. The size of the triggering footprint informs the level of refinement. If the refined shoot is not successful in achieving the desired footprint size, or new footprints are too large, the MBRDC algorithm attempts further, recursive refinements. The **MBRDC Maximum Split Depth** setting limits how aggressively the refinement algorithm is applied by specifying the maximum number of split recursions. For this reason, linear

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

increases in Maximum Split Depth can yield a geometric progression in the total number of rays shot and associated solution time.

When both MBRDC and UTD are enabled, first-bounce UTD rays and bright points on wedges are recomputed during the MBRDC splitting process, but only the original raytrack is rendered up to the 1st bounce. Similarly, length-based ray filters are applied to the original UTD initial ray track up to 1st bounce rather than using the information in the additional MBRDC tracks. This is a known limitation.

8. Advanced Option for Geometrical Optics (GO) Blockage

SBR+ normally applies a physical optics (PO) blockage model where blockage effects emerge from partial cancellation of the incident field by the scattered field. PO blockage not only occurs in connection with the incident field from the Tx antennas, but also within the scattered field itself when reflected fields from the previous bounce are diminished by a blocking surface along the geometrical-optics (GO) reflection path at the current bounce by evaluating the scattering contribution of the incident ray field at that bounce.

In cases of significant blockage by a large obstruction, the PO blockage model requires accurate currents over the extent of the obstruction. Sometimes, this is hard to achieve in the ray tracing approximation. For example, the obstruction may not be well illuminated by direct rays from the Tx antenna or multi-bounce rays after an earlier reflection. An alternative is to use a GO blockage model where scattered field contributions are added to an observation point subject to a line-of-sight (LOS) blockage check performed by the ray tracer. In some cases, this is more accurate than the default PO blockage formulation, while in others it is less accurate. GO blockage also entails a non-trivial computational cost, as the blockage check with the ray tracer must be performed for each observation angle (or point) for each ray hit point.

In addition to GO Blockage for SBR and UTD ray tracks, this feature is also available for PTD and CW.

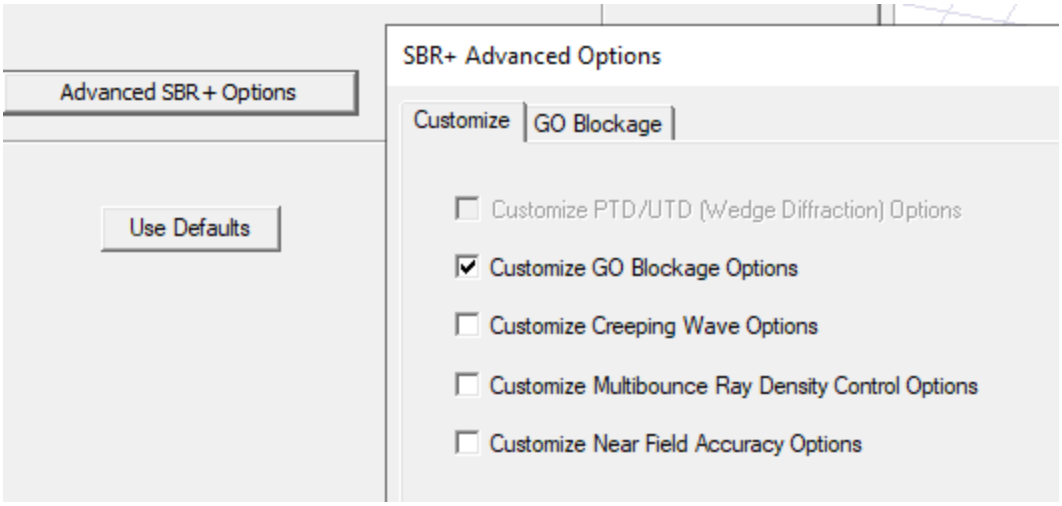
Limitations for GO Blockage:

GO Blockage formulation is suited specifically for SBR+ ray tracks, including UTD ray tracks. While GO Blockage for CW and PTD has been implemented for the first time in this release, results can be surprising and/or inaccurate, as a limitation of the methodology.

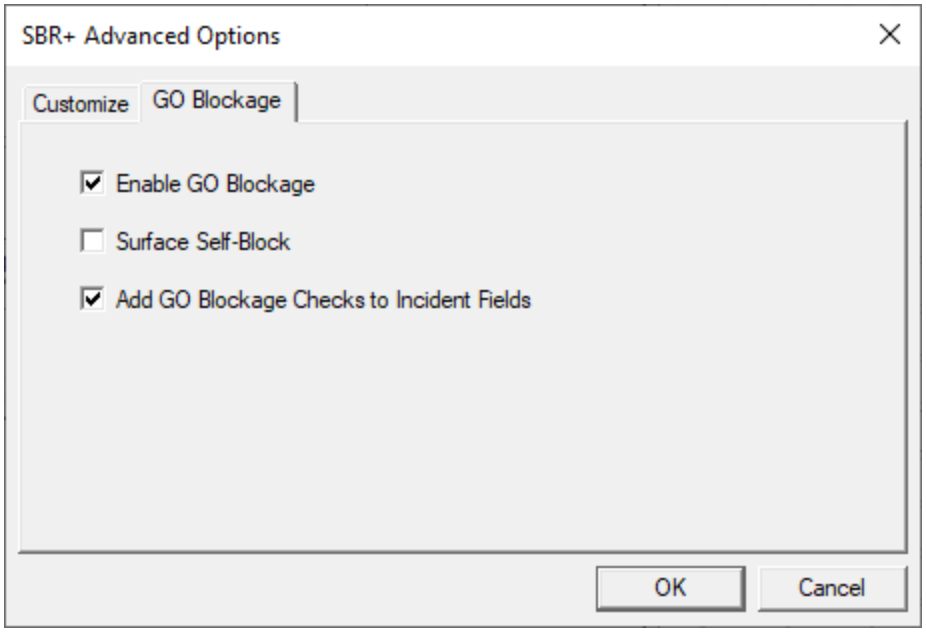
Summary Workflow for GO Blockage:

Click the **Advanced SBR+ Options** button to open the **SBR+ Advanced Options** window. On the **Enable** tab, Check GO Blockage Options to enable the **GO Blockage** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



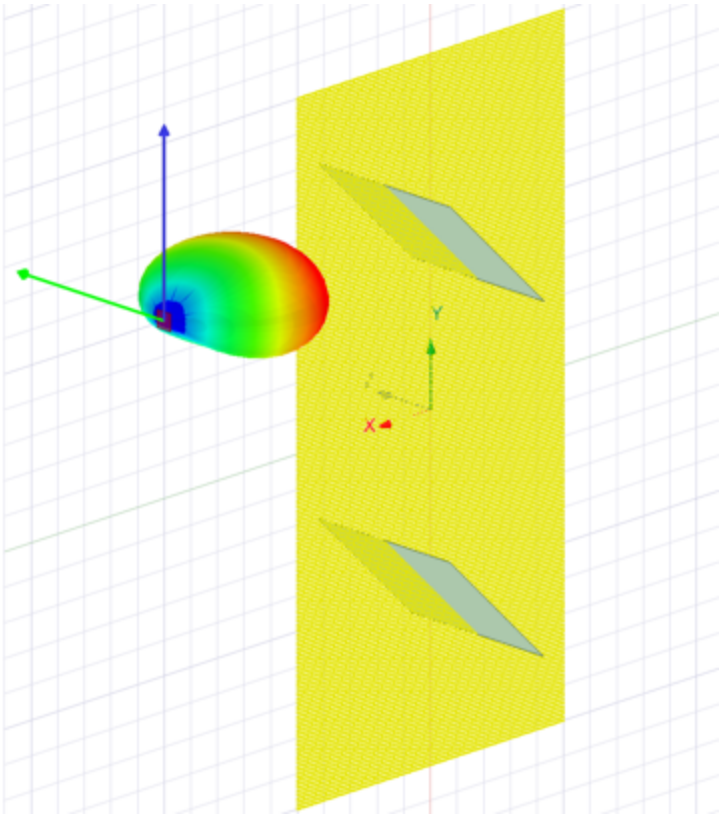
On the **GO Blockage** tab, you have check box options to enable or disable GO Blockage, Surface self-block, and GO Blockage Checks for Incident Fields..



Surface Self-Block: if enabled, the surface where the footprint is radiated from is used for blockage check, that is, anything below the surface with respect to the incoming ray is not in line of sight. If the setting is disabled, the footprint surface is not used for blockage checks (anything below the surface with respect to the incoming ray is in line of sight), but other surfaces are used for blockage.

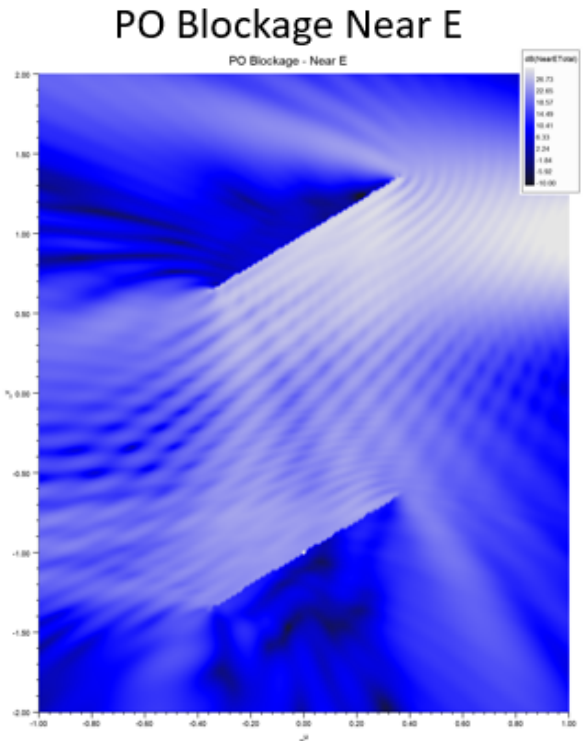
For example, consider a parametric beam antenna illuminating a series of plates.

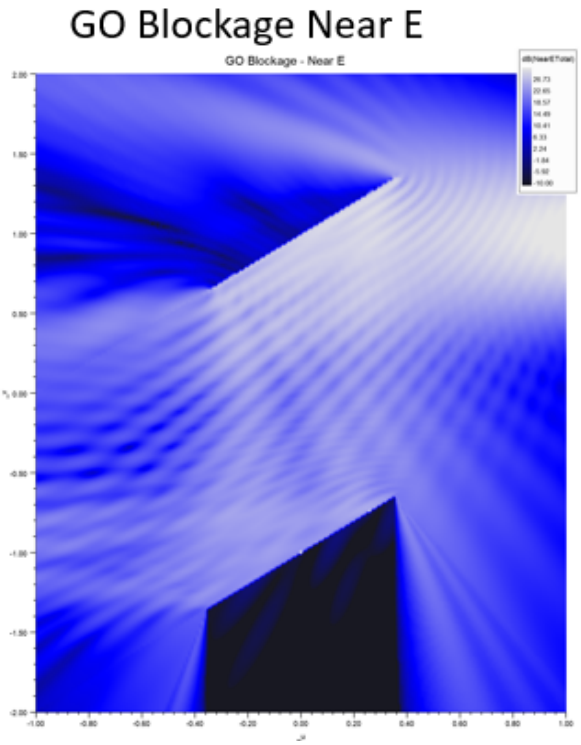
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the plots, the shadows behind the plates are much more pronounced with GO Blockage enabled, but the fields have non-physical discontinuities.

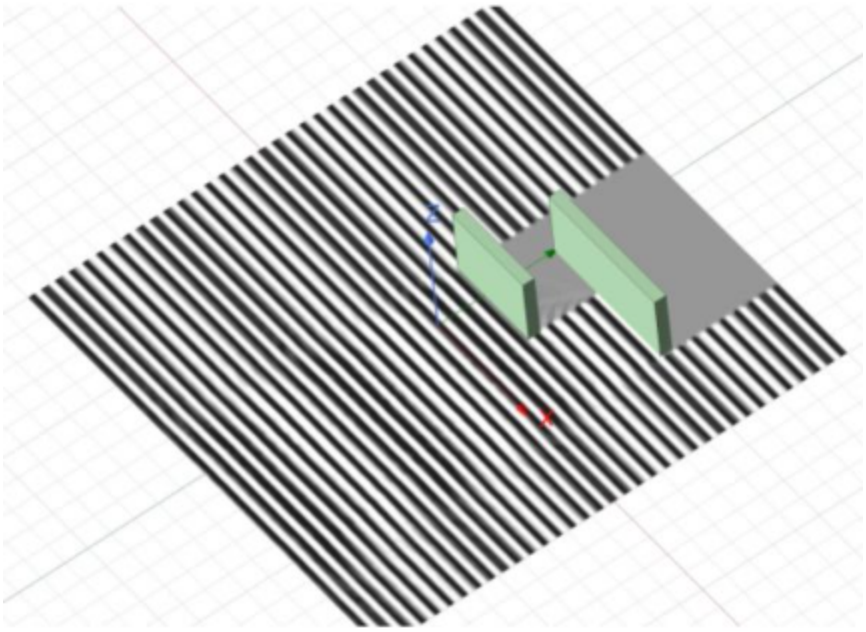
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



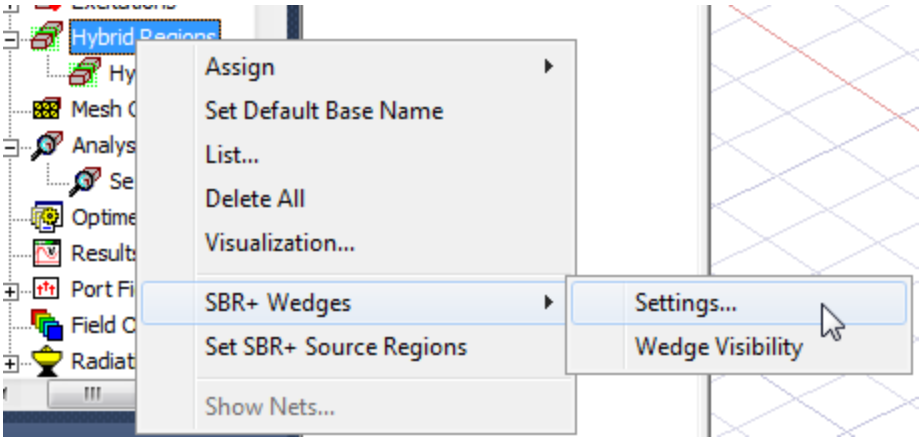


Add GO Blockage Checks to Incident Fields: if enabled, GO blockage effects will also be calculated for the incident field via line-of-sight checks from sources to field points, effectively computing a direct field in the Geometrical Optics sense. For example, the figure below shows the total E-field of a planewave incident on PEC bodies with this option enabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



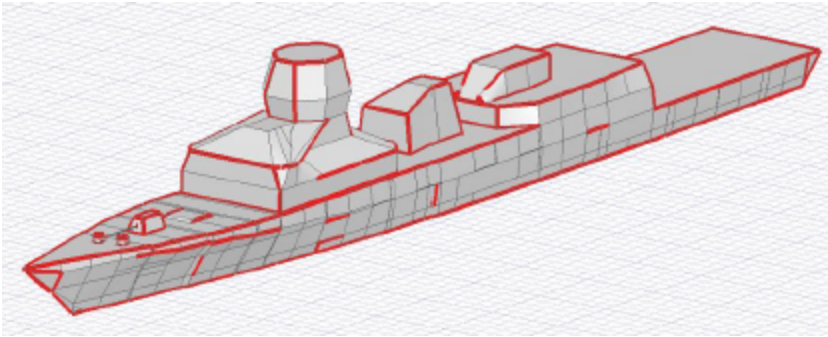
9. Wedge settings for the SBR+ simulation with PTD/UTD are only accessible for an SBR+ design. Wedges are based on theAutomatic Wedge Generation settings, optional Interactive Wedge Editing, and the initial mesh. The initial mesh is automatically generated on launch of the SBR+ Wedge Settings dialog box, if needed. The dialog will not launch if an initial mesh is not available and cannot be generated. The **SBR+ Wedge**s submenu can be accessed through a right-click the Hybrid Regions icon in the Project tree.



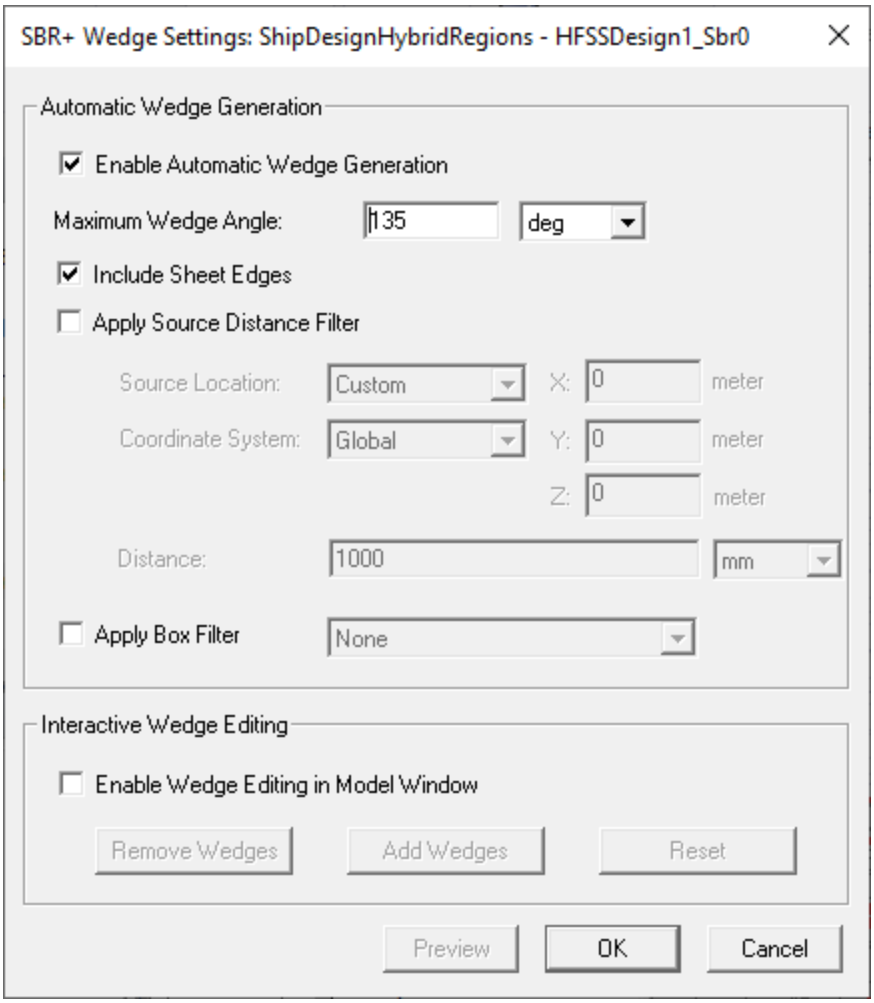
The submenu contains the option for opening the **SBR+ Wedge>Settings** dialog and a checkable option for enabling or disabling **Wedge Visibility**. If visible, wedges are shown as thick red lines in 3D model window for automatically generated wedges, and as blue

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

lines for interactively edited wedges. Visualization is turned off when there is design edits that invalidate the initial mesh.



The **SBR+ Wedge Settings** dialog contains the Automatic Wedge Generation parameters and the Interactive Wedge Editing feature.

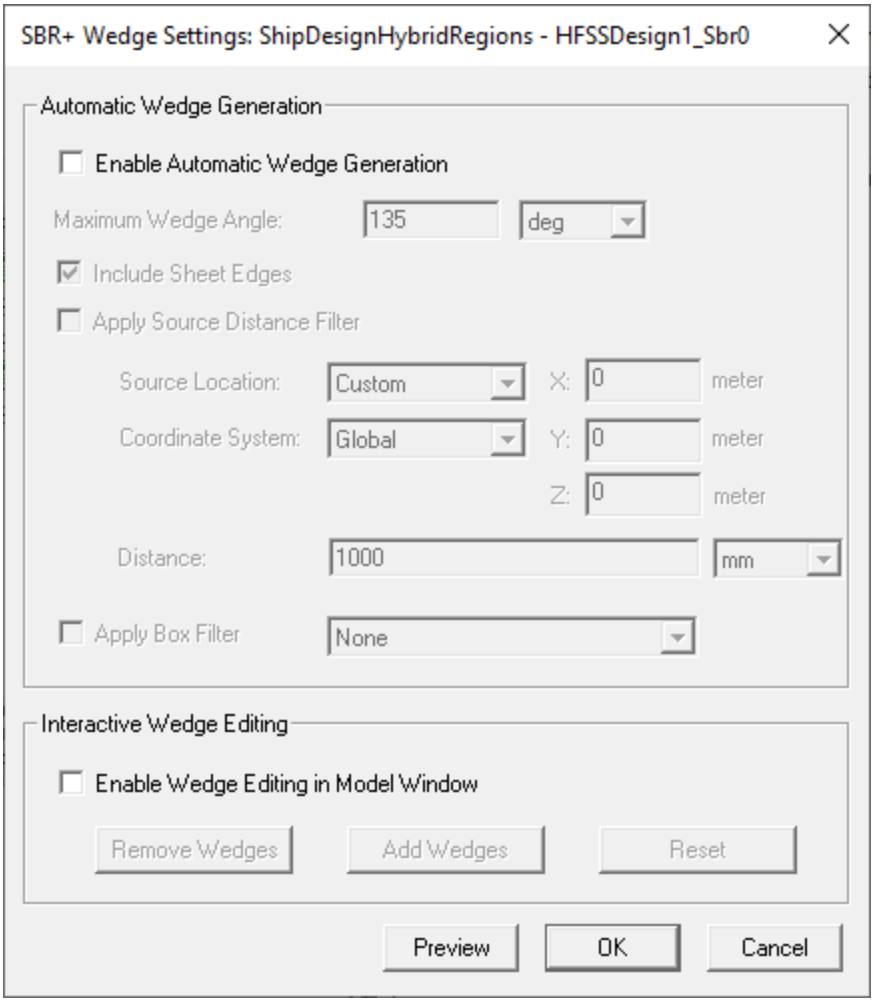


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Wedge Generation parameters are used to determine the candidate set of wedges to be included, with optional filters that can restrict their location on the geometry. The final set of wedges to be used in the SBR+ simulation is the intersection of all the wedge generation criteria, and the effects of any optional Interactive Wedge Editing. complete flexibility in specifying where the wedges are to be used for the simulation. This is important since adding many wedges on very large geometry structures (aircraft, vehicles, etc.) can significantly impact the SBR+ simulation time.

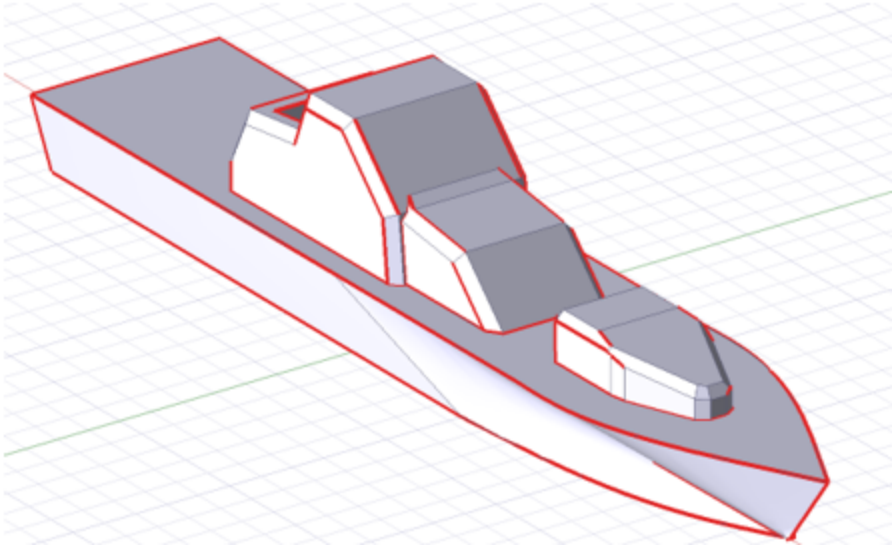
Enable Automatic Wedge Generation.

By default, the checkbox is enabled, allowing access to the automatic features and parameters. If you un-check it, the automatic features are disabled. You can independently enable Wedge Editing.



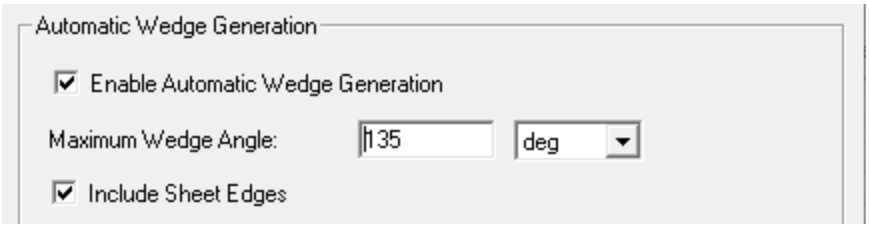
Wedges added through Automatic Wedge Generation are displayed in red.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Maximum Wedge Angle:

An angular criteria can be specified for the maximum wedge angle to help filter the desired set of wedges - default value is 135 deg. This wedge angle ranges from minimum 0 deg to maximum 180 deg. All wedges are included that are less than or equal to the specified maximum angle. An angle equal to 0 deg would be created by a “collapsed” wedge (like a closed hinge), while an angle equal to 180 deg would be created by a completely “open” wedge (like a fully open hinge). You can assign a design variable to Maximum Wedge angle, but you cannot create a design variable within the dialog.



Include Sheet Edges:

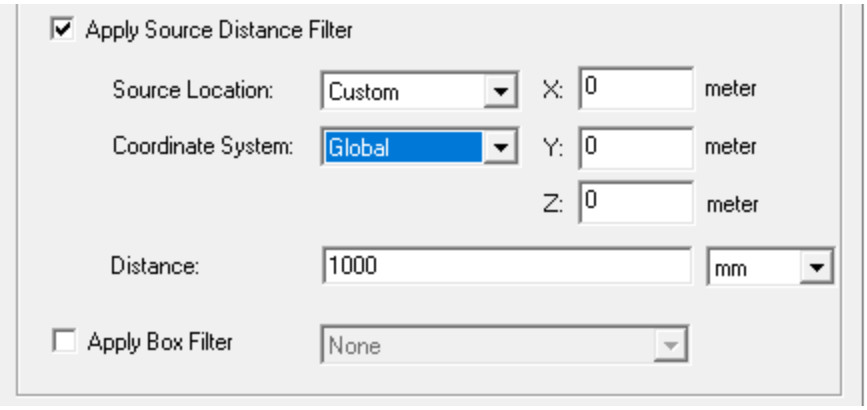
You can choose to include or exclude the edges from sheets (non-connected edges, also known as “knife edges”). The default value is true which includes these edges. Sheet edges are triangle edges with no adjacent/connected triangle along that edge. In the physical object we are modeling these may represent extremely thin surfaces such as metal “fins”, etc.

Optional parameters:

Apply Source Distance filter:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A “source distance filter” allows you to specify that you want to include wedges within a finite 3D distance from a point (the “source”).



You can choose model Points from drop down as source location, or specify the absolute X, Y, Z in model units. X, Y, Z can be specified in global coords or relative to a custom CS. When you select a geometry Point, its coordinate system and X, Y, Z are displayed.

The Distance default is 1 <model unit>. You can assign an existing design variable to the Distance. Default value is false (no Source Distance Filter).

Apply Box Filter:

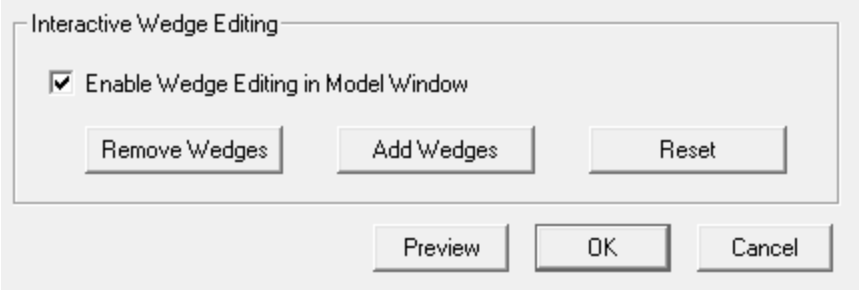
You can select any non-model box that is aligned with the Global Coordinate System axes via a drop-down menu, within which wedges will be included. Default value is false (no Box Filter).

Optional Interactive Wedge Editing

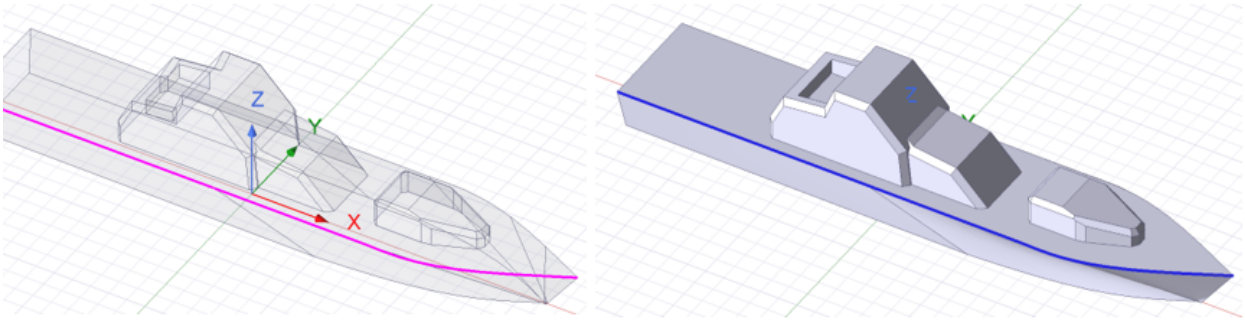
This feature provides complete flexibility in specifying where the wedges are to be used for the simulation. This is important since adding many wedges on very large geometry structures (aircraft, vehicles, etc.) can significantly impact the SBR+ simulation time.

Checking the Enable Wedge Editing the Model Window box enables the buttons for removing wedges from selected geometries (faces, objects, etc.) , adding wedges to selected geometries, and resetting edits.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After you select a geometry and **Add Wedges**, the interactively added wedges appear in blue.



If you select **Remove Wedges**, all wedges on the selected geometry objects are removed. These can be automatically generated wedges or interactively added wedges.

Click **Reset Wedges** to undo all interactive wedge editing operations. A confirmation dialog appears. Press Yes to reset all wedge editing operations. The reset button is disabled if no wedge edit operations are present.

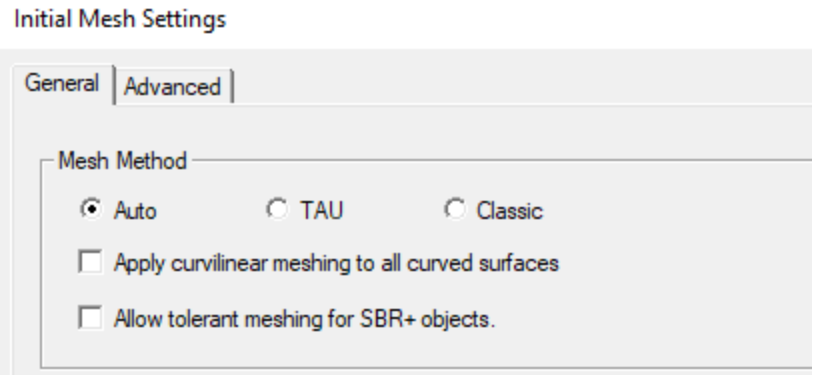
Current Limitations on PTD/UTD Wedge Generation/Editing

Situations where wedges cannot be generated:

Light Weight Imported STL Geometries: STL models imported as light weight geometries are currently not supported for interactive wedge editing (as of R24.1).

In the Initial Mesh Settings for SBR+ Solution type or for HFSS with Hybrid and Arrays that include SBR+ regions, you can turn off tolerant meshing.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



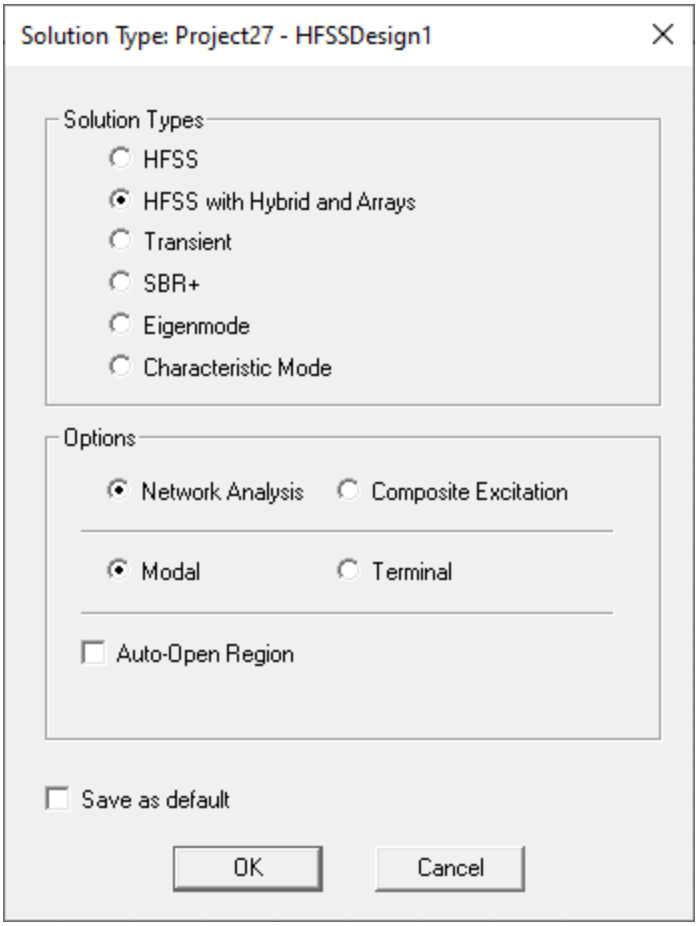
In this case you should consider Current Limitations on PTD/UDT Wedge/Generation Editing.

Configure Simulation with SBR+ Antenna Geometry Blockage (HFSS with Hybrid and Array Solution Type):

The typical configuration begins with a fully integrated model with all geometry. You then select the near antenna region including any nearby geometry interacting with the antenna for analysis by HFSS – either by identifying these objects by hybrid region assignment, or by copying them out to a separate source antenna design to be solved with the HFSS solver, etc. The nearby antenna geometry could even be something different than what is used for the integrated model, such as a flat plate or other structure for considering the nearby antenna interactions. This source antenna design should likely be enclosed in a FE-BI region to allow for easier downstream processing, whether we are in a hybrid or SBR+ solution type. If you use a separate source antenna design, you can re-introduced it to the target design with the full geometry model by creating a 3D component for an antenna that is a linked design to the source design. This means you no longer need to designate any “Model Blockage” when setting up the linked design. You do this as a separate step outside creation of the linked design by selecting geometry from the SBR+ region to be designated as “Model Blockage” associated with a particular source antenna. Note there is no restriction on what objects and/or materials can be designated as “model blockage” geometry. Also note that for either solution type, SBR+ or hybrid HFSS-SBR+, that this selection will be from the available SBR+ region geometry objects, which is different than the previous workflow. Once you select these antenna blockage geometries for each source antenna, HFSS with SBR+ can solve the combined target SBR+ design.The preferred work flow is this.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

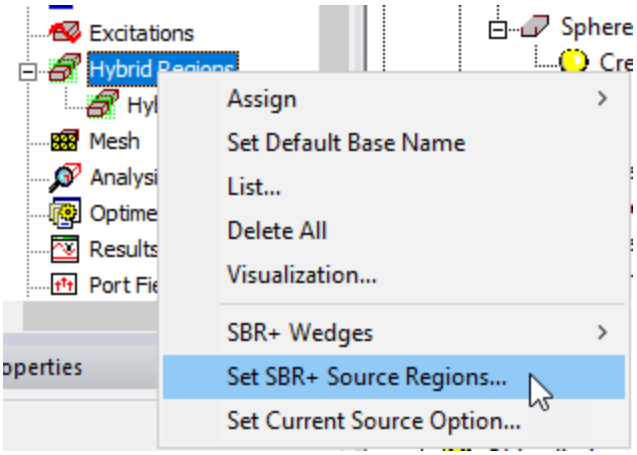
1. Create/open an design with HFSS with Hybrid and Arrays solution type.



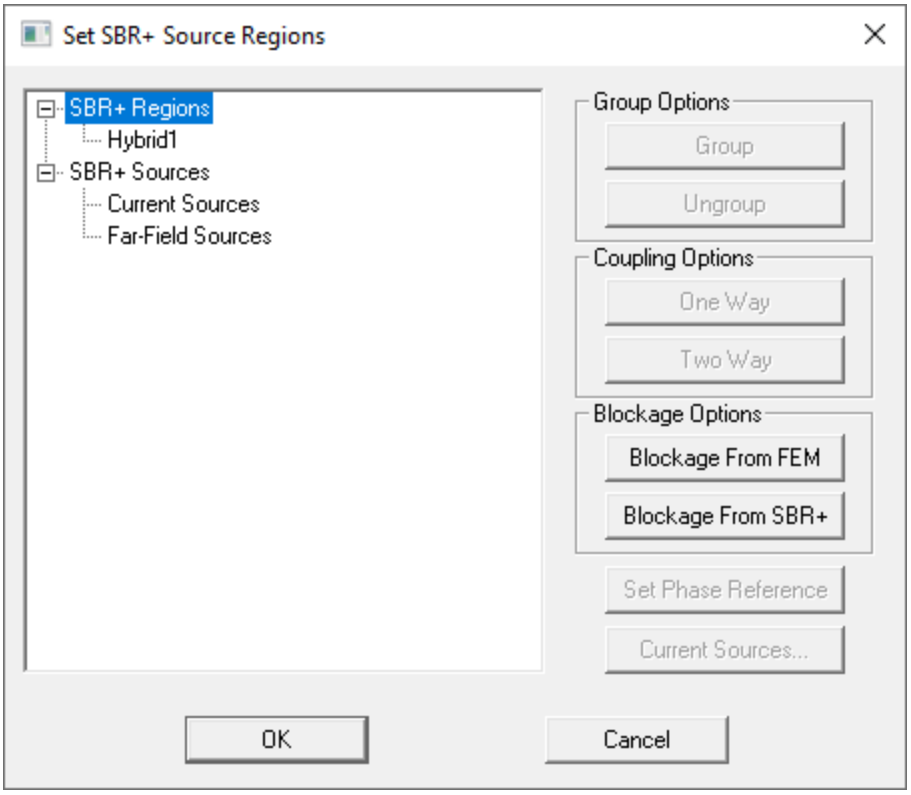
2. Configure the source excitation, including adding or creating any nearby geometry.
3. Create a hybrid FE-BI region to enclose the source excitation and all the nearby geometry objects to be part of the HFSS simulation.
4. Add all the additional geometry objects that will be part of the hybrid analysis, configuring all materials/boundaries as desired.
5. Create/assign a hybrid SBR+ region to all the desired geometry objects to the design.
6. If you want to designate multiple SBR+ geometry objects as blockage, select them in the 3D Modeler window and create an Object List by selecting **Modeler > List > Create > Object List**. You may want to name the list appropriately. If only a single object is to be designated as a geometry blockage, skip this step..

   Only object lists that contain exclusively SBR+ can be selected for antenna blockage.
7. Open the dialog to configure SBR+ source regions (which includes antenna blockage) via the menu option for **Hybrid Regions > Set SBR+ Source Regions...**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

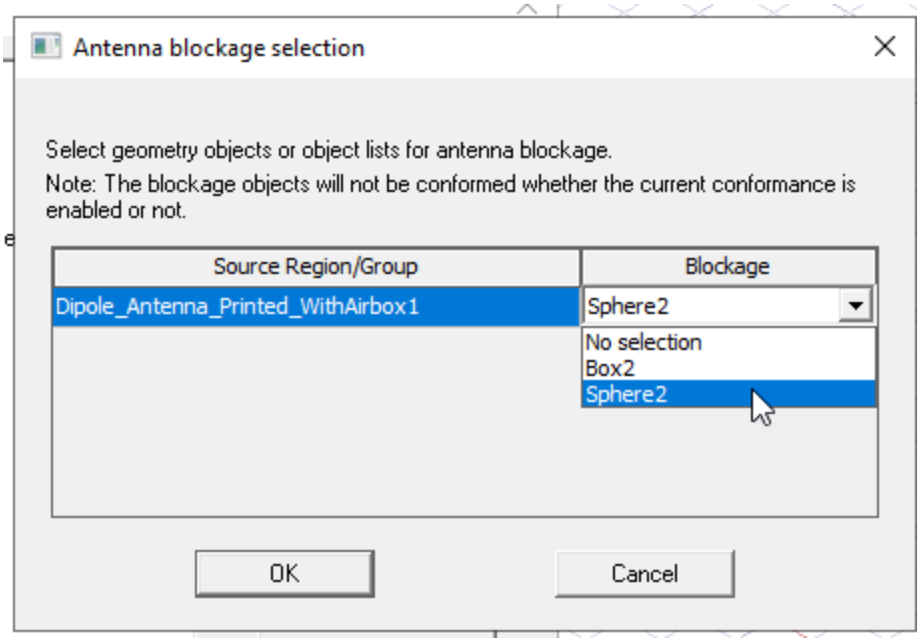


This opens the **Set SBR+ Source Regions** dialog.



Click the **Blockage from SBR+** button to bring up the antenna blockage dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8. For each antenna source, use the “Blockage" column to select the geometry blockage from the dropdown. Each cell in the column contains a dropdown list of all geometry objects and all user created object lists.
9. Click **OK** to save the configuration.
10. Create an **Analysis Setup**.
11. Run the simulation.

Assigning a Dielectric Cavity

To assign a region as a Dielectric Cavity, the object must have a dielectric material assignment. The Lambda refinement setting in the Solution Setup is ignored in a dielectric cavity. Two dielectric cavities can be in contact.

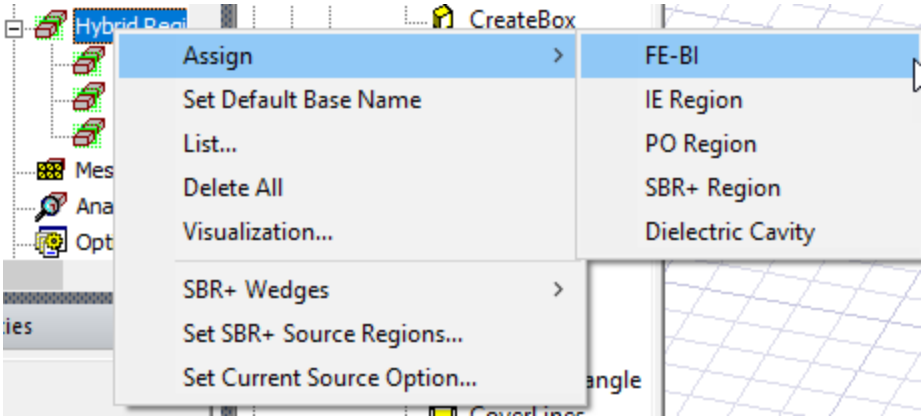
1. Select the object you want to assign as an Dielectric Cavity, and use one of the following menus:

   HFSS>Assign Hybrid>Assign>Dielectric Cavity

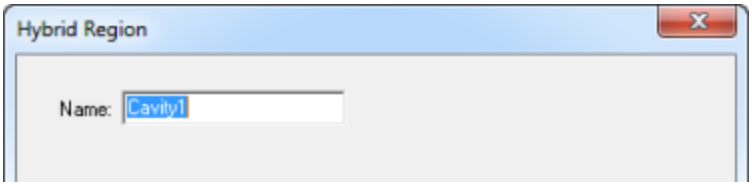
   Right-click in the Modeler window and select **Assign Hybrid>Dielectric Cavity**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select the Hybrid Regions icon in the Project tree, right-click, and click **Assign>Dielectric Cavity**.



The **Hybrid Region** dialog opens.



2. When you OK the dialog box, the assignment appears in under Hybrid Regions icon in the Project tree.

Set Coupling for Hybrid Regions

The **Set Coupling** Command for Hybrid Regions opens a menu that lets you specify one-way, two-way or advanced region coupling. Two-way coupling is the default for all HFSS designs, and fully models the field interaction between regions. One-way coupling includes propagation from source to target regions, and a single bounce of interaction between all regions. This is more efficient than two-way coupling, but also more accurate than traditional one-way links. The Advanced command on the **Set Coupling** menu lets you place regions in two-way coupled groups in a design that is otherwise one-way coupled..

In some cases the coupling between regions is restricted:

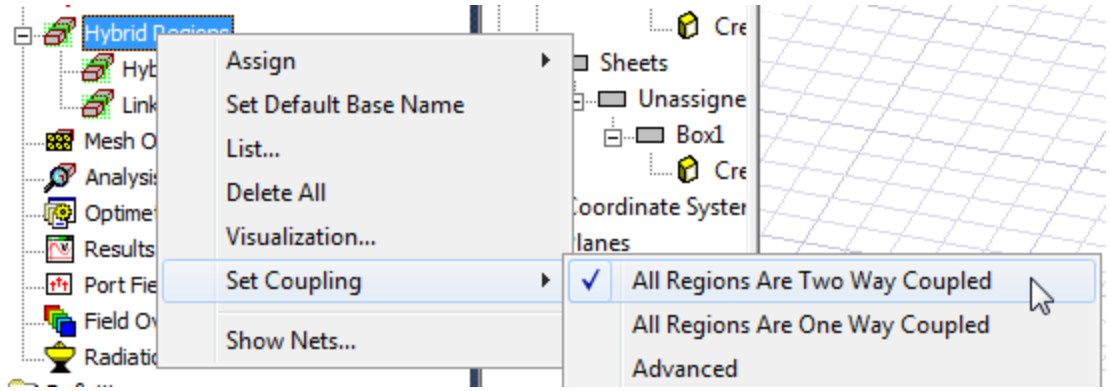
- Two-way coupling is always used for:
  - Regions of the same mesh type that are in contact:
    - Contact between FE-BI / Dielectric Cavity regions
    - Contact between IE/PO Regions

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

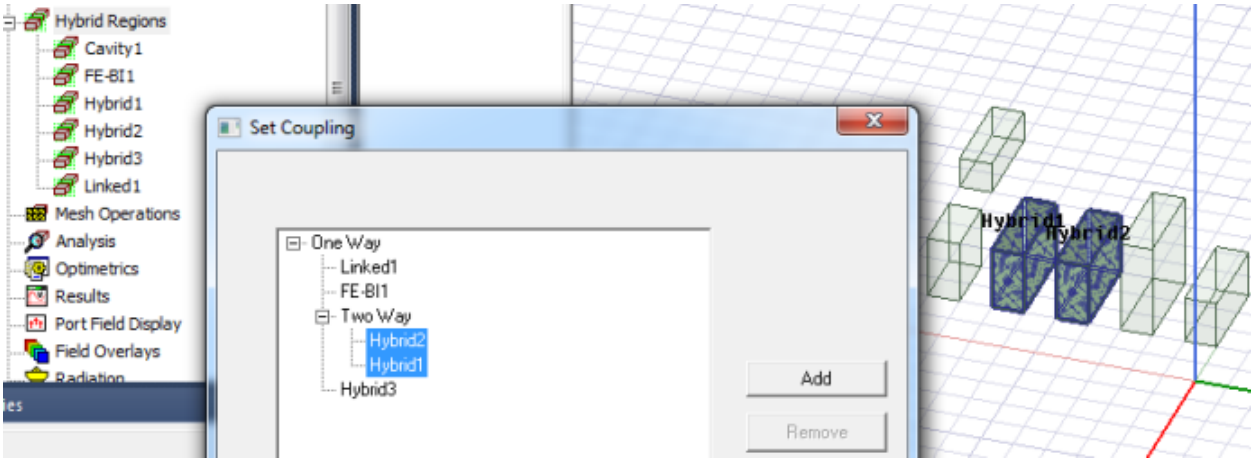
    - Physically separated objects that are part of the same region assignment, for example, two boxes that don't touch, but are in one FE-BI region
- One-way coupling is always used between:
    - Linked regions and all other regions
- Designs containing SBR+ regions will always be set to All One-Way and the Set Coupling menu will be replaced by the **Set SBR+ Source Regions** command. Non-SBR+ regions will serve as sources, and interact with each other through the SBR+ regions.

These restrictions are either enforced by design validation or implicitly handled by the solver.

The Set Coupling menu appears in the right-click menu of the Hybrid folder at the project tree and at the **HFSS>Hybrid** menu.

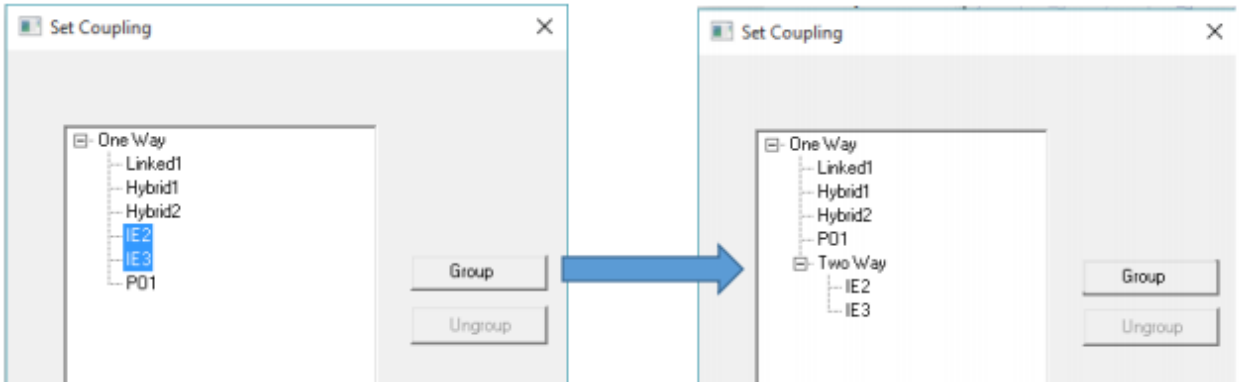


Selecting **Set Coupling>Advanced** opens a **Setup Coupling** dialog setup that lists the coupling settings of all the hybrid regions in the design. Any Selected regions in this dialog are highlighted in the view window. You can edit groupings (as shown below) but remember that editing groups invalidates existing solutions.

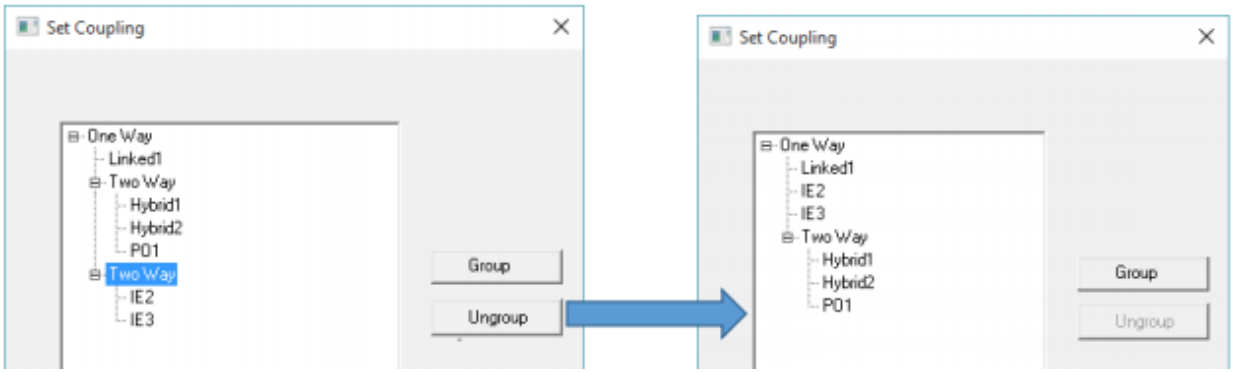


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can add a Two Way coupling group by first selecting the regions to be coupled and then click **Group** and Ok.



You can un-group a Two Way Coupled group by selecting it and clicking **Ungroup** and clicking Ok.



You can also edit grouping by dragging and dropping from one group to another. On Ok, any empty Two Way groups are automatically removed. You will be asked to fix the coupling if you have a Two Way group with only one region.

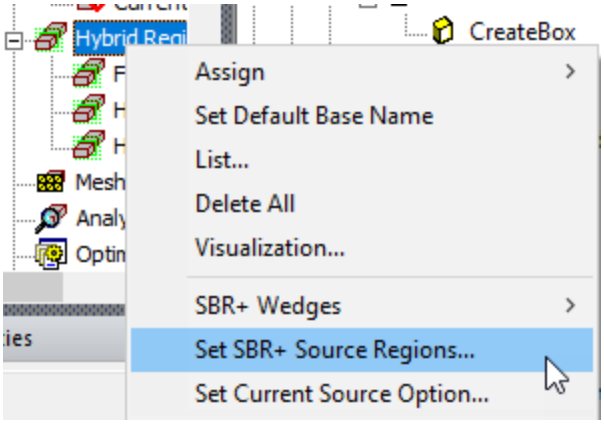
Set SBR+ Source Regions

The **Set SBR+ Source Regions** Command for Hybrid Regions opens a dialog that lets you group source regions for SBR+. Designs containing SBR+ regions will always be set to All One-Way and the **Set Coupling** menu is replaced by the **Set SBR+ Source Regions** command. Non-SBR+ regions will serve as sources, and interact with each other through the SBR+ regions. All source regions interact as one-way links to SBR+ and coupling between the source regions are handled by SBR+ using reciprocity theory.

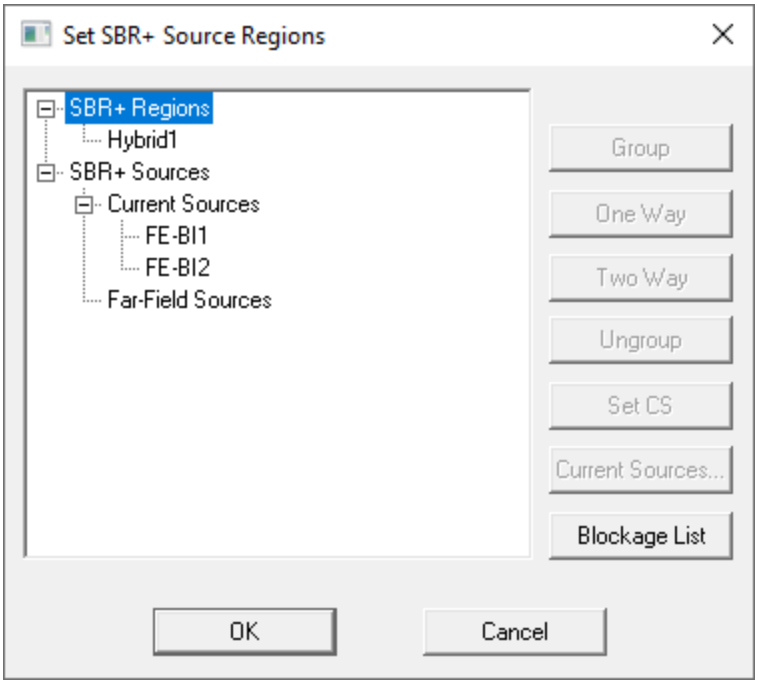
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** The result using far field source option for SBR+ region is sensitive to the phase reference defined when the excitation is close to the SBR+ Region. Phase reference needs to be carefully adjusted to obtain reasonable result.

The **Set SBR+ Source Regions** command appears in the right-click menu of the Hybrid folder at the project tree and at the **HFSS>Hybrid** menu if an SBR+ Region is assigned.

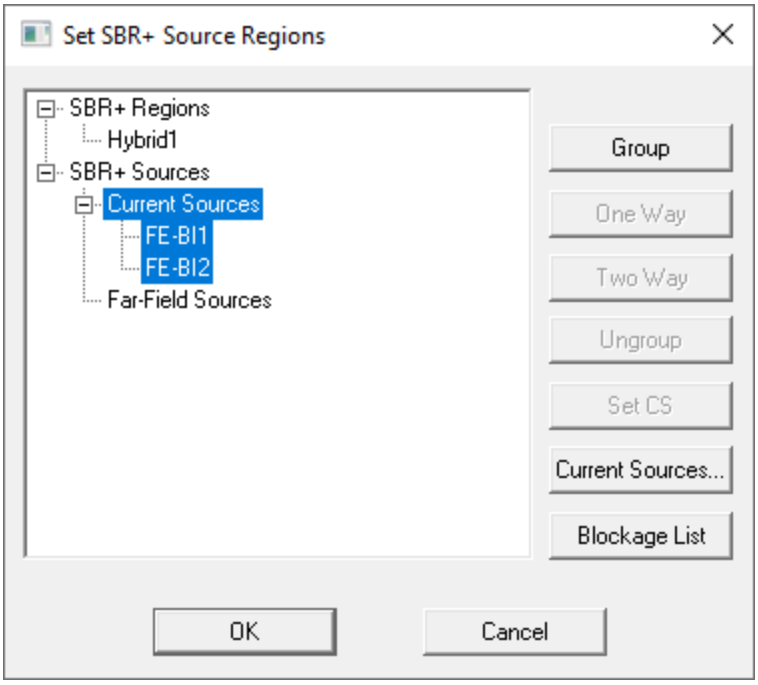


Selecting **Set SBR+ Source Regions** opens a **Set SBR+ Source Regions** dialog that lists the SBR+ Regions and the SBR+ Sources in the design. If the design includes an Incident Wave excitation, you see a message that such designs do not support advanced coupling. However, designs with Current Sources and Far Field Sources will be listed in the dialog box.



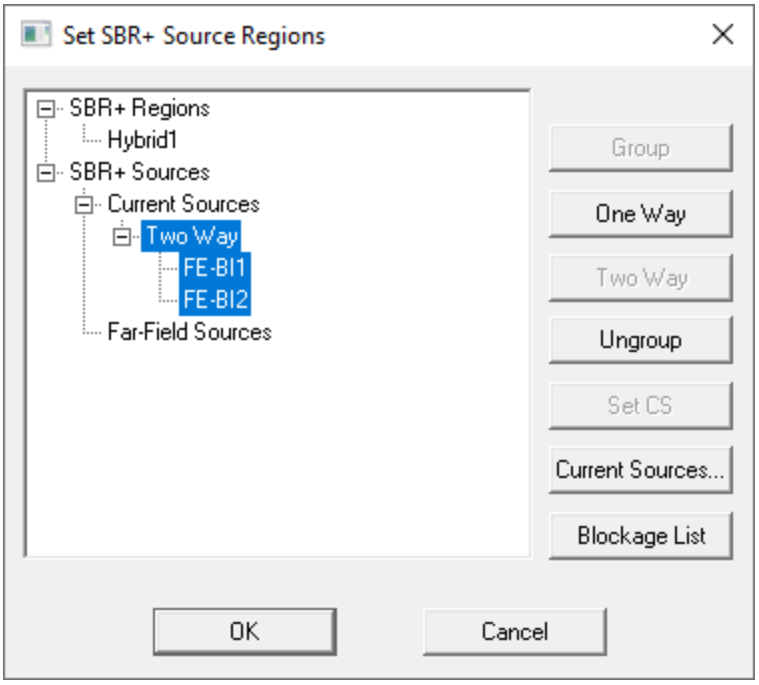
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Any Selected regions in this dialog are highlighted in the view window. You can edit groupings (as shown below) but remember that editing groups invalidates existing solutions.

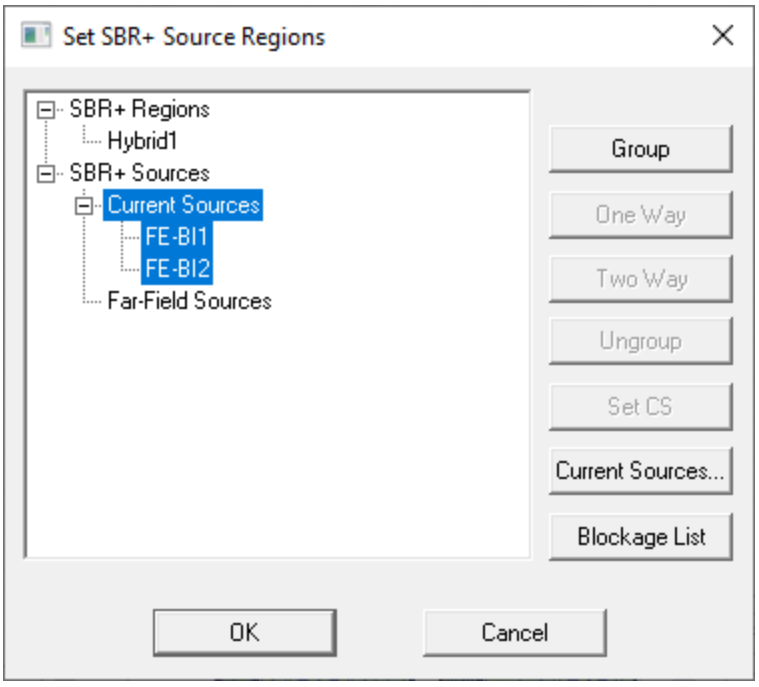


Selecting one or more sources enables the **Group** button provides a means to designate two-way couplings within a group. The regions so grouped are assigned Two Way coupling. The **One Way** button is enabled so that you can change their assignment as One Way couplings. The **Ungroup** button lets you remove coupling groupings.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



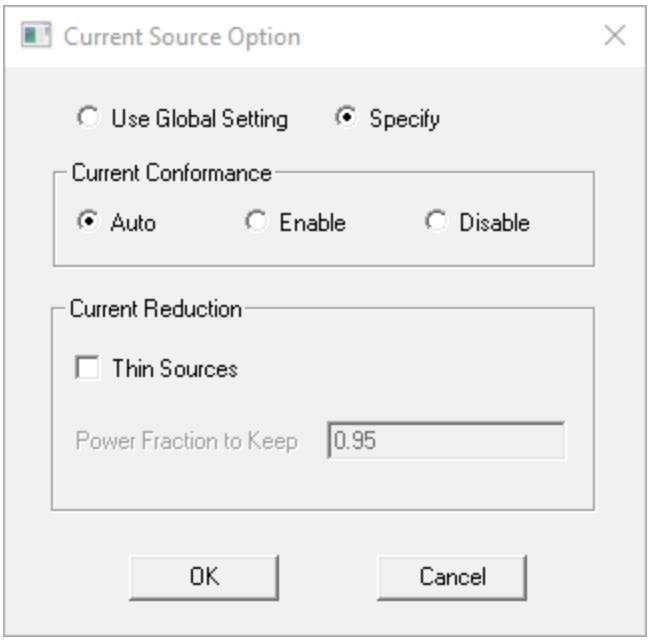
If the design includes Current Sources or Far Fields, these are listed.



The **Current Sources...** button allows you to specify customized power fraction. Use Global is the default. Enabling Thin Sources lets you specify a Power fraction should be greater than 0.1

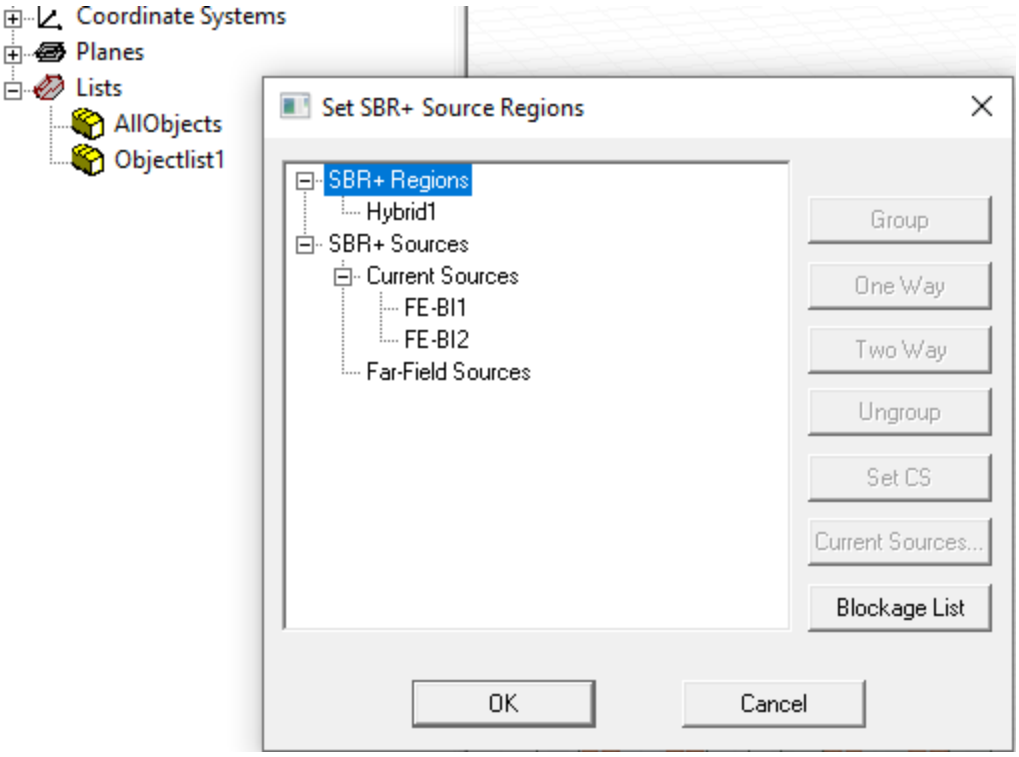
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and less than 1. Solutions are invalidated if you change current source conformance, use Thin Sources, Power Fraction, or change the Global option.



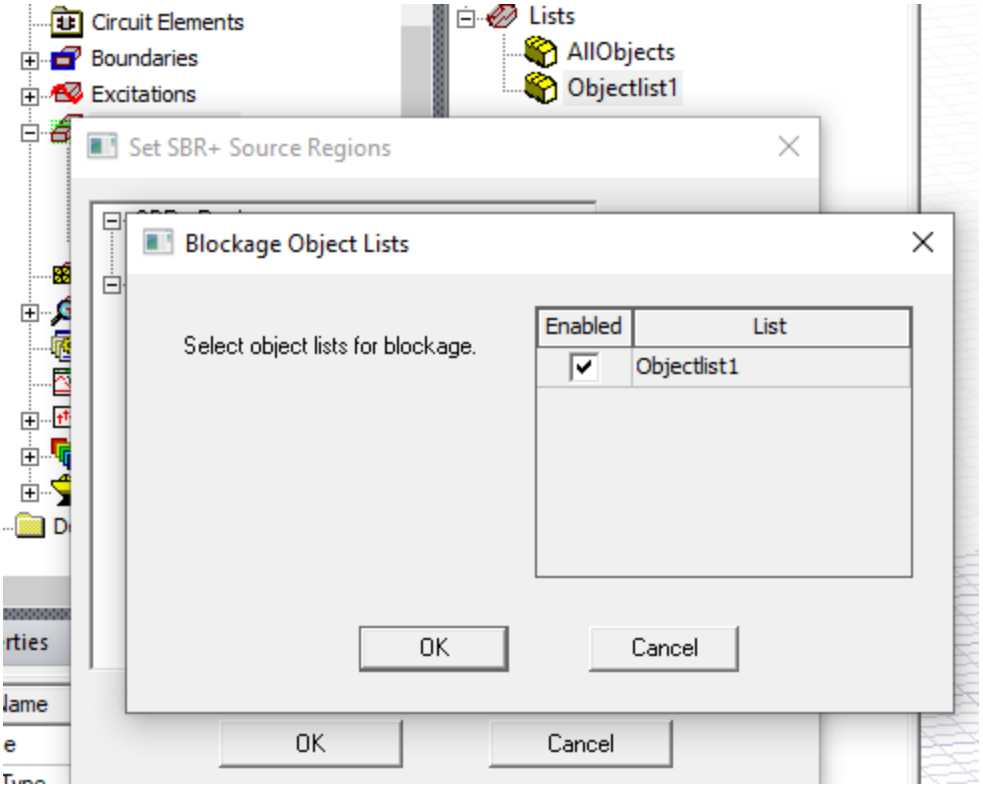
The Blockage List button opens a dialog that lets you view blockage lists. Blockages are allowed when an SBR+ region exist in driven modal and terminal designs. You can easily define an object list for simulation, and automatically assign blockage structures for each antenna.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Click the **Blockage List** button to open a table of available Objectlists. Such lists must contain at least one conducting object and have no SBR+ Regions. Enable those lists to treat the objects as blockages.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

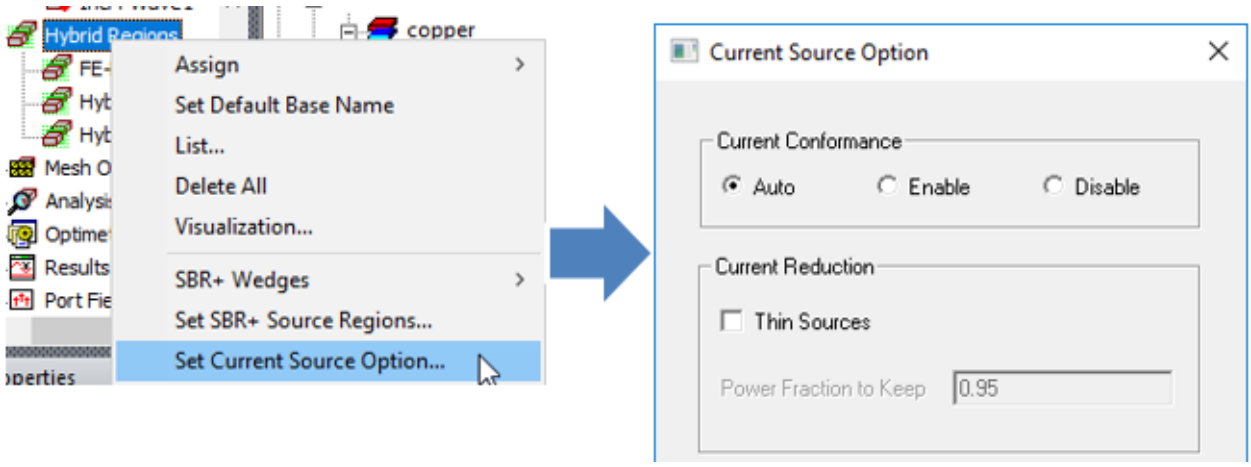


Set Current Source Option

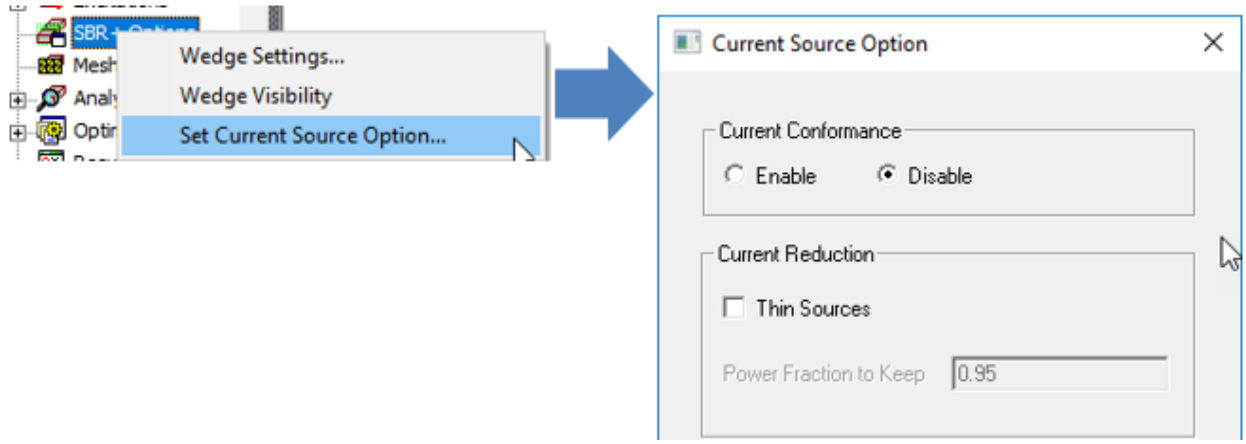
This feature provides a global current source reduction setting, which allows a customized power fraction. You can also override the global setting for each current source in an HFSS Driven design, and for each field linked antenna in an SBR+ design. Use Global is the default. Enabling Thin Sources lets you specify a Power fraction. It should be greater than 0.1 and less than 1. Solutions are invalidated if you change current source conformance, use Thin Sources, Power Fraction, or change the Global option.

For Modal and Terminal Solutions, you can right click on the Hybrid Regions icon in the Project tree and access the **Set Current Source Option...** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

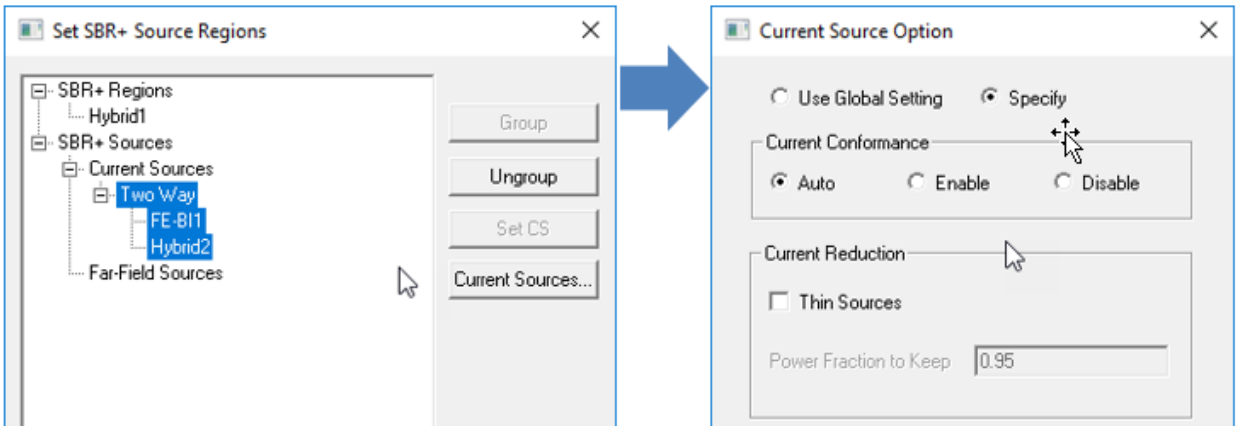


For SBR+ Designs, you can also access **Set Current Source Option** through the SBR+ Options.

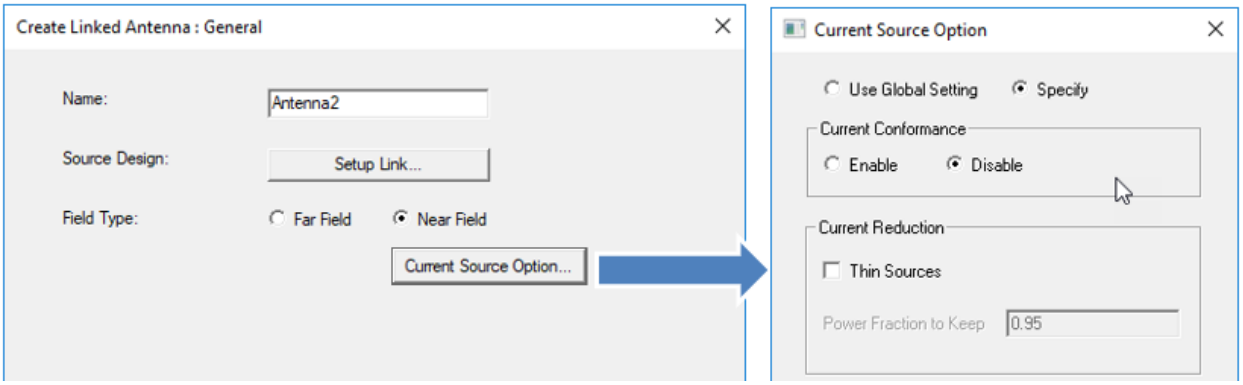


You can set Current Sources when you Set SBR+ Source Regions, but selecting the **Current Sources...** button in the dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also set specific sources when you create a create a linked a Near Field antenna from a driven Modal design to SBR+.

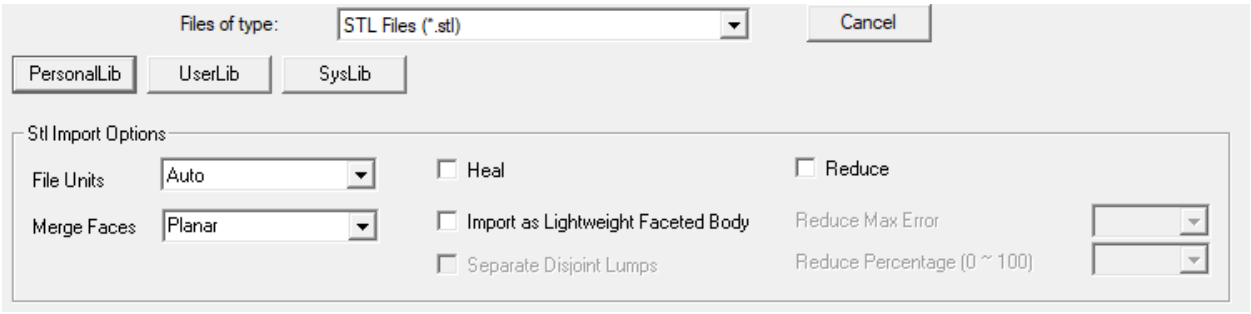


Lightweight Geometry for HFSS Hybrid Regions Designs

Importing large stl files into Ansys Electronics Desktop can be time consuming. STL files contain triangulations of objects, and importing them to the modeler creates a face out of every triangle, and an edge out of every triangle edge. This creates a large number of faces and edges. which can slow down operations. The lightweight geometry feature gives you the option to import objects as one face, without edges or vertices in them. Internally, the modeler does not create bodies for these objects. The Beta lightweight geometry feature supports all hybrid design, including IE, PO and SBR+.

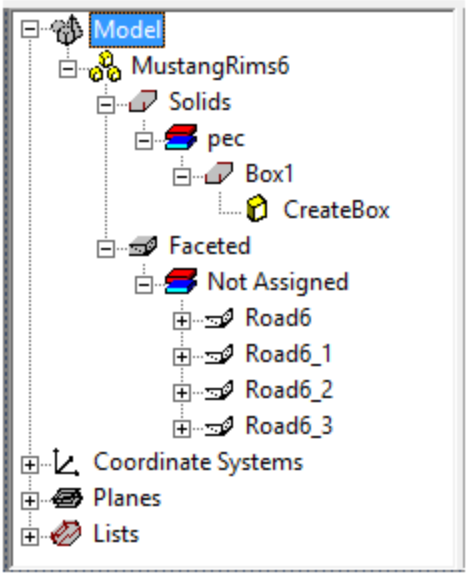
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the Beta Feature for Lightweight Geometry for Hybrid Regions is enabled for HFSS with Hybrid and Arrays Solution type, the Import File dialog for STL files contains several options.



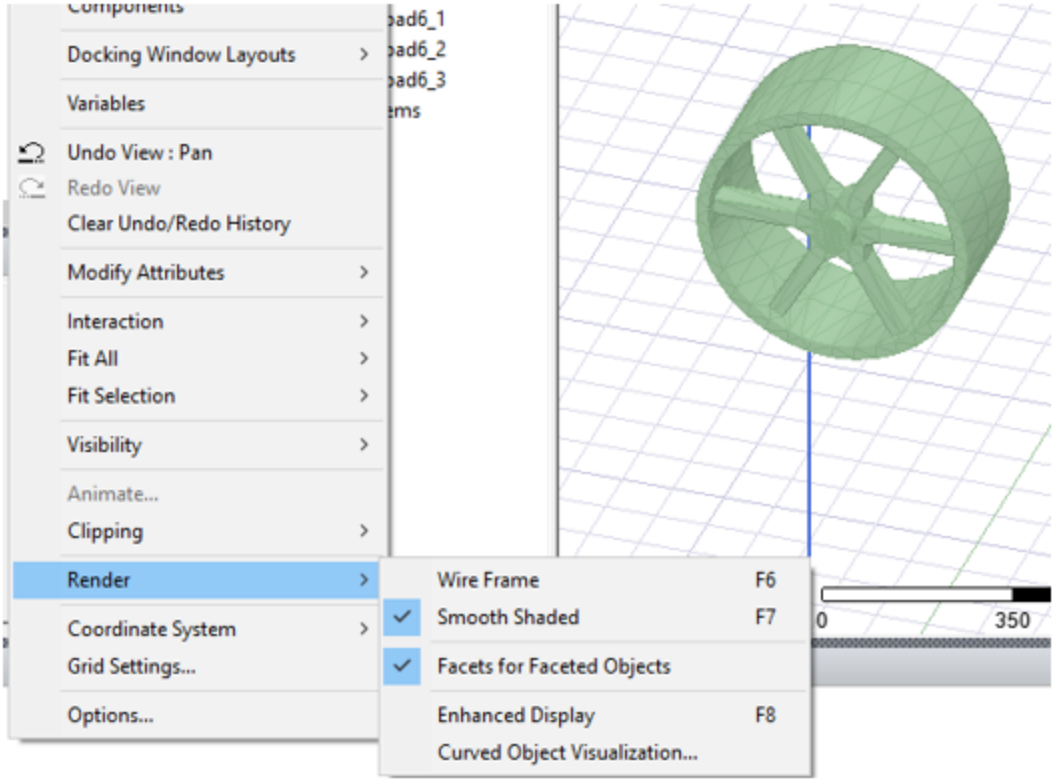
- **Heal** attempts to remove common issues while importing stl files, including closing solids when they are open due to minor cracks in triangulation and removing self-intersections.
- **Import as Lightweight Faceted Body**, if checked, creates faceted bodies from imported objects and enables the Separate Disjoint Lumps option.
- **Separate Disjoint Lumps** option, if checked, create multiple parts if the .stl files have objects having disjoint lumps.
- **Reduce** enables lets you reduce the number of facets in faceted objects while keeping the overall shape of the object. Checking **Reduce** enables fields for specifying Reduce Max Error and Reduce Percentage.

Once imported, faceted objects have distinct icons in history tree. They can also be grouped together, as shown in the following figure.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

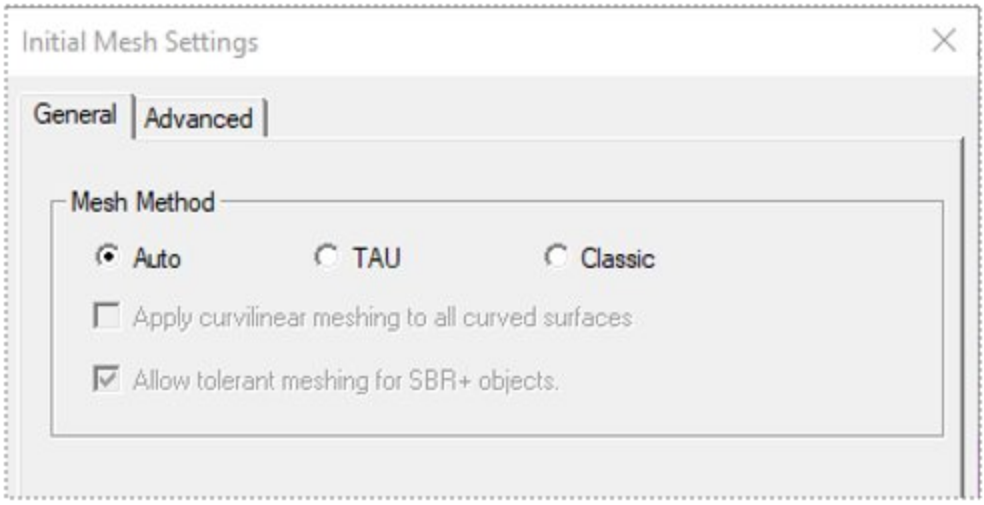
A new option, **Facets for Faceted Objects**, is provided in **View>Render** menu for Hybrid solution types. When this option is check on, the modeler renders edges of triangle contained in the faceted bodies. These edges cannot be selected. See the following figure:



Initial Mesh Operations for Lightweight Geometries

Importing a lightweight geometry into an HFSS with Hybrid design will internally use Tolerant Meshing (add link) technique and disable the "Apply curvilinear meshing to all curved surfaces" option.

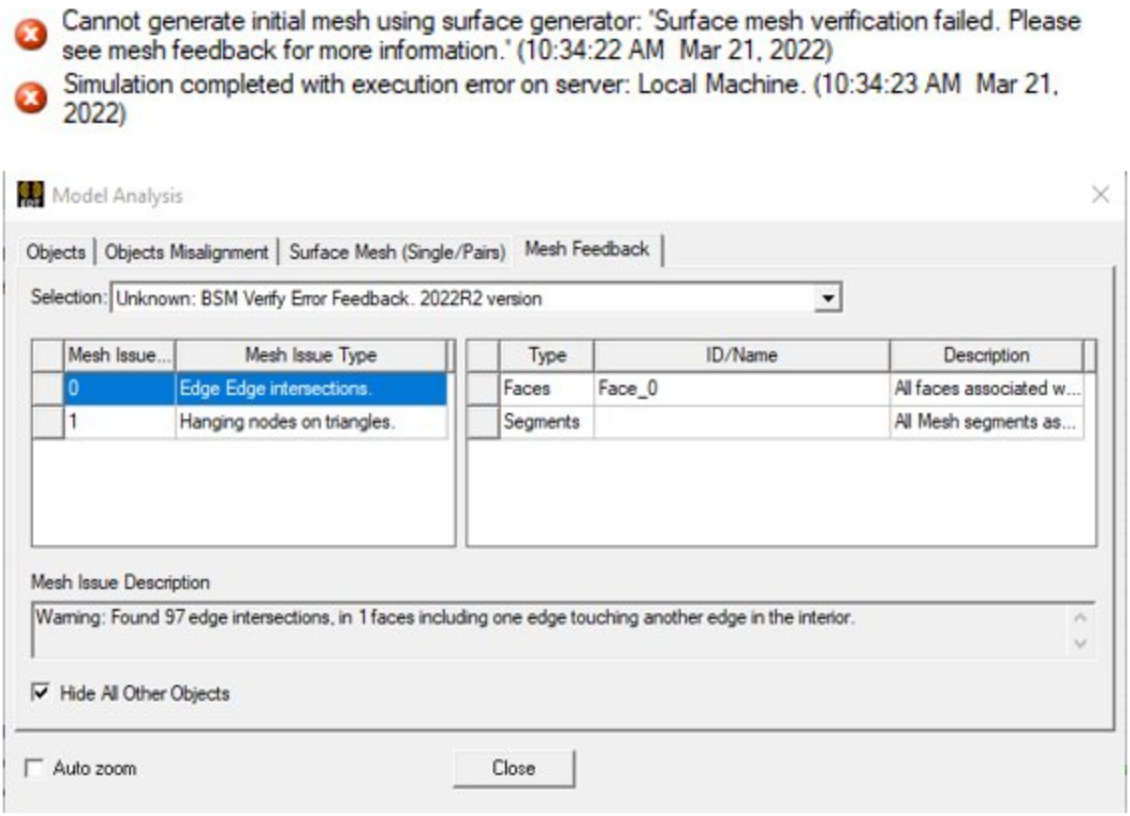
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Lightweight Geometry Verification for the IE Solver

The IE solver has stricter requirements toward lightweight geometry. For example, any triangle intersection or overlapping is not allowed. Mesh feedback is generated when performing tolerant meshing to help you better identify any geometry issue.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Permitted Modeler Operations on Faceted Objects

- Arrange operations such as Move, Rotate, Mirror.
- Duplicate along line, around axis, with mirror.
- Scale.
- Copy-Paste.
- Create object CS - you can snap to vertices of facets to create object CS.
- Create face CS - you can pick any point, determine the triangle on which the picked point lies, and store the position relative to the triangle vertices. **Note:** this capability is particularly useful for antenna placement on lightweight geometry models in SBR+ designs.
- Create 3D components out of faceted objects.
- Selecting faces for assigning boundaries, coordinate systems, excitations or hybrid regions. **Note:** the entire object boundary is selected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Other Characteristics of Faceted Objects

- Solid and sheet objects are identified in the History tree.
- Object Measure dialog shows the correct volume, and surface area of faceted objects.
- Faceted objects are skipped during intersection checks and containment checks done during geometry validation.
- Faceted objects cannot be exported from the Modeler.

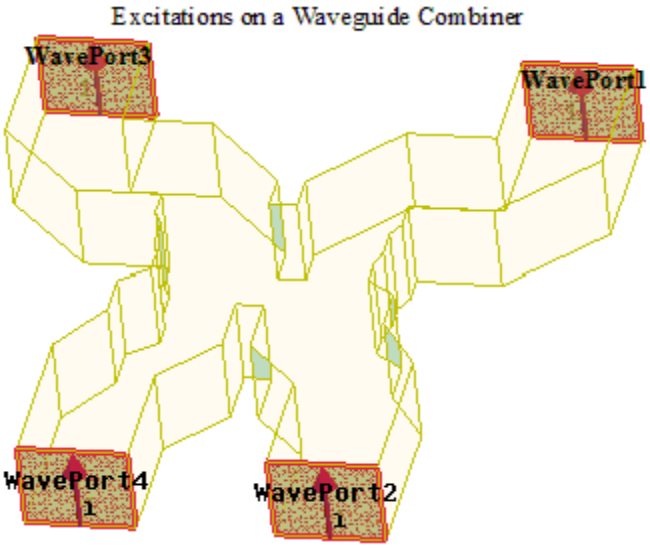
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19 - Assigning Excitations for HFSS or HFSS-Transient

Excitations are sources of electromagnetic fields in the design. HFSS has various options to generate incident fields that interact with a structure to produce the total fields. Some of these excitations are local sources residing within the structure such as wave ports and voltage sources, while other excitations such as plane waves are created from local sources away from the structure.The available excitations depend upon the product. They are as follows:

- HFSS Excitations including SBR+
- HFSS Transient Excitations



Also see Circuit Elements in HFSS.

HFSS Excitations

You can assign the following excitations to an HFSS design:

Circuit Port	Represents an HFSS 3D Layout feature implemented in HFSS. Circuit ports can be thought of as Lumped Ports assigned to a pair of edges. In the driven modal problem, the integration line for the circuit port will be the vector from the middle of the second edge to the middle of the first edge. In

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	the driven terminal problem, the two edges are populated as terminal and reference under circuit port. For designs with only lumped ports or a combination of lumped and circuit ports, the Advanced Solution Setup, **Options** tab includes an option to use Enhanced low frequency accuracy.
Wave Port	Represents the *external* surface through which a signal enters or exits the geometry. It is effectively a semi-infinite waveguide attached to the model. This waveguide has the same cross-section and material properties as the port. Wave ports are placed on this interface to provide a means to link the model device to the external world.
Lumped Port	Represents an *internal* surface through which a signal enters or exits the device. It is effectively a lumped element for exciting the device and measuring S-parameters. For designs with only lumped ports or a combination of lumped and circuit ports, the Advanced Solution Setup, **Options** tab includes an option to use Enhanced low frequency accuracy.
Terminal	A terminal is defined by one or more conductors in contact with the port. HFSS treats microwave structures as a black box that may have one or more terminals, each of which has a voltage/current pair. Terminals are assigned automatically.
Floquet Port	Floquet Ports are used exclusively with periodic structures defined by Primary and Secondary (Coupled Lattice Pair) boundaries. They contain plane waves whose frequency, phasing, and the geometry of the periodic structure determine the propagation direction. Chief examples are planar phased arrays and frequency selective surfaces when these may be idealized as infinitely large and analyzed using a unit cell.
Incident Wave	Represents a propagating wave impacting the geometry.
Linked Field	Represents a Far Field Wave or Near Field Wave or Cable Network. You can also link a driven Modal HFSS design to SBR+.
Voltage Source	Represents a constant electric field across feed points.
Current Source	Represents a constant electric current across feed points.
Magnetic Bias	Used to define the net internal field that biases a saturated ferrite object.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Multipaction Charge Region	In HFSS Multipaction analysis, charges are added to the simulation domain as excitations.
Multipaction DC Bias	In HFSS Multipaction analysis, sets up DC bias fields for a multipaction analysis
SBR+ Antenna Components	For SBR+ Solution types, use Antenna components to represent port excitations using the available Parametric Antennas, Parametric Arrays, or via Linked Source design. Individual transmit/receive antenna port couple pairs can be assign for the SBR+ simulation by using the Tx/Rx Selection dialog.
Circuit Elements	For driven Modal and Terminal solutions, you can convert Lumped Ports and Terminals to Circuit Elements, based on Touchstone files or linked projects.

HFSS Transient Excitations

You can assign the following types of excitation. For HFSS Transient designs, the properties for each excitation include a **Transient** tab, enabling you to designate each excitation as Active or Passive.

Wave Port	Represents the *external* surface through which a signal enters or exits the geometry. It is effectively a semi-infinite waveguide attached to the model. This waveguide has the same cross-section and material properties as the port. Wave ports are placed on this interface to provide a means to link the model device to the external world.
Lumped Port	Represents an *internal* surface through which a signal enters or exits the device. It is effectively a lumped element for exciting the device and measuring the S-parameters.
Terminal	A terminal is defined by one or more conductors in contact with the port. HFSS treats microwave structures as a black box that may have one or more terminals, each of which contains a voltage/current pair. Terminals are assigned either manually or automatically.
Plane Wave	Represents a wave that propagates in one direction and is uniform in the directions perpendicular to that of its propagation.
Voltage Source	Represents a constant electric field across feed points.
Current Source	Represents a constant electric current across feed points.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This release offers a beta feature to allow you to mix wave ports and terminals in a transient waveguide simulation.

Assigning Circuit Ports

Circuit Ports in HFSS are equivalent to those in HFSS 3D Layout. These ports provide major benefits for HFSS:

- Good low frequency accuracy (Hz range)
- Good for multi-physics work flows as there is no restriction about being planar
- For designs with only lumped ports or a combination of lumped and circuit ports, the Advanced Solution Setup, **Options** tab includes an option to use Enhanced low frequency accuracy.

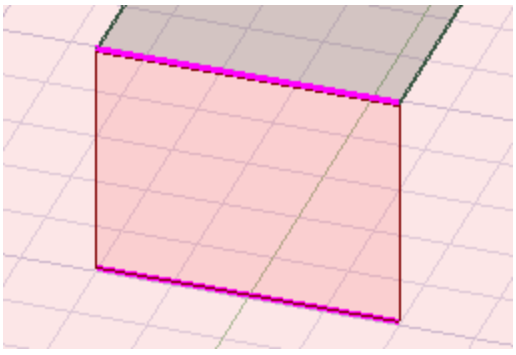
Circuit ports can be thought of as Lumped Ports assigned to a pair of edges. In a driven modal solution, the integration line for the circuit port will be a vector from second edge to the first edge. In the driven terminal problem, the two edges are populated as terminal and reference under circuit port. Deleting either edge will delete the circuit port. You can create circuit ports, translate between circuit ports and lumped ports, and include circuit ports in N-port circuit elements. They are treated exactly as lumped ports.

If you change the solution type between driven modal and driven terminal, the integration line will be converted to terminal and reference under circuit port and vice versa.

Addition, deletion, modification of circuit ports will invalidate solutions. Editing the impedance will invalidate. Changing or deleting the assignment will invalidate.

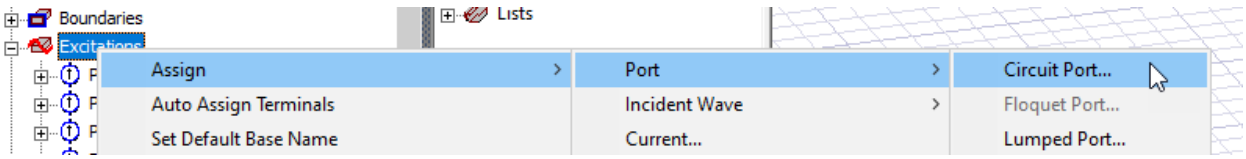
To assign a Circuit Port:

1. Select two edges in the modeler window. The order of selection is significant. For a driven modal design, an integration line will go from the second edge to the first edge. For a driven terminal design, the first edge will be assigned a terminal, and the second edge will be assigned a reference.

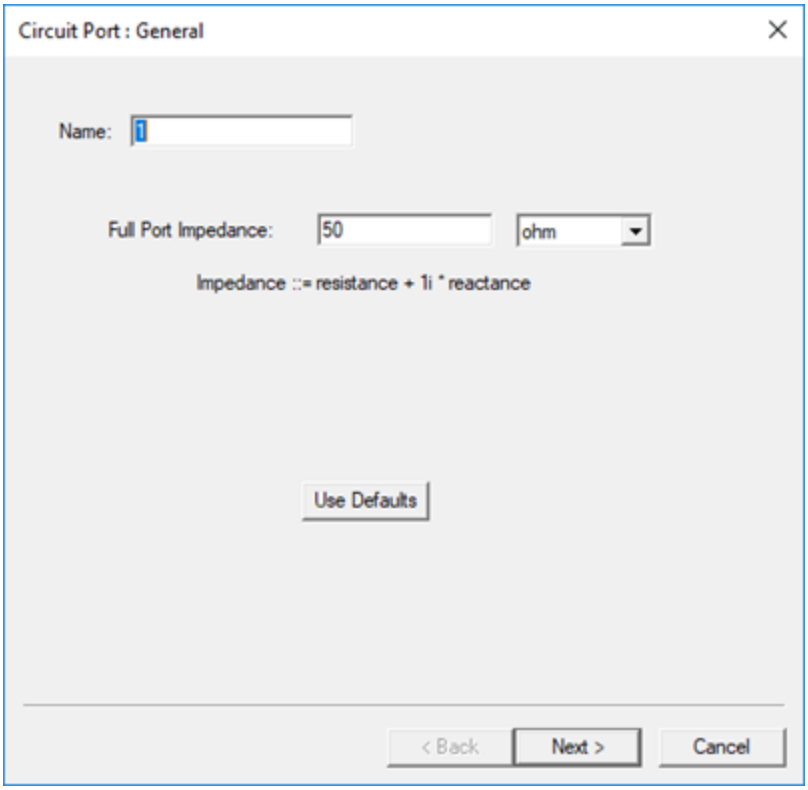


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Right-click the **Excitations** Icon in the Project Manager and select **Assign> Port>Circuit Port...** You can also use the HFSS toolbar menu to select **Excitations> Assign>Port> Circuit Port...**, or right-click in the Modeler window and click **Assign Excitation>Port> Circuit Port...**.

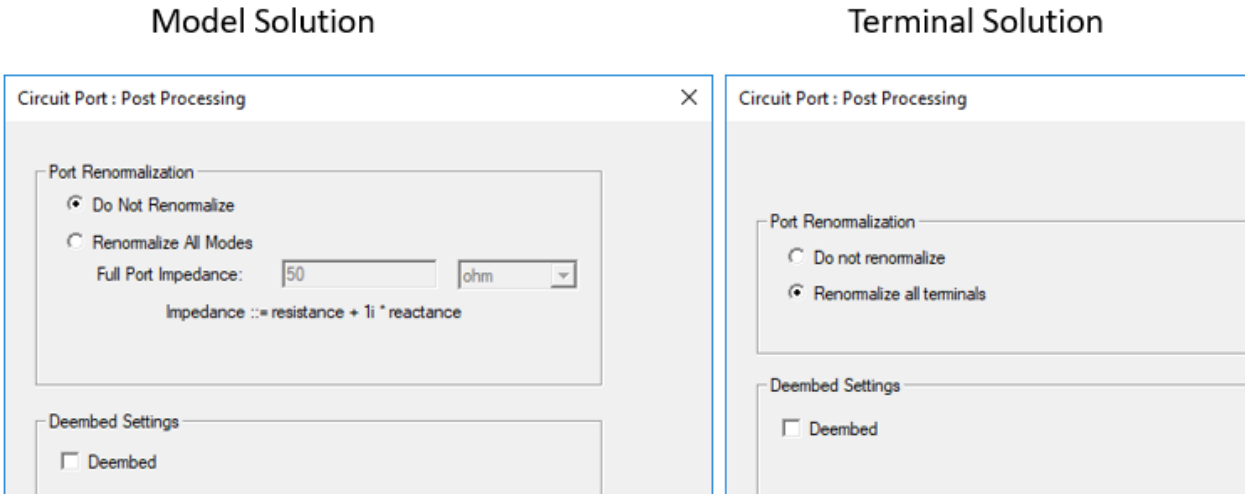


The **Circuit Port: General** dialog box appears.



3. You can set the Full Port Impedance. Click Next to view the Port Processing tab. These are slightly different for Modal and Terminal solutions.

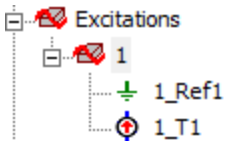
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For Modal solutions, the default is *Do Not Normalize*, but you can choose *Renormalize All Modes* and specify the *Full Port Impedance*. For Terminal solutions, the default is *Renormalize all terminals*. You can also enable *Deembed*.

4. Click Finish.

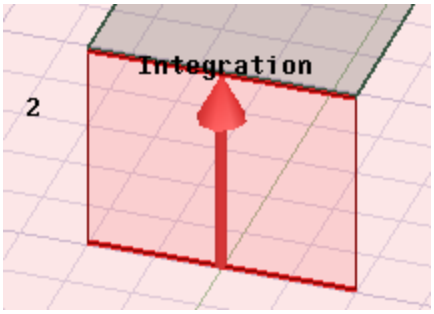
   For Terminal solutions, the History Tree will show the Circuit Port with Terminal and Reference. Selecting the Circuit Port highlights both edges in the Modeler window. Selecting the Terminal or the Reference highlights the associated edge. Terminal is assigned to the first edge selected, and the reference is assigned to the second edge selected. The renormalization impedance belongs to the terminals. Terminal and reference from the circuit port cannot be added to or deleted from the circuit port; exactly one terminal and one reference is required by the circuit port. However, you can reassign terminal and reference independently. Reassigning a circuit port means that its assignment is then populated into the terminal reference.



   For Modal solutions, selecting the Circuit port in the Project Manager highlights both edges and shows the integration line from the second edge selected to the first edge

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

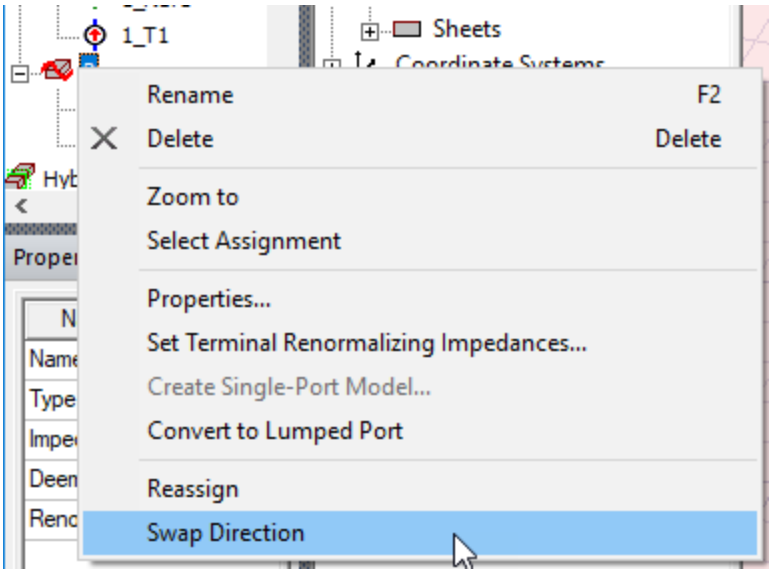
selected.



When you create circuit ports, you can do the following:

Swap Assignment

Right click on the Circuit Port and click **Swap Assignment** to swap the direction of an integration line or a terminal reference assignment.



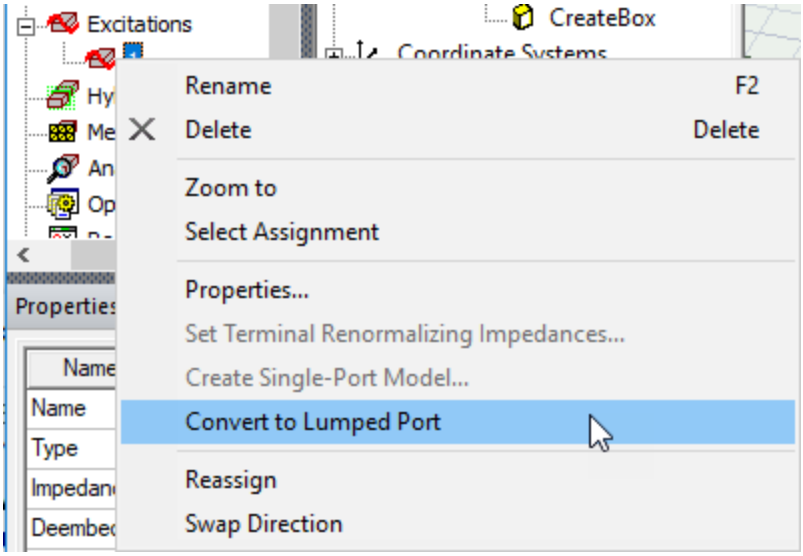
Create a Circuit Element from a Circuit Port

See Circuit Elements in HFSS.

Convert a Circuit Port to a Lumped Port

Select a circuit port, right-click and select **Convert to Lumped Port** from the short-cut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For driven modal solutions:

- If the two edges are on the same face, the lumped port will be assigned to that face, and the integration line is preserved.
- If the two edges are not on the same face but on the same plane, then a sheet will be created, and the lumped port is assigned to that sheet.
- Otherwise, the conversion fails.

For terminal solutions:

- The terminal is transferred to the lumped port.
- The lumped port face determination is similar to driven modal using the edges from the terminal and the reference.

Convert a Lumped Port to a Circuit Port

Select a lumped port, right click and select **Convert to Lumped Port.**

For driven modal solutions:

- The integration line is preserved
- If the integration line is defined by two end points on edges, the two edges are the assignment of the circuit port.
- Otherwise, the conversion fails.

For driven terminal solutions:

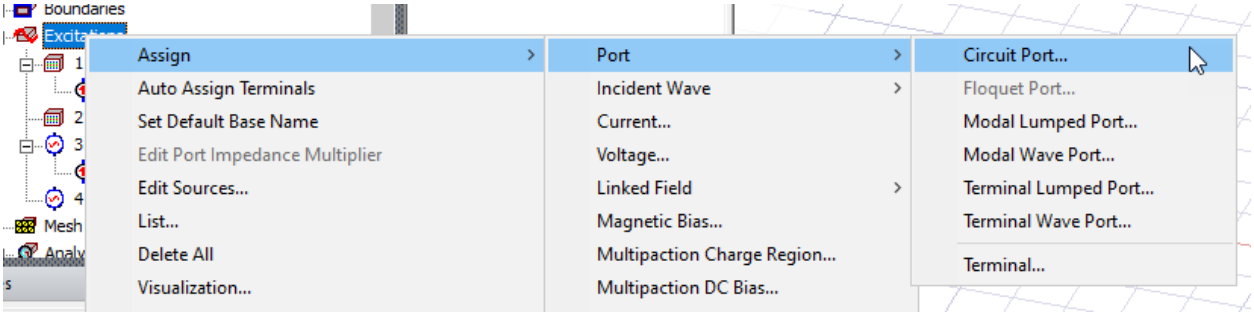
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The terminal is transferred to the circuit port.

  The reference edge is determined by the terminal assignment and the lumped port face assignment. An edge touching a conductor from the face, and not touching the terminal will be the reference edge.

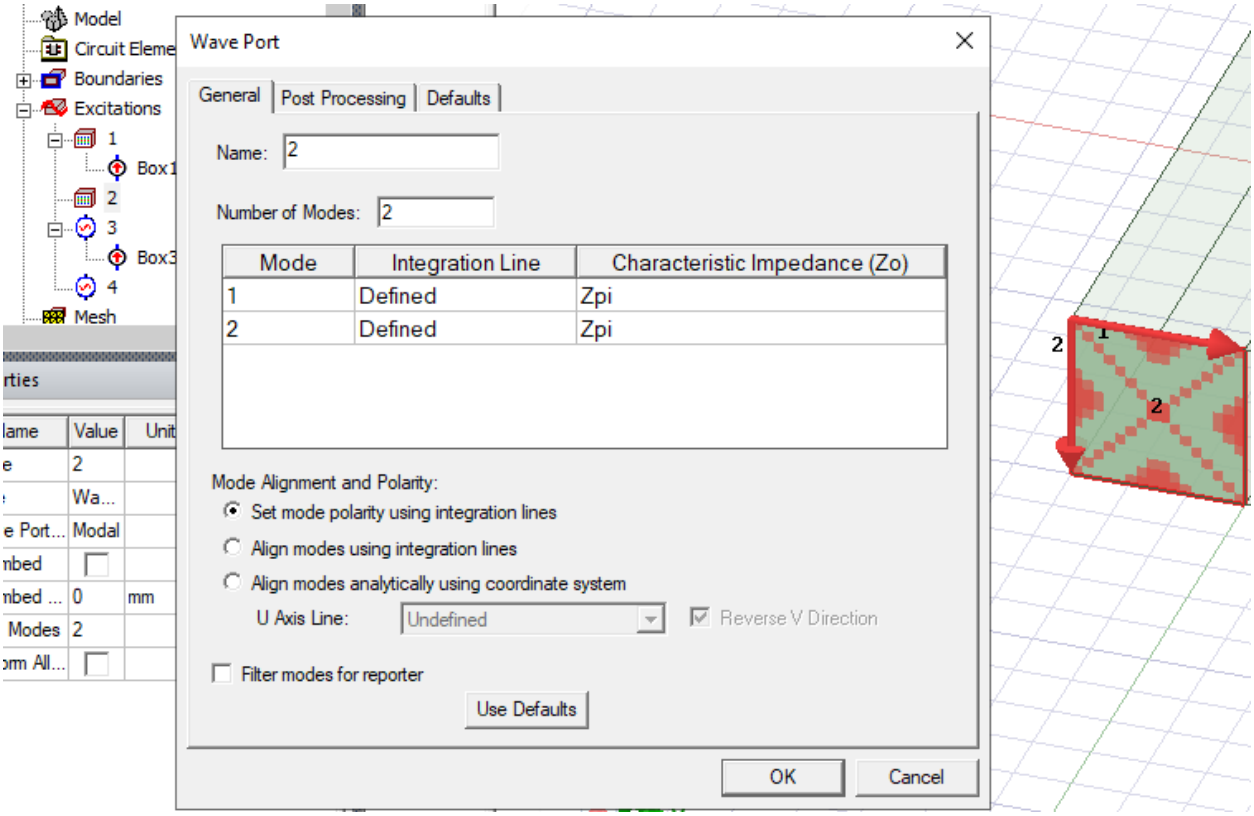
Multimode Simulations in HFSS Terminal Solutions

This release allows you to mix modal and terminal ports in an HFSS Terminal Solution. You can add Modal Wave Ports, Modal Lumped Ports, Terminal Wave Ports, and Terminal Lumped Ports to a single simulation. This is particularly helpful for component assembly where one component cones from modal design, and another from terminal design. You can now assemble them in a single design without having to manually edit component definition as the same solution type. The Excitations menu shows these ports as available.



For mixed modal/terminal in HFSS Terminal simulation, the modal Wave Port dialogs are the same as before.

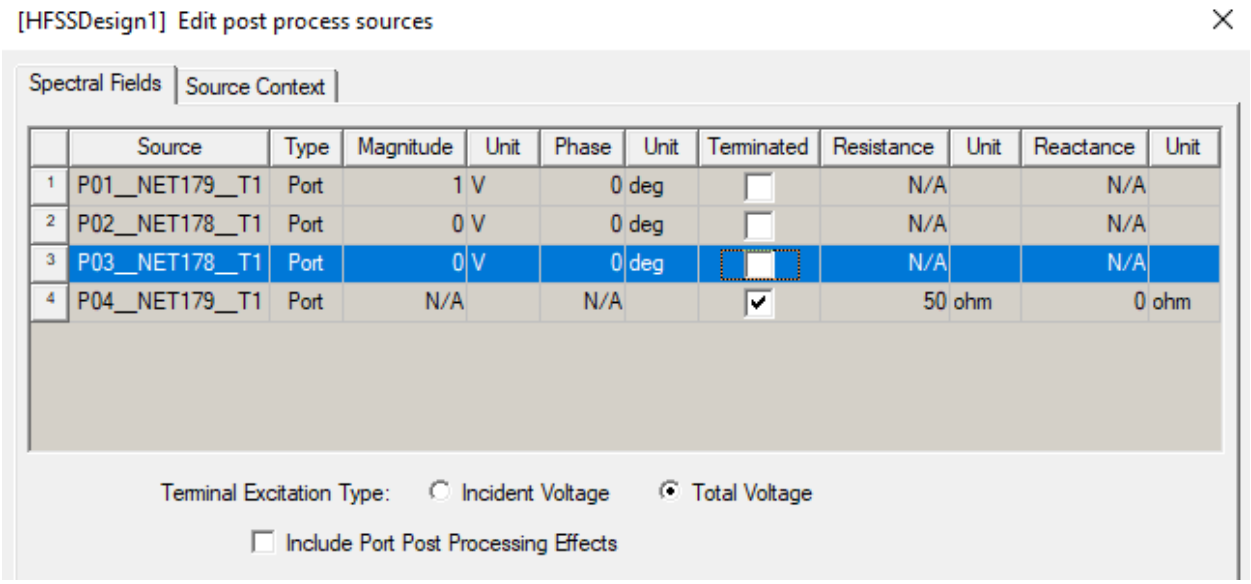
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



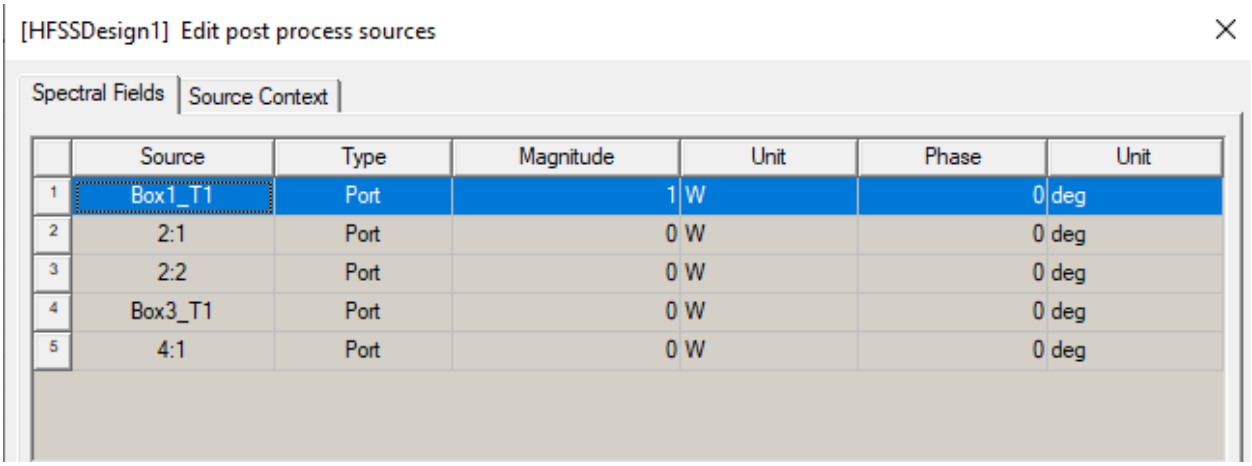
You can mix each kind of wave and terminal ports with the others.

For an HFSS Terminal simulation with only terminal ports, the transient results in reporter are in Volts units, and you edit sources is in Volts units.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For an HFSS Terminal simulation with all modal or mixed modal/terminal, the results in are in Watt units.

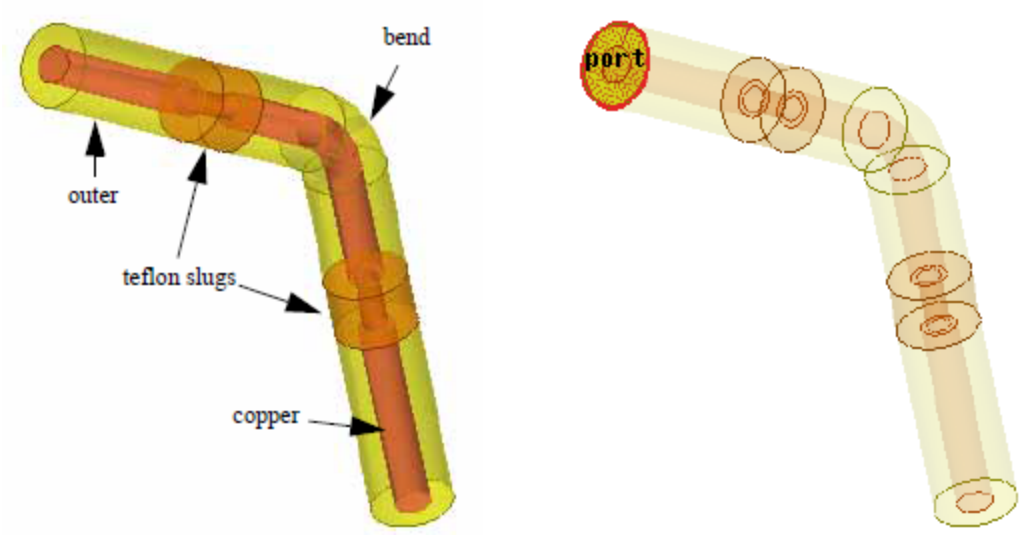


Consequently, for all modal, and mixed modal/terminal simulations, you edit sources in Watt units.

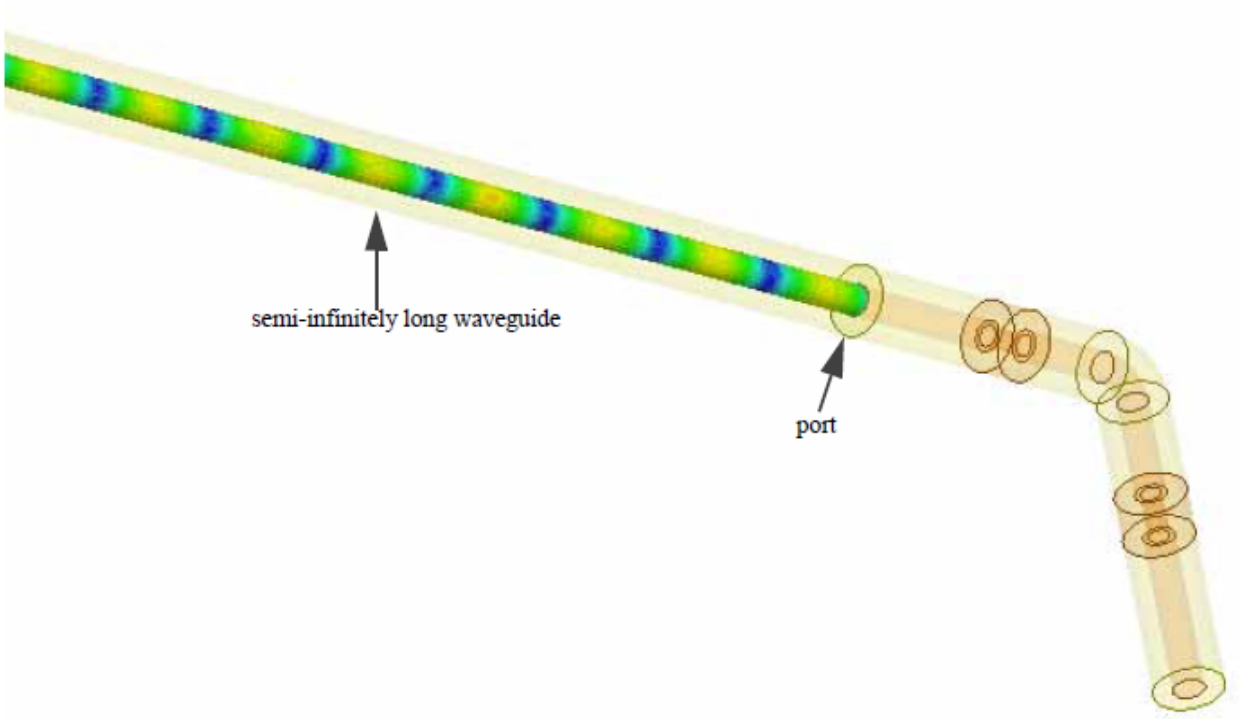
Wave Ports

The objective of this section is to provide a thorough description of wave ports. You must know what a wave port represents in HFSS to understand its capability. To illustrate we will use an HFSS model of a coaxial bend as an example where the wave port is assigned on the outer face.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



HFSS treats this wave port as though a waveguide or a transmission line of the exact same cross-section (in this case the cross-section of the coaxial bend) and material properties, comes from infinity and ends at the port as shown in the following figure.



The properties and the cross-section of the waveguide or the transmission line determine the natural field patterns called modes that excite the model and the HFSS port solver determines the propagating modes that the waveguide or transmission line will carry.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

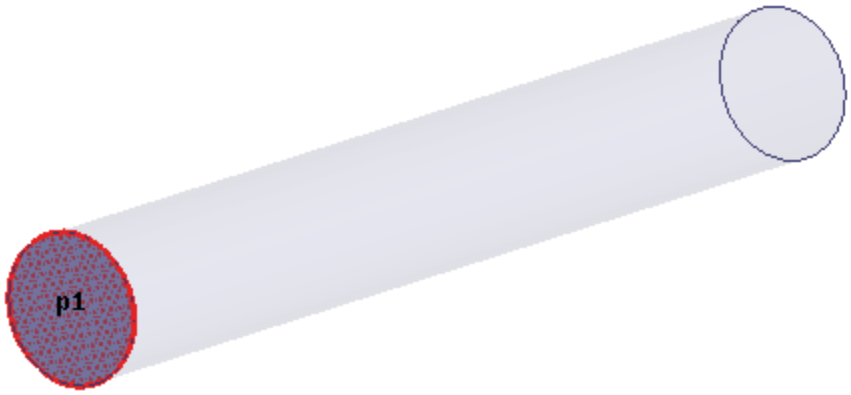
> **Note:**
>
> A wave port can be placed internal to a model as long as it is backed by a PEC object.

Wave Port Size

Wave ports are used for exciting transmission lines and waveguide structures. Wave ports that are assigned on waveguide structures are naturally defined by the cross-section of the waveguides. However, for transmission lines (e.g., microstrip, CPW, slotline) ports should be defined carefully. Sometimes transmission lines are part of a large PCB structure which makes it your responsibility to define the port size properly. This section provides guidelines for the appropriate port sizes for transmission lines. Examples of different waveguide and transmission line structures are as follows:

Circular Waveguide

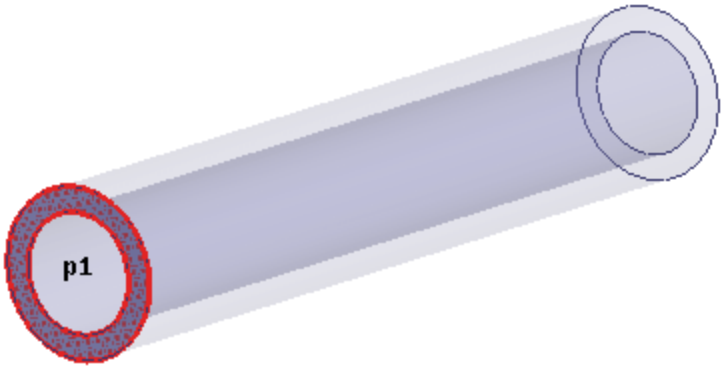
An HFSS model of a cylindrical waveguide of uniform circular cross-section along its length excited with a wave port *p1* is shown below. You do not need to define the port size because it is naturally defined by the cross-section.



Coaxial Cable

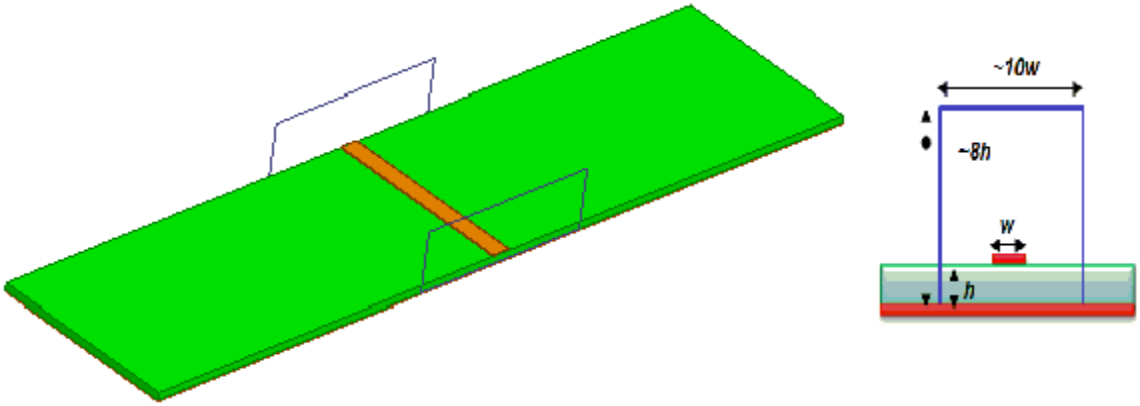
The following figure represents a coaxial cable. You do not need to define the port size as it is naturally determined by the inner and outer radii of the shield.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

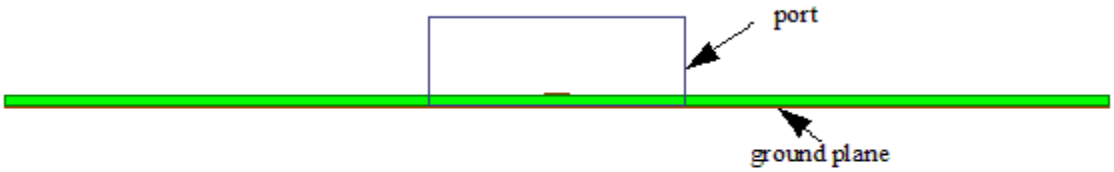


Microstrip Transmission Line

The figures below show an HFSS model of a microstrip transmission line and guidelines for setting the wave port dimensions.



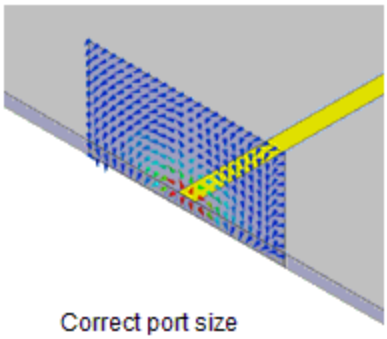
See the HFSS model at scale in the following figure. The bottom of the port touches the ground plane of the microstrip.



The port width affects the port impedance and the propagating modes. If the defined port is too narrow more fields will couple to the side walls. The height of the port is affected by the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

permittivity of the substrate. If the permittivity is too high less fields will propagate in the air, so the wave port can be made shorter.

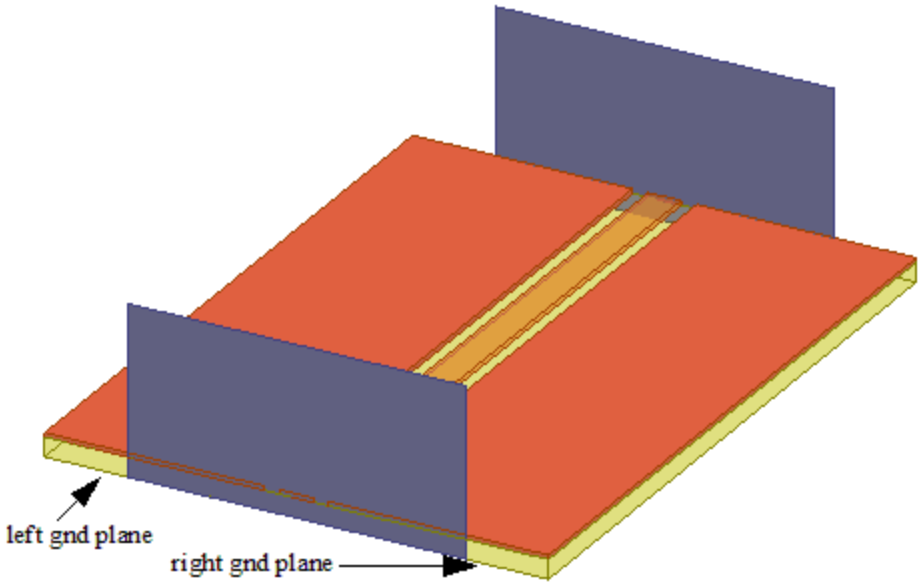


Correct port size

Port too narrow
(fields coupled to sidewalls)

Coplanar Transmission Line

The following figure shows an HFSS model of a coplanar transmission line. The left and right edges of each port must touch the left and right ground planes. We recommend that you make the port size 8h x 10w where "w" represents the width of the trace and "h" represents the height of the substrate.

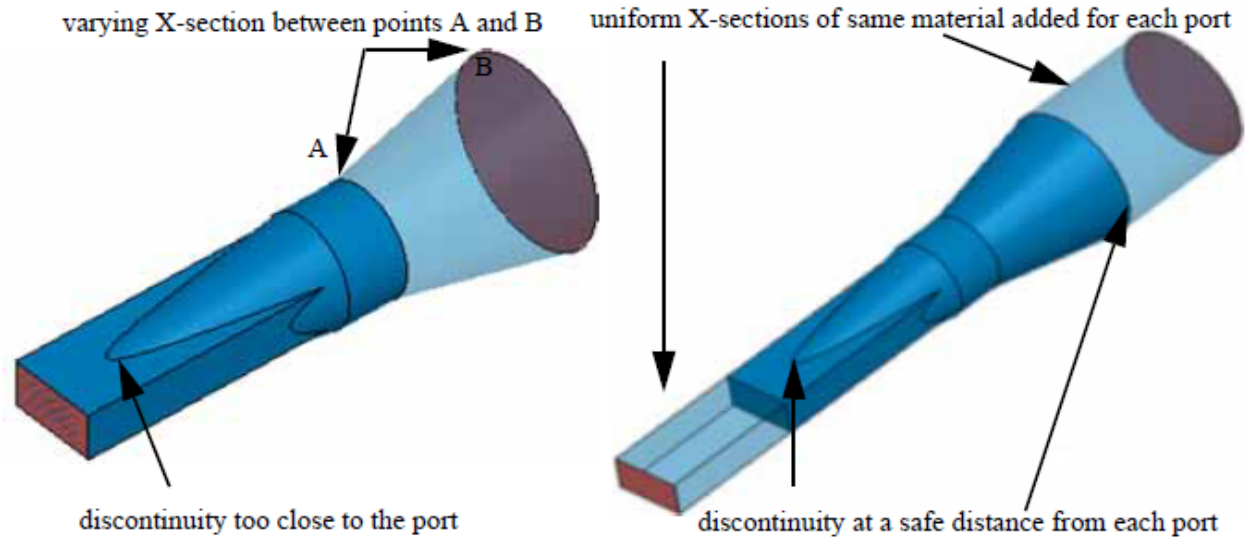


Wave Port Placement

HFSS treats each port that you define to be connected to a semi-infinitely long waveguide or transmission line that has the same cross-section and properties as the wave port. The placement of the wave port is critical because it can affect the accuracy of the solutions. If the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

wave port is close to a discontinuity due to change in shape, dimension, or material of a structure the resulting 3D field can be a superposition of propagating and non-propagating modes.



Hence, the first discontinuity should be at an appropriate distance away from the port surface. This allows any energy reflected into an evanescent mode of the port to decay before reaching the port. Without this separation the resulting accuracy of the S-parameters will be compromised. To precisely determine a distance you can solve the 'port only' with one additional mode (for a correct port definition this mode will be evanescent) and extract the decay length of this mode from its complex gamma. Then, as a rule of thumb, place the first discontinuity at least three times this decay length. Such rigor is not usually necessary and experience over time will give you a solid understanding of typical spacings for their port geometries and frequencies of analysis.

Number of Modes

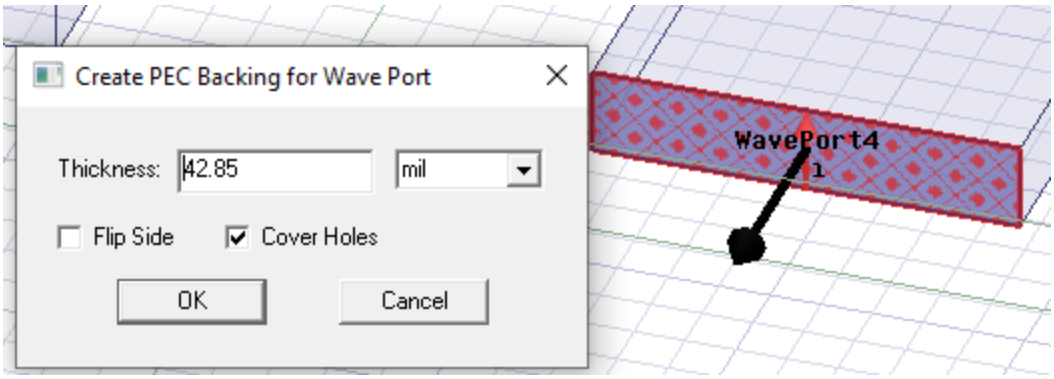
The electrical size of a port at the highest frequency of interest determines the number of modes to be included in the port definitions. If the guidelines on discontinuity discussed above are followed, the number of modes equals the number of propagating modes at the highest frequency; if they are not followed then, the port as a boundary condition will not be accurate. Regardless of whether they are propagating or not, the solver will solve these modes for calculating the S-parameters.

Automatic PEC Creation on Wave Ports

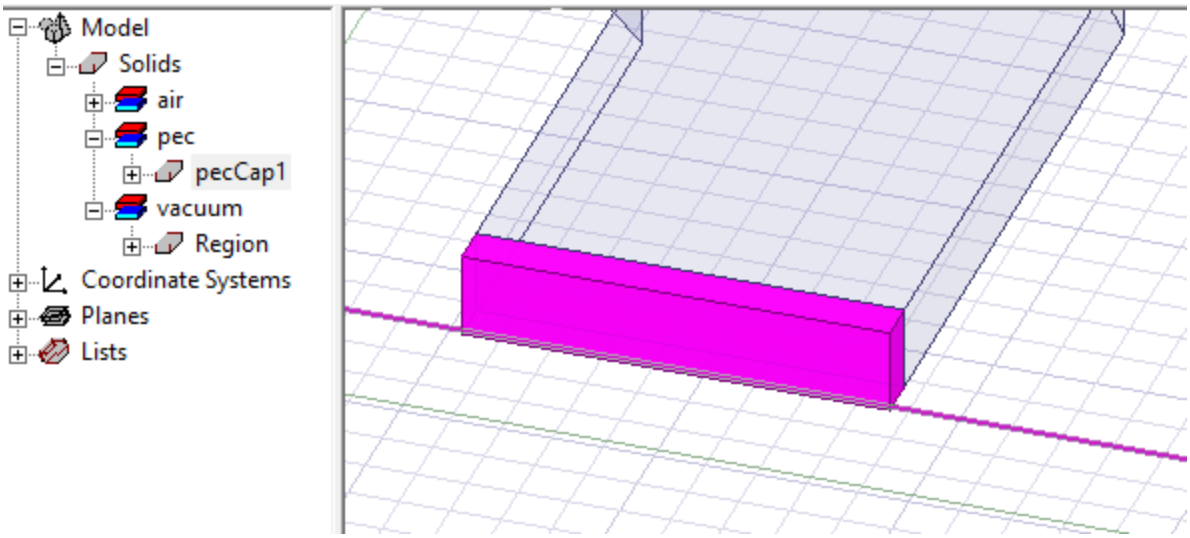
In HFSS, A wave port must have 'solve inside' objects on one side only. An internal wave port would require a PEC object to be attached to and cover the opposite side before the solution can proceed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This HFSS Boundary Assignment Option automatically adds a PEC backing to every newly created wave port when necessary. When a new wave port is created and requires a PEC backing, a dialog window opens to let users create such backing by extruding the port face with specified thickness. You have the option to flip the side of the PEC backing. If the wave port has holes, you have the option to create backing that covers the holes. You can also set the thickness as a variable.



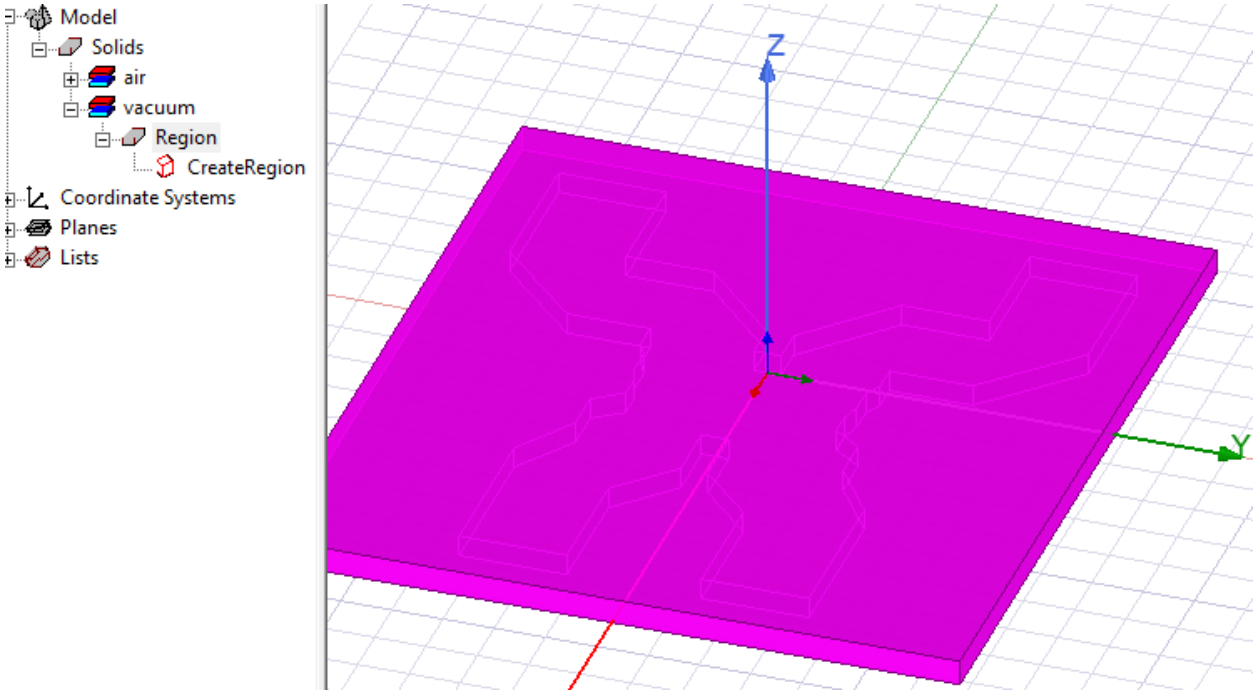
After clicking OK, the PEC backing is created and the new PEC object appears in the History tree.



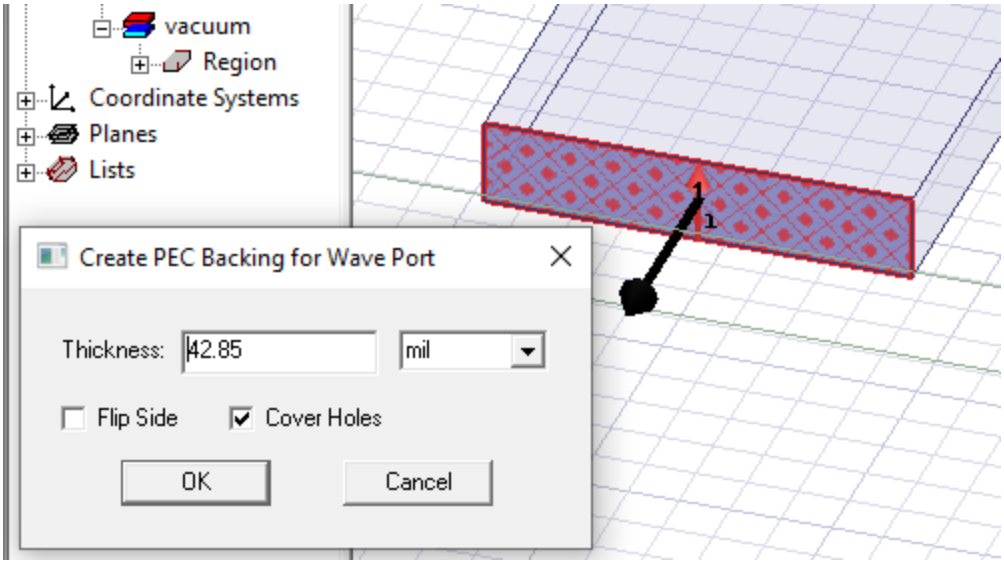
Example Using the Waveguide Combiner Example

Open the HFSS example project “wg_combiner.aedt”. Add a region with padding around the design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

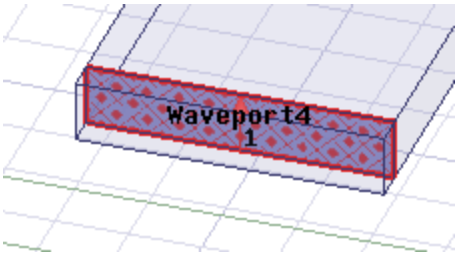


This will make all the wave ports internal to the design and invalid. Now delete one of the wave ports and create a new one on the same location, you will see this dialog after it is created.

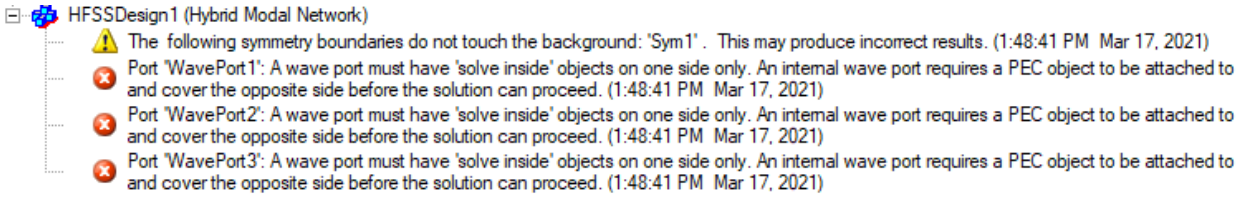


You can edit the Thickness and units. The black arrow on the selected wave port shows the side on which the backing is applied. If necessary, you can check Flip side to change the arrow direction. In this example, this is the correct side, so leave that unchecked. Click OK to see the PEC backing.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Validation will show that the other four wave ports are invalid and require PEC backing.

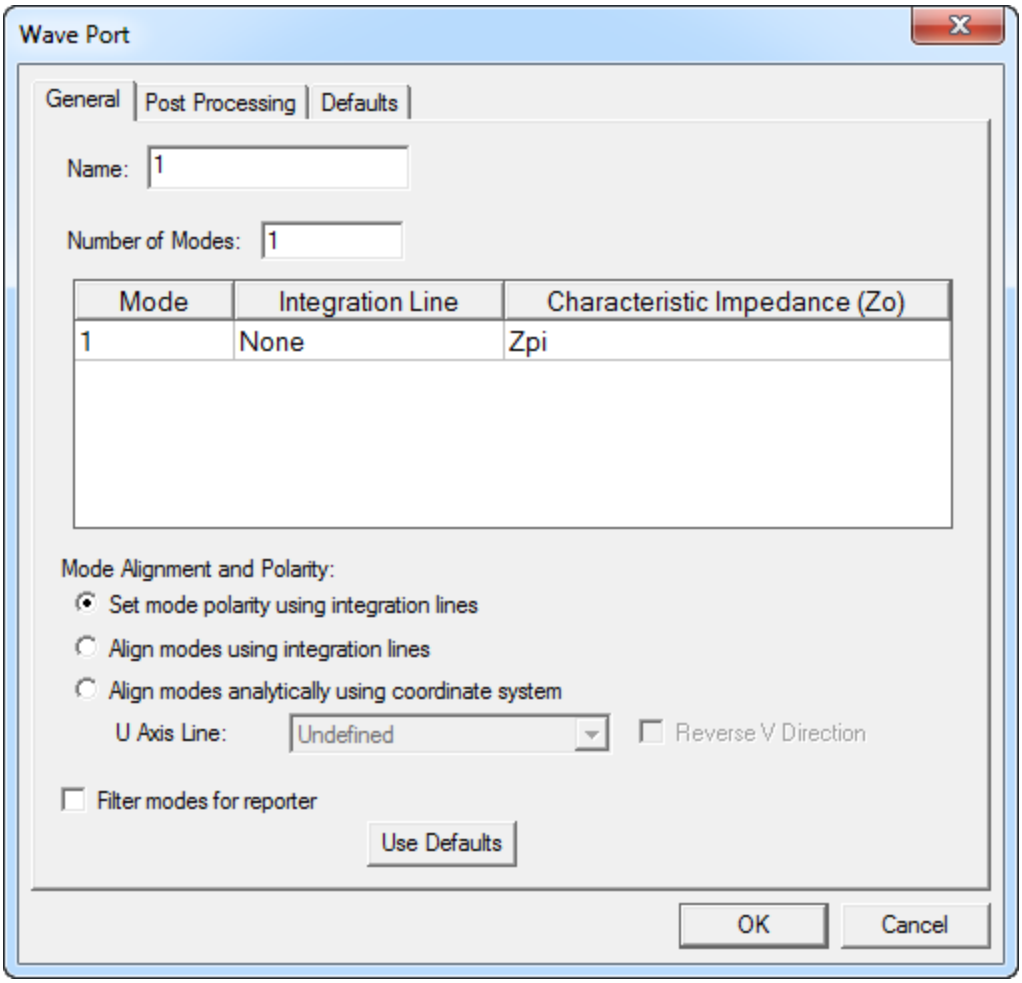


Wave Port Dialog for Modal Solutions

A **Wave Port** dialog box with its default settings is shown below. On this dialog box you can set the number of modes, their alignment, mode polarity, the choice of characteristic impedance etc.

> **Note:**
>
> The option **Use Wizards for data input when creating new boundaries** under **Tools** > **Options** > **HFSS Options** >**Boundary Assignment** is unchecked; so the dialog box appears in the tabbed format. For more information about the appearance of the **Wave Port** dialog box, see **HFSS Options**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



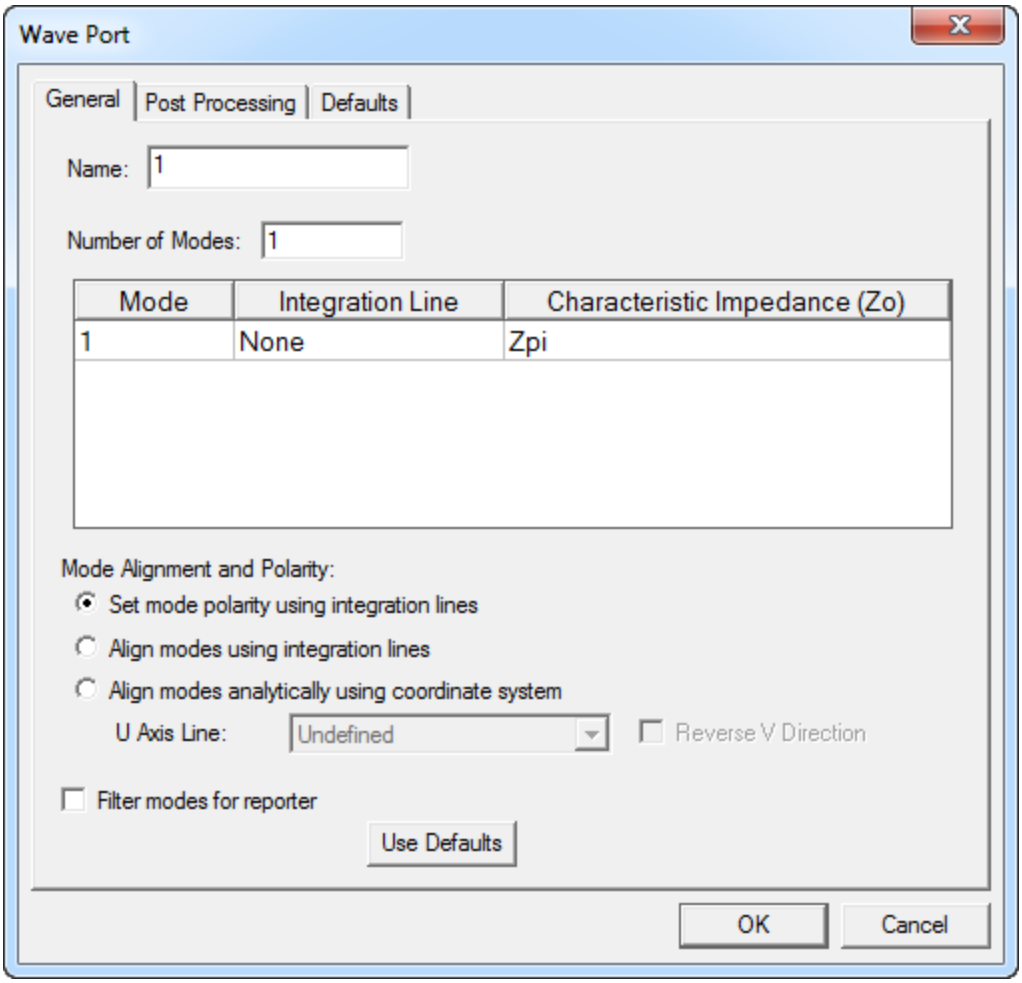
The different tabs and their sub-panels are described in the following subsections.

- General Tab
- Post Processing
- Defaults

General Tab: Wave Ports

This panel contains a field where you can enter the name of the wave port, as well as the number of modes, set integration lines, and define mode alignment, polarity, and characteristic impedance. Subsequent sections describe all these options and the effect they have on the port field.

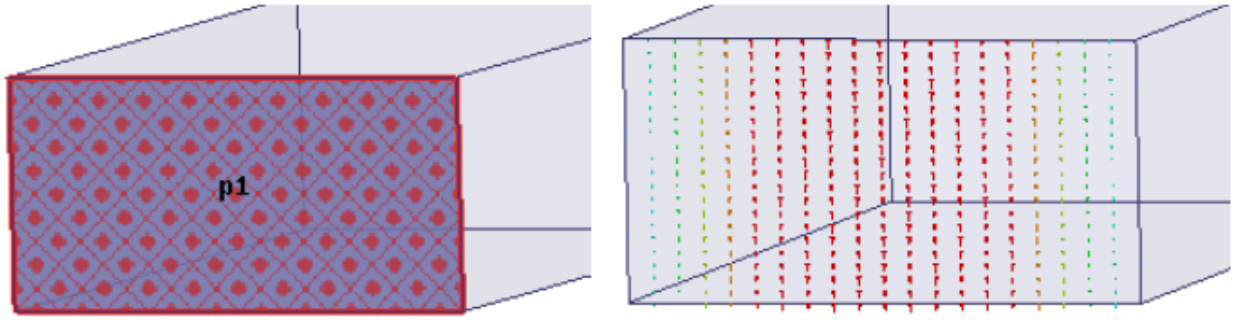
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

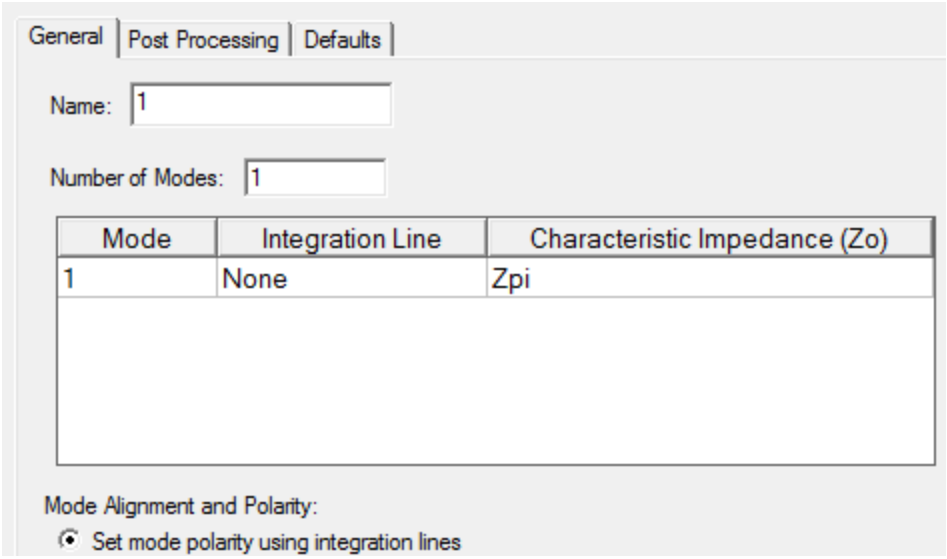


Modes with Default Settings

The following figure shows a wave port assigned on the face of a rectangular waveguide. The default settings in the **General** tab of the **Wave Port** dialog box are used and the port is named *p1.* No integration line has been defined. From the **Project Tree** if you select the option **Mode1** under **Port Field Display** you can see the electric field patterns for that mode. The direction of mode propagation for the corresponding wave port is shown below. Notice that the intensity is highest in the middle (indicated by the red arrows) and tapers to zero on the sides for the TE01 mode for the rectangular waveguide. Since no integration line has been defined, the polarity is arbitrarily up or down.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

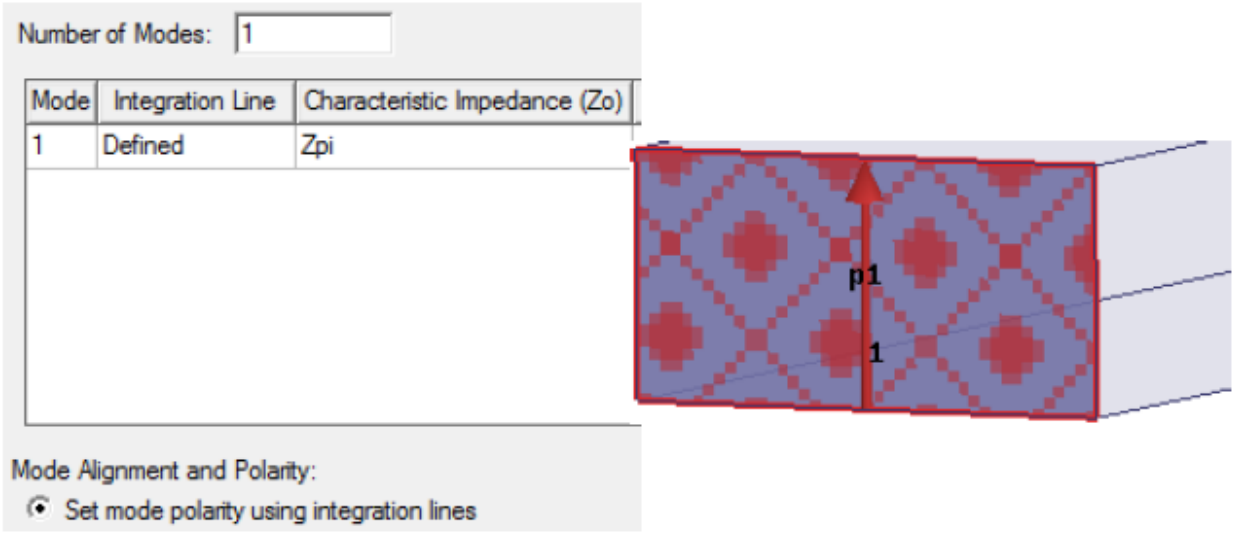




Set Mode Polarity Using Integration Line

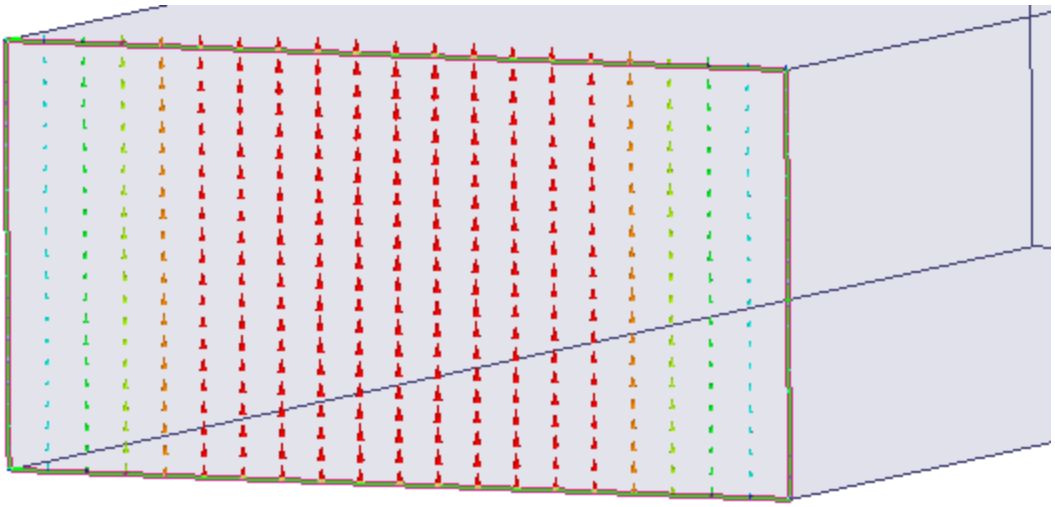
By drawing integration lines you can set mode polarity. The settings in the **General** tab of the **Wave Port** dialog box for a single mode and the integration line drawn for a WR-90 rectangular waveguide are shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



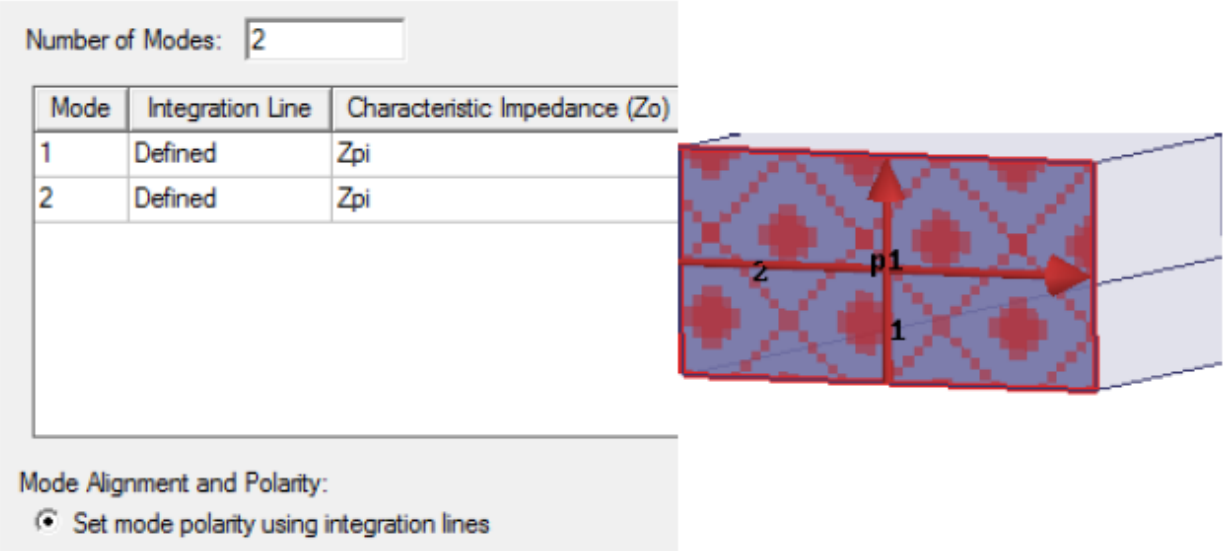
The mode alignment can also be set using the integration lines. In this case the direction is flipped upward as set by the direction of the integration line.

.

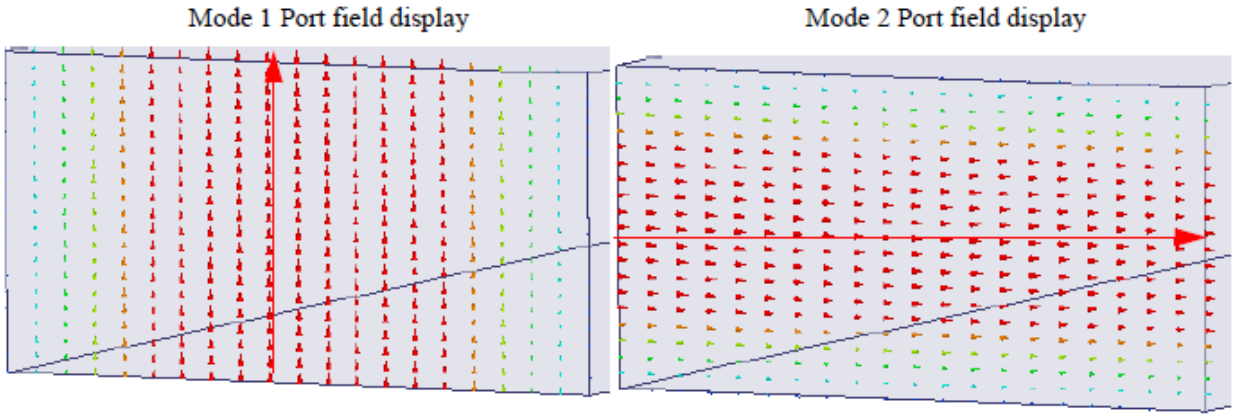


See the settings for another WR-90 waveguide with the integration lines 1 and 2 defined for two modes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The corresponding port field display are as follows:



Note:

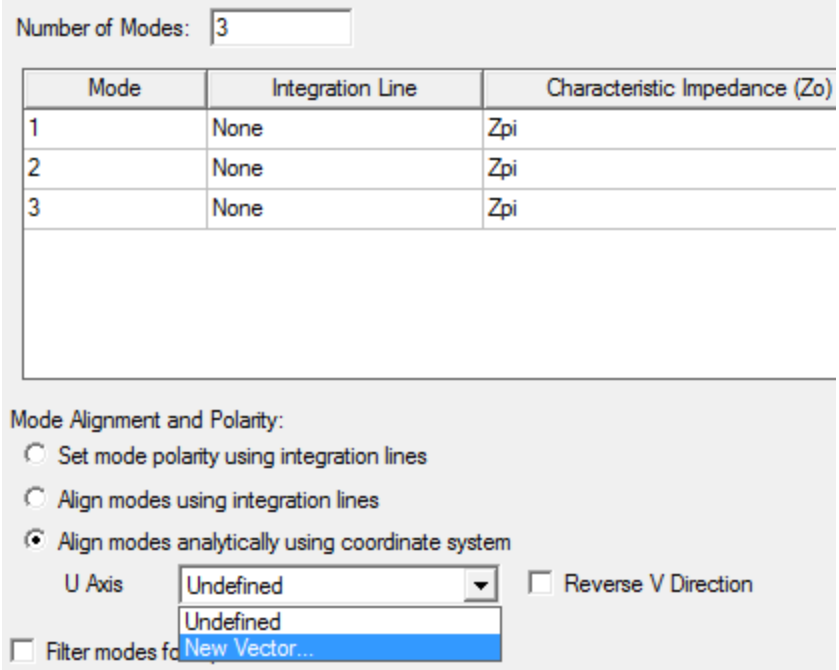
The voltage along integration line 1 is positive for Mode 1 and the voltage along the integration line 2 is positive for Mode 2.

Align Modes Analytically Using Coordinate System

This option is used to align modes for standard waveguides such as square waveguides, rectangular waveguides, coaxial waveguides etc. (For a complete list, see the section Analytic Port Types). On the **General** tab of the **Wave Port** dialog box, select the last radio button and **New Vector** from the drop-down menu. Draw the U-V axes to determine the mode alignment.

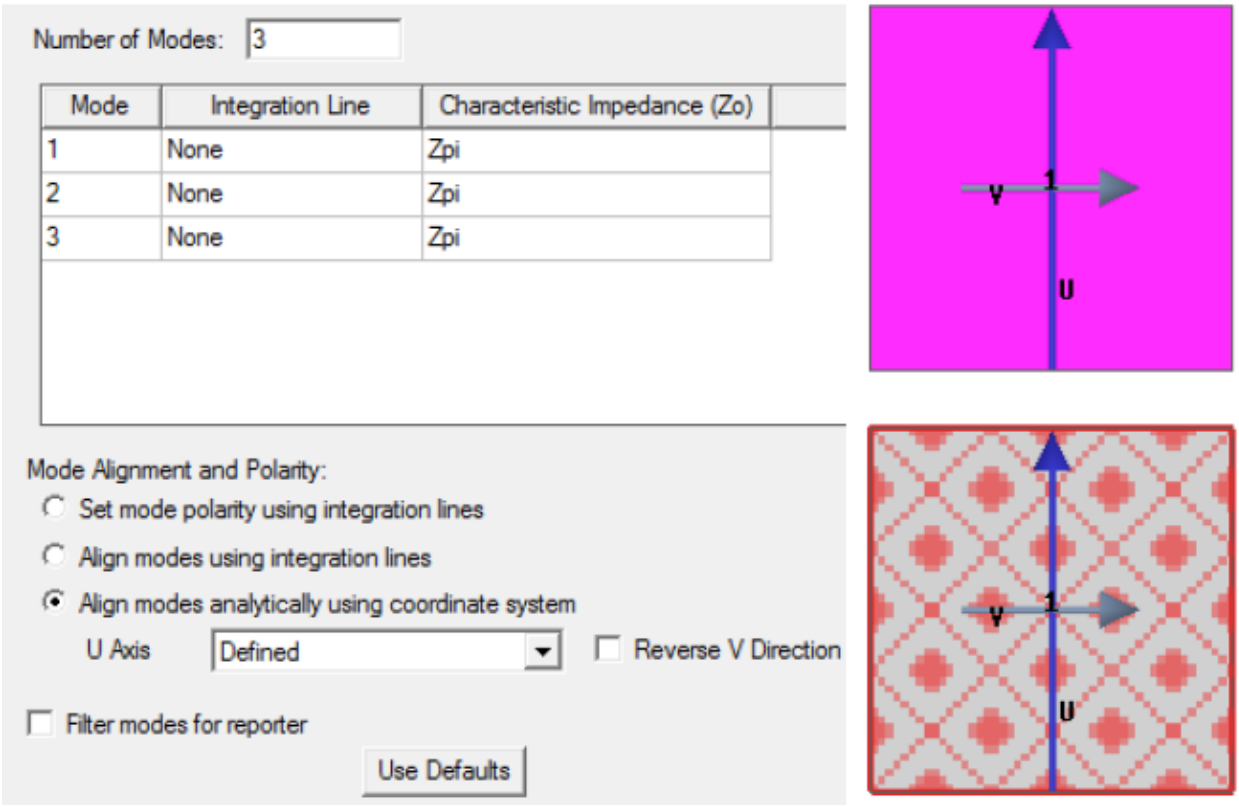
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

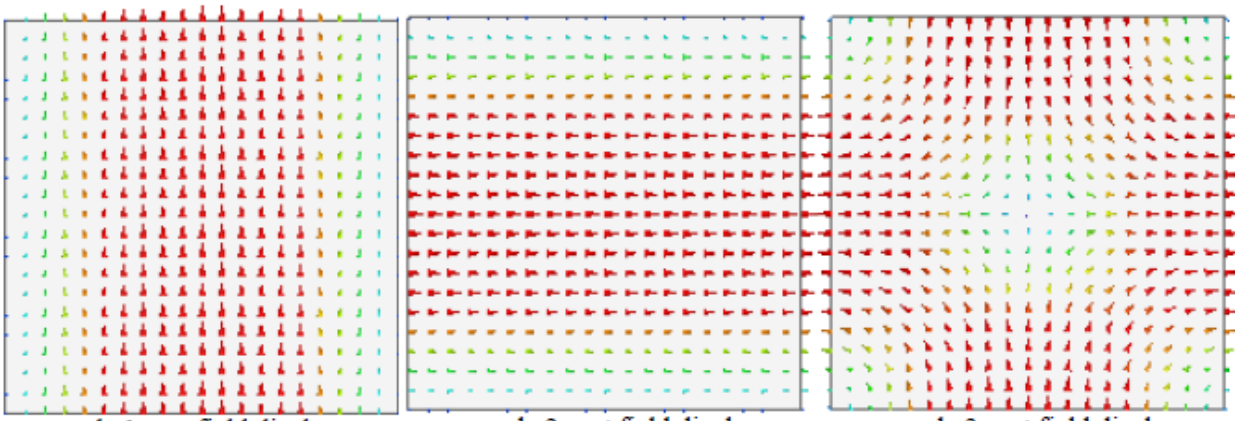
The U axis must lie on the center of the port plane and split the port into equal haves. Otherwise an error message will pop up.



The figures below show the cross-section of a square wave guide for which the U-V axes are set and no integration lines defined for 3 modes. The U-axis must lie along the port plane and split the port into two equal halves. After you define the **Solution Setup** and run the simulation, the **Port Field Display** for the 3 modes are also shown in the figures below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





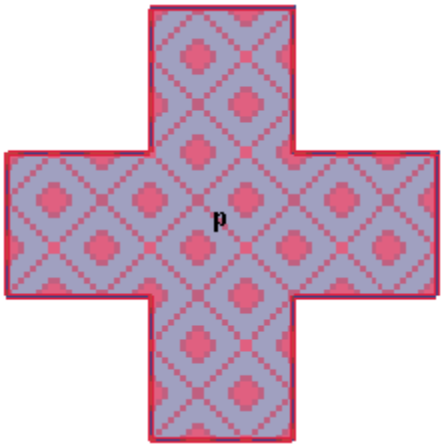
mode 1 port field display | mode 2 port field display | mode 3 port field display

Align Modes Using Integration Lines

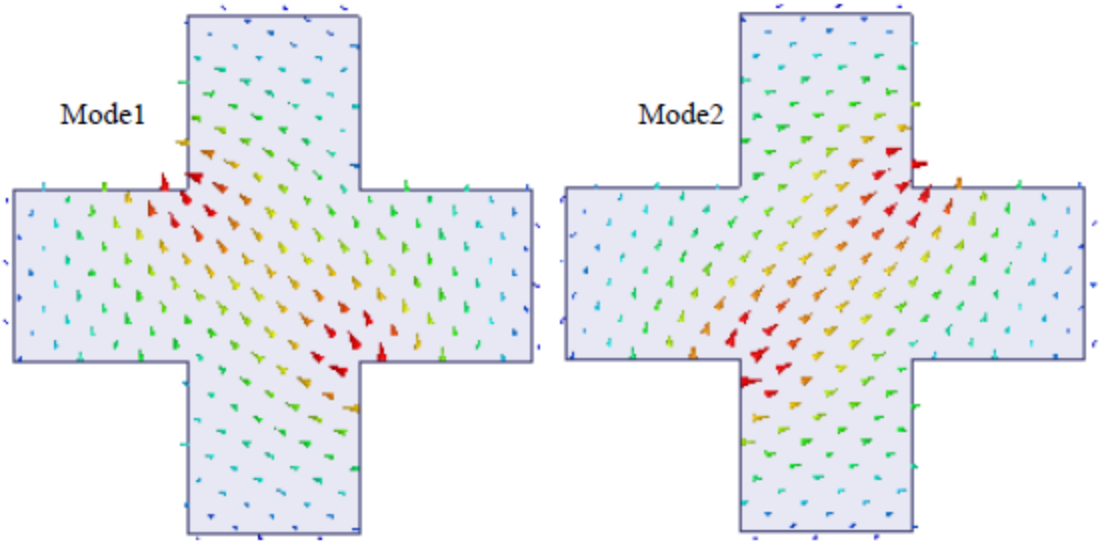
This option is for advanced users working with non-standard waveguides which may have degenerate modes. First we will investigate the fields for a wave port assigned on the surface of a symmetric cross-waveguide (shown below) using the default settings on the **General** tab of

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the **Wave Port** dialog box. (**Set mode polarity using integration lines**) for 2 modes. No integration line is defined.

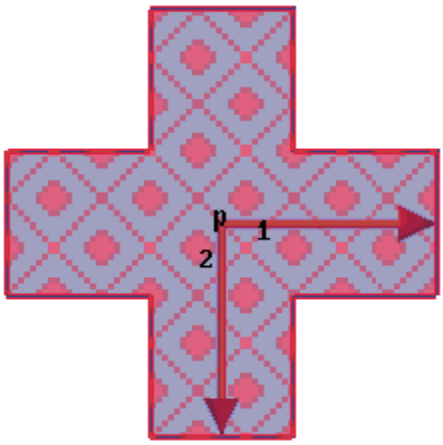


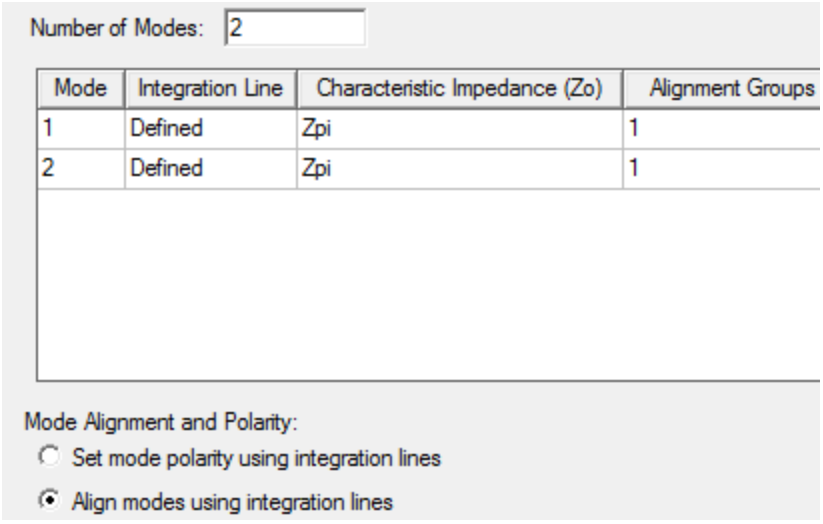
The solver arbitrarily assigns the polarity for the modes 1 and 2.



Any combination of degenerate modes results in just another combination of degenerate modes. There is no uniqueness in the orientation of the modes. To get a unique orientation advanced users can select this option **Align modes using integration lines** and define the integration lines appropriately. Prior knowledge of the mode pattern is necessary before you define the Integration Lines for mode alignment. You cannot draw the integration lines randomly. They are extremely important to get the mode alignment that you want. See the integration lines 1 and 2 defined on a port face shown below and the settings in the **Modes** sub-panel.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

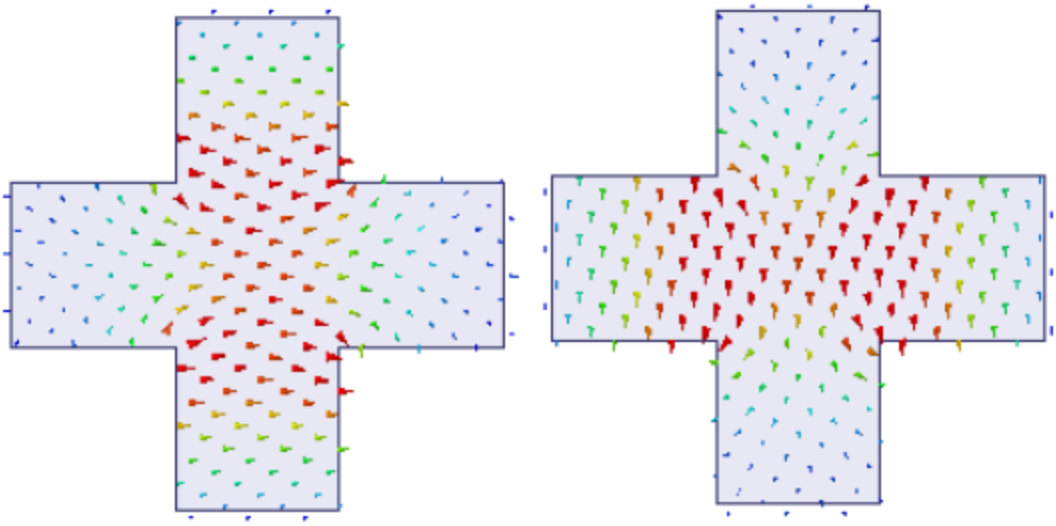




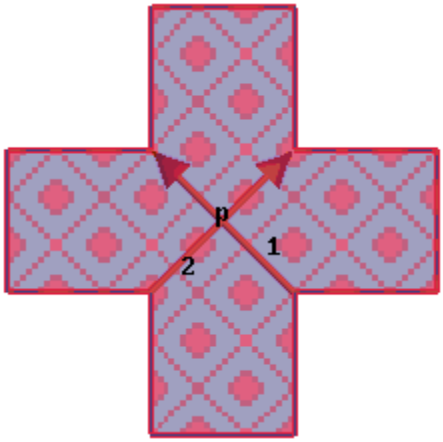
Alignment Groups

This column appears only when you select the option **Align modes using integration lines**. The modes that have the same **Alignment Group** number are degenerate. To help the solver choose which modes are degenerate you must assign the **Alignment Groups.** When you set the alignment group numbers the solver accumulates the modes of a specific group number and causes each mode within that group to have a positive voltage along its own integration line and zero voltage along all other integration lines. For example, defining integration lines 1 and 2 at right angles to each other for the port face (shown above) will cause the modes to be aligned as illustrated in the following figures.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

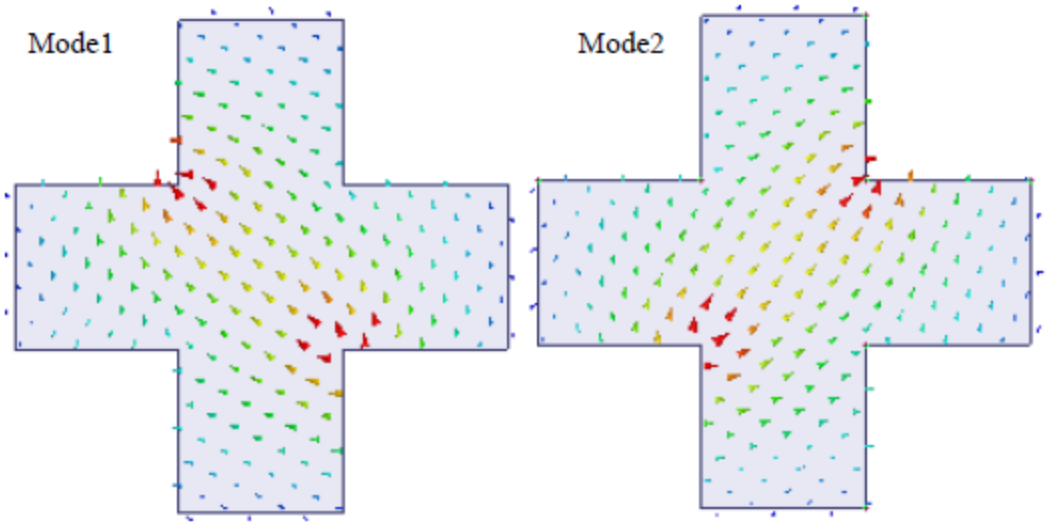


Suppose you want the modes to be aligned along the integration lines 1 and 2 defined diagonally as shown in the following figure.



According to this definition of the integration lines, the solver aligns the modes within a given group (1 in this case) such that Mode *n* has a positive voltage along integration line *n* and zero voltage along all other integration lines.

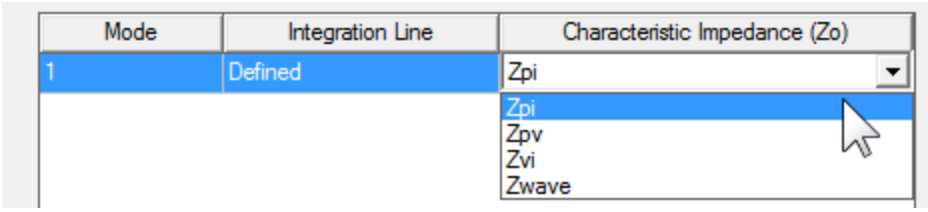
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> For more information, see Mode Alignment.

Characteristic Impedance Column

The **Characteristic Impedance Column (Zo)** lists four options (**Zpi, Zpv**, **Zvi, and Zwave)** in its drop-down menu. The options **Zpv**and **Zvi** will appear only if you provide a voltage through defining an integration line for a specific mode such that a unique voltage for that mode can be computed. These quantities **Zpi, Zpv**, and **Zvi** need not be equal to each other. Other than TEM modes, these three quantities will give different results since the voltage is not unique and the result depends on the path used to compute the voltage. Zwave is strictly only applicable for homogeneous waveguides but we use this equation for any ports which should be OK for weakly inhomogeneous waveguides as well.



In case you do not define an integration line only **Zpi** is shown.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mode	Integration Line	Characteristic Impedance (Zo)
1	None	Zpi
2	None	Zpi

Note:

For more information, see calculate the characteristic impedance.

Electromagnetic power **P** = **E** x **H** (with magnitude of **P**) is injected into the 3D model to calculate the fields, voltages, and currents. The basic equations for characteristic impedance are as follows:

$$Zo = \frac{V}{I} \quad ; \quad P = V \cdot I$$

The expressions for characteristic impedance in terms of power vary as follows:

When using **Zpi:**

$$Zpi = \frac{P}{I^2}$$

When using **Zpv:**

$$Zpv = \frac{V^2}{P}$$

For **Zvi** the following expression holds

$$Zvi = \sqrt{Zpi \cdot Zpv}$$

For **Zwave**:

$$Zwave = \int \frac{(nxE \bullet nxE)}{P}$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

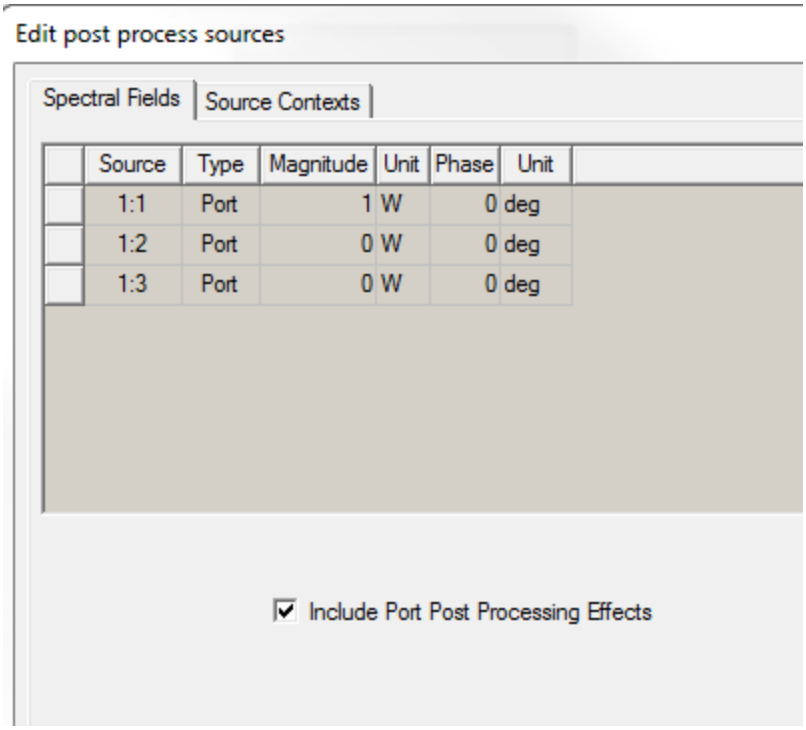
For more information about how HFSS defines these values, see Calculating the Z_{PI} Impedance, and Calculating the Z_{PV} Impedance.

Port Post Processing Tab: Modal Solutions

De-embedding and renormalization are the two post processing options available in HFSS. De-embedding changes the reference plane where the port is placed rather than literally defining a new port on a different reference plane. Reference plane is the location where the S-parameters are calculated. De-embedding helps to calculate the S-parameters near or on the plane of a discontinuity or when a long transmission line is attached to the port plane instead of explicitly modeling it in HFSS.

Renormalization changes the load impedance of the port.

Both renormalization and deembedding affect the network parameters. They will also affect the fields if the **Include Port Post Processing Effects** option is selected in the **Edit Sources** dialog box.

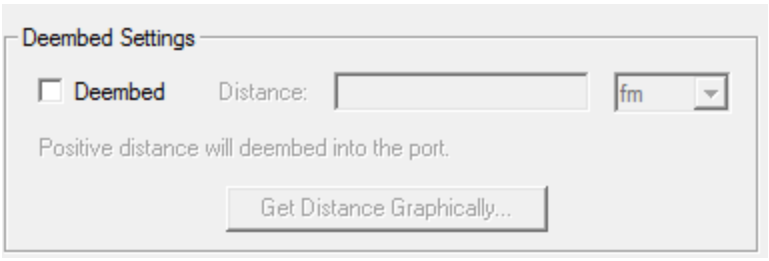


We will now explain the concept of **De-embedding** and **Renormalization**.

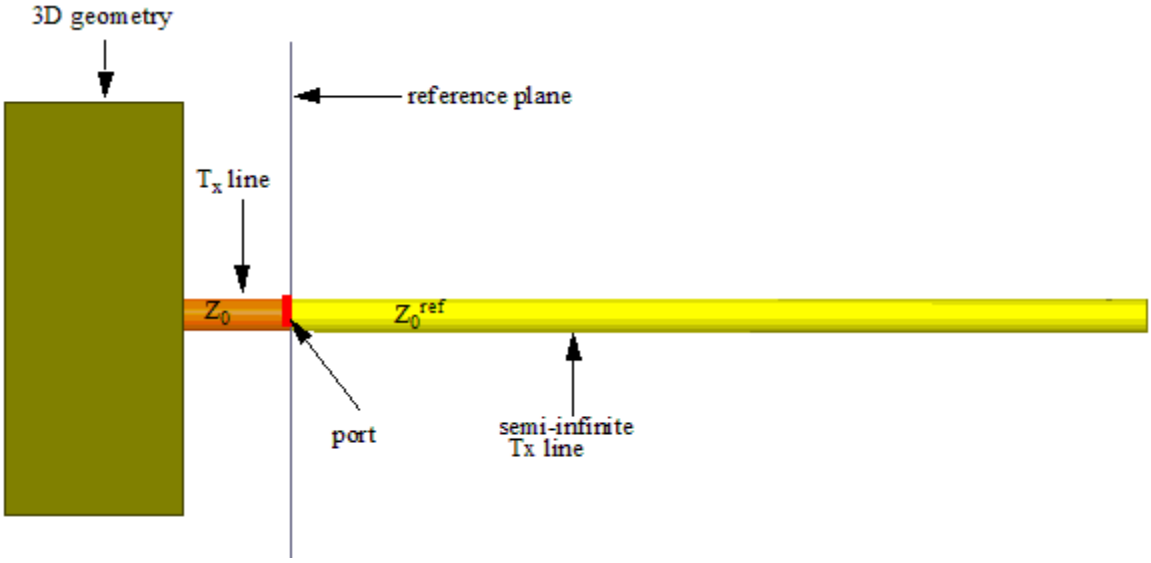
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Deembed Settings

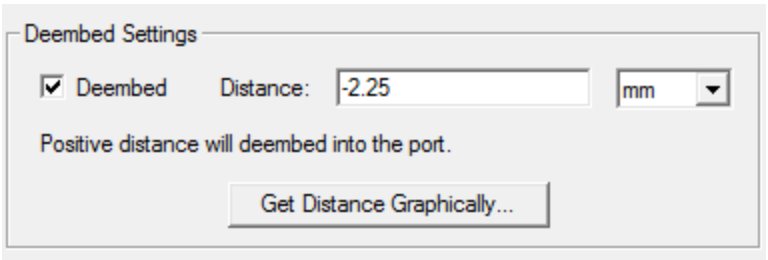
First we will not employ deembedding as shown under **Deembed Settings** of the wave port post processing dialog box.



In this situation, by default the reference plane will be along the port surface as shown below.

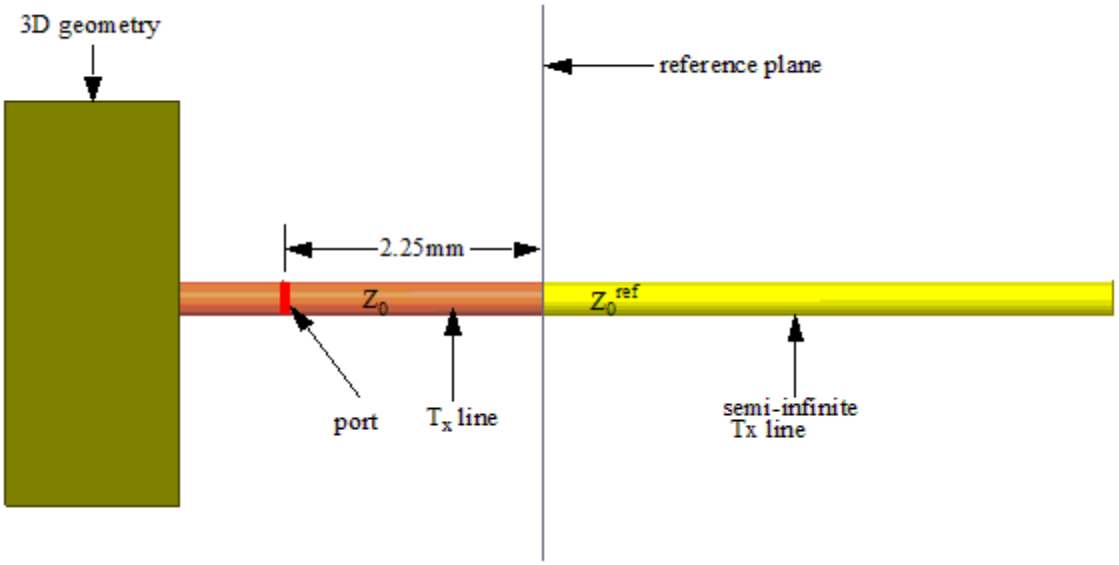


We will now employ deembedding.



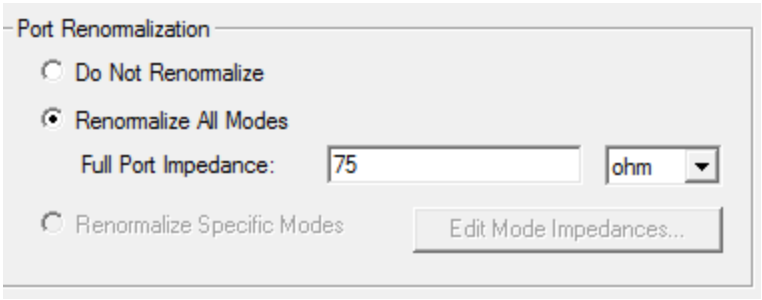
HFSS shifts the reference plane when you deembed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Renormalization

This option lets you vary the value of Z_0^{ref}. When you do not renormalize Z_0^{ref} is Z_0 . When you renormalize as shown in the following figure, Z_0^{ref} will no longer be Z_0 but it will be assigned the value that you enter in the **Full Port Impedance** field. The settings in the following figure makes Z_0^{ref} = 75 ohm.



If you want to enter a complex impedance, enter it in the form *<re> + <im>j.*, for example 50ohm + (-5ohm) * 1i . You can assign a variable to these values, (for example " resistance + (reactance) * 1i"). This variable can be dependent on the frequency, which allows use of a dataset for frequency dependent impedance, (for example, pwl(ds1,freq) + (pwl(ds2,freq)) * 1i).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> You do not need to re-run a simulation in order to renormalize a port. Post-processing reports are automatically updated to reflect the renormalized S-matrix.

Assign Wave Ports for Modal Solutions

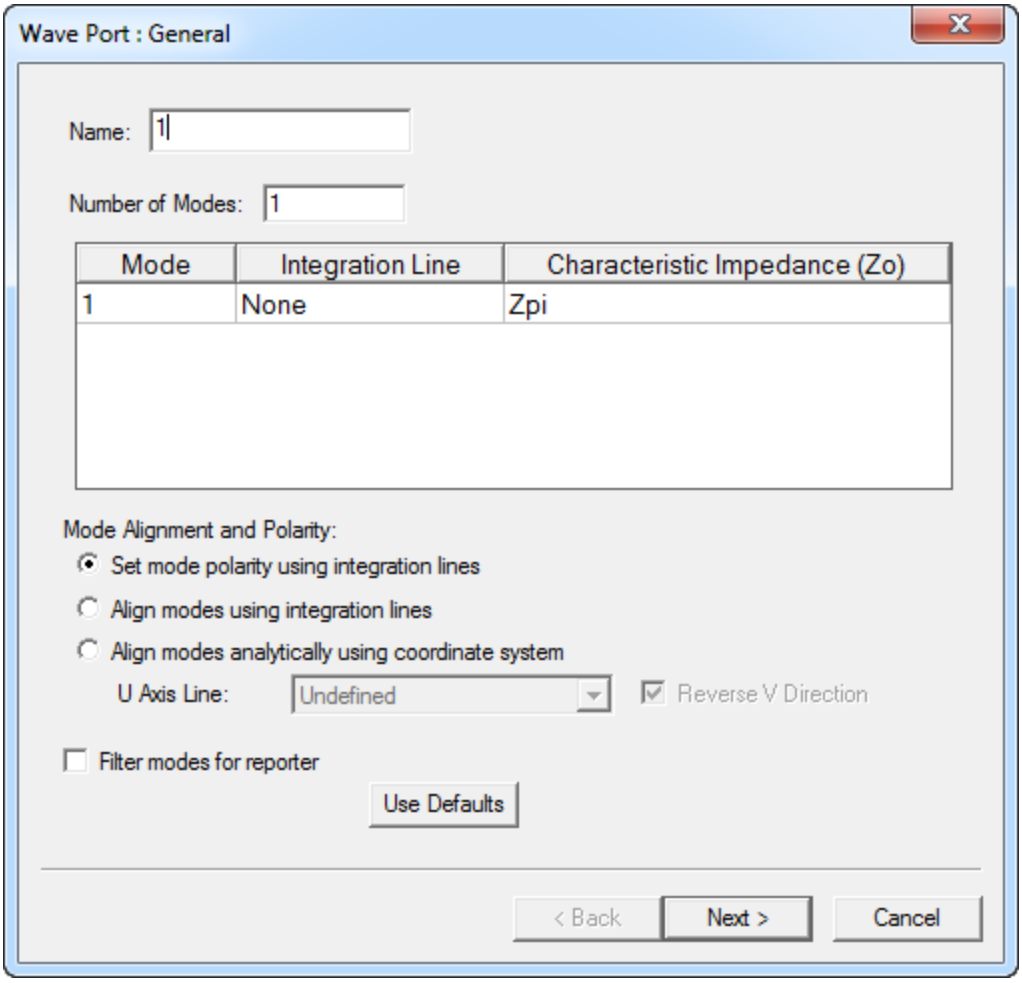
This section outlines the steps for defining excitations for modal solutions.

1. Select coplanar face(s) that you want to excite.
2. Click **HFSS>Excitations>Assign>Wave Port**

   The **Wave Port** dialog menu appears with the **General** tab selected.
3. Enter the name of the wave port.
4. Specify the number of modes for a port.
5. If necessary define the integration lines for each mode, set the characteristic impedance, and specify whether to filter modes for the reporter.

   > **Note:**
   >
   > For more information about the **Integration Lines**, see Defining an integration line.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



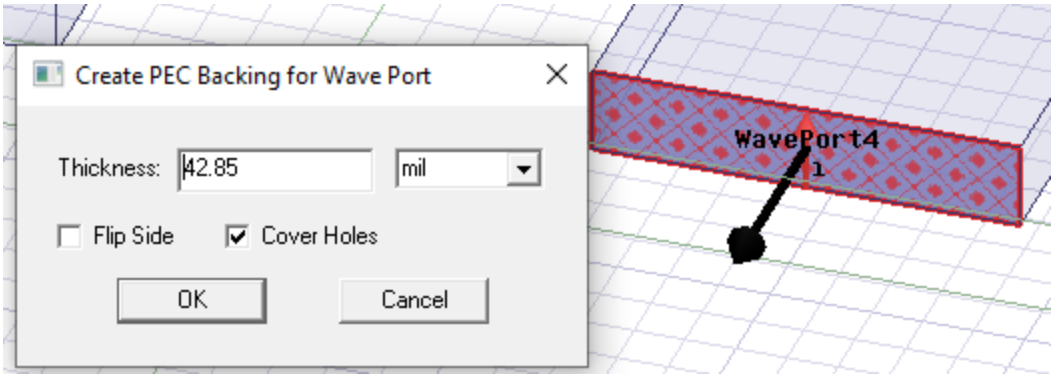
See Define Mode Alignment and Polarity.

Automatic PEC Creation on Modal Wave Ports

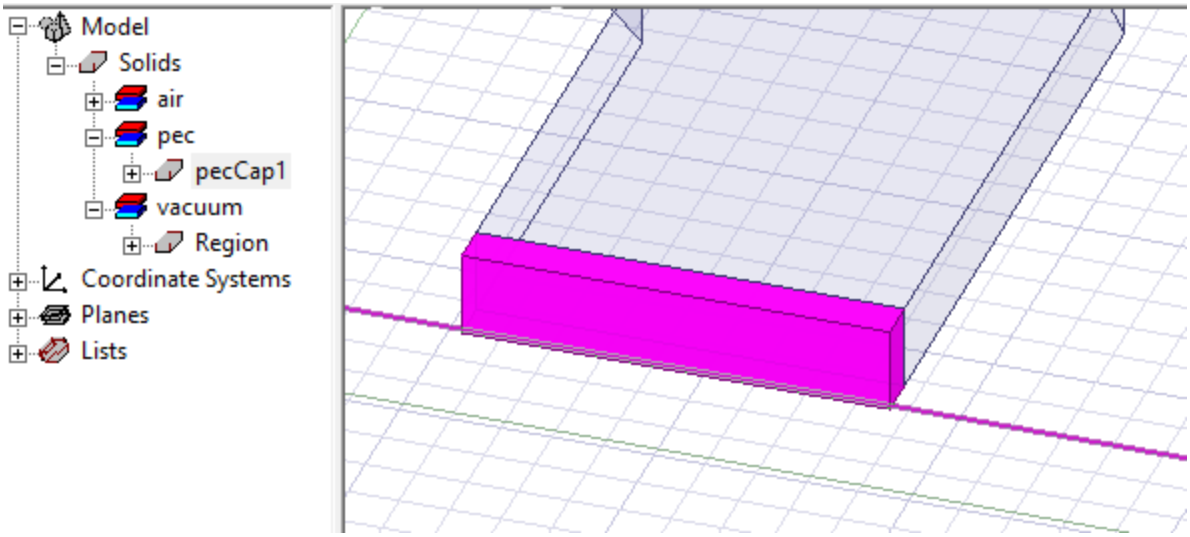
In HFSS, A wave port must have 'solve inside' objects on one side only. An internal wave port would require a PEC object to be attached to and cover the opposite side before the solution can proceed.

This HFSS Boundary Assignment Option automatically adds a PEC backing to every newly created wave port when necessary. When a new wave port is created and requires a PEC backing, a dialog window opens to let users create such backing by extruding the port face with specified thickness. You have the option to flip the side of the PEC backing. You can also set the thickness as a variable.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After clicking OK, the PEC backing is created and the new PEC object appears in the History tree.



Once the PEC cap is created it is associated with the port. If you edit the geometry of the port or the geometry/material of the cap, association is deleted and a message is issued. If you reassign or delete the port, the PEC cap is deleted, and a warning is issued.

For an example, see Automatic PEC Creation for Wave Ports.

Define Mode Alignment and Polarity

This section discusses the options under Modes alignment and Polarity that appears on the **Wave Port** dialog box. First bring up the **Wave Port** dialog box as follows:

1. Select the face that you want to excite, right-click, and select **Assign Excitations**>**Wave Port**.
2. Name the port.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

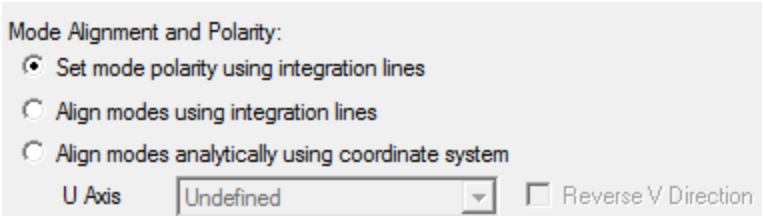
The three options under **Mode Alignment and Polarity** are described below.

Use Set mode polarity using integration lines

This is the default. Use this option as follows:

1. Bring up the **Wave Port** dialog box, access the Modes panel.
2. Enter the number of modes, select the radio button **Set mode polarity using integration lines** to control the polarity.

> **Note:**
>
> HFSS will arbitrarily align the E fields of the modes.

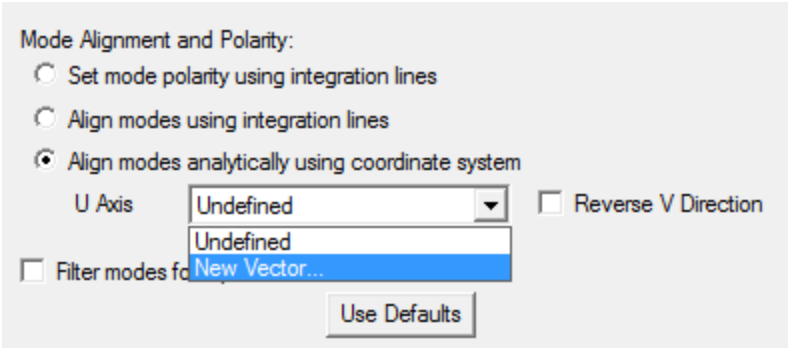


> **Note:**
>
> For more information, see Modes with Default Settings and Set Mode Polarity Using Integration Lines.

Use Align modes analytically using coordinate system

Use this option if the port corresponds to an analytic port type.

1. Bring up the **Wave Port** dialog box and select this option.



2. Select **New Vector** from the U Axis drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Draw the U axis to split the port symmetrically.

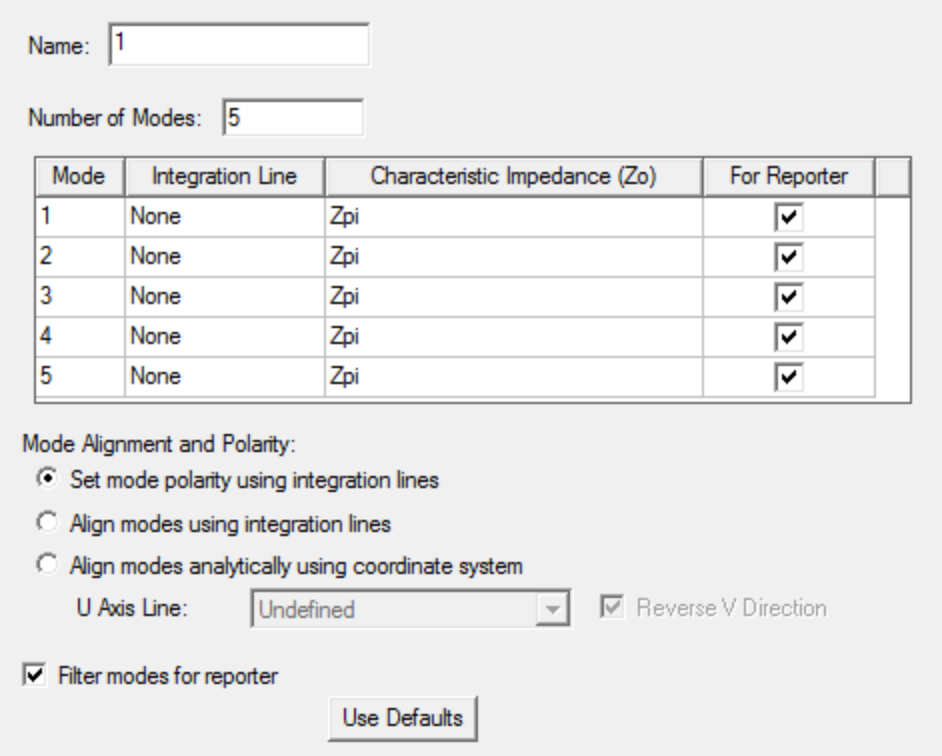
   The V direction is computed automatically and can be flipped using the **Reverse V Direction** check box. The solver polarizes the fields by aligning them with analytic mode patterns that are generated on the U-V coordinate system.

   > **Note:**
   >
   > For legacy projects, when reading in a port, if alignment is requested but only one mode exists, HFSS turns alignment OFF. This prevents an error in validation, and preserves the behavior of having the fields polarized but not aligned.

   If desired, you can check the **Filter Modes for Reporter** check box on the lower left (see below).

   This adds a new column to the Mode table, which lets you use a check box to designate a mode **For Reporter**. For designs with multiple modes, this function will simplify your selections when you create traces for reports.

   For more information, see Align Modes Analytically Using Coordinate System.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use Align modes using integration lines

This option meant for advanced users causes polarization for non-analytic ports. There is no restriction on the port geometry, materials, or integration lines.

1. Bring up the **Wave Port** dialog box.
2. Specify more than one mode.
3. Select **Align modes using integration lines**.

   A column **Alignment Groups** is added in the Modes table.

4. Select **Alignment Groups** from the corresponding drop-down menu.
5. Define the **Integration Lines** on the port surface as needed.

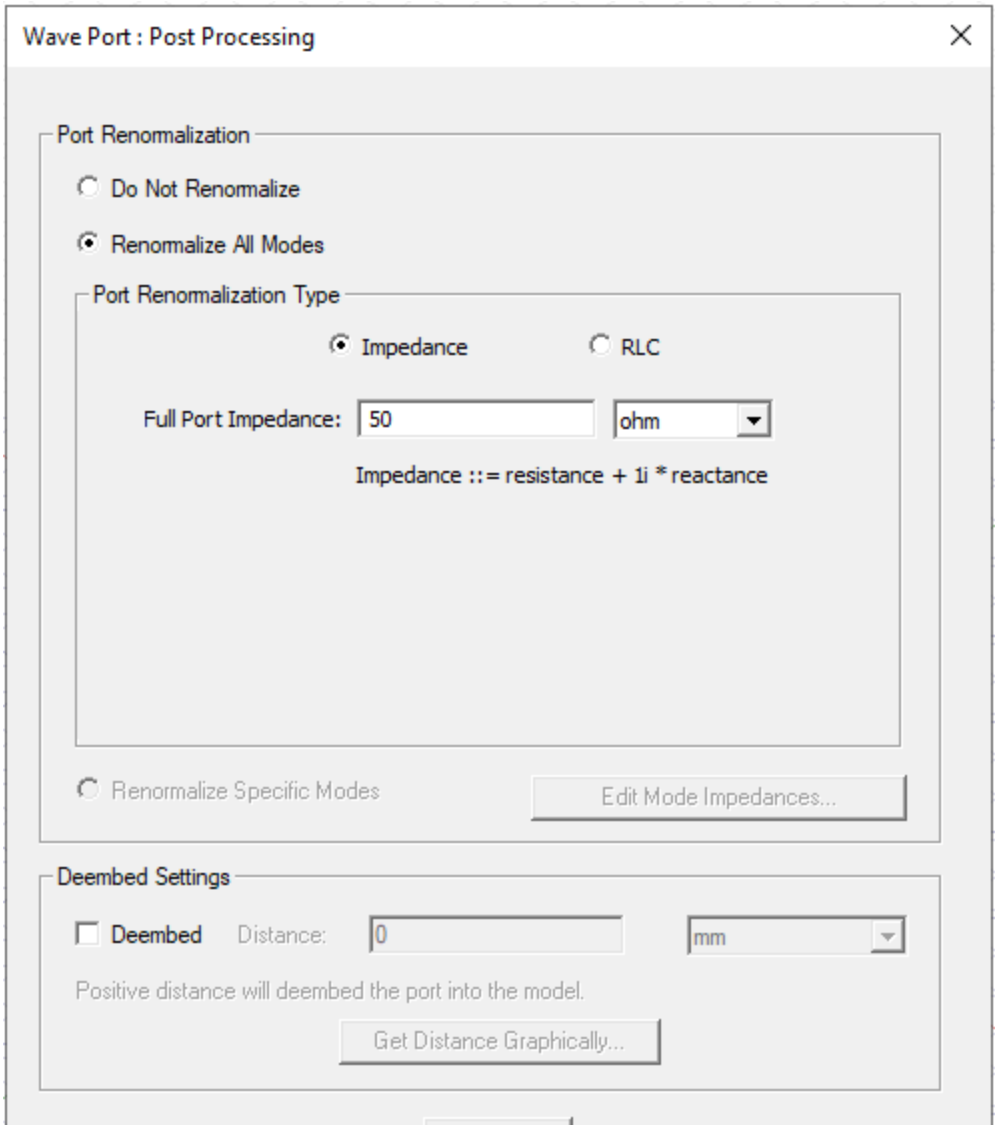
For more information about this option and the effect of **Integration Lines**, see Align Modes Using Integration Lines.

Post Processing: Wave Ports

The **Post Processing** sub-panel has options to renormalize the port to 50 ohms (or any other desired value) or just accept the default option **Do Not Renormalize** and also the **Distance** field to de-embed a port into or out of the model as needed.

A **Wave Port: Post Processing** panel is shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



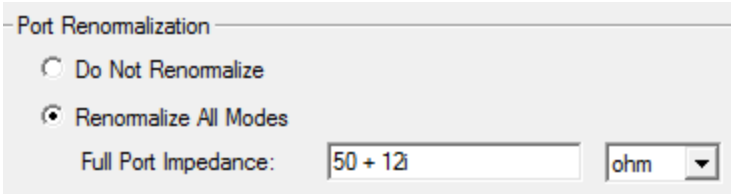
The Port Renormalization choices include:

- **Do Not Renormalize:** This is the default.
- **Renormalize All Modes**: Enables the **Port Renormalization** selections for **Impedance** and **RLC**.
  The default impedance for re-normalization of each mode is 50 ohms.

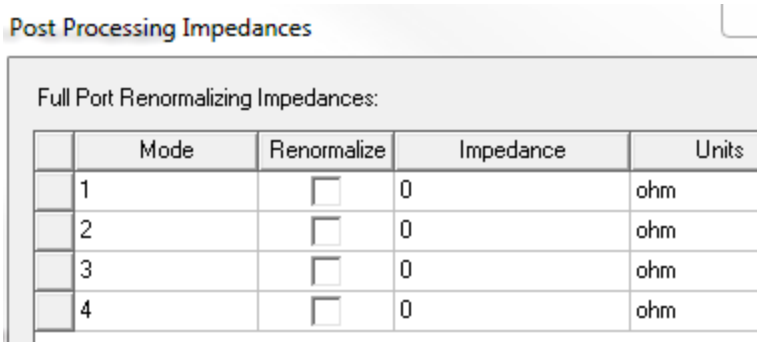
  If you select Impdence, complex impedance should be entered in the form: *<re> + <im>j* or *<re> + <im>i*. Either *j* or *i* entered into the field is displayed as *i* in the field and in a

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

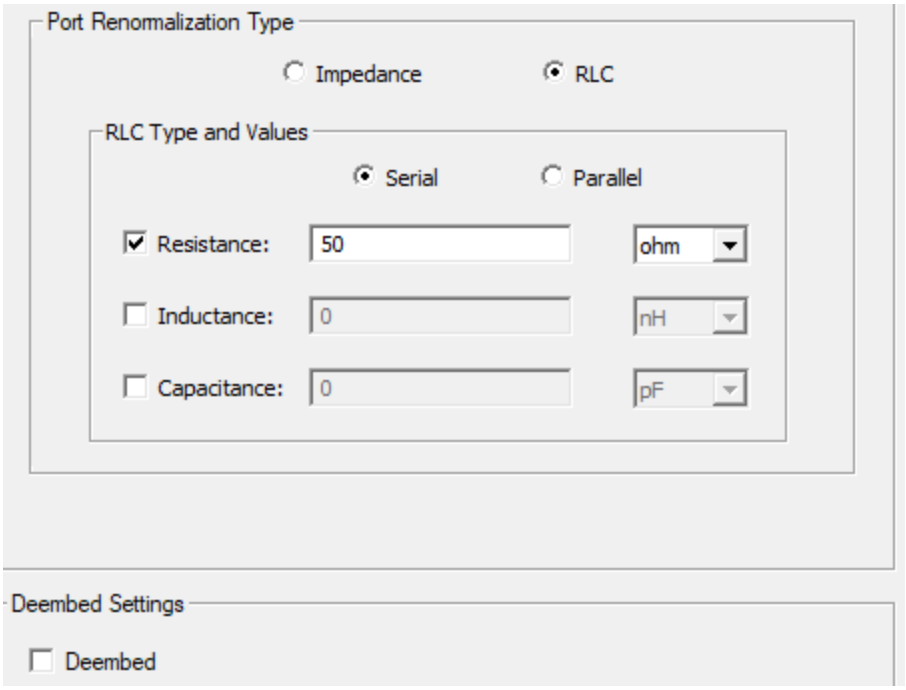
recorded script. See figure below.



- For multiple modes, the **Renormalize Specific Modes** is enabled, causing the **Edit Mode Impedances** button to be active. Click this button to renormalize the impedance of desired modes.

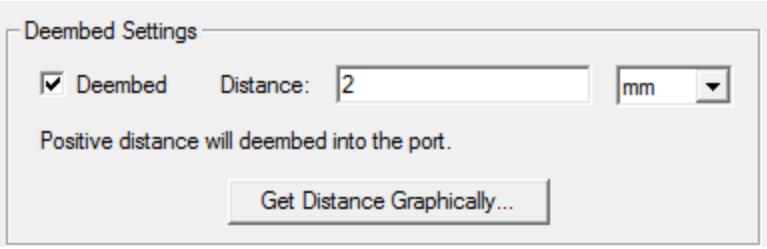


If you select RLC, you can specify Resistance, Inductance, and Capacitance and designate the RLC type as Serial or Parallel.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enter the **Deembed** distance in the field or draw a line on the model to represent the distance and get it graphically. You can also assign a variable in the **Distance** field.



Important:

The orientation of the **Deembed** arrows in the modeler is determined by the modeler direction of the normal of the face where the port is defined. From the standpoint of the S-parameters what is correct is the sign of the deembed distance.

Note:

- A positive distance value will deembed the port into the structure. A negative distance value will deembed the port out of the structure.
- Make sure you are not creating a non physical situation when you deembed cutoff modes. For example, when you enter a positive distance to deembed a port with a cutoff mode into the model the resulting field values for that mode can be very large.

Wave Ports for Terminal Solutions

HFSS treats a wave port as a semi-infinitely long waveguide or transmission line. For terminal solutions, we deal with transmission lines only. Terminals are the ends of signal traces that intersect the plane of a port.

To fit circuit theory HFSS has options for assigning terminals and expressing the relationship between currents and voltages. Terminals are especially useful when you are dealing with circuit simulators since the S-parameters obtained in terms of currents and voltages are in a format that is compatible with most circuit simulators.

Terminals and Modes

Example of Terminals

Reference Conductors

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Terminals and Modes

A terminal is assigned on a conductor in contact with the port. A port with *n+1* distinct conductors that do not touch each other will support *n* terminals, where the remaining conductor is the local reference ground. The solver generates an error if the number of terminals exceeds *n* conductors.

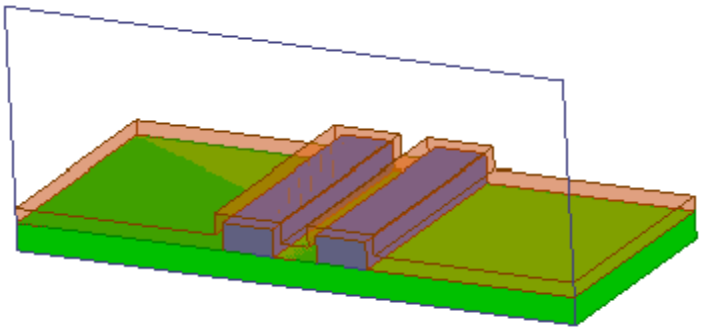
The solver uses as many modes as terminals. These modes are not eigen modes and sometimes referred to as terminal modes. The solver computes terminal modes where all the terminal modes are a superposition of eigen modes such that the S-parameters computed by the solver constitute the corresponding terminal S-Matrix instead of the eigen S-Matrix.

For terminal projects we will deal only with transmission lines and assume that the currents and voltages correspond to quasi-TEM modes. The number of terminal modes is dictated by the number of conductors on the port in the project. Proper port definitions should ensure that no higher order propagating modes exist in the transmission lines.

Example of Terminals

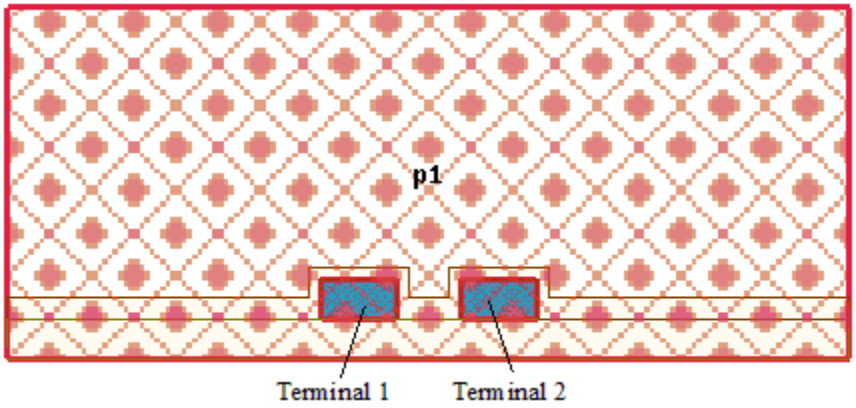
A terminal is essentially an insertion point for voltage and current on a multi-trace transmission line.

An HFSS model of a coupled microstrip transmission line is shown below.



The terminals are defined at the ends of the signal traces where they intersect with the surface of the port as shown below.

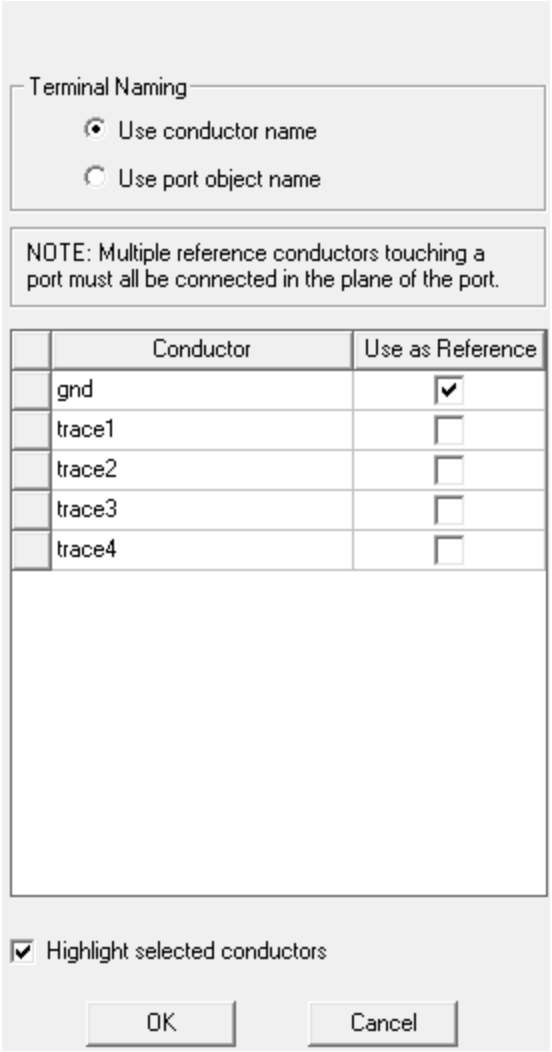
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Reference Conductors

The **Reference Conductors**dialog box contains a table that lists all the conducting objects used in a model. In this dialog box you will choose the conductor(s) that will act as reference. You can name terminals based on the name of their conductors or their associated ports.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

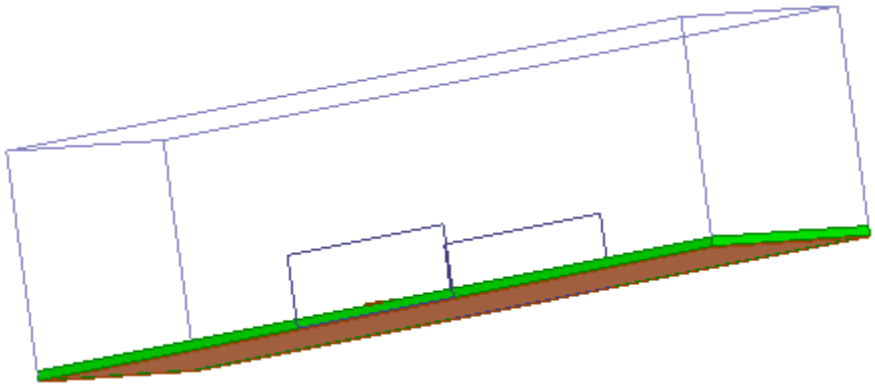


We will depict a couple of scenarios that you may encounter while assigning terminals to conductors in your model. For both scenarios we will consider the same microstrip line with modifications to the model.

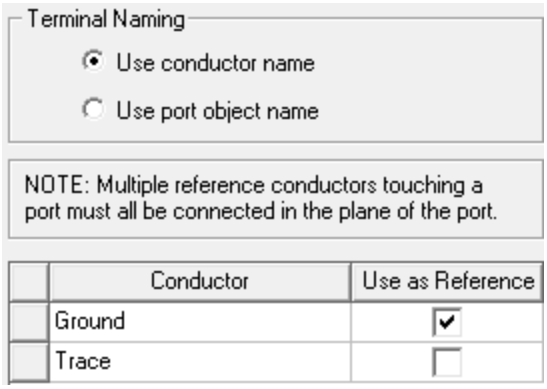
Scenario 1: Explicit Reference Conductor

The following figure shows a microstrip line enclosed by an air box. The model has an explicit ground plane that is orange in color.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



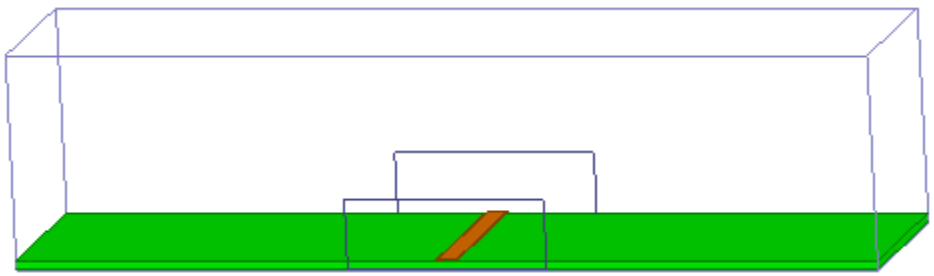
The ground is an explicit object and so it will appear as a conductor in the **Reference Conductors for Terminals** dialog box when assigning terminals to the trace.



Select **Ground** as reference so that the terminal appears only on the *Trace* and not on the *Ground*.

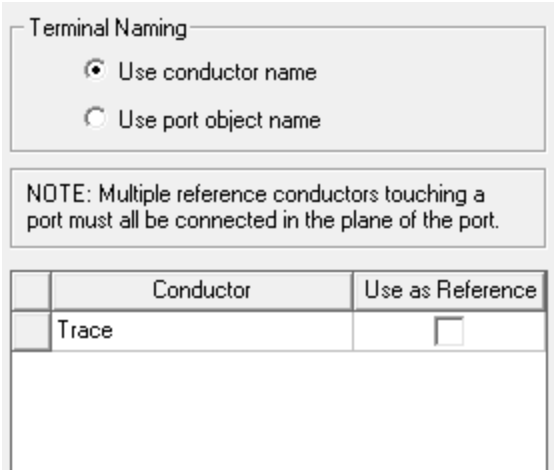
Scenario 2: Implicit Reference Conductor

The following figure illustrates the same microstrip line enclosed by an air box without the radiation boundary. This model does not have the explicit ground plane.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

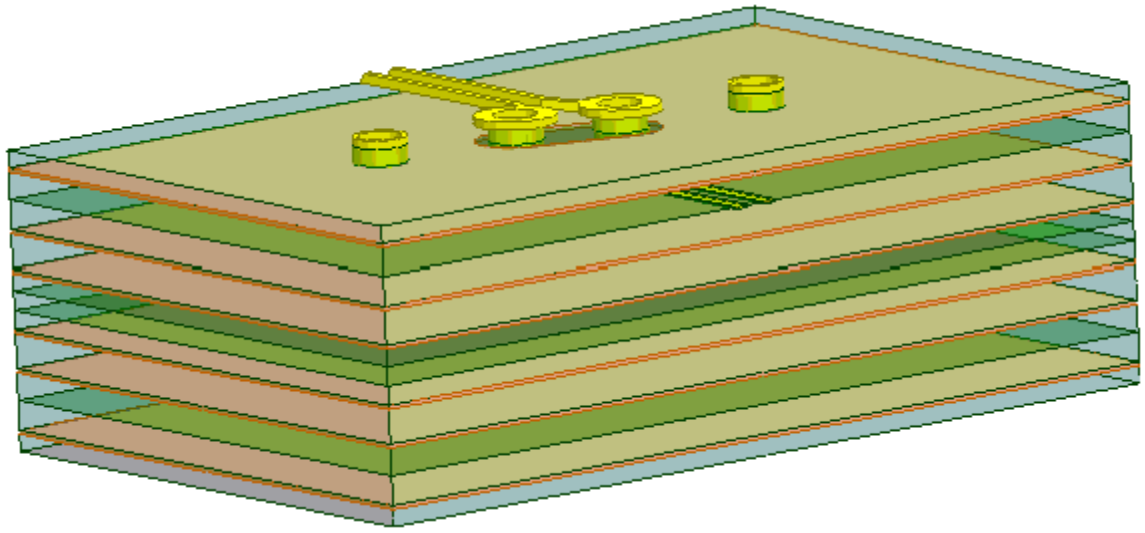
The solver treats the boundary of the air box as PEC because no other explicit boundary condition is applied to it. The air box acts as a reference conductor but will be treated as an implicit conductor that does not appear in the dialog box. In this case do not define the reference conductor as it is already assumed by the solver.



Assign Wave Ports for Terminal Solutions

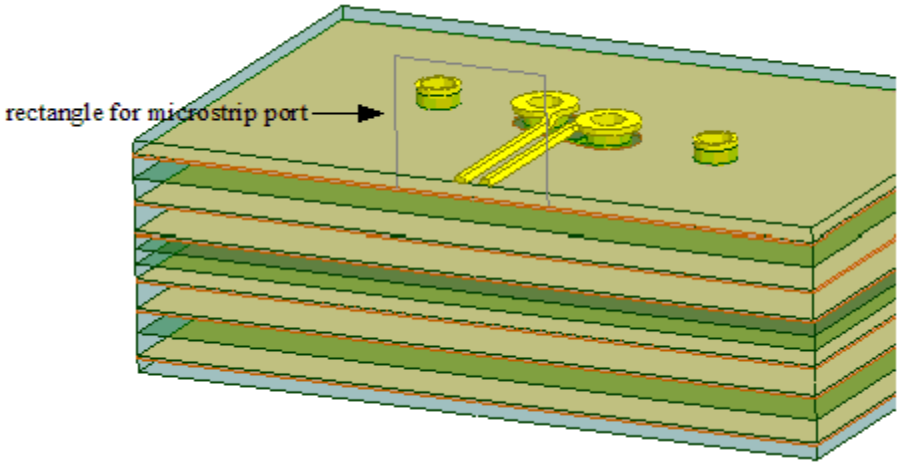
A terminal is a geometry intersection of a conducting object with a port face. The intersection can be a face or an edge in a conducting geometry. The edge can even extend outside the port and though it can be defined as a terminal in the user interface, only the portion of the edge that overlaps with the port is used to define the terminal edge for the solver.

In this section we will assign wave ports on a differential pair via model shown below.

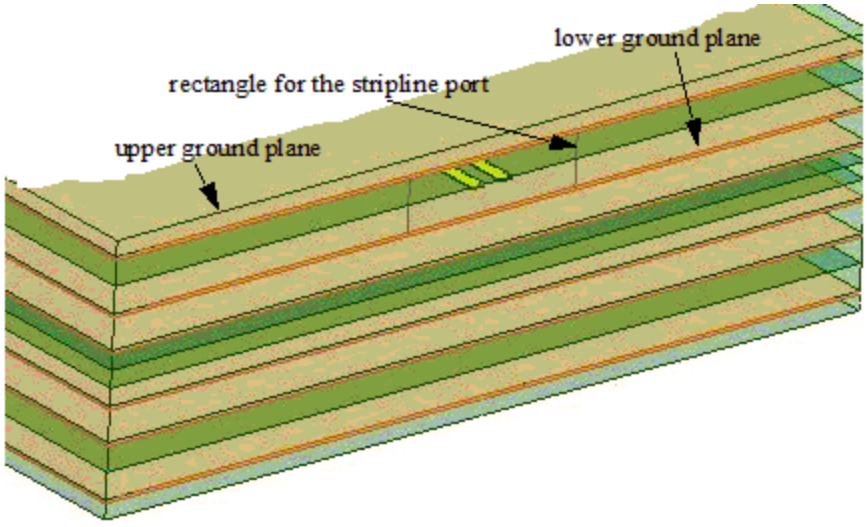


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

We will assign a microstrip port and a stripline port. The guidelines for defining port size introduced in the Microstrip Waveguide section are also applicable for the differential pair model. The microstrip port size is defined as (10w+s + 2w) x 8h where w = microstrip trace width, s = separation between the traces, and h = height of the substrate.



The stripline port is defined as (10b + c + b) x d where b = stripline trace width, c = separation between the traces and d = the distance between the upper and the lower ground plane.

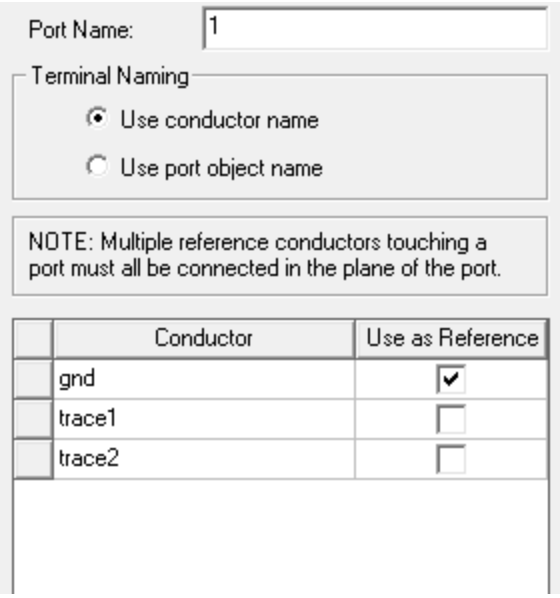


Perform the following steps for terminal assignment.

1. Right-click the rectangle for the microstrip port and select **Assign Excitations**>**Wave Port.**

   The **Reference Conductors** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select the **gnd** conductor as reference and click **OK**.

   > **Note:**
   >
   > Number of terminals = Number of conductors - 1 = 3-1 = 2.

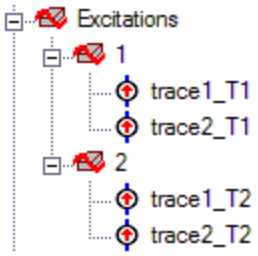
   The port gets assigned and the terminals appear automatically on the microstrip.

3. Similarly assign a wave port on the stripline rectangle.

   The port gets assigned and the terminals appear automatically on the stripline.

   > **Note:**
   >
   > - In rare cases when automatic terminal assignment fails, you can manually assign a terminal by selecting the face of the conductor.
   > - Identification of conductors depends on a threshold conductivity value. The threshold is based on the material assignment or the boundary assignment if a conducting boundary is assigned to the object.
   > - The project manager window gets populated as shown in the following figure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

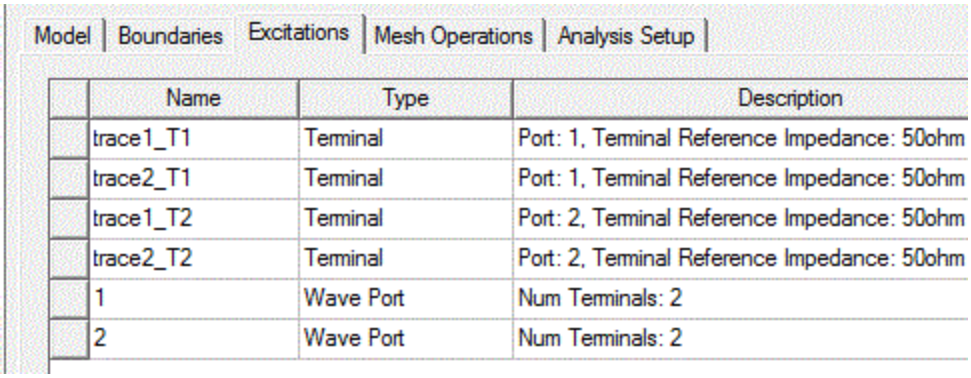
Terminal naming conventions are based by default on the first geometry in the assignment selection for the terminal. You can change the names from the Set Default Boundary/Excitation Base Name dialog box. For auto assign terminals, you can also specify whether naming uses the conductor or the port object name.

Multi-Select Ports and Terminals

To view all of the assigned excitations at the same time, perform the following steps:

1. Double-click **Excitations** from the **Project Manager** window.

   This brings up the **Design List** dialog box.



	Name	Type	Description
	trace1_T1	Terminal	Port: 1, Terminal Reference Impedance: 50ohm
	trace2_T1	Terminal	Port: 1, Terminal Reference Impedance: 50ohm
	trace1_T2	Terminal	Port: 2, Terminal Reference Impedance: 50ohm
	trace2_T2	Terminal	Port: 2, Terminal Reference Impedance: 50ohm
	1	Wave Port	Num Terminals: 2
	2	Wave Port	Num Terminals: 2

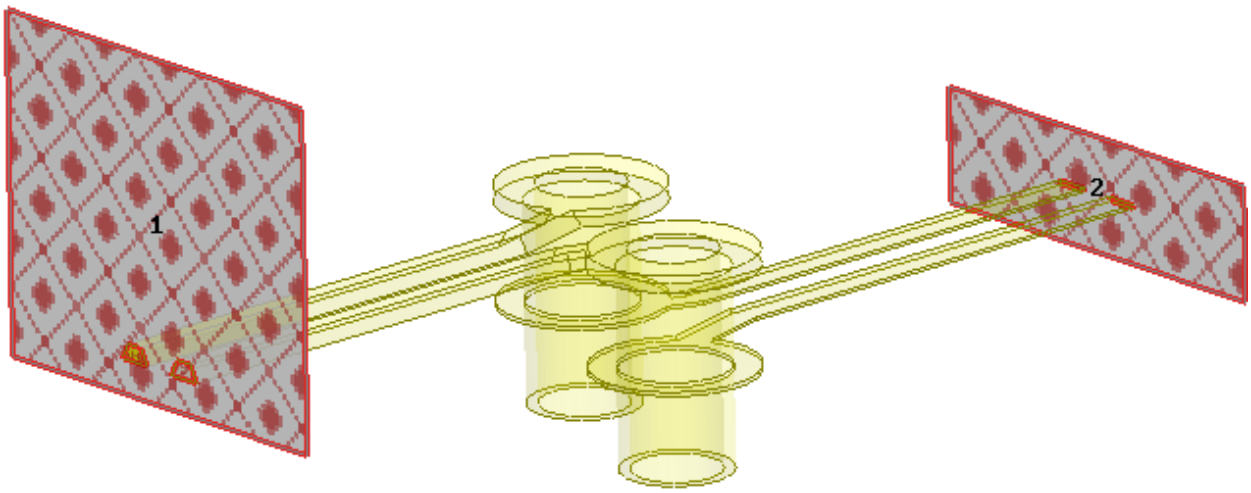
2. Press **Ctrl** and select the excitations that you want to see in the model.

   The selected excitations get highlighted in the model.

3. Click **Done** to close the dialog box.

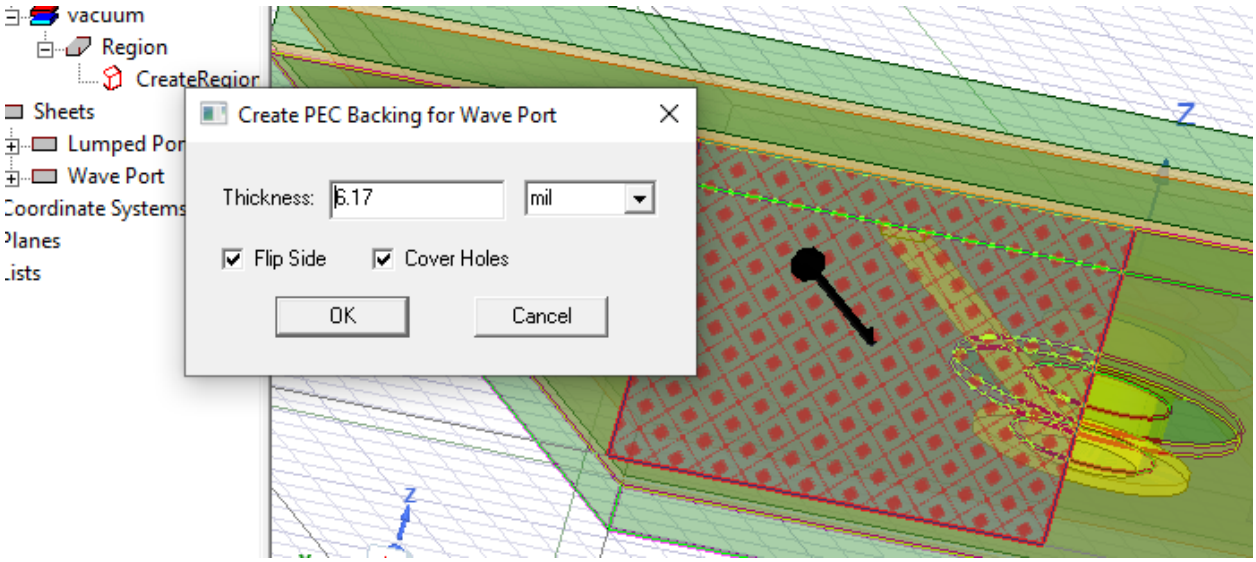
   For example, the following figure shows all the excitations on the differential pair.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



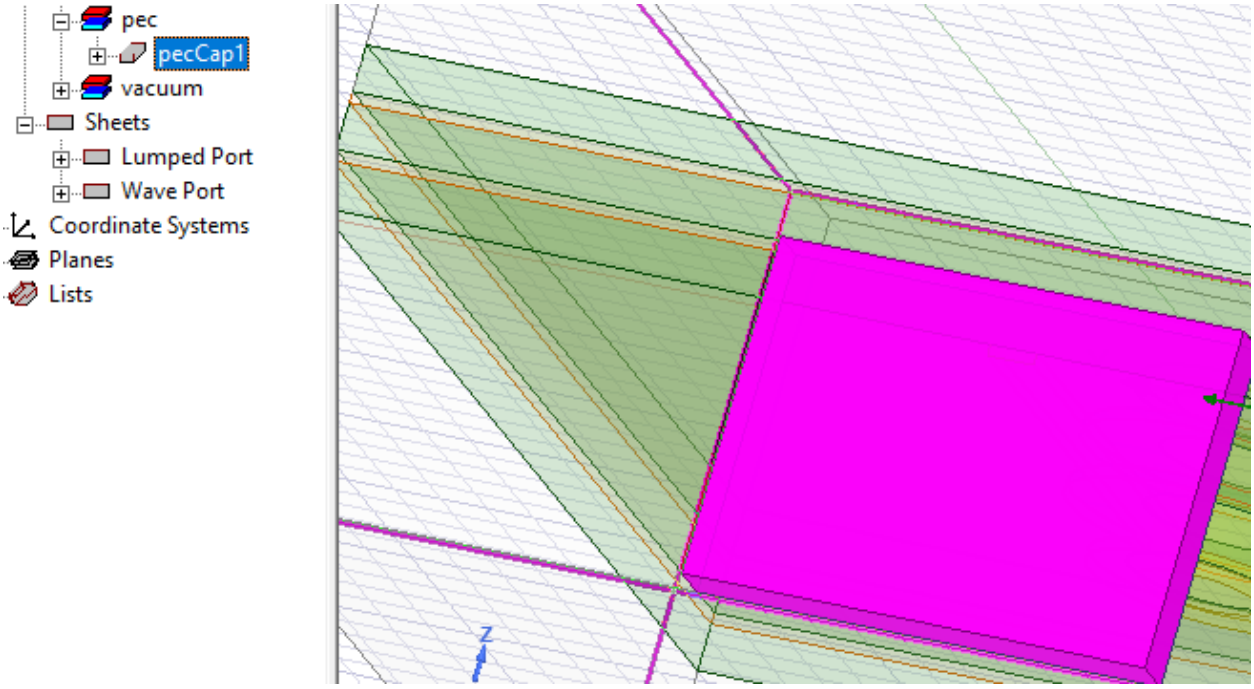
Automatic PEC Creation on Terminal Wave Ports

In HFSS, A wave port must have 'solve inside' objects on one side only. An internal wave port would require a PEC object to be attached to and cover the opposite side before the solution can proceed. This HFSS Boundary Assignment Option automatically adds a PEC backing to every newly created wave port when necessary. When a new wave port is created and requires a PEC backing, a dialog window opens to let users create such backing by extruding the port face with specified thickness. You have the option to flip the side of the PEC backing. You can also set the thickness as a variable.



After clicking OK, the PEC backing is created and the new PEC object appears in the History tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once the PEC cap is created it is associated with the port. If you edit the geometry of the port or the geometry/material of the cap, association is deleted and a message is issued. If you reassign or delete the port, the PEC cap is deleted, and a warning is issued.

For an example, see Automatic PEC Creation for Wave Ports.

Associating Terminals and Ports

HFSS can automatically associate terminals with the ports whether you assign the ports first and then define the terminals or vice versa. You can define terminals from the **Excitations** level (in the **Project tree**) when the ports already exist. Regardless of the order in which they are assigned, a terminal is associated with the port containing the signal and reference conductors that define the terminal.

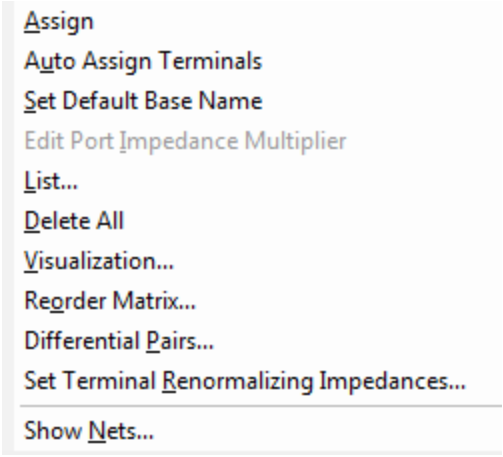
> **Note:**
>
> At either the port level or for all excitations, you can set the renormalizing impedance for all terminals.

the Auto-assign terminals on ports option on the **Tools>Options>HFSS Options** dialog on the **General** tab or use the Auto Assign Terminals command.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Auto Assign Terminals

Right-click **Excitations** from the project tree to bring up the shortcut menu that contains the **Auto Assign Terminals** option.



This option assigns terminals automatically. When you change the **Solution Type** of a model from **Driven Modal** to **Driven Terminal**, you can use this option to define the terminals for the conductors on the existing ports.

In exceptional cases with complicated arrangement of conductors or geometry with slight coordinate misalignments, auto assign may create either too few or too many terminals on a port. In such cases, you can manually assign the terminals.

Post Processing Operations

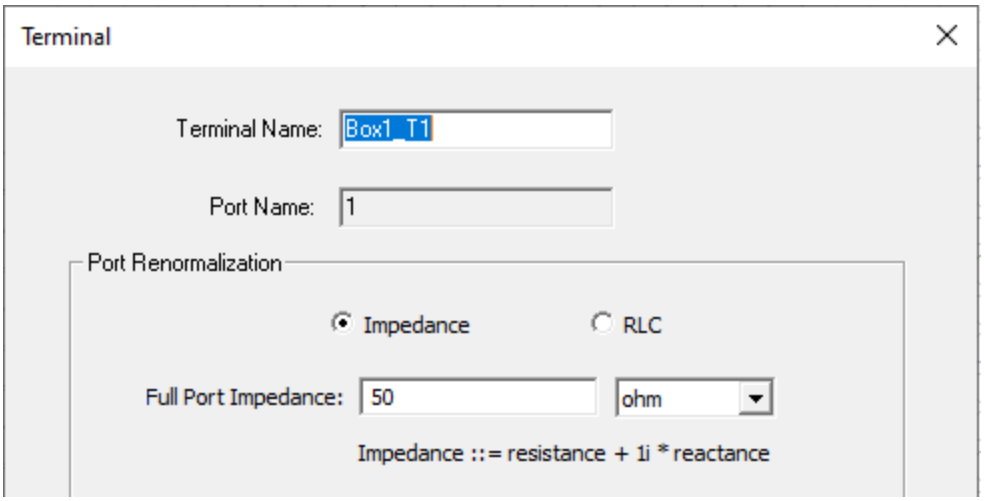
As a post processing operation you can renormalize the impedance of the individual terminals that you assigned. This is done as follows.

1. Double-click the terminal from the **Project Tree**.

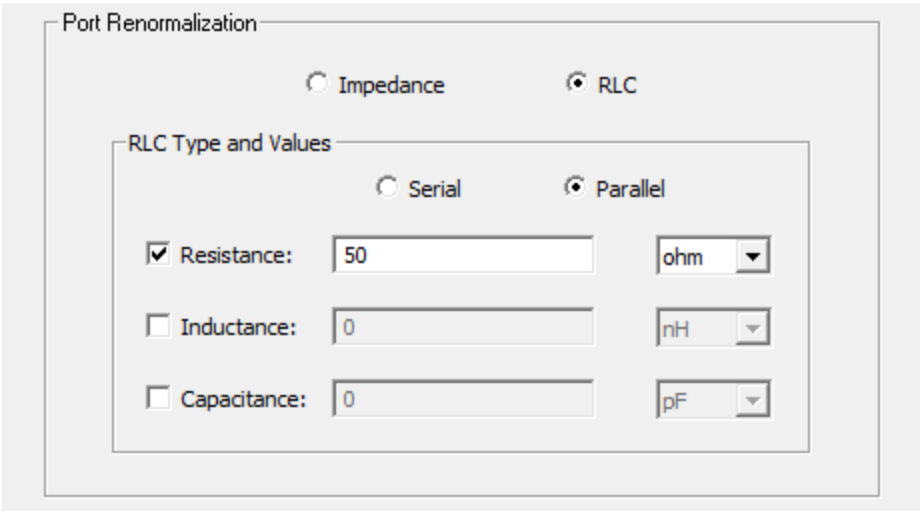
   The **Terminal** dialog box appears.

2. If you select Port Renormalizations as Impedance, enter the value in the **Resistance** and set the units from the drop-down menu and click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



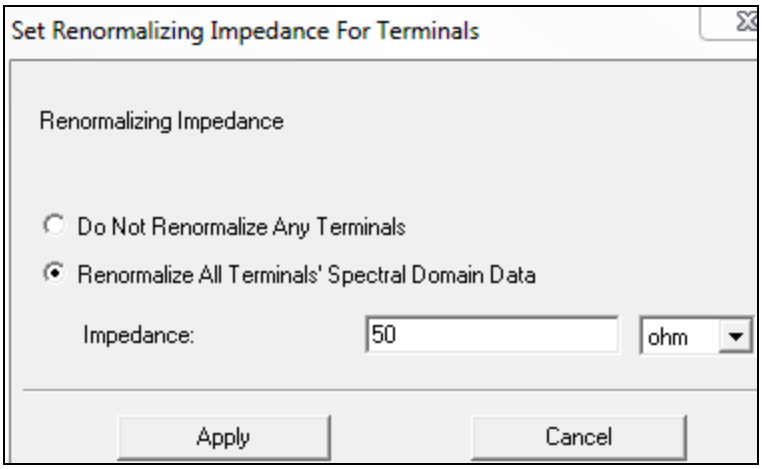
If you select RLC, you can specify Resistance, Inductance, and Capacitance and designate the RLC type as Serial or Parallel.



Set Renormalizing Impedance for Terminals

The setup panel for each terminal includes its post processing renormalizing impedance. You can set this value either for all excitations, or for a specific port. If a design includes at least one wave port, the setup panel also includes the radio buttons **Do Not Renormalize Any Terminals**, or **Renormalize All Terminals' Spectral Domain Data** .

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To set the renormalizing impedance for **all excitations**:

1. Right-click **Excitations** in the Project tree and select **Set Terminal Renormalizing Impedances**.

   The **Set Renormalizing Impedance for Terminals** dialog box appears.

2. In the field for **Impedance**, set the value, and select the units from the pull down.

   The format is "<real_part> + <complex_part>*i ohm", for example 50ohm + (-5ohm) * 1i

   > **Note:**
   >
   > The reference impedance is meant to represent the component modeled by the lumped port. You can assign a variable to these values, (for example " resistance + (reactance) * 1i"). This variable can be dependent on the frequency, which allows use of a dataset for frequency dependent impedance, (for example, pwl (ds1,freq) + (pwl(ds2,freq)) * 1i).

3. Click the **Apply** button to close the dialog and apply the change.

To Set the Reference for All terminals on a Port:

1. Right-click the port icon in the Project tree and click **Set Terminal Renormalizing Impedances**.

   The **Set Renormalizing Impedance for Terminals** dialog appears. It differs from the related command for all excitations by specifying that the **Renormalizing Impedance** is for terminals on the selected port.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the field for **Impedance**, set the value, and select the units from the pull down.

   This value can be a variable. This variable can be dependent on the frequency, which allows use of a dataset for frequency dependent impedance.

3. Click the **Apply** button to assign the impedance value.

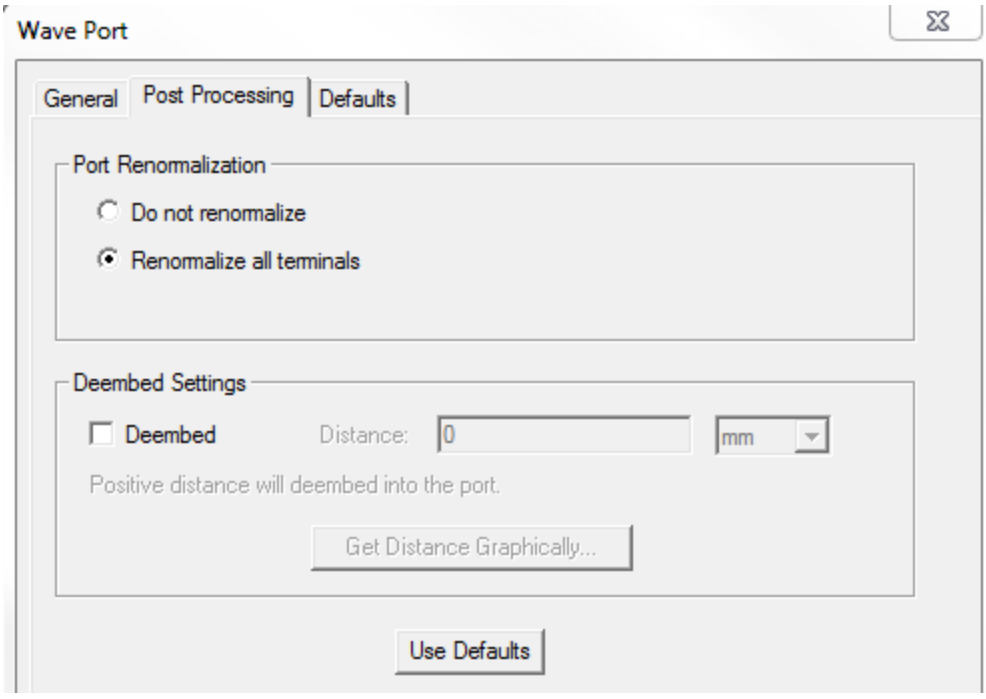
> **Note:**
>
> For more information, see Scaling a Source Magnitude and Phase.

You can also set the Terminal Reference Impedance on a port by selecting the port and editing the value in the **Properties** dialog.

In designs with at least one wave port, where you want to view un-renormalized Zo impedance and/or the corresponding S parameters in either the Matrix data or in a report, you can select the **Do Not Renormalize Any Terminals** radio button.

Renormalize Terminals from the Port Level

1. Double-click the wave port in the **Project** tree to display the **Properties** window for the wave port and select the **Post Processing** tab.



   Values here affect S-Parameters only. Port Renormalization choices include:

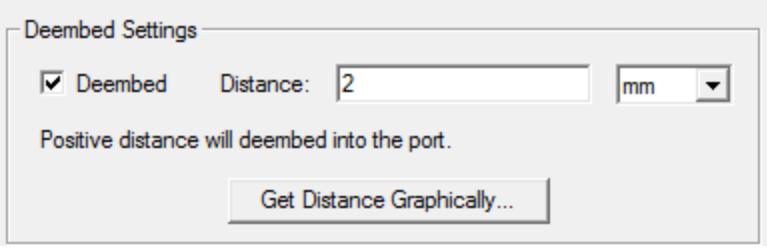
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Do Not Renormalize** (the default). Selecting this disables the Impedance fields for the port and terminals.
- **Renormalize All Terminals**. The default impedance for re-normalization of each port is 50 ohms. To specify a different impedance, you must open the Properties for the terminal and specify a value in the **Terminal Renormalizing Impedance** field.

If you want to enter a complex impedance, enter it in the following form: *<re>* + *<im>j*, for example 50ohm + (-5ohm) * 1i . You can assign a variable to these values, (for example " resistance + (reactance) * 1i"). This variable can be dependent on the frequency, which allows use of a dataset for frequency dependent impedance, (for example, pwl(ds1,freq) + (pwl(ds2,freq)) * 1i).

- If there are multiple modes, **Renormalizing Impedance for Specific Terminals** is enabled. Click this to enable the **Edit Terminal Impedances** button. This opens an editable table with the impedances for each terminal.

2. Enter the **Deembed** distance in the **Distance** field or draw a line on the model to represent the distance and get it graphically. You can also assign a variable in the **Distance** field.



Important:

After you enter the value, a blue arrow depicts the deembedding distance in the graphics window when the port is selected. When a wave port has been defined on a sheet, the inward or the outward normal will dictate the direction of the arrows. The orientation of the deembedding arrows in the modeler is determined by the direction of the normal of the face where the port is defined upon. From the standpoint of the S-parameters what is correct is the sign of the deembed distance.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

- A positive distance value will de-embed the port into the model. A negative distance value will de-embed the port out of the model.
- Make sure you are not creating a non physical situation when you deembed cutoff modes. For example, when you enter a positive distance to deembed a port with a cutoff mode into the model the resulting field values for that mode can be very large.

3. For HFSS Transient solutions, you can designate a port as Active or Passive.

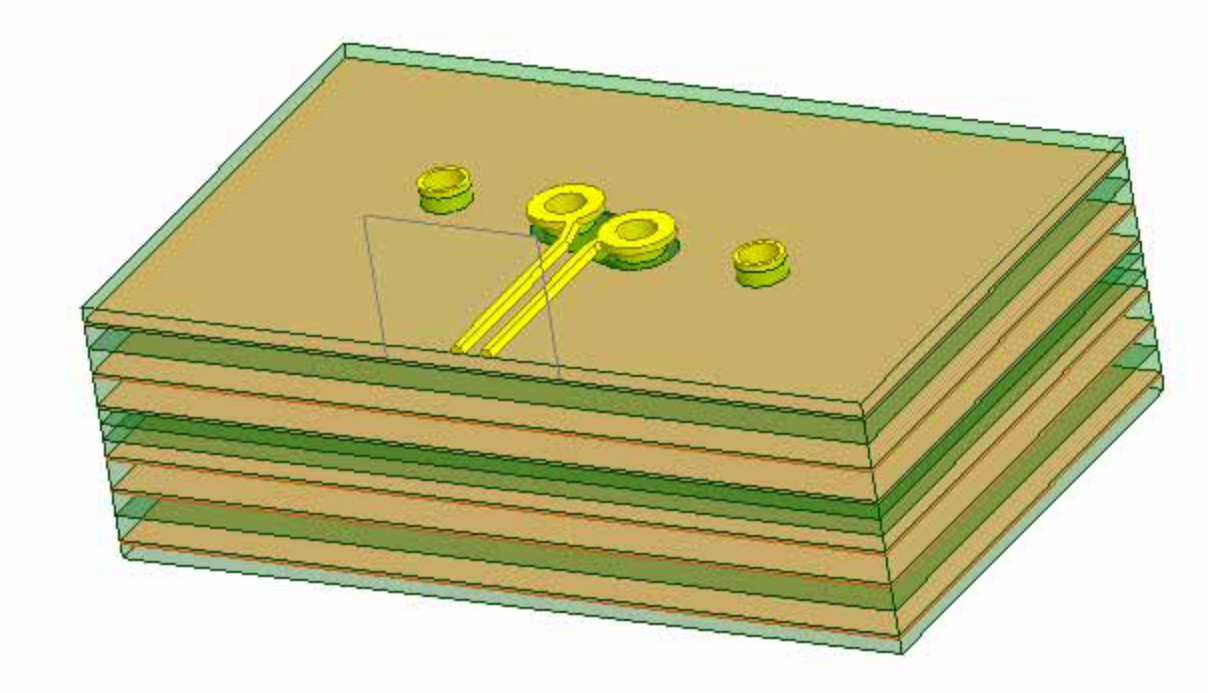
Set Differential Pairs

A differential pair represents two circuits, one positive and one negative routed close together so they will pick up nearly the same amount of noise. The two signals are subtracted from each other by a receiver, yielding a near "noise-free" version of the signal.

You can define one or more differential pairs from terminal excitations assigned on existing wave ports. Differential pairs can span ports, use lumped ports, and be enabled or disabled. To allow automated calculation of differential S-parameters from lumped ports, you can select terminals from two arbitrary ports, whether wave ports or lumped ports, for use in a differential pair.

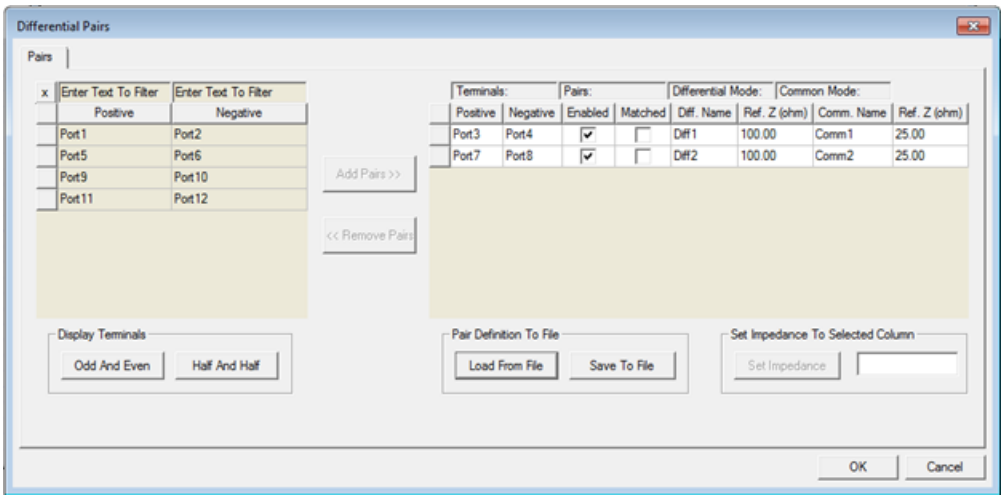
Because differential pairs can span ports or occur within a port, the **Differential Pairs** command is accessible at corresponding levels in the Project tree via the right-click menu both at the **Excitations** level, and at the port name level. If a differential pair involves terminals from two different ports, the **Differential Pairs** command for those ports can only be accessed at the **Excitations** level. If an individual wave port has multiple terminals defined, the **Differential Pairs** command is enabled when you select that port and right-click to display the shortcut menu. In order to combine differential pairs across ports, both ports must have the same renormalization setting; that is, either ports have **Do not Renormalize** on, or both have it off. For Transient Network solutions, differential pairs cannot include passive terminals. We will use a differential pair via model to assign the differential pairs. See Figure below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



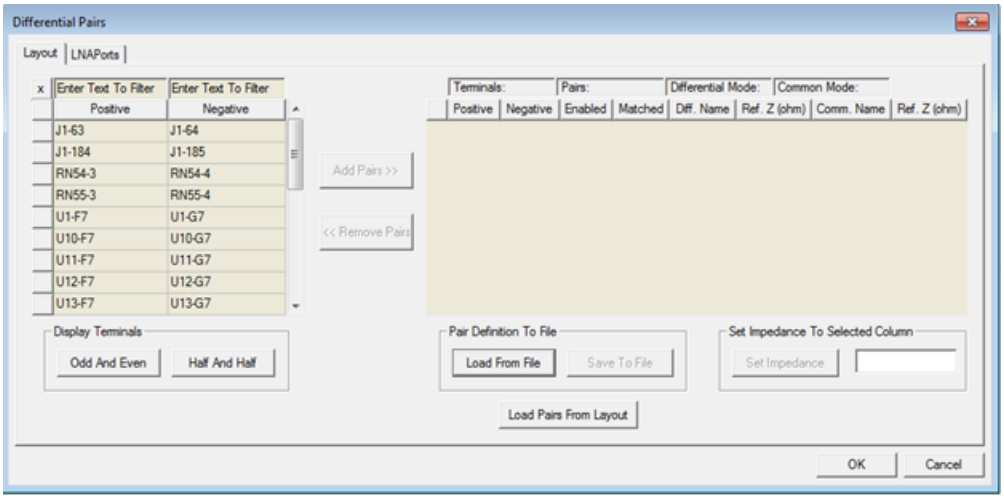
Differential Pairs Dialog can be opened from multiple places:

- Network Data Explorer: Edit > Define Differential Pairs
- Ports Folder in a Nexxim Design: right-click Ports > Differential Pairs
- Excitations Folder in a 3D Layout Design: right-click Excitations > Differential Pairs



Differential Pairs can be defined for both design layout and LNA setups if both are available. Both pairs will be available in two tabs of the same dialog as shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To set up a differential pair:

1. All terminals that are not already in pairs are shown in columns Positive and Negative of the Terminals list (on left):

- Clicking on a cell, selects that cell; Multiple selection is allowed using Ctrl+click and Shift+click; if one cell in each column is selected, Add Pairs button is enabled; clicking that button will add the pair to the bottom of the Pairs list (on right)
- Clicking on a row button in Terminals list selects that row; user can select multiple rows using Ctrl+click and Shift+click; when one or more rows are selected, Add Pairs button is enabled; clicking that button will add the pairs to the bottom of the Pairs list
- If anything is selected in Terminals list, accelerator Ctrl+A will select all rows
- Click-dragging a cell in Terminals list to different position in same column or to any position in the other column will move that terminal to the drop position
- Entering text into the first cell in each column will be used to filter which pins are shown in the column; * wildcard character is supported. See the Layout List dialog for an example of this.

2. Display Terminals group:

- Clicking Odd and Even will move odd position terminals to Positive column and even position terminals to Negative column; the filters will still be applied
- Clicking Half and Half will move terminals in the first half of the original list to the Positive column and terminals in the second half to the Negative column; filters will still be applied

3. Pairs list shows all terminal pairs that have been defined:

- Clicking Enabled or Matched header will toggle the check boxes in those columns; if Matched check box is checked, remove text from both Z Ref cells and disable those cells; if check box is subsequently unchecked restore Z Ref cells to previous values

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Clicking Positive, Negative, or Name column headers will toggle sorting the pairs alphanumerically on that column
- Clicking the Ref Z column headers will select the column
- Clicking on a row button in Pairs list selects that row; user can select multiple rows using Ctrl+click and Shift+click; when one or more rows are selected, Remove Pairs button is enabled; clicking that button will remove the pairs and add the corresponding Terminals back into the Terminals list
- If anything is selected in Pairs list, accelerator Ctrl+A will select all rows; if Del key is clicked, it will remove the selected pairs
- Clicking in a cell for Name or Ref Z will allow user to edit that value directly; names must satisfy pin name restrictions; Ref Z must be positive number with optional resistance units (typing “k” is acceptable for kohm)

4. Set impedance group:

- Entering a positive number with option units into the edit box will enable the Set impedance button
- If a Ref Z column is selected in the Pairs dialog box, clicking the Set impedance button will change all values in that column to the value in the edit box

5. Pair Definition file group:

- Clicking Save To File button will open Save File dialog with library location buttons; entering a name and clicking Save will save the Pair configuration to that file in csv format; this button is not enabled if there are no pairs defined
- Clicking Load from File button will open Open File dialog with library location buttons; choosing a .txt file will load the previously saved Pair configuration; this will show an error and NOT load if the file syntax is incorrect. Note that the file may contain pair definitions that do not reference pins that exist in the circuit; those definitions won’t be applied. Only definitions where both terminals actually exist will be applied.
- Saving a file and then subsequently loading a file should open a window already showing the last file saved or loaded.

6. Load Pairs from Layout button:

- This button will only be shown when the pairs dialog is launched from HFSS 3D Layout; clicking it will load pairs corresponding to Differential Pair net assignments

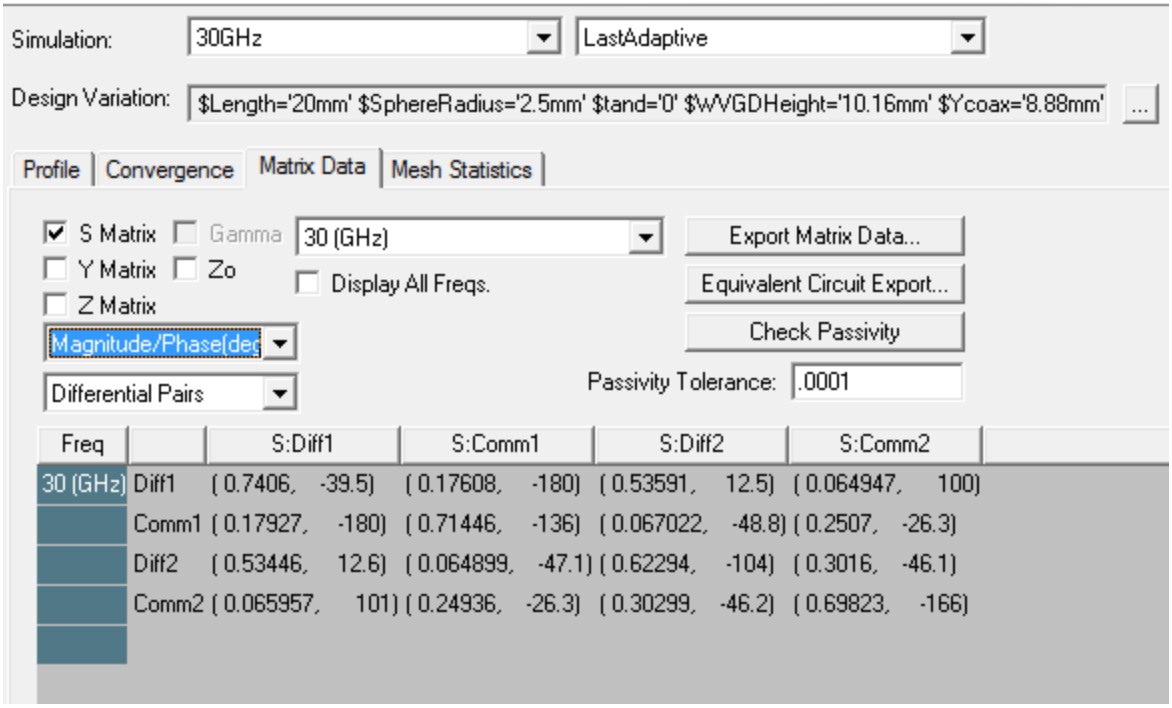
Note:

Scripting for loading/saving differential pairs from/to a file will not be recorded as part of the scripting in the dialog box. The user can hardcode the scripting as follows:

- LoadDiffPairsFromFile(filename)
- SaveDiffPairsToFile(filename)

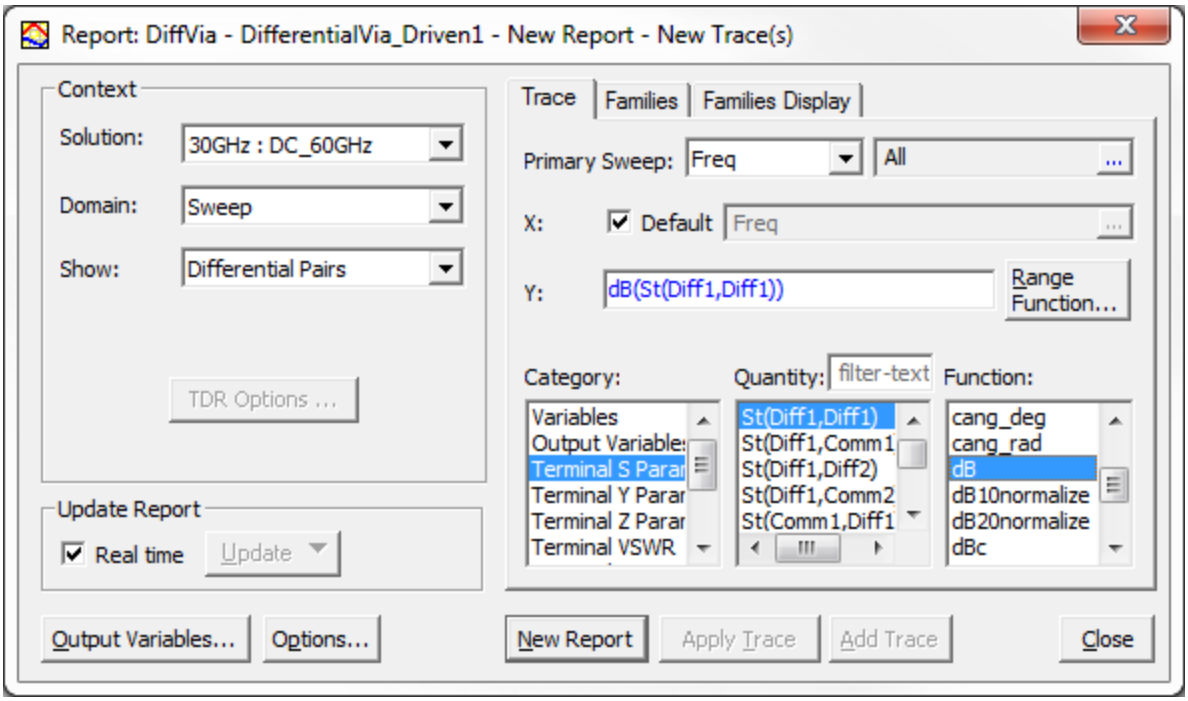
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After a solution has been generated, view the common and differential quantities of the differential pair under the **Matrix** tab of the **Solution Data** window as shown below.



When the design has differential pairs (link), the reporter can display quantities for the defined pairs or for the single-ended terminals upon which they are based. A drop-down menu will appear in the **Context** area of the **Report** creation dialog which allows the user to select which quantities will be displayed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can freely mix differential and single-ended terminal quantities. However, single ended quantities are computed as if no differential pairs existed. So, in the unlikely case of several terminals where only a subset are combined into pairs, the results may not be as expected.

Lumped Ports

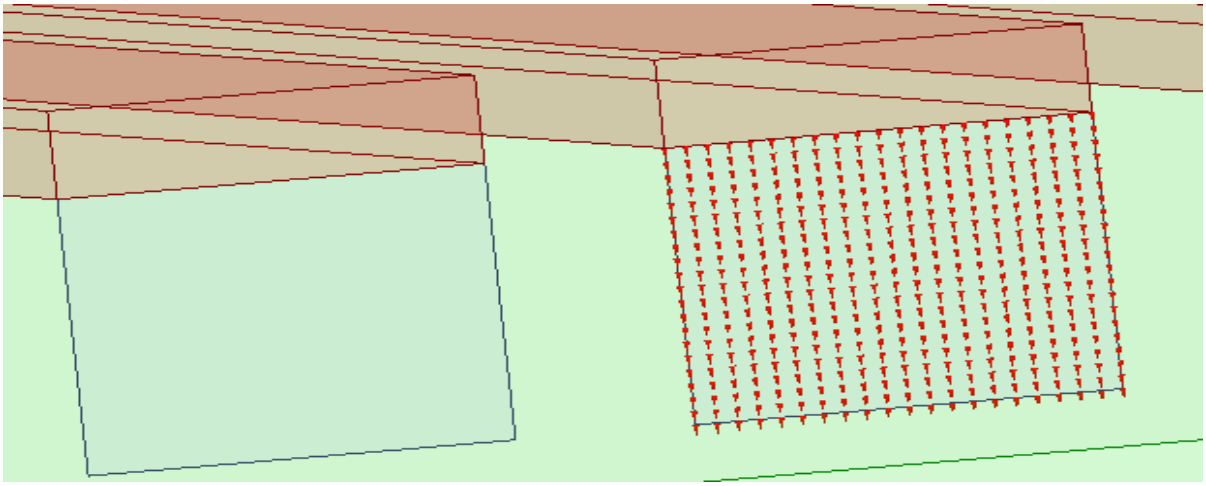
Lumped ports support single mode excitations when S-parameters have to be extracted at internal locations of a model. It can also be used to represent a terminal of a passive component to be subsequently optimized in a circuit simulator using S-matrix description of the model.

For designs with only lumped ports or a combination of lumped and circuit ports, the Advanced Solution Setup, **Options** tab includes an option to use Enhanced low frequency accuracy. The previous port solution procedure can experience failures at low frequencies. Once enabled, the feature is transparent to the user. There may be slight variations in the number of triangles, but these are not reported for lumped ports. Reported 3D mesh size may be different, and the number of adaptive passes may change.

For lumped ports all edges that do not touch metal are treated as perfect H boundaries. From this definition the resulting field distribution on the lumped port geometry is solved with the wave port solver. For a rectangular lumped port this results in electric fields orientated parallel to these perfect H sides. See figures below. The physical geometry of the rectangular lumped port carries current with the corresponding H fields resulting in parasitic inductance. For these same

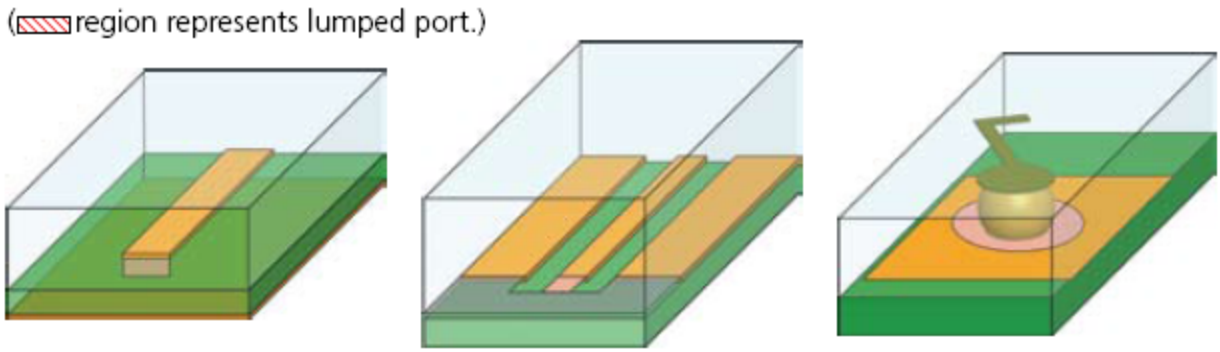
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

rectangular lumped ports the parasitic inductance can be calibrated out of the s-parameter response with the deembedding option for lumped ports.



Examples of Lumped Port

Lumped ports can be used in a variety of ways as shown below.



They vary as follows:

- For 1: The 2D port rectangle touches the signal trace with one edge while the opposite edge touches the ground plane.
- For 2: The 2D port rectangle touches the signal trace with one edge while the opposite edge touches the PEC objects (drawn in grey).
- For 3: the port is an annular ring around the BGA ball where the resulting field patterns closely resemble those for a coaxial TEM mode.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Use lumped ports only for those surfaces that are internal to the model and wave ports to model *exterior* surfaces through which a signal enters or exits the geometry.

Assign Lumped Ports for Modal Solutions

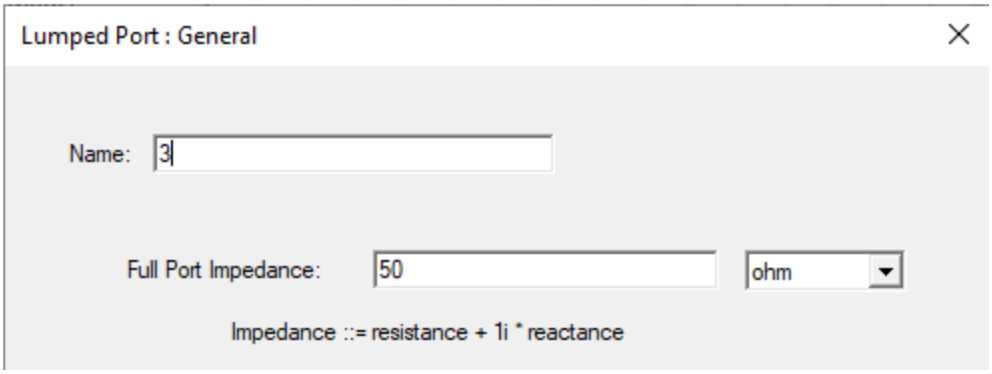
While assigning lumped ports for modal solutions, you will be prompted to set the complex full port impedance which must be non-zero, set the non-negative resistance, and finally define the integration line for the single mode. You can assign lumped ports in 3 different ways:

- Selecting a Face or Sheet and Manually Drawing an Integration Line
- Selecting Two Linear Edges that Do Not Intersect
- Selecting a Face and a Linear Edge

Selecting a Face or Sheet and Manually Drawing an Integration Line

The steps are as follows:

1. Select a surface to which you want to assign the port and click **HFSS>Excitations>Assign>Lumped Port** to bring up the **Lumped Port: General**



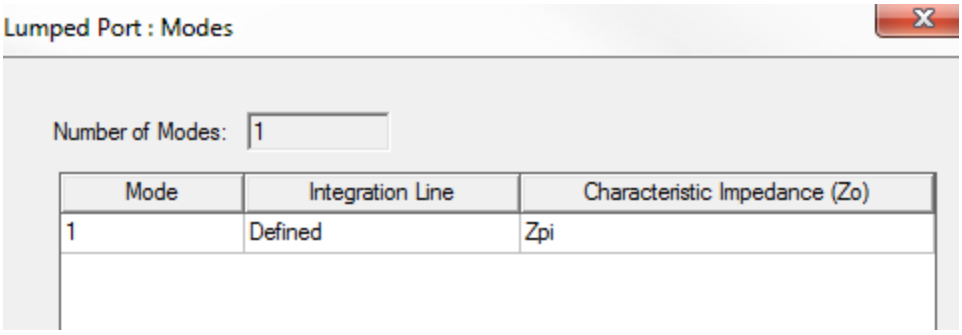
   dialog box.
2. Define the complex **Full Port Impedance** in the text field. The format is "<real_part> + <complex_part>*i ohm", for example 50ohm + (-5ohm) * 1i

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The reference impedance is meant to represent the component modeled by the lumped port. You can assign a variable to these values, (for example " resistance + (reactance) * 1i"). This variable can be dependent on the frequency, which allows use of a dataset for frequency dependent impedance, (for example, pwl (ds1,freq) + (pwl(ds2,freq)) * 1i).

3. Click **Next** to display the **Lumped Port: Modes** dialog box.

> **Note:**
>
> The field **Number of Modes** is not editable because only one mode is allowed.



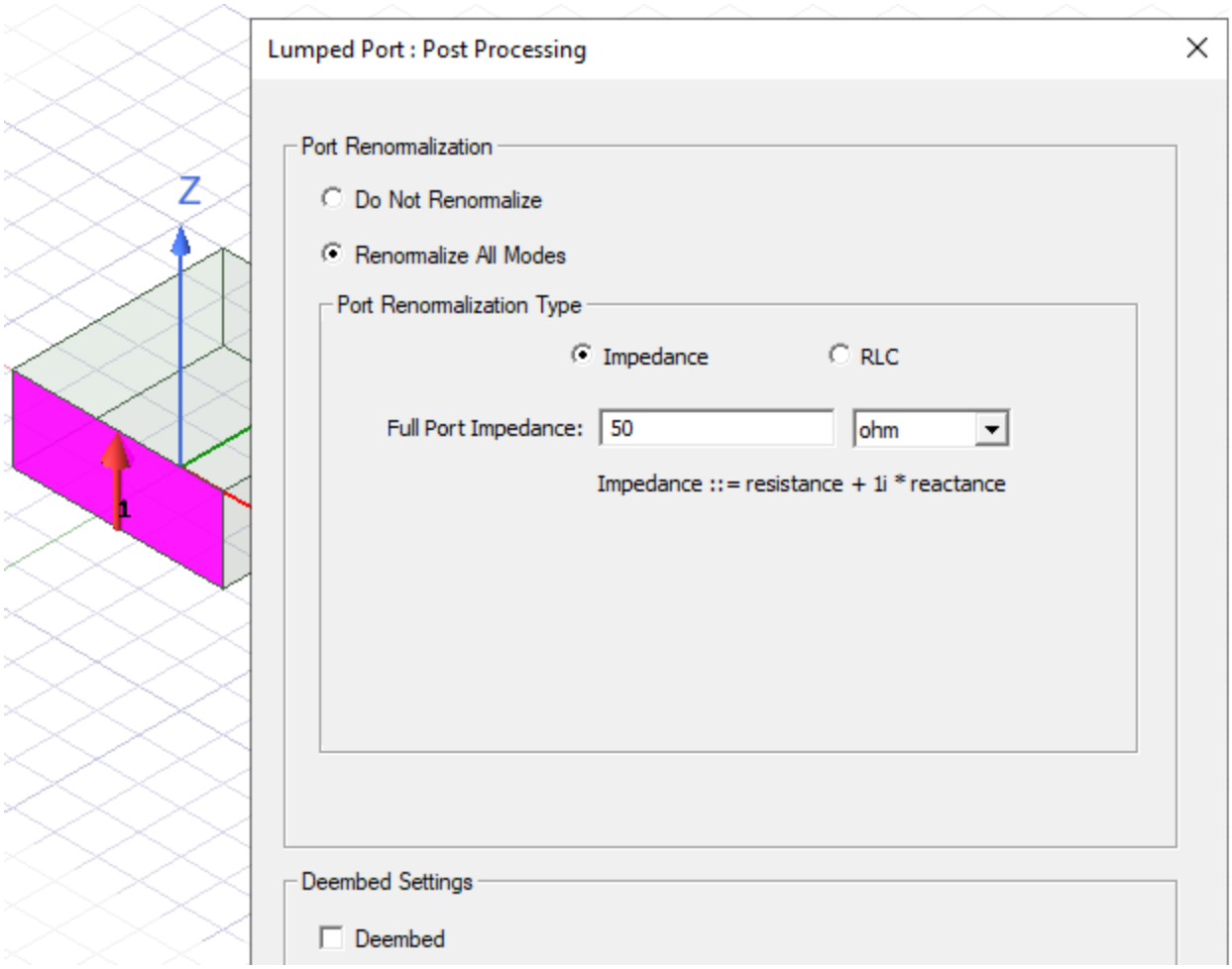
4. Integration line must be drawn for a lumped port.

   The **Characteristic Impedance (Zo)** column shows the Zpi method usually used to calculate the characteristic impedance. If Zpi is zero, HFSS uses Zpv.

   For definitions of how HFSS defines these values, see Calculating the Zpi , and Calculating the Zpv.

5. Click **Next** and the Post Processing settings appear.

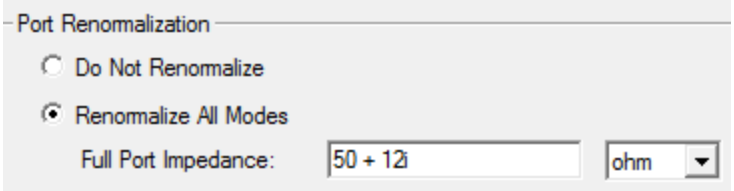
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Port Renormalization choices include:

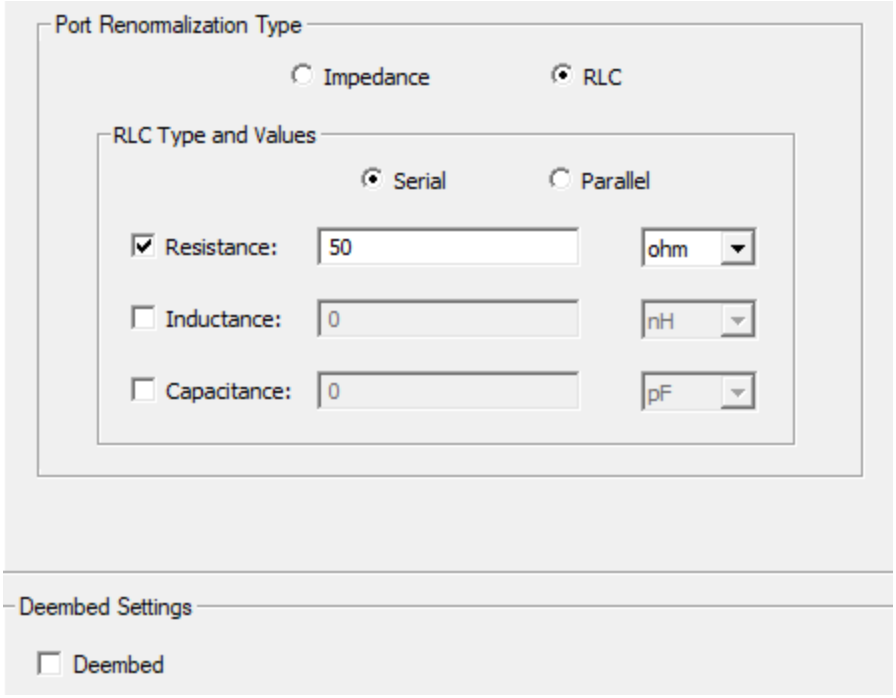
- **Do Not Renormalize:** This is the default.
- **Renormalize All Modes**: Enables the **Full Port Impedance** text box. The default impedance for re-normalization of each mode is 50 ohms.

  Complex impedance should be entered in the form: *<re> + <im>j* or *<re> + <im>i*. Either *j* or *i* entered into the field is displayed as *i* in the field and in a recorded script. See figure below.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you select RLC, you can specify Resistance, Inductance, and Capacitance and designate the RLC type as Serial or Parallel.

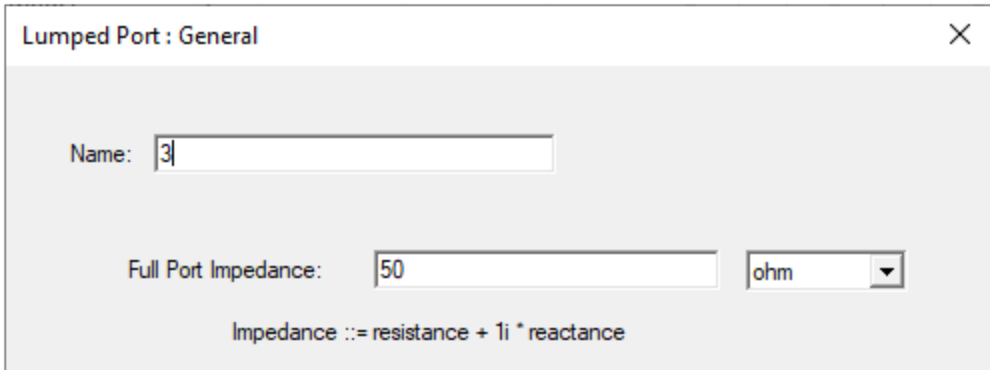


For designs with only lumped ports or a combination of lumped and circuit ports, the Advanced Solution Setup, **Options** tab includes an option to use Enhanced low frequency accuracy.

Selecting Two Linear Edges that Do Not Intersect

An integration line will be automatically drawn from the second linear edge selected to the first. The steps are as follows:

1. Set the selection mode to Edge or Multi, select the edge to which you want to assign the port and click **HFSS>Excitations>Assign>Lumped Port** to bring up the **Lumped Port: General** dialog box.

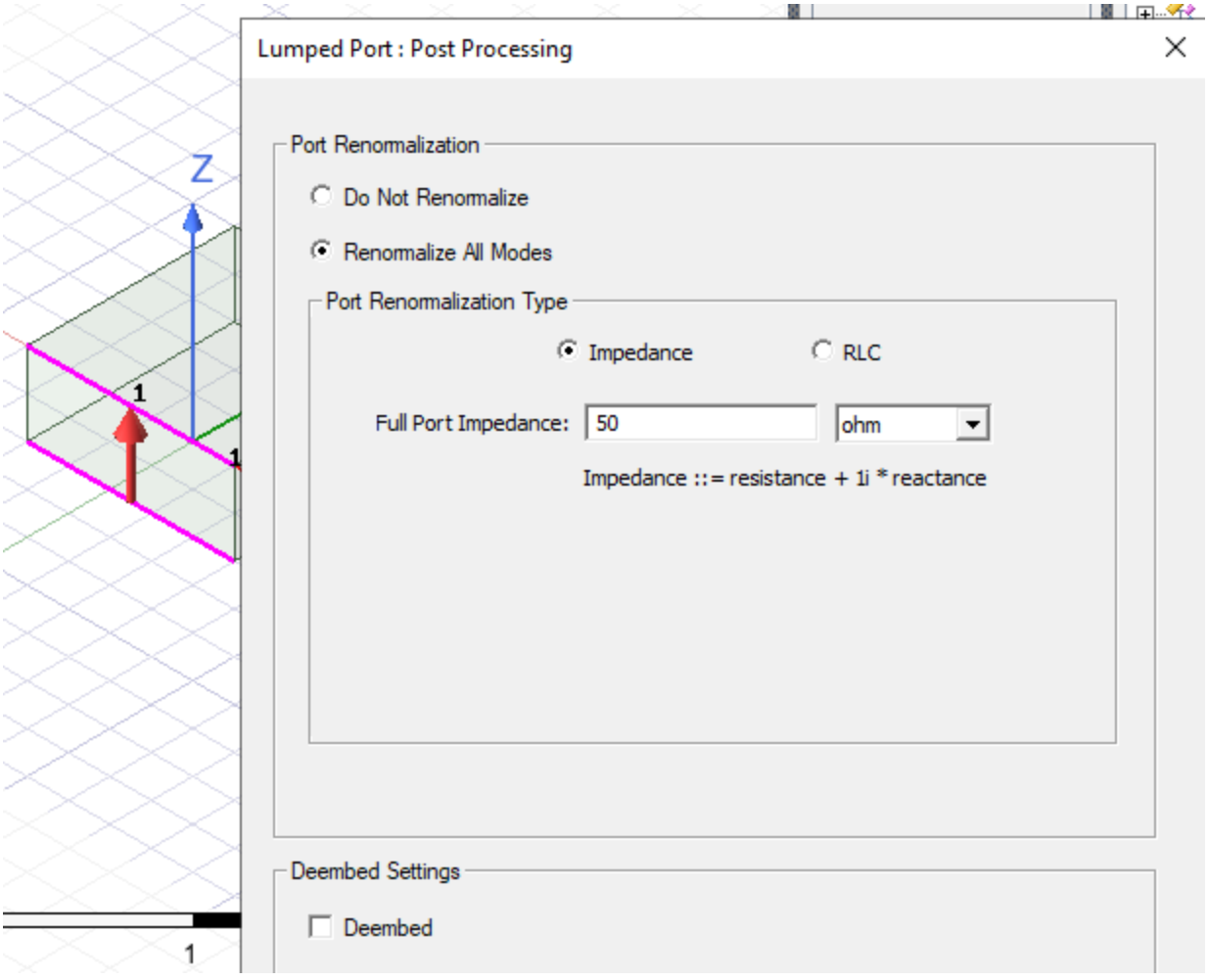


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Define the complex **Full Port Impedance** in the text field. The format is "<real_part> + <complex_part>*i ohm", for example 50ohm + (-5ohm) * 1i

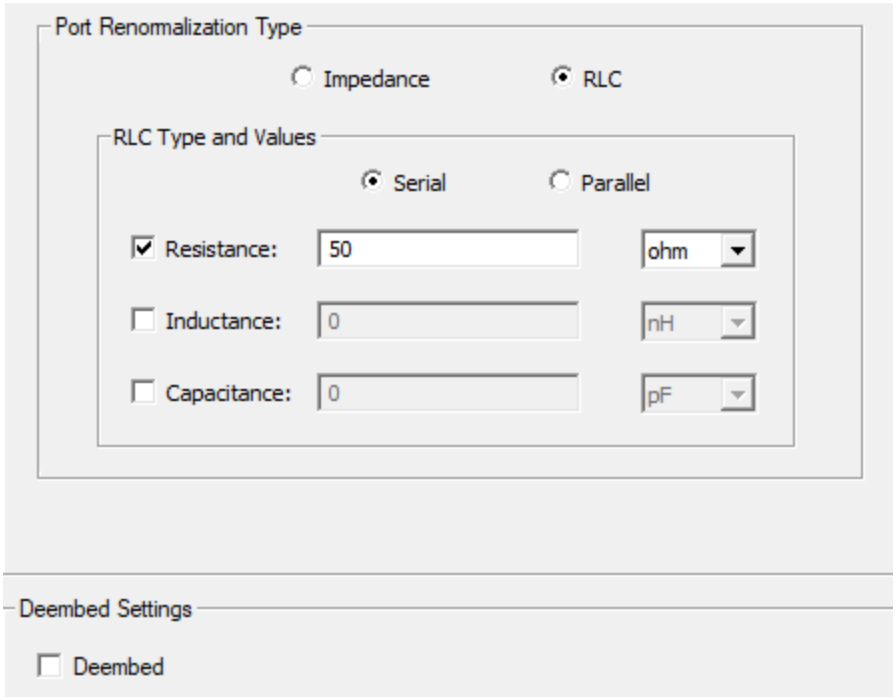
> **Note:**
>
> The reference impedance is meant to represent the component modeled by the lumped port. You can assign a variable to these values, (for example " resistance + (reactance) * 1i"). This variable can be dependent on the frequency, which allows use of a dataset for frequency dependent impedance, (for example, pwl (ds1,freq) + (pwl(ds2,freq)) * 1i).

3. Click **Next**. An integration line is automatically drawn and the Post Processing settings appear.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you select RLC, you can specify Resistance, Inductance, and Capacitance and designate the RLC type as Serial or Parallel.

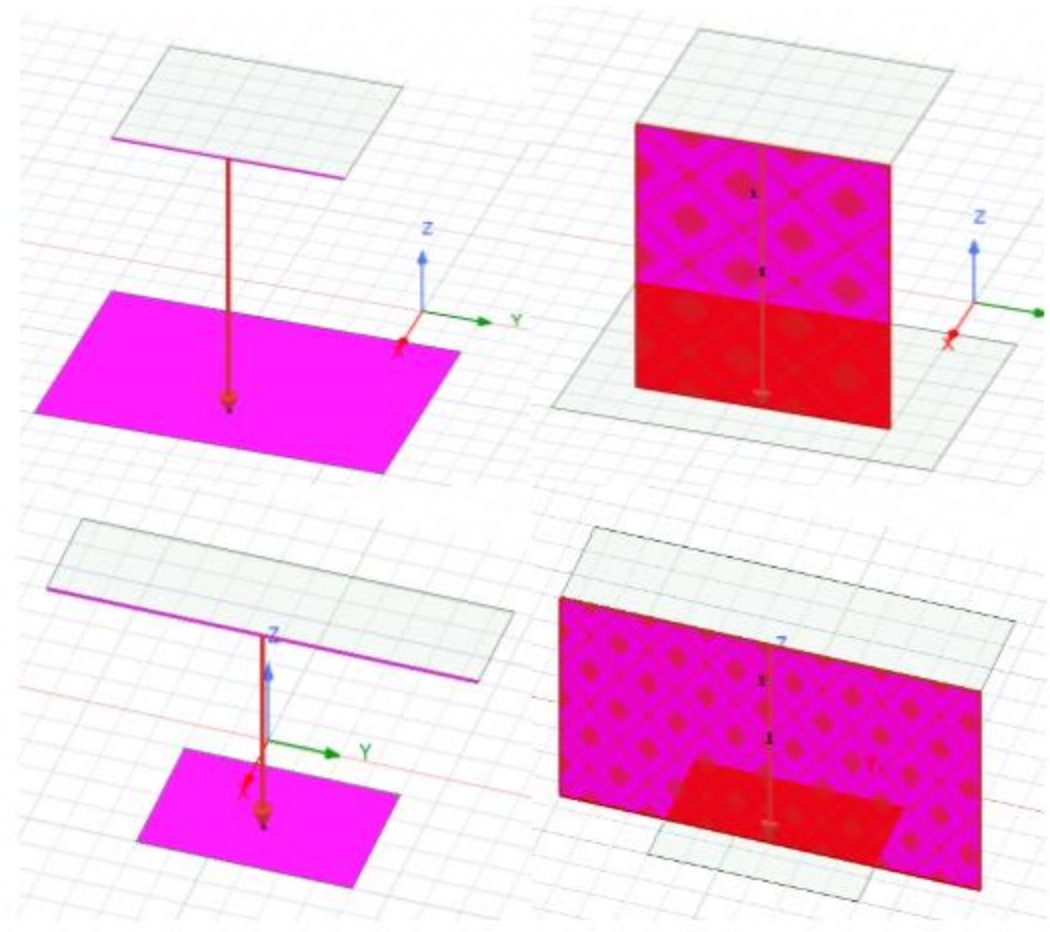


For designs with only lumped ports or a combination of lumped and circuit ports, the Advanced Solution Setup, **Options** tab includes an option to use Enhanced low frequency accuracy

Selecting a Planar Face and a Linear Edge

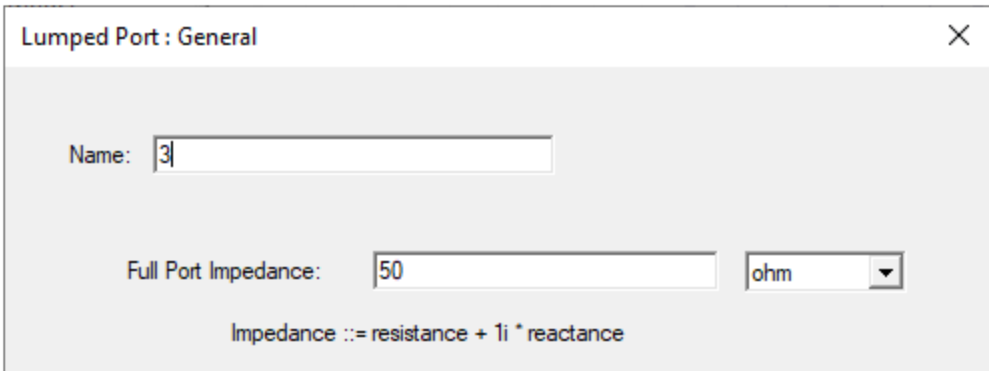
You can select exactly one linear edge and one planar face and create a lumped port to launch the dialog. For a modal lumped port, the Mode Page is skipped and an integration line is automatically created. As to the associated sheet created for the lumped port, if the edge intersects with the face, or the edge is vertical to the face, or most part of the edge cannot be projected on the face, an error message is issued. Otherwise, the 2 endpoints of the edge and the 2 projected point of the face are used to create a sheet that is vertical to the selected face.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The steps are as follows:

1. Select a face and a linear edge where you want to assign the lumped port and click **HFSS>Excitations>Assign>Lumped Port** to bring up the **Lumped Port: General**



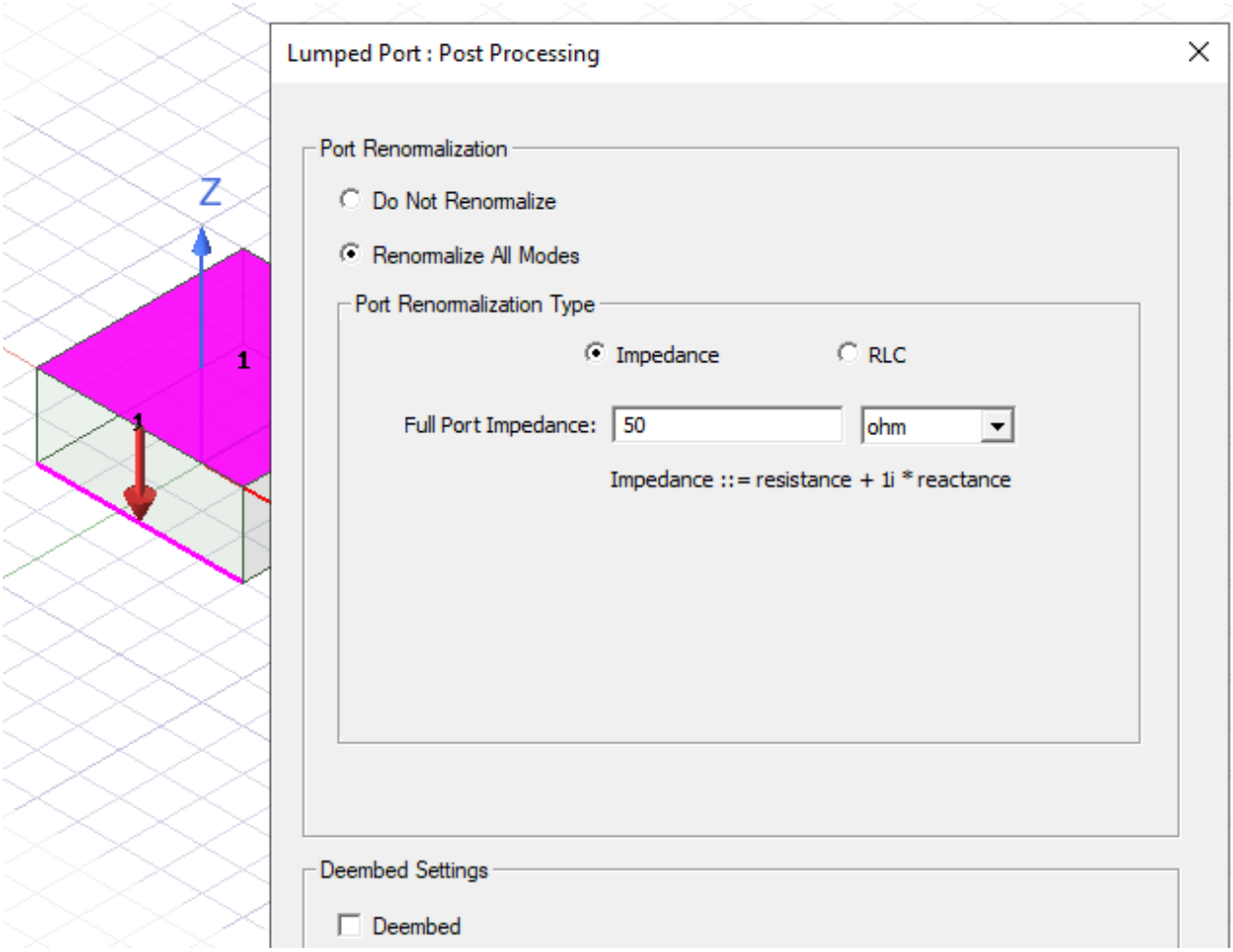
dialog box.

2. Define the complex **Full Port Impedance** in the text field. The format is "<real_part> + <complex_part>*i ohm", for example 50ohm + (-5ohm) * 1i

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

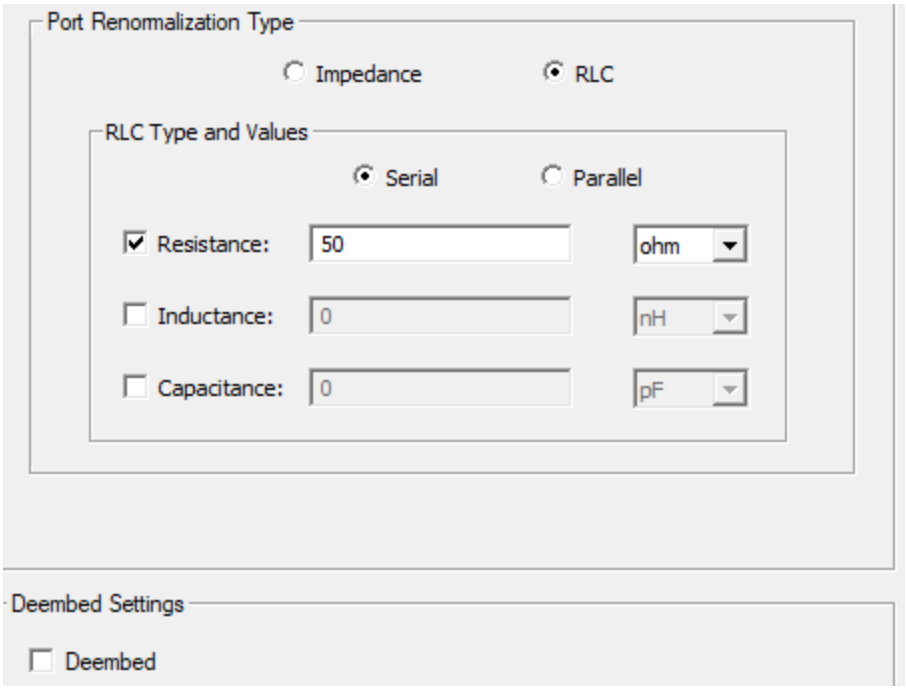
> **Note:**
>
> The reference impedance is meant to represent the component modeled by the lumped port. You can assign a variable to these values, (for example " resistance + (reactance) * 1i"). This variable can be dependent on the frequency, which allows use of a dataset for frequency dependent impedance, (for example, pwl (ds1,freq) + (pwl(ds2,freq)) * 1i).

3. Click **Next**. An integration line is automatically drawn. Click **Next** and the Post Processing settings appear.



If you select RLC, you can specify Resistance, Inductance, and Capacitance and designate the RLC type as Serial or Parallel.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

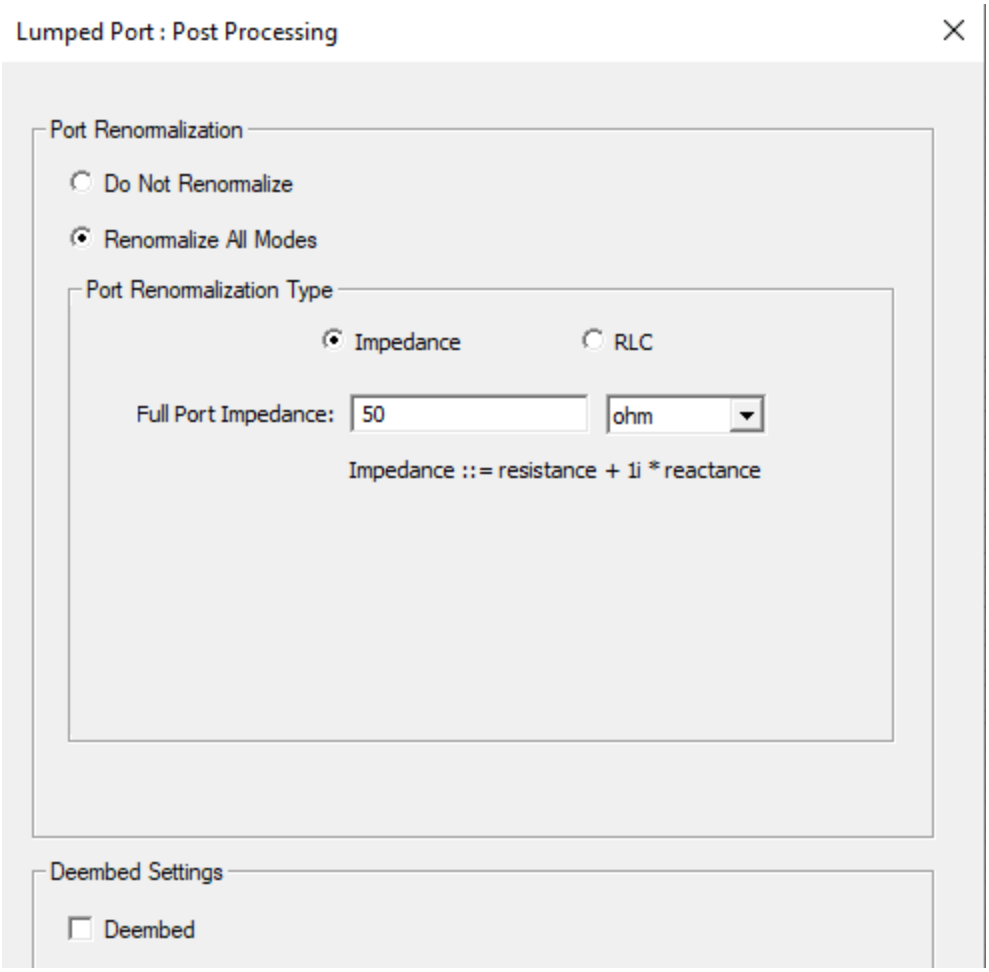


For designs with only lumped ports or a combination of lumped and circuit ports, the Advanced Solution Setup, **Options** tab includes an option to use Enhanced low frequency accuracy.

Lumped Port: Post Processing

The parasitic inductance can be calibrated out of the s-parameter response with the deembed option for lumped ports. Click **Next** on the **Lumped Port: Modes** dialog box, to bring up the **Lumped Port: Post Processing** dialog.

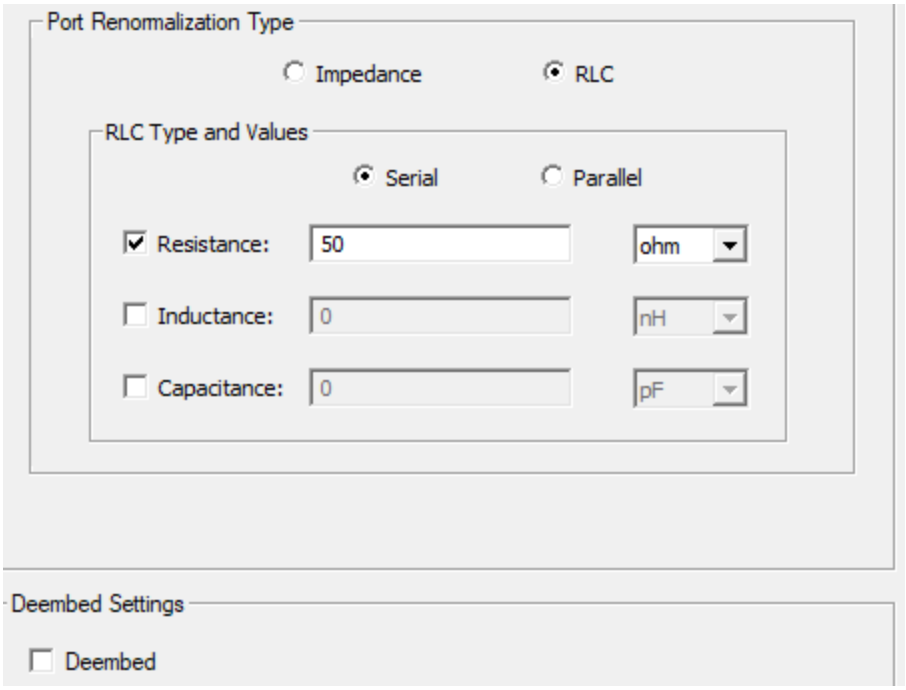
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Values here affect S-Parameters. They will also affect the spectral fields if you selected the **Include Post Processing Effects** on the **Edit post process sources** dialog box. By default, lumped ports are renormalized to a 50 Ohm full port impedance. You can override the default by entering a value of your choice in the **Full Port Impedance** field. If you want to enter a complex impedance, enter it in the form *<re> + <im>j.*, for example 50ohm + (-5ohm) * 1i . You can assign a variable to these values, (for example " resistance + (reactance) * 1i"). This variable can be dependent on the frequency, which allows use of a dataset for frequency dependent impedance, (for example, pwl(ds1,freq) + (pwl(ds2,freq)) * 1i).

If you select RLC, you can specify Resistance, Inductance, and Capacitance and designate the RLC type as Serial or Parallel.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For designs with only lumped ports or a combination of lumped and circuit ports, the Advanced So

Lump port de-embedding available on any shaped lump ports assigned. This capability de-embeds from the resulting s-parameters the parasitic inductance associated with the port's geometry

> **Note:**
>
> For more information about the port calibration technique, see the topic When HFSS Needs Port Calibration (Deembedding).

The Deembed option can be turned on or off as a post processing operation without invalidating solutions. If multiple lumped ports are selected in the Excitations List panel, changes to the

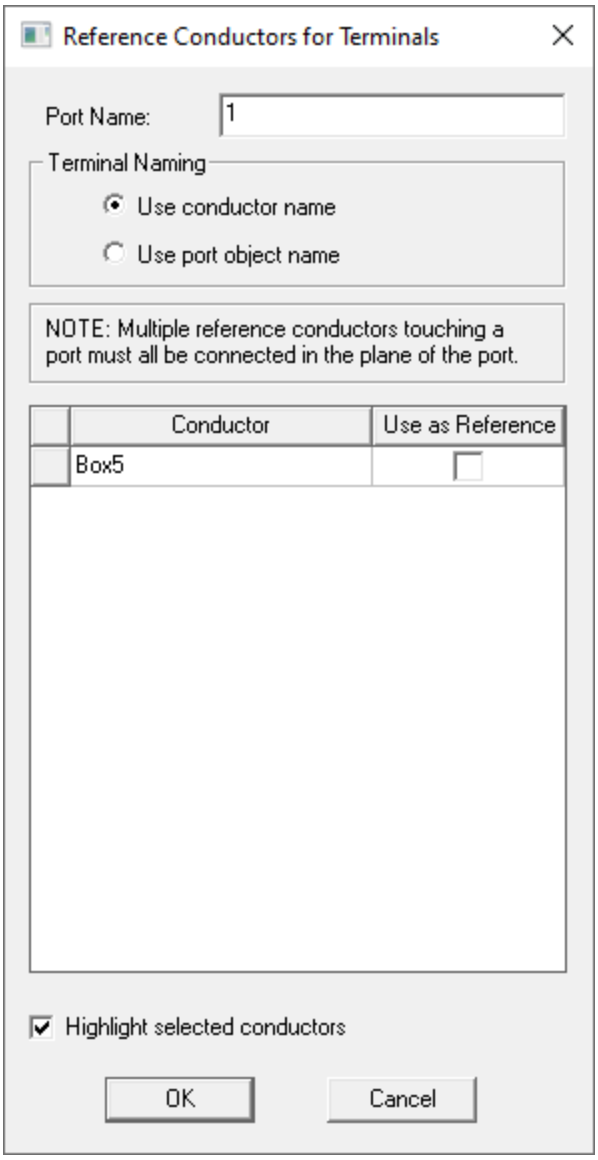
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

common Deembed property change the setting for all lumped ports at once, but only for those that support calibration.

If a port is selected for calibration but the validation checks fail for some variation (which could happen with a parametric solve), then the matrix data will not load for that variation. Also, if the fields are to include port post processing effects, then the fields do not load.

Assign Lumped Ports for Terminal Solutions

After making selections for assigning a lumped port, and selecting **HFSS>Excitations>Assign>Lumped Port**, you see the **Reference Conductors for Terminals** dialog.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For details, see Reference Conductors.

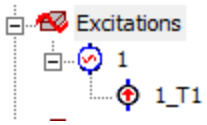
As with lumped ports for modal solutions, while assigning lumped ports for terminal solutions, you will be prompted to set the complex full port impedance. The real part (resistance) must be positive. The imaginary part (reactance) must be non-zero if the resistance is zero. There are three ways to assign lumped ports for terminal solutions.

- Selecting a Face or Sheet
- Selecting Two Linear Edges that Do Not Intersect
- Selecting a Face and a Linear Edge

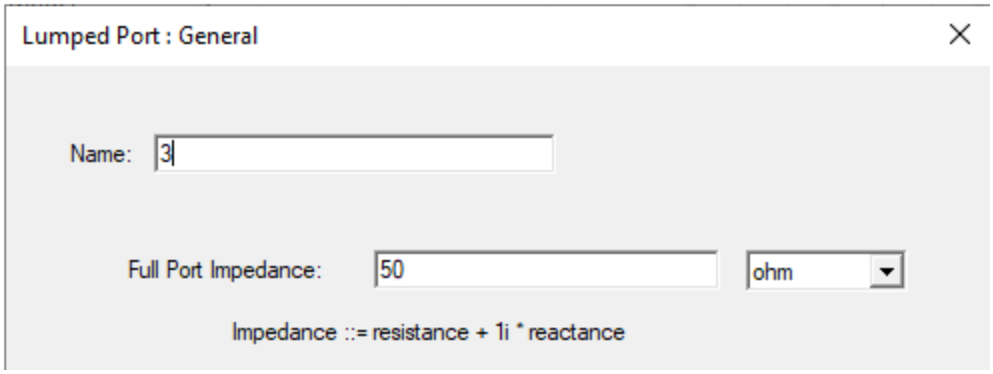
Selecting a Face or Sheet

The steps are as follows:

1. Select a surface to which you want to assign the port and click **HFSS>Excitations>Assign>Lumped Port** to bring up the Reference Conductors dialog. After you OK the Reference Conductors dialog, you see the Port assigned in the Project tree.



2. Double click on the Port to open **Lumped Port: General** dialog box.

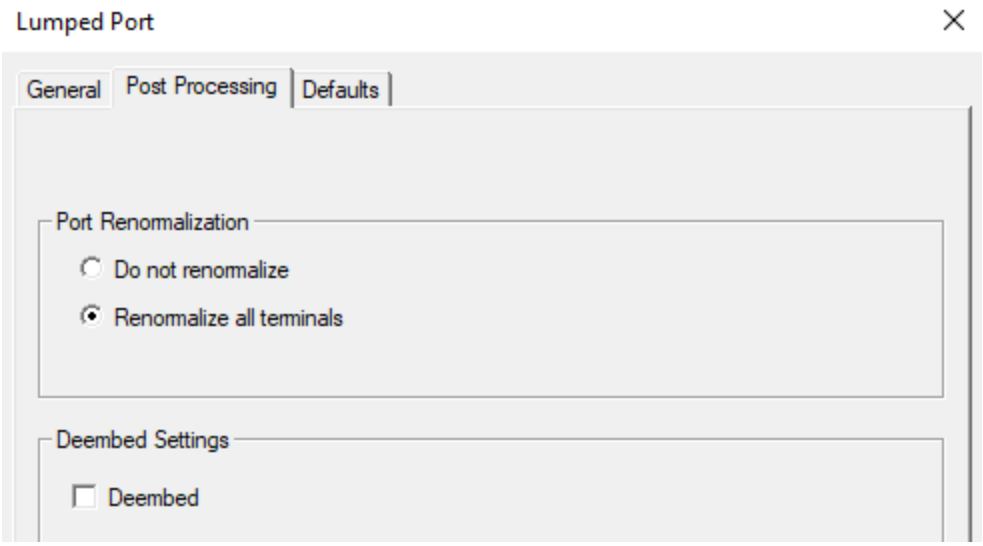


2. Define the complex **Full Port Impedance** in the text field. The format is "<real_part> + <complex_part>*i ohm", for example 50ohm + (-5ohm) * 1i

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> The reference impedance is meant to represent the component modeled by the lumped port. You can assign a variable to these values, (for example " resistance + (reactance) * 1i"). This variable can be dependent on the frequency, which allows use of a dataset for frequency dependent impedance, (for example, pwl (ds1,freq) + (pwl(ds2,freq)) * 1i).

3. Click the **Post Processing** tab to see the Post Processing settings.



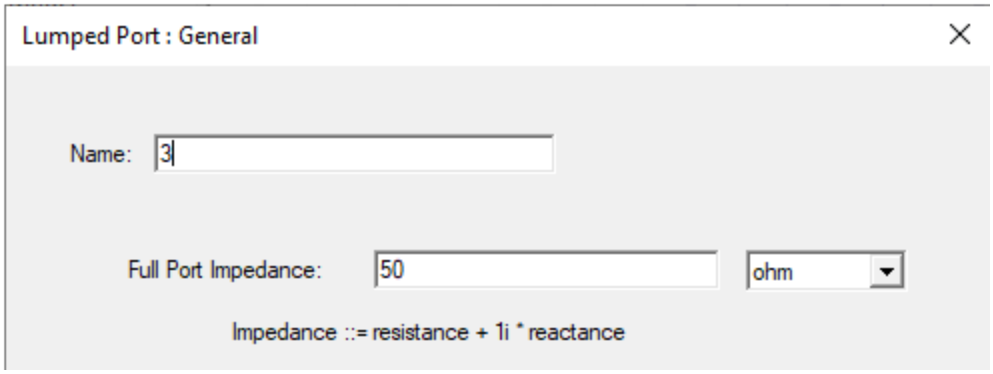
For designs with only lumped ports or a combination of lumped and circuit ports, the Advanced Solution Setup, **Options** tab includes an option to use Enhanced low frequency accuracy.

Selecting Two Linear Edges that Do Not Intersect

A terminal will be assigned to the first edge selected. The steps are as follows:

1. Set the selection mode to Edge or Multi, select the edge to which you want to assign the port and click **HFSS>Excitations>Assign>Lumped Port** to bring up the Reference Conductors dialog. After you **OK** the Reference Conductors dialog, you see the **Lumped Port: General** dialog box.

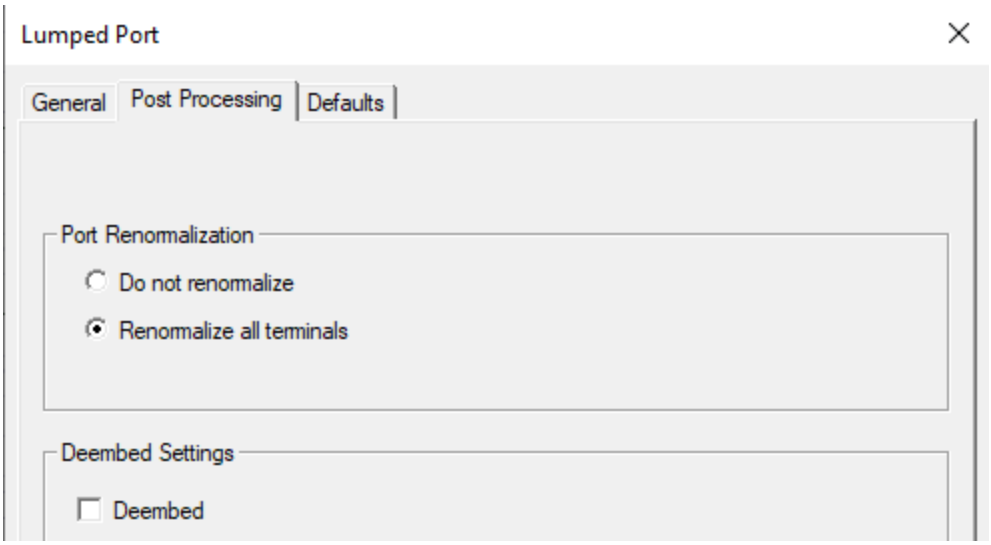
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Define the complex **Full Port Impedance** in the text field. The format is "<real_part> + <complex_part>*i ohm", for example 50ohm + (-5ohm) * 1i

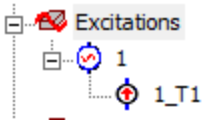
> **Note:**
>
> The reference impedance is meant to represent the component modeled by the lumped port. You can assign a variable to these values, (for example " resistance + (reactance) * 1i"). This variable can be dependent on the frequency, which allows use of a dataset for frequency dependent impedance, (for example, pwl (ds1,freq) + (pwl(ds2,freq)) * 1i).

3. Click **Next**. The Post Processing settings appear.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

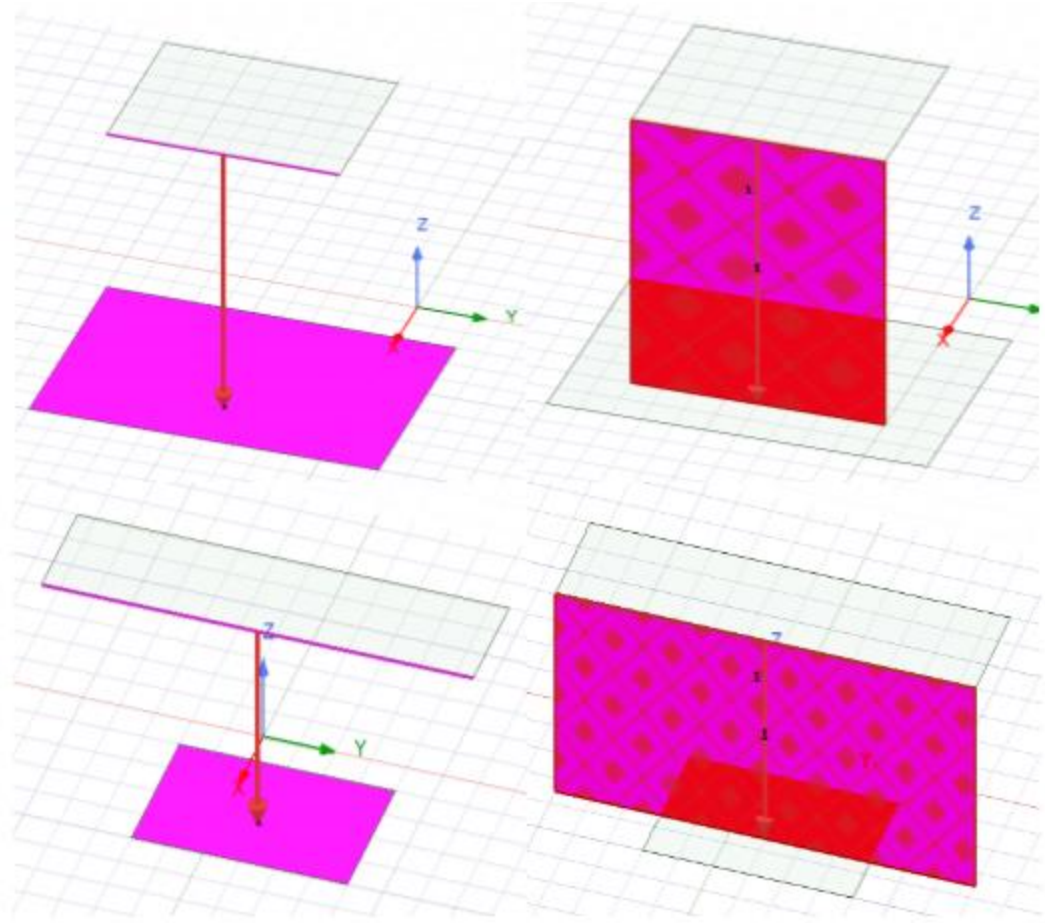
4. Click **Finish** to see the Port in the Project tree.



For designs with only lumped ports or a combination of lumped and circuit ports, the Advanced Solution Setup, **Options** tab includes an option to use Enhanced low frequency accuracy

Selecting a Planar Face and a Linear Edge

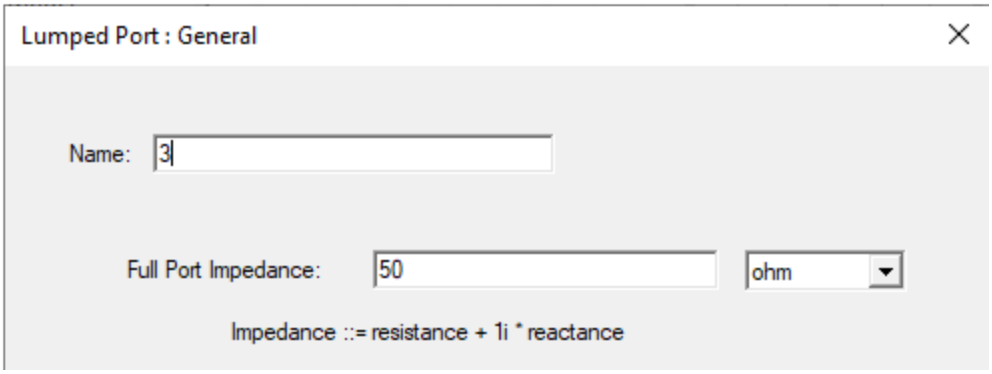
You can select exactly one linear edge and one planar face and create a lumped port to launch the dialog. As to the associated sheet created for the lumped port, if the edge intersects with the face, or the edge is vertical to the face, or most part of the edge cannot be projected on the face, an error message is issued. Otherwise, the 2 endpoints of the edge and the 2 projected point of the face are used to create a sheet that is vertical to the selected face.



The steps are as follows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select a face and a linear edge where you want to assign the lumped port and click **HFSS>Excitations>Assign>Lumped Port** to bring up the **Lumped Port: General**



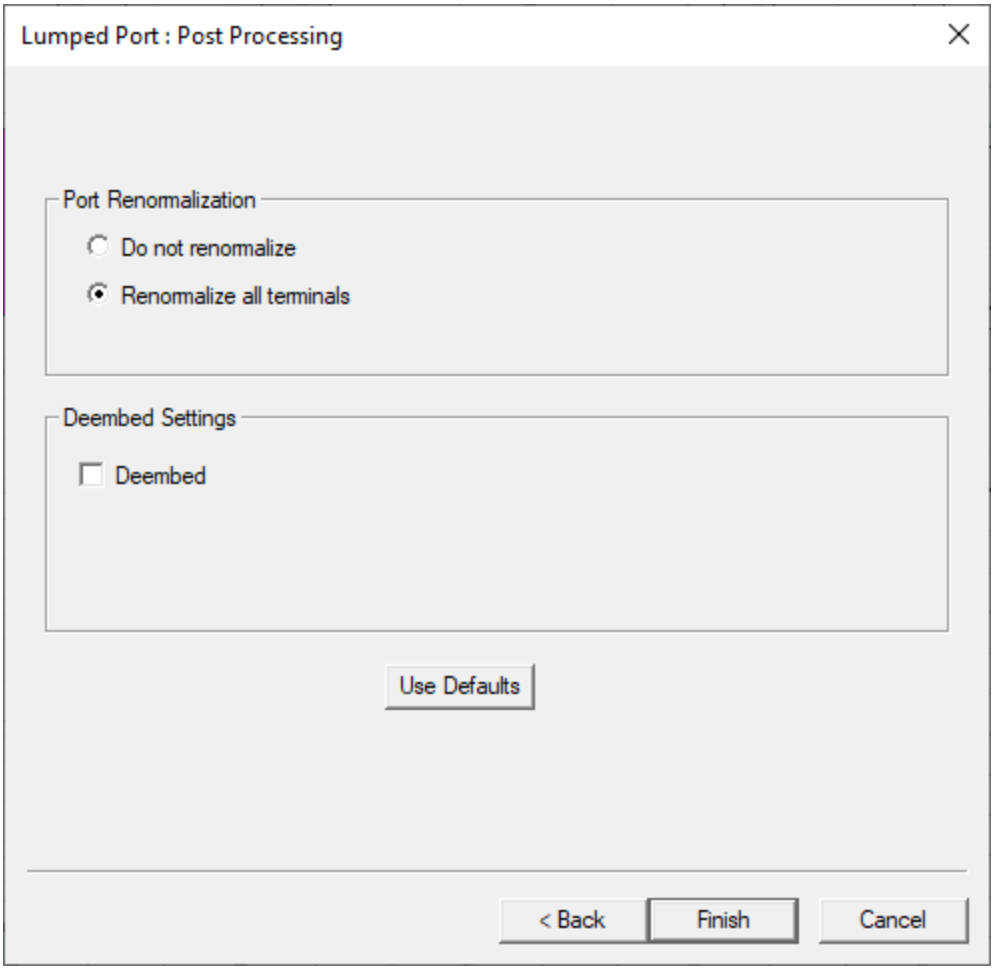
dialog box.

2. Define the complex **Full Port Impedance** in the text field. The format is "<real_part> + <complex_part>*i ohm", for example 50ohm + (-5ohm) * 1i

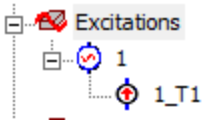
> **Note:**
>
> The reference impedance is meant to represent the component modeled by the lumped port. You can assign a variable to these values, (for example " resistance + (reactance) * 1i"). This variable can be dependent on the frequency, which allows use of a dataset for frequency dependent impedance, (for example, pwl (ds1,freq) + (pwl(ds2,freq)) * 1i).

3. Click **Next**. An integration line is automatically drawn, and the Post Processing settings appear.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click **Finish** to see the Port in the Project tree.



For designs with only lumped ports or a combination of lumped and circuit ports, the Advanced Solution Setup, **Options** tab includes an option to use Enhanced low frequency accuracy.

Floquet Ports

The Floquet port in HFSS is used exclusively with planar-periodic structures. Chief examples are planar phased arrays and frequency selective surfaces when these may be idealized as infinitely large. The analysis of the infinite structure is then accomplished by analyzing a unit cell. Linked boundaries most often form the side walls of a unit cell, but in addition, a boundary condition is

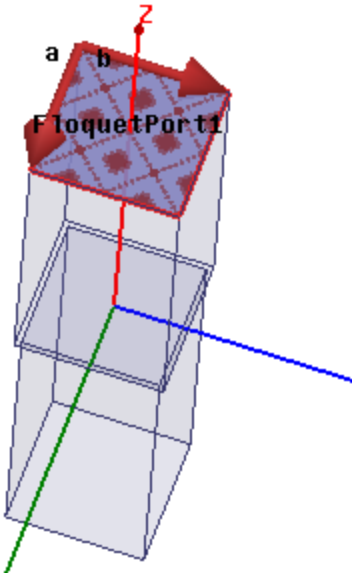
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

required to account for the infinite space above. Floquet ports must be placed along the Z axis. The Floquet port is closely related to a wave port in that a set of modes ("Floquet modes") represents the fields on the port boundary. Fundamentally, Floquet modes are plane waves with propagation direction set by the frequency, phasing, and the geometry of the periodic structure. Just like Wave modes, Floquet modes too have propagation constants and experience cut-off at low frequency. When a Floquet port is present, the HFSS solution includes a modal decomposition that gives additional information on the performance of the radiating structure. As in the case of a wave port, this information is cast in the form of an S-matrix interrelating the Floquet modes. In fact, if Floquet ports and wave ports are simultaneously present, the S-matrix will interrelate all Wave modes and all Floquet modes in the project.

For further discussion and a sample exercise, click **Help> HFSS Getting Started Guides> Floquet Ports**.

Example of Floquet Ports

As a simple example, consider an infinite array of radiating rectangular apertures in a ground plane. The following figure depicts a simple HFSS model for the unit cell of the infinite array. The model consists of two boxes. The lower box represents the feeding waveguide and the upper box is the unit cell for the region above the plane. The dimensions and geometry of the unit cell reflect the lattice vectors of the array. Linked boundaries are defined on the cell walls and a wave port provides the array element excitation. Floquet ports must be placed along the Z axis, as shown in this example.



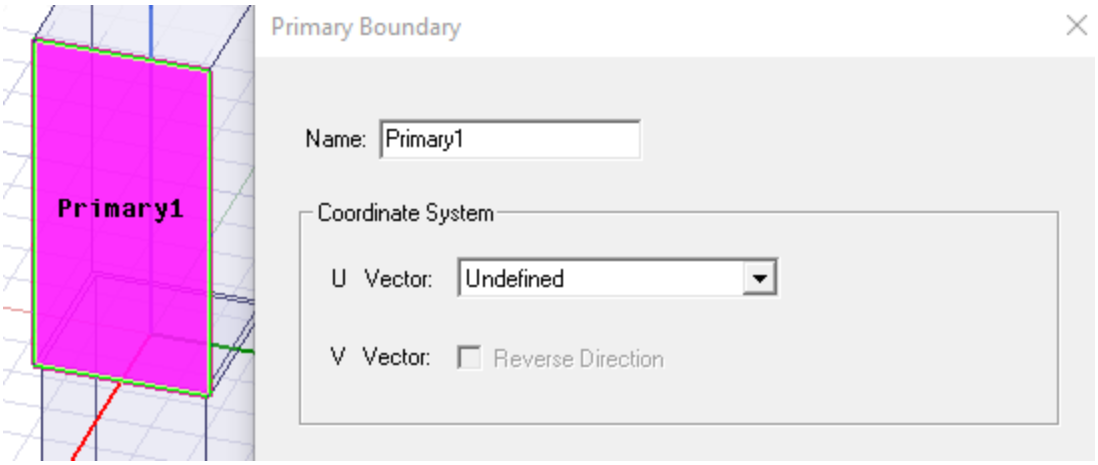
For further discussion and a sample exercise, click **Help> HFSS Getting Started Guides> Floquet Ports**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

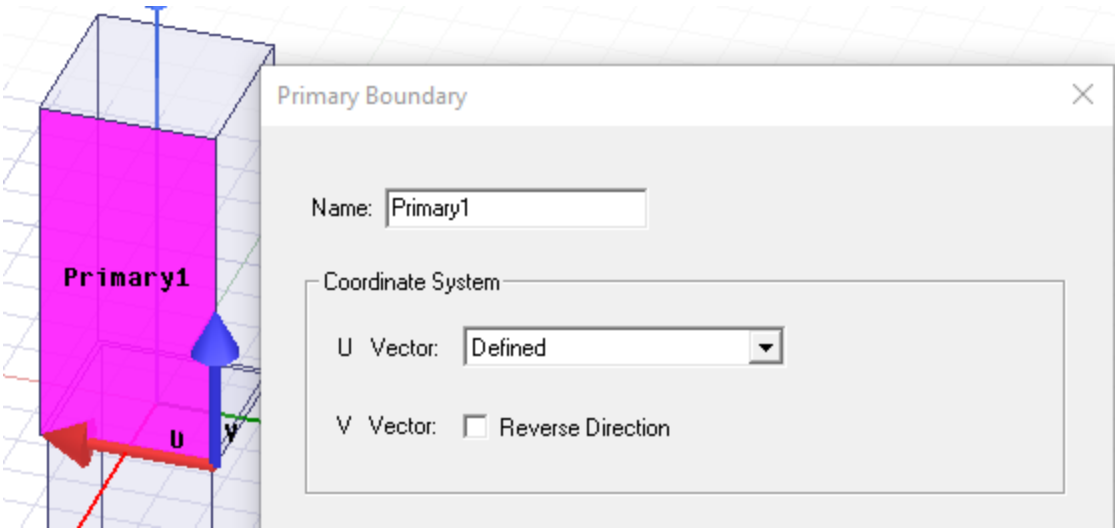
Assign Primary and Secondary Boundaries

In this section we will illustrate a key requirement for setting up a unit cell--the perimeter of a Floquet port must be covered by Primary and Secondary boundaries. You will assign the boundaries as follows:

1. Select the face of the box and click **HFSS**> **Boundaries**> **Assign**> **Coupled**> **Primary**.

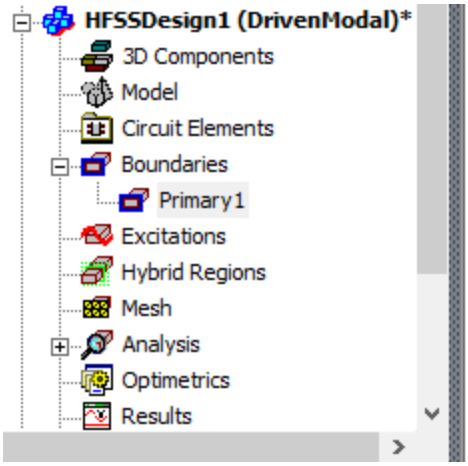


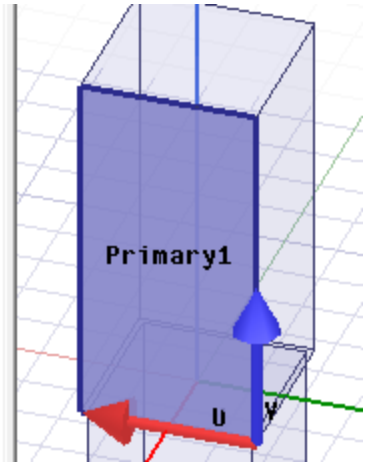
2. Select New Vector from the the U Vector menu and draw the U-V vectors as shown below, from right edge to left.



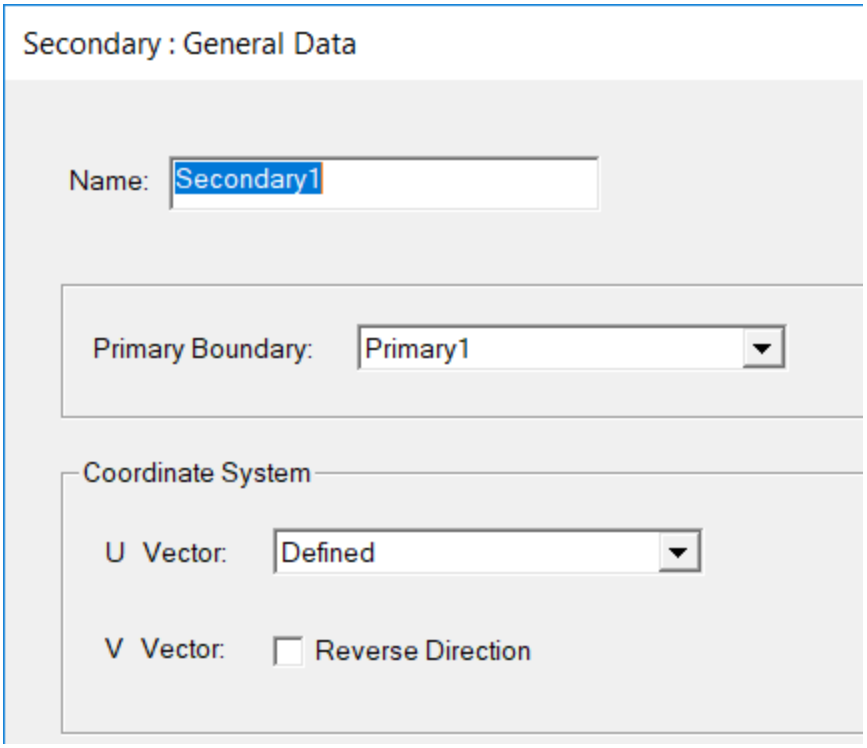
The Primary1 boundary gets assigned as shown in the following figure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



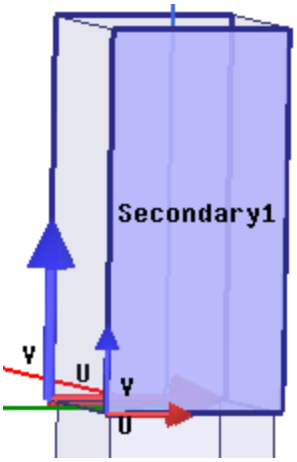


3. Access the face opposite to Primary1 and click **HFSS**> **Boundaries**> **Assign**> **Coupled**> **Secondary**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Assign the secondary boundary as shown in the following figure.



Direction of the U-V Vectors

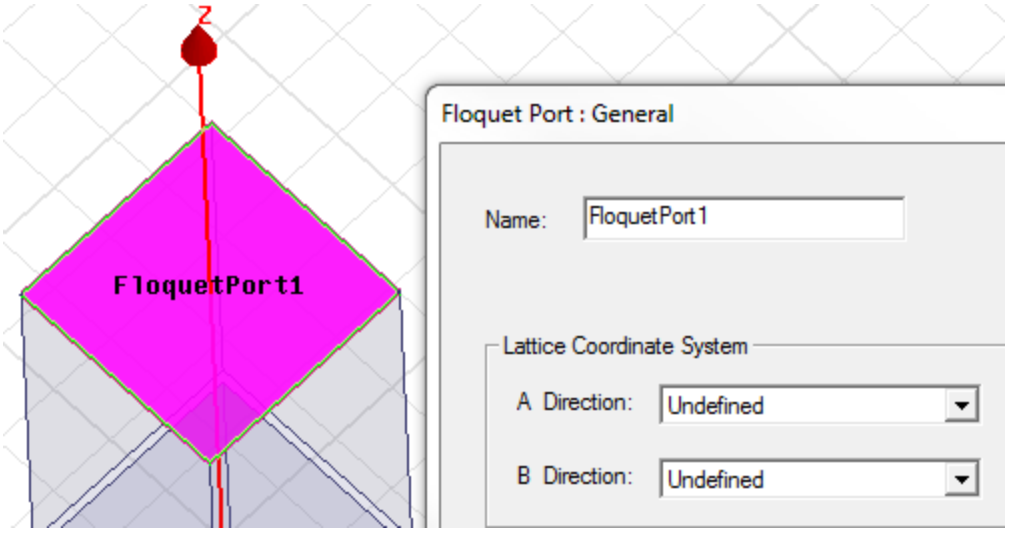
HFSS uses the U-V vectors to set up a local co-ordinate system. A point on the primary boundary must correspond to that on the secondary. A point on the secondary needs to be paired on the primary so that a one-to-one correspondence can be established. Such a co-ordinate system should be constructed and aligned properly in the same direction so that the mapping is accomplished successfully. Therefore, when the V-vector is directed downwards (when it is below the U vector), you must check the **Reverse Direction** option to keep it on the face of the box and not pointing outside of it.

Assign Floquet Ports

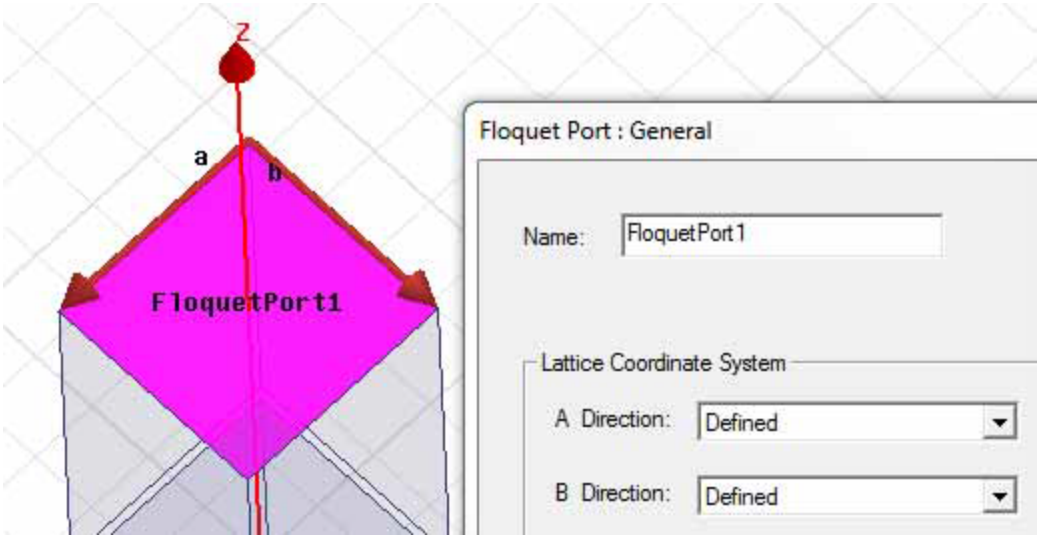
This section describes how to assign Floquet Ports on the unit cell after you define the Primary and Secondary boundaries. To assign the Floquet port perform the following steps.

1. Floquet ports must be placed along the Z axis, as shown. Select the top face of the unit cell, right-click, and select **Assign**> **Excitation**> **Floquet Port** from the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

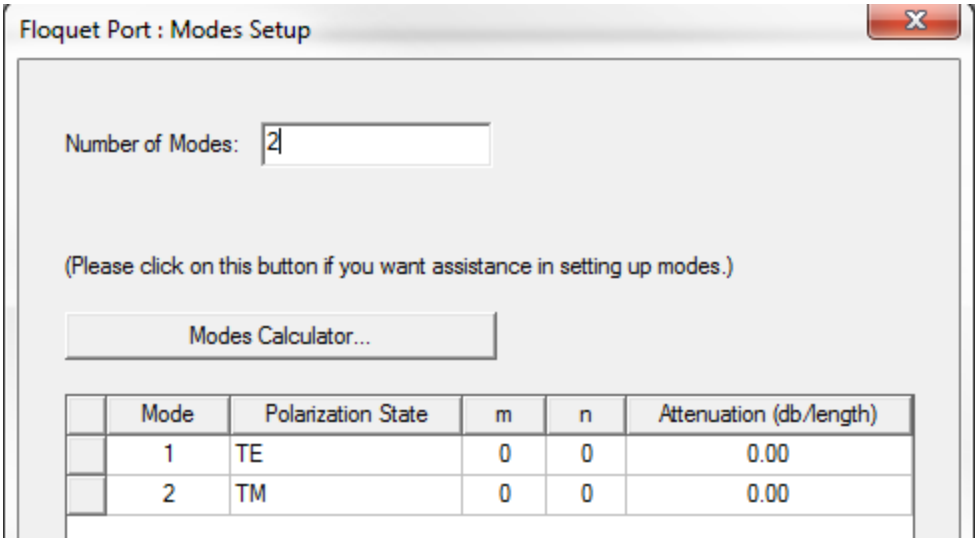


2. Specify the A and B directions for the Lattice coordinate system. These define the periodicity of the planar lattice. The vector arrows must start and end at points on the face of the Floquet port and must have a common initial point.



3. The **Modes Setup** window displays a field for the **Number of Modes**, a button for access to the **Modes Calculator**, and a table.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. On the post processing panel, if needed, the deembed distance can be specified.

Floquet Port Dialog Box

This section describes the settings on each of the tabs available on the **Floquet Port** dialog box.

Topics in this Section:

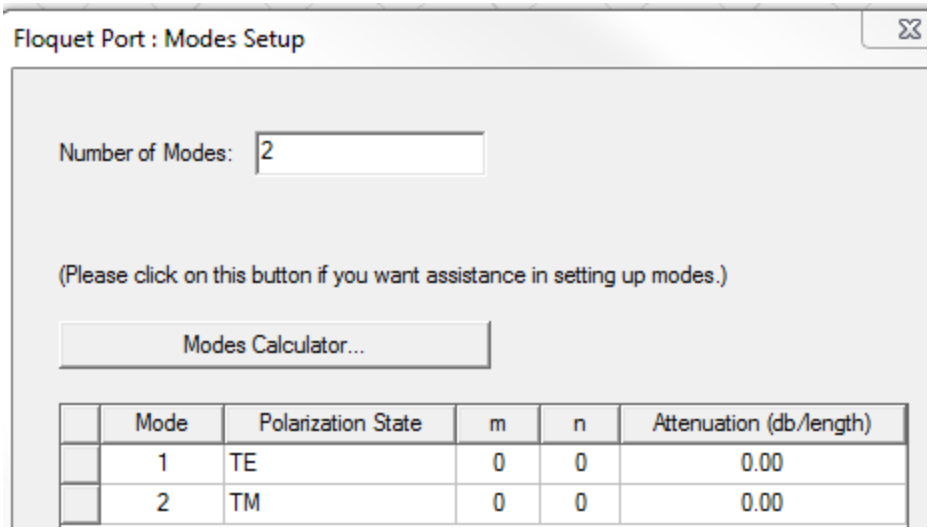
- Floquet Port: Mode Setup
- Floquet Port: Post Processing
- Floquet Port: 3D Refinement
- Floquet Ports: Lattice Coordinate System
- Floquet Port: Modes Calculator

For further discussion click **Help> HFSS Getting Started Guides> Floquet Ports**.

Floquet Port: Mode Setup

In general Floquet modes are specified by two modal indices and a polarization setting. These designations resemble the textbook notation for rectangular waveguide modes, such as ``TE10."

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The default mode table specifies a pair of Floquet modes. The default modes both have modal indices equal to zero and are sometimes called "specular" modes, which are always an essential part of the Floquet mode set. For general frequency and scan conditions, other higher-order Floquet modes will be needed. A modes calculator, invoked by selecting the **Modes Calculator** button, is available to set these up for the user.

The values under **Attenuation** represent the modal loss in amplitude along the direction normal to the Floquet port plane. The numbers in this column are computed by the modes calculator that will help you decide which modes to keep.

Attenuation for a mode is a function of both the frequency and the scan angle specified in the modes calculator. When the latter includes more than one scan direction, the least amount of attenuation experienced by the mode over all the specified scan directions.

Thus when the table gives a value of 0 dB, at one or more scan directions specified in the modes calculator the particular mode propagates without attenuation. Similarly, when the table displays say 60 dB, a 60 dB per unit length is the least amount of attenuation at all specified scan directions. At any given direction, only the same or larger attenuations (for example, 70 dB per length) will occur. To improve simulation efficiency and interpret the results easily eliminate any modes that are not necessary. Do this by editing the **Number of Nodes** field in the **Modes Setup** tab.

> **Note:**
>
> The list is trimmed from the bottom.

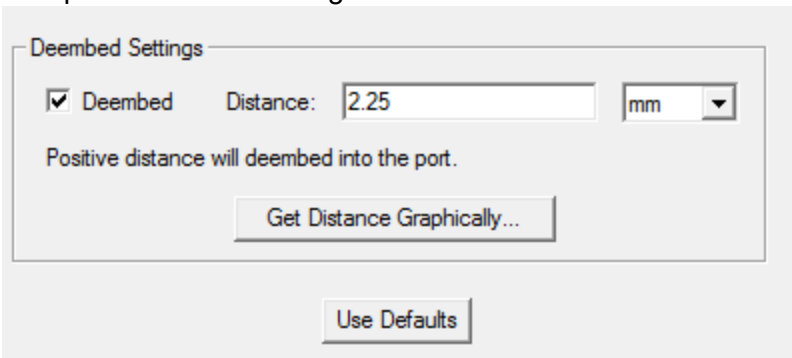
To change the order of items in the final Modes list, drag each corresponding line by the square box at the left of each row.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Floquet Port: Post Processing

This tab lets you specify a de-embedding distance. This optional post processing step is employed when you are interested in the phase of the S parameter elements. The interface for deembedding a Floquet port is the same as that for a wave port.

This panel contains settings that affect the fields once the field solution is complete.

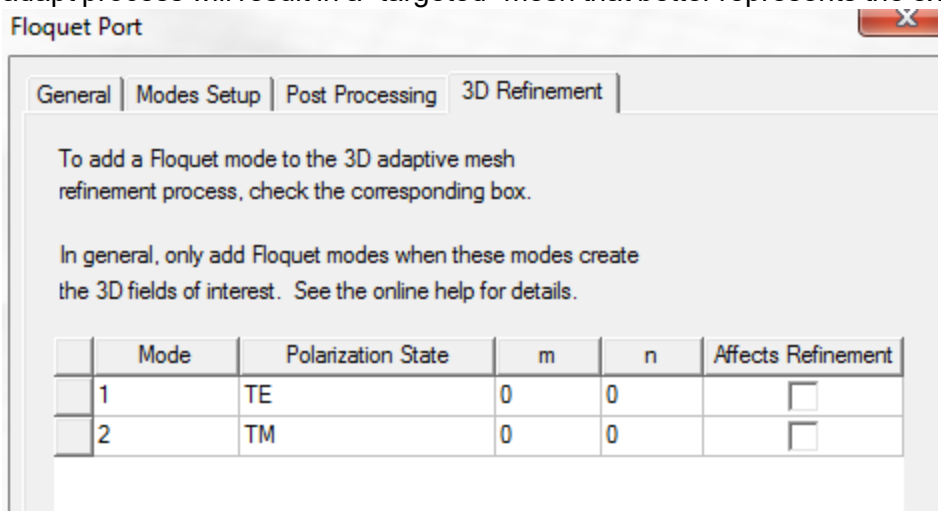


This enables the distance and units field for the positive value to deembed the port into the mode, and for a negative value to deembed the port out of the model.

Floquet Port: 3D Refinement

This panel contains the **Affects Refinement** check boxes which allows you to specify **Floquet** modes in the 3D adaptive refinement process. Typically, you select no modes or only one or both specular (TE00 or TM00) modes. Selecting more than this may affect the efficiency and accuracy of the solution process.

In 3D adaptive refinement the generated mesh is a compromise which simultaneously represents the 3D field patterns of every mode included in the adapt process. If the field patterns of certain modes represent the fields of interest and others do not, excluding the latter from the adapt process will result in a "targeted" mesh that better represents the excitation field pattern.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For antenna array simulations in which the active impedance or embedded-element pattern is sought, the Wave or Lumped ports modeling the feed structure provide the fields of interest. In this case, no Floquet modes should be included in the adapt process. On the other hand, if the per-cell RCS is of interest, one or both specular Floquet modes provide the fields of interest and should be included in the 3D adapt process by checking the corresponding **Affects Refinement** boxes. Similarly, for an FSS simulation with two Floquet ports, specular modes provide the fields of interest and should be selected to participate in the 3D adapt process.

In certain simulations (for example, a frequency-selective surface) you will set up a second Floquet port. Note that when you do this HFSS automatically copies the lattice vectors, modes table, and 3D refinement settings from the first Floquet port to the second.

Floquet Ports: Lattice Coordinate System

In the **Floquet Port** dialog box, **General** tab, for each Vector (A and B Direction):

1. Select the drop-down menu and click **New Vector**.

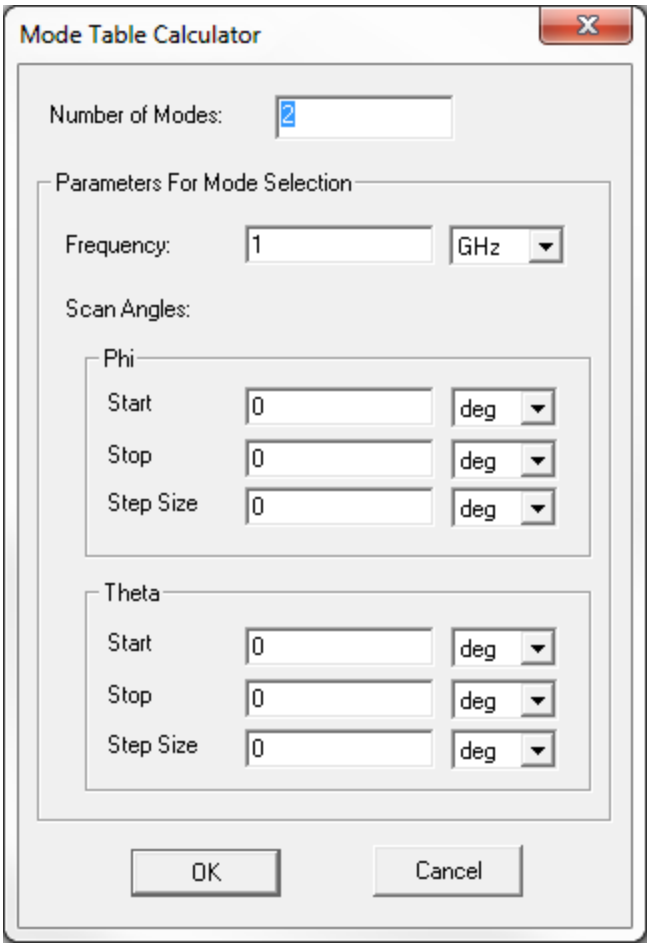
   This opens a **Measure Data** dialog and causes the cursor to drag a visual marker that drops a dashed line to the reference plane, and shows a location indicator on the Floquet plane.

2. Drag the marker to select a location for the Direction vector. Click to set the origin point, and drag and click to specify the position 2 point that defines the direction from that origin. The vector arrows must start and end at points on the face of the Floquet port and must have a common initial point.
3. Clicking the second point closes the **Measure Data** dialog box, and exits the New Vector mode. The drop-down menus for Position A and Position B now include an entry called "Defined" along with "Undefined" and "New Vector."

Floquet Port: Modes Calculator

1. In the **Floquet Port: Modes** tab, if you select the **Modes calculator** button, the **Mode Table Calculator** window displays.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Enter the following fields:

- **Number of modes** - you can trim this value later, as you learn which modes are needed and which are not.
- **Frequency** - if the problem setup contains one or more frequency sweeps, usually you will set this value to the highest frequency.
- **Scan Angles -**enter the values and select units for Phi and Theta, including Start, Stop, and Step size. The **Mode Table Calculator** calculates a set of Floquet modes on the basis of all the angles defined.

These inputs constitute the information required to create a set of recommended modes for the Floquet port. The inputs are used by the mode selection algorithm but do not affect the problem setup.

1. Click **OK** to leave the **Mode Table Calculator** and to compute the recommended list of modes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The new modes table appears on the **Modes Setup** tab of the **Floquet Port** properties/setup window. The attenuation associated with a listed mode represents the minimum (or worst case) for that mode over the range of scan angles.

Incident Waves

An incident field is the electromagnetic field in the absence of any scatterers. We suppose that the incident field is present everywhere and it comes from a source residing in some location. The source can even be another HFSS project or an SIwave project.

Incident waves can be of the following types:

- Plane Wave
- Hertzian-Dipole Wave
- Cylindrical Wave
- Gaussian Beam
- Linear Antenna Wave

For incident analytical waves (plane waves through linear antenna waves) HFSS supports two basic methods: scattered field formulation and total field formulation.

Etotal = **E**incident + **E**scattered

The total field formulation is useful for viewing weak total fields while scattered field formulation is useful for viewing weak scattered fields.

For near field and far field waves we are merely specifying the fields on the radiation surfaces. So, we cannot view only the total fields.

Radiation Boundary Panel

When assigning an incident wave either a radiation boundary, PML boundary, or Hybrid Region Finite Element Boundary Integral (FE-BI) method must be defined at least on one section of the surface of the model. The Radiation Boundary panel is shown below.

For more information see Assigning Radiation Boundaries

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To assign a FE-BI, use Hybrid>Assign>FE-BI.

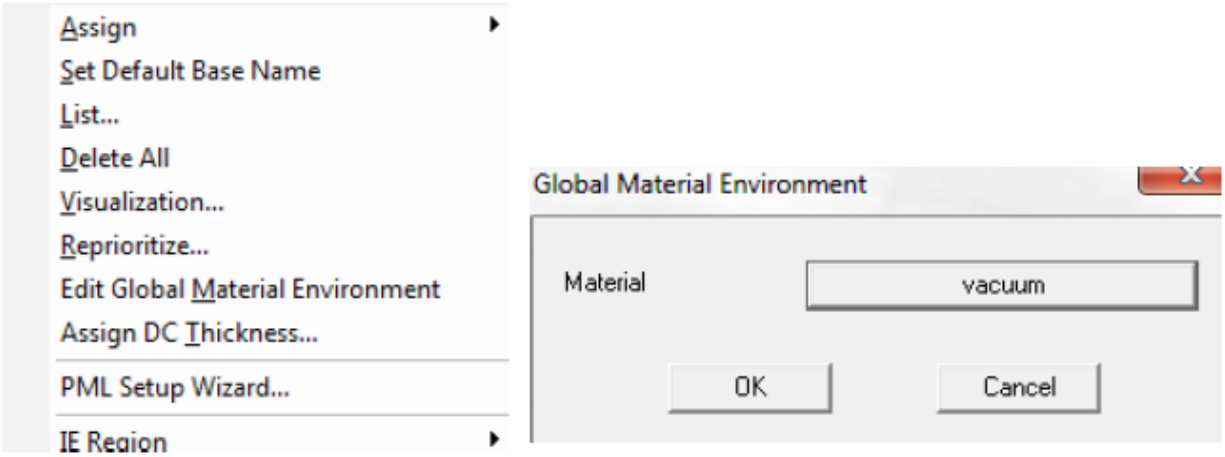
Global Material Environment

A plane wave is described by the following equation.

$$\vec{E} = \vec{E}_0 \, e^{-j\vec{k}\cdot\vec{r}}$$

The **k** vector also depends on the material of the radiation boundary. More precisely it is the wave number of the global background material.

You can set the global material environment which is otherwise vacuum by default. Right-click **Boundaries** and select **Edit Global Material Environment** to bring up the dialog box.



The material specified in the **Global Material Environment** panel can impact an HFSS design in the following three areas:

- All incident waves except evanescent plane waves and near and far field links are assumed to have been generated in the global background material.
- All fields outside the computational domain are obtained using the global background material.
- The currents on exterior HFSS-IE regions are computed assuming the global background material.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The PML and simple ABC are constructed assuming the interior material touching the radiation surface is the same as the exterior global background material. The actual material used is the interior in this case in order to allow different materials to be in contact with the radiation surface as is the case when truncating the substrate of a PCB design. To obtain consistent results when viewing fields outside the computational domain, ensure that the interior material touching the radiation surface is the same as the global background material.

Incident Plane Wave

A plane wave is a wave that propagates along a fixed direction where the electric and magnetic fields are in the transverse plane and perpendicular to each other.

A plane wave is defined by the following equation:

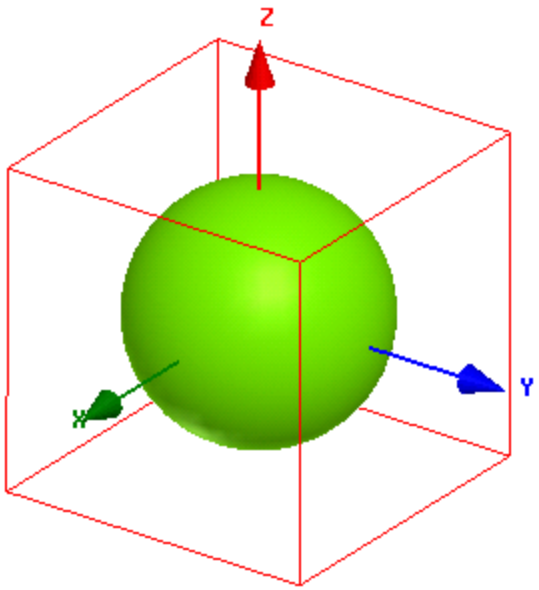
$$\vec{E} = \overrightarrow{E_0} e^{-j\vec{k}\cdot\vec{r}} \qquad \text{where} \qquad \vec{k} = k\hat{k}$$

$$\text{and} \qquad \overrightarrow{E_0} = E_0\hat{e}_0$$

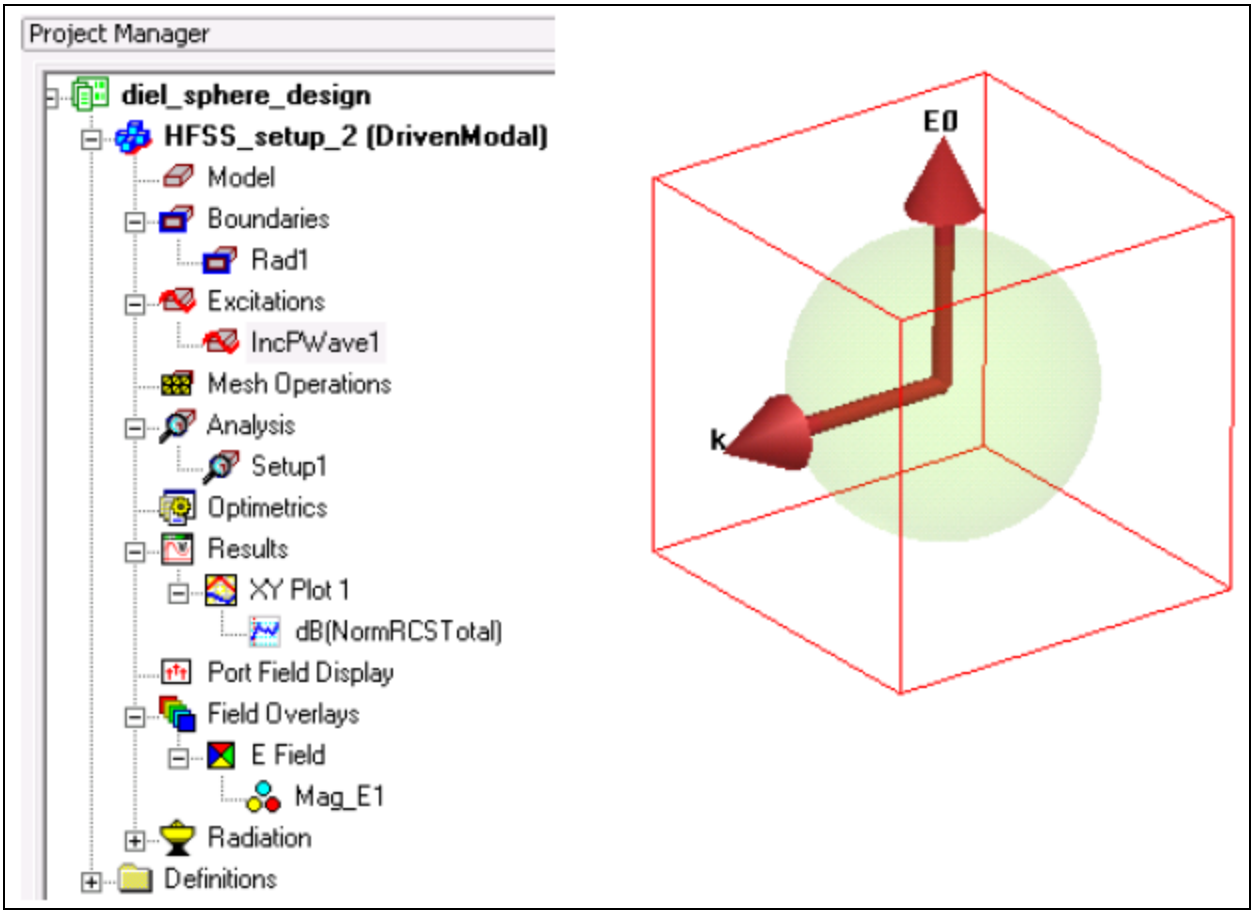
The term *k* represents the wave number of the global background material for regular/propagating plane waves.

An HFSS model of a dielectric sphere is shown to illustrate the incident plane wave setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

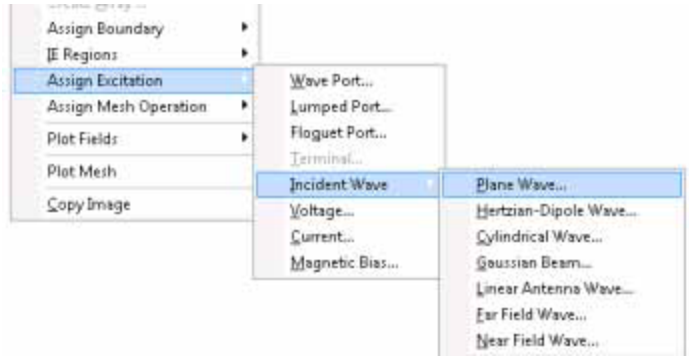


After a plane wave is defined, the propagation direction and the electric field direction can be visualized as shown in the following figure.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To define a plane wave, right-click anywhere in the Modeler and select **Assign Excitation> Incident Wave> Plane Wave.**



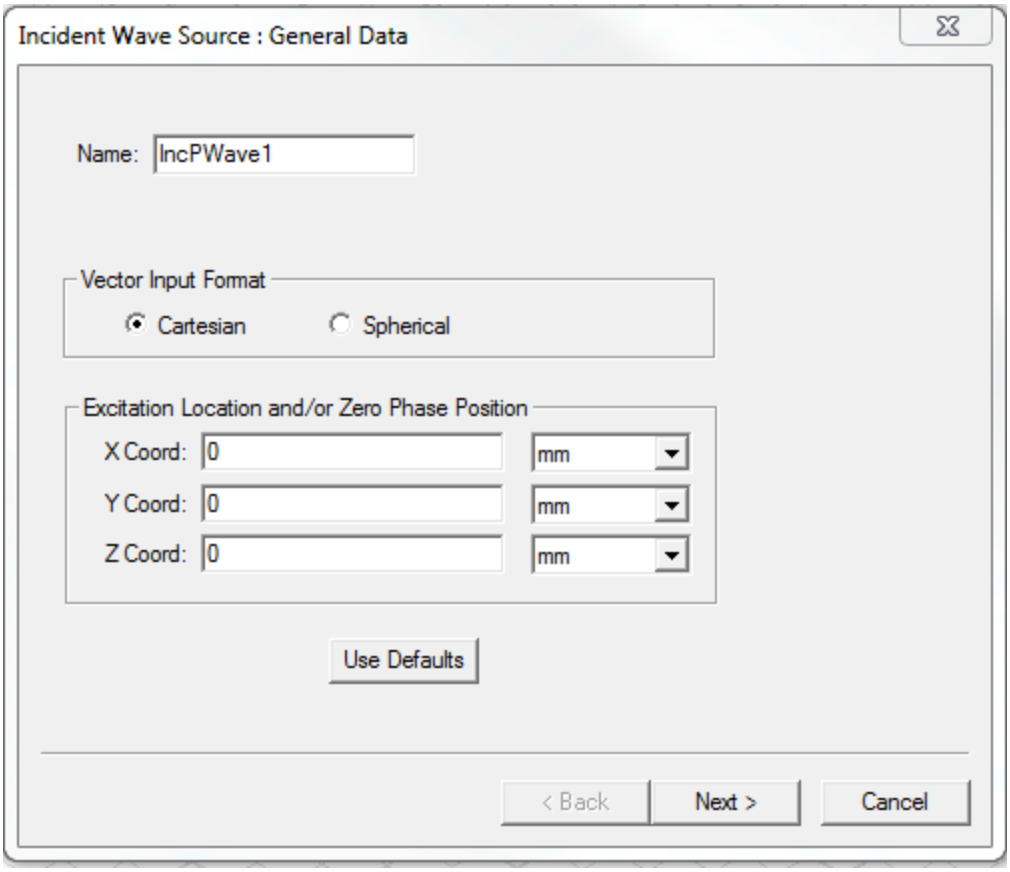
The **Incident Wave Source: General Data** dialog box appears. You can select either **Cartesian** or **Spherical** for the **Vector Input Format**.

Cartesian Vector Setup

Enter the Cartesian co-ordinates on the **Incident Wave Source: General Data** dialog box to set the zero phase location for propagating wave and click **Next**.

> **Note:**
>
> For evanescent waves, Cartesian co-ordinates are defined for the Excitation location.

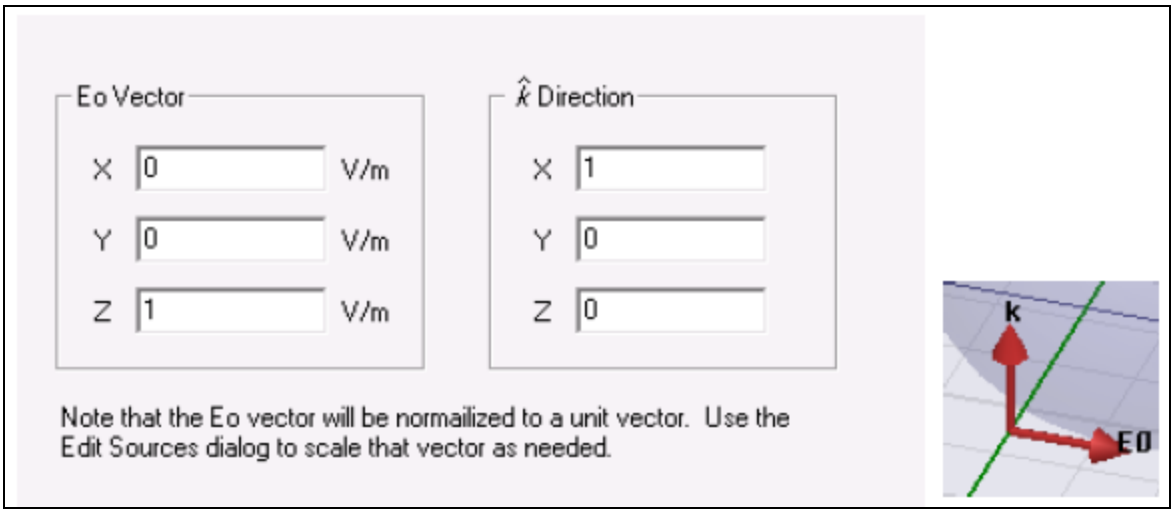
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following figure shows the **Cartesian Vector Setup** panel. Define the direction of **Eo**. Regardless of the magnitude of the field vector **Eo**, HFSS normalizes it to 1. However, the magnitude of **Eo** can be scaled to the desired value on the **Edit post process sources** panel.

Also, define the direction of the unit vector of propagation, $\hat{k}$. It is your responsibility to ensure that the direction of propagation, $\hat{k}$ of the plane wave is perpendicular to **E0.L**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Click **Next** to specify the the type of plane wave on the **Plane Wave Options**panel.

Recall the equation that describes a plane wave:

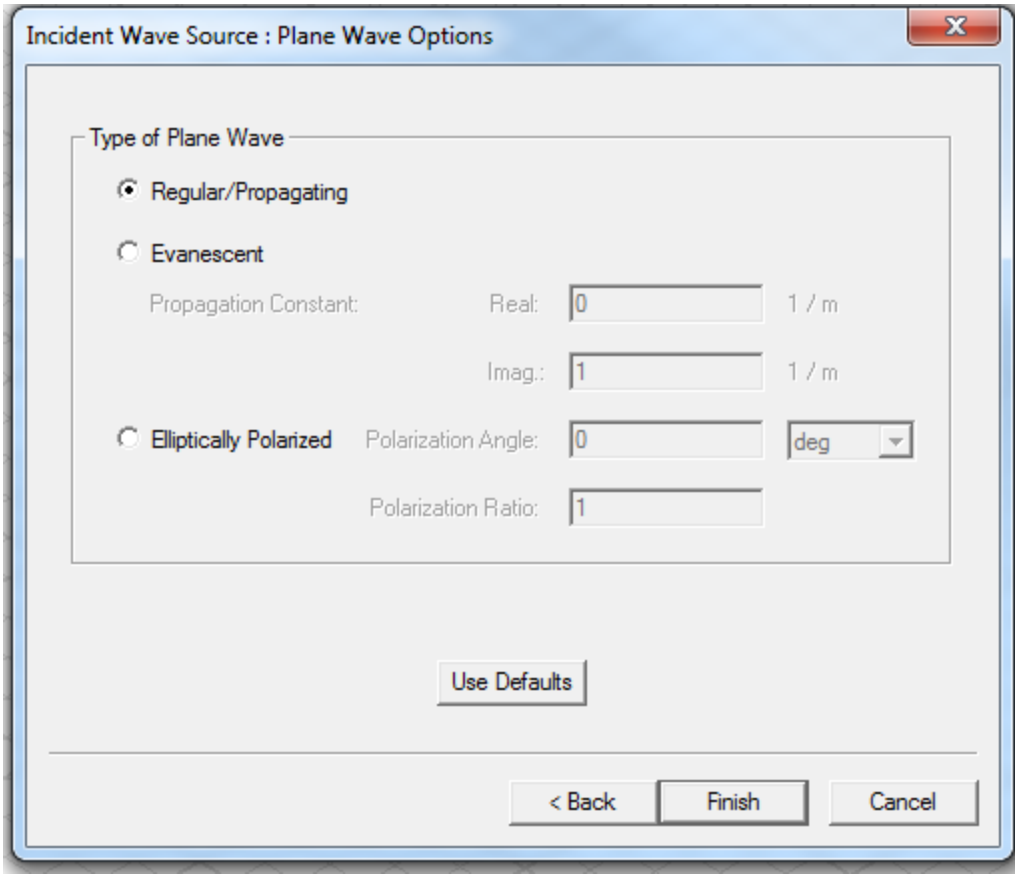
$$\vec{E} = \overrightarrow{E_0} e^{-j\vec{k}\cdot\vec{r}} \qquad \text{where} \qquad \vec{k} = k\hat{k}$$

$$\overrightarrow{E_0} = E_0\hat{e}_0$$

where k is the wave number of the global background material for regular/propagating plane waves. In most cases, the values defined for **Eo** and **k** are meant for regular/propagating plane waves. For evanescent waves, since $k = \beta + j\alpha$ it overrides the magnitude of the complex propagation constant.

> **Note:**
>
> In HFSS, evanescent waves do not depend upon the global background material. However, the post-processed near or far fields depend upon the global background material. For more information see the section Global Material Environment.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



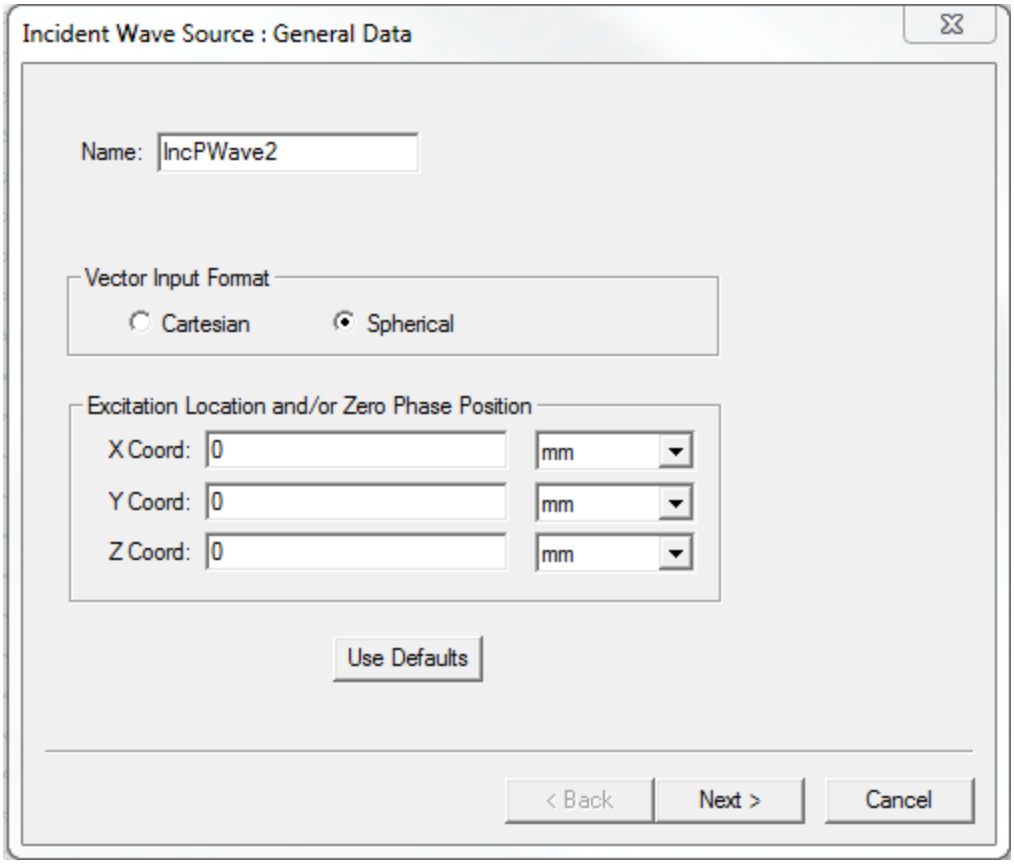
If you select the type of plane wave as **Elliptically Polarized**, specify the ratio of the large axis to the small axis of the ellipse and the phase angle of the large axis. For more information, see the section Elliptically Polarized Plane Wave.

Spherical Vector Input Format

This section demonstrates the same incident plane wave by defining the vector input format for a Spherical setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

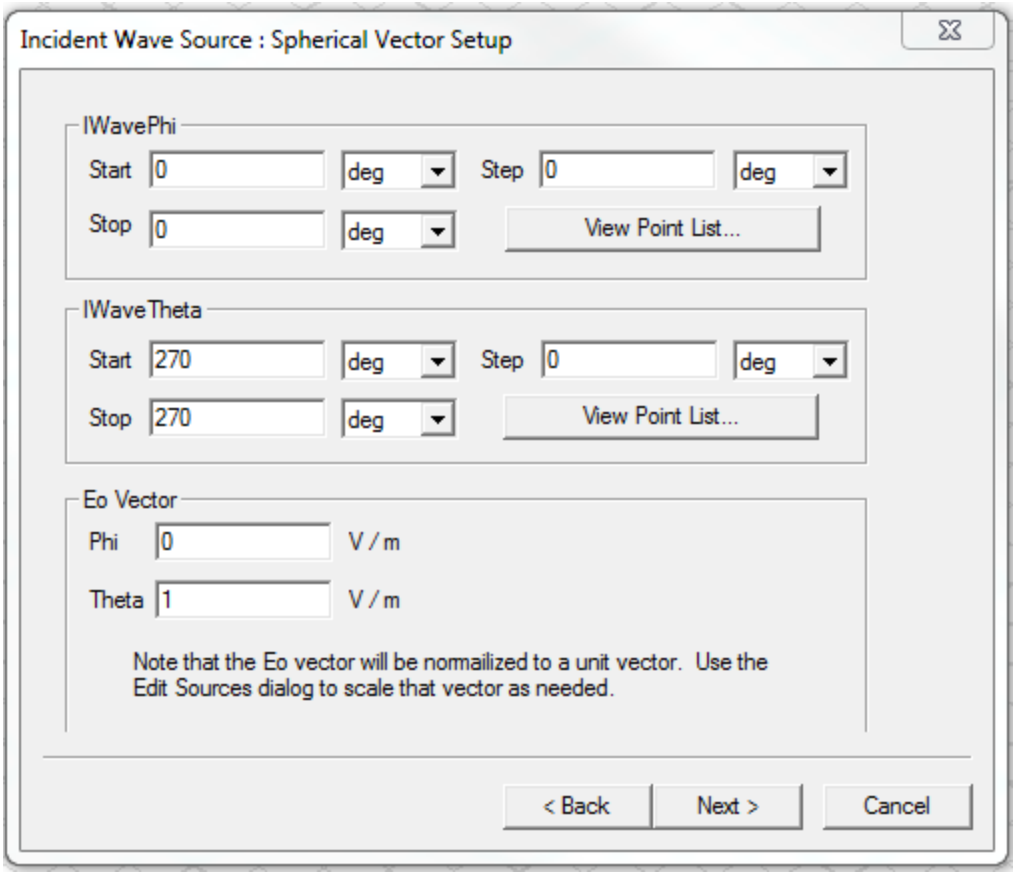
1. On the **Incident Wave Source: General Data** dialog box, select the option **Spherical**.



2. Click **Next** and edit the **IWavePhi**, **IWaveTheta**, and **Eo Vector** fields as shown in the following figure.

   This setup provides an alternate way of defining plane waves using spherical vector format. The orthogonality of $\hat{k}$ and **Eo** is automatically satisfied since $\hat{k}$ is in the $\hat{r}$ direction. Regardless of the magnitude of **Eo**, HFSS normalizes it to 1. However, the magnitude of **Eo** can be scaled to the desired value on the **Edit post process sources** panel.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



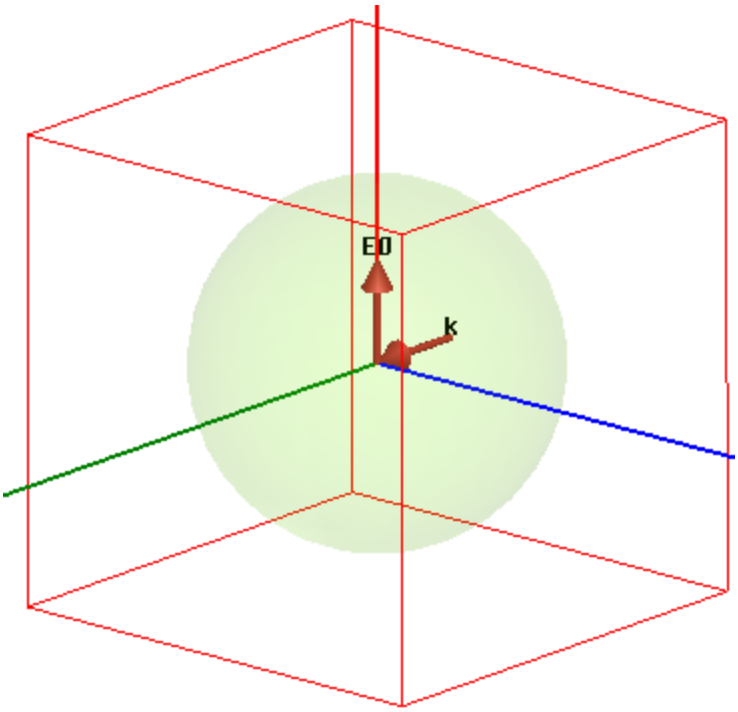
Note:

If you enter values in the **Step** fields and click the **View Point List** button, you can see all the phi or theta values.

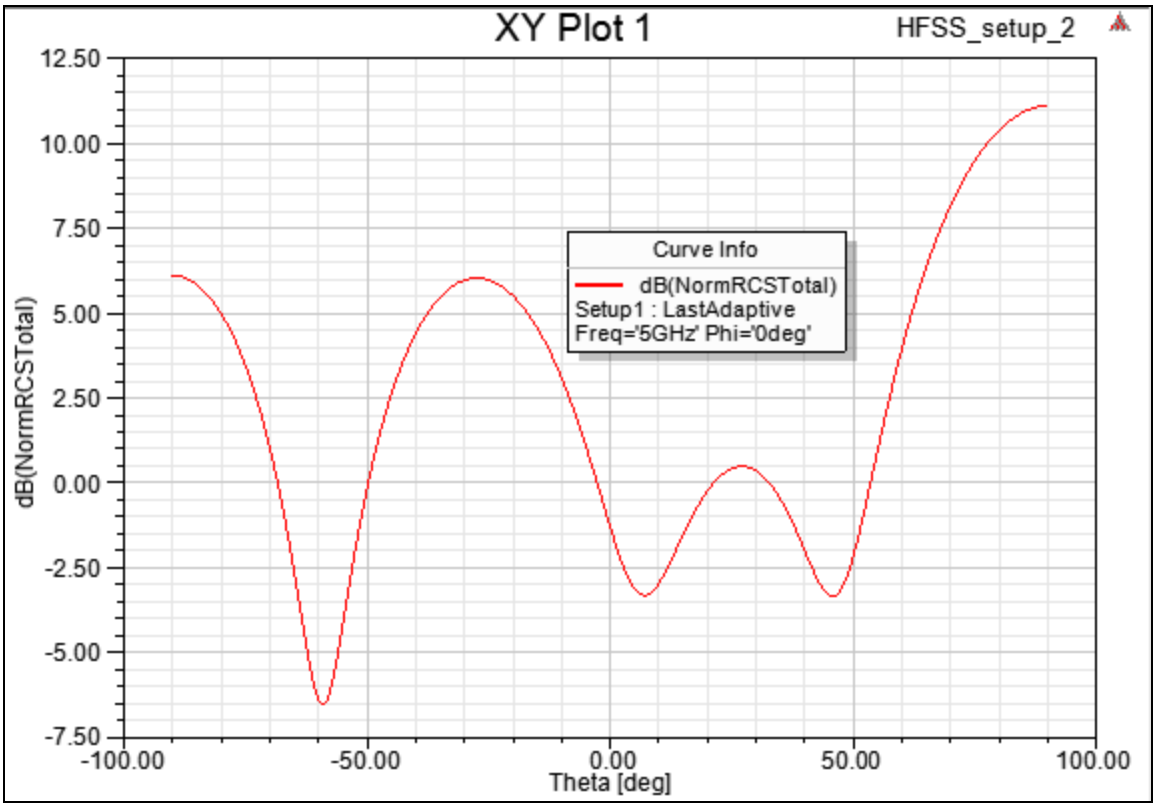
For a spherical incident wave you can specify an expression to define an angle in the start field only. If use an expression, the dialog disallows any stop/step values. In other words, we allow only a single angle if you choose to parametrize the start angle.

You can visualize the propagation direction and the electric field direction for a plane wave defined for spherical vector setup as shown in the following figure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

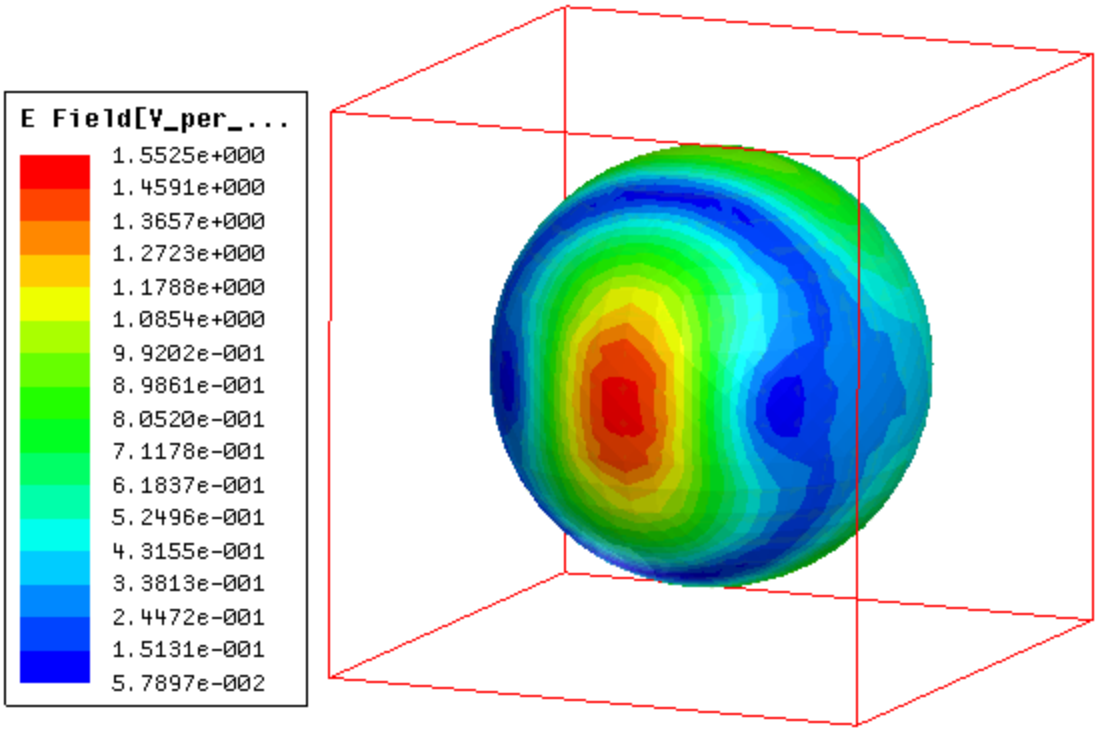


The normalized bistatic RCS plot for theta scan at phi = 0 plane is shown below.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following figure shows the scattered E-field plot on the surface of a dielectric sphere (radius = 30mm, relative permittivity = 10, and solution frequency = 5 GHz.)



Note:

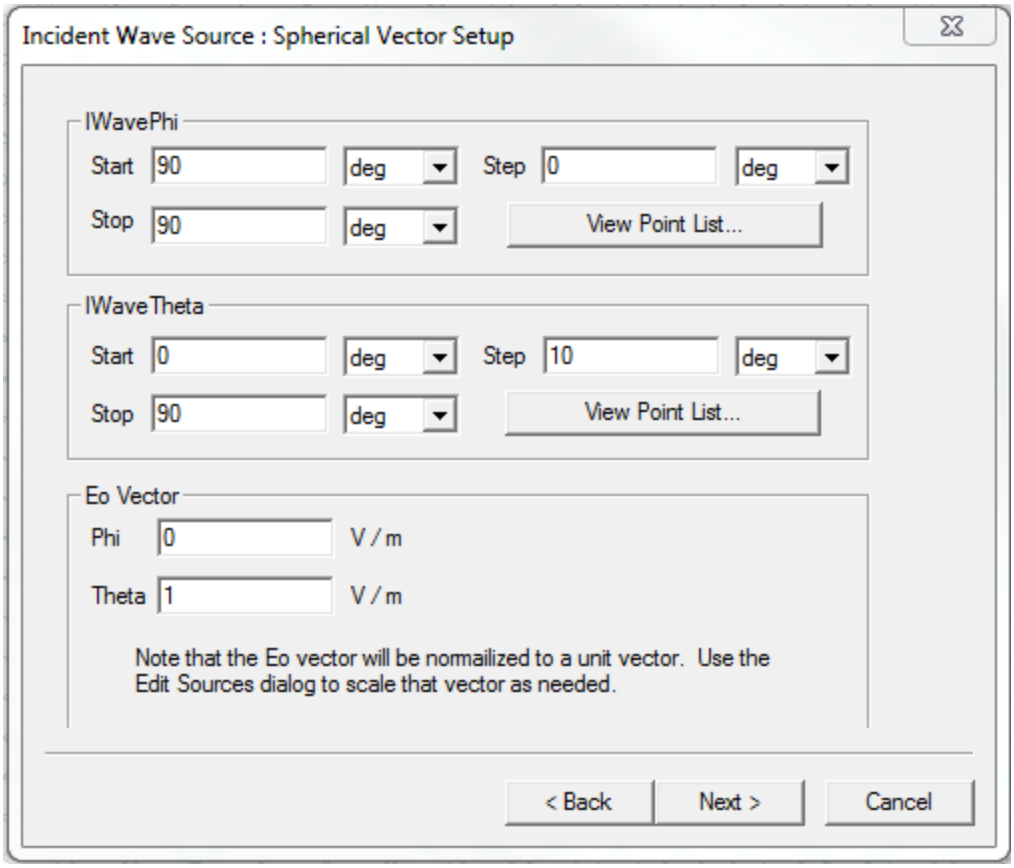
The Spherical vector input format provides a convenient way of specifying multiple incident angles.

Defining Multiple Plane Waves

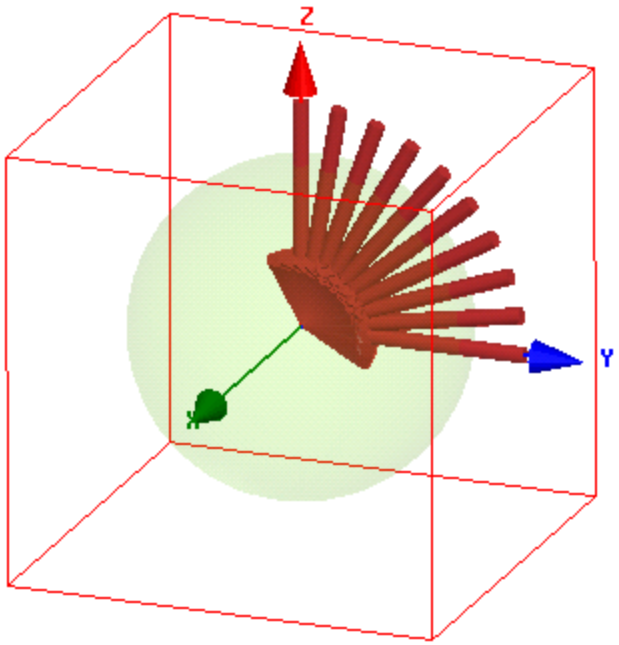
This section describes how to define multiple plane waves using **Spherical Vector Setup**.

The following figure shows the **Spherical Vector Setup** panel. An incident wave sweep is defined in the phi = 90 degrees plane for theta, with step size of 10 degrees.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

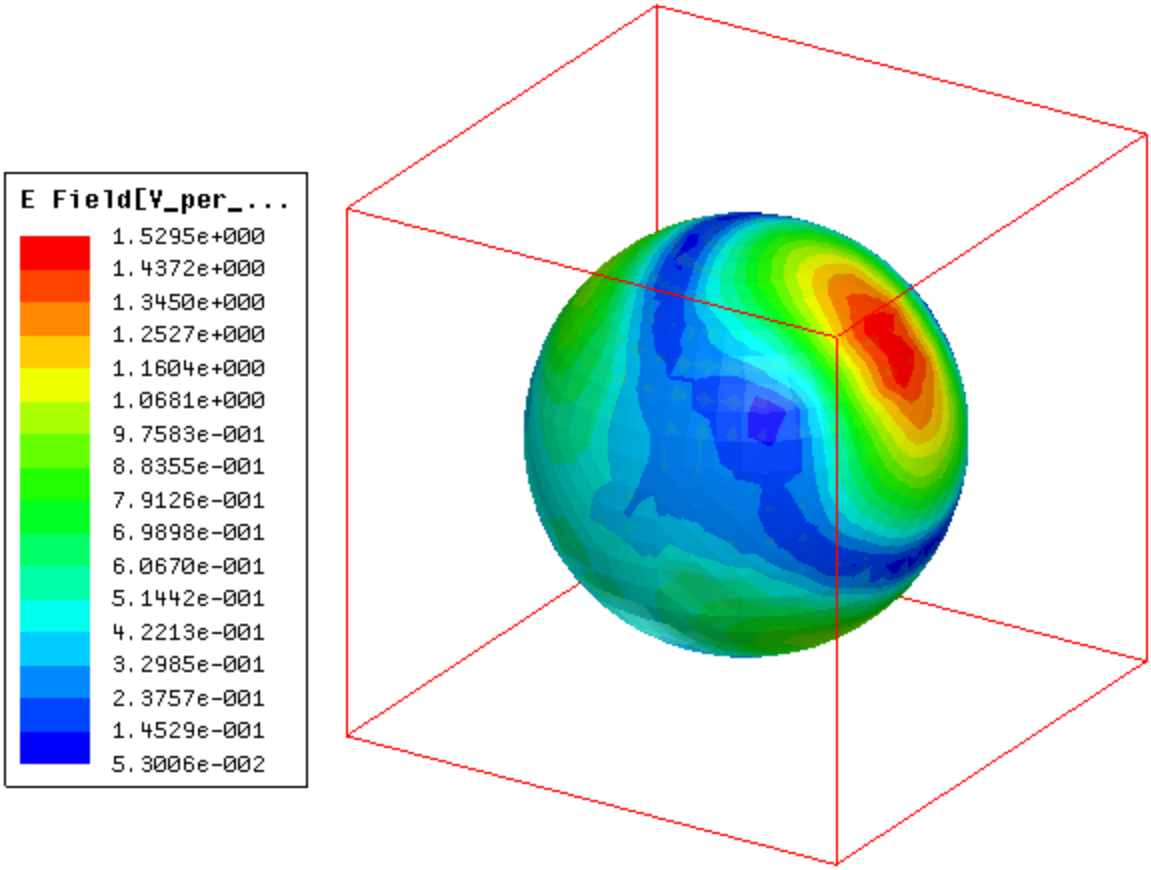


This setup defines eleven incident plane waves as source as shown in the following figure.



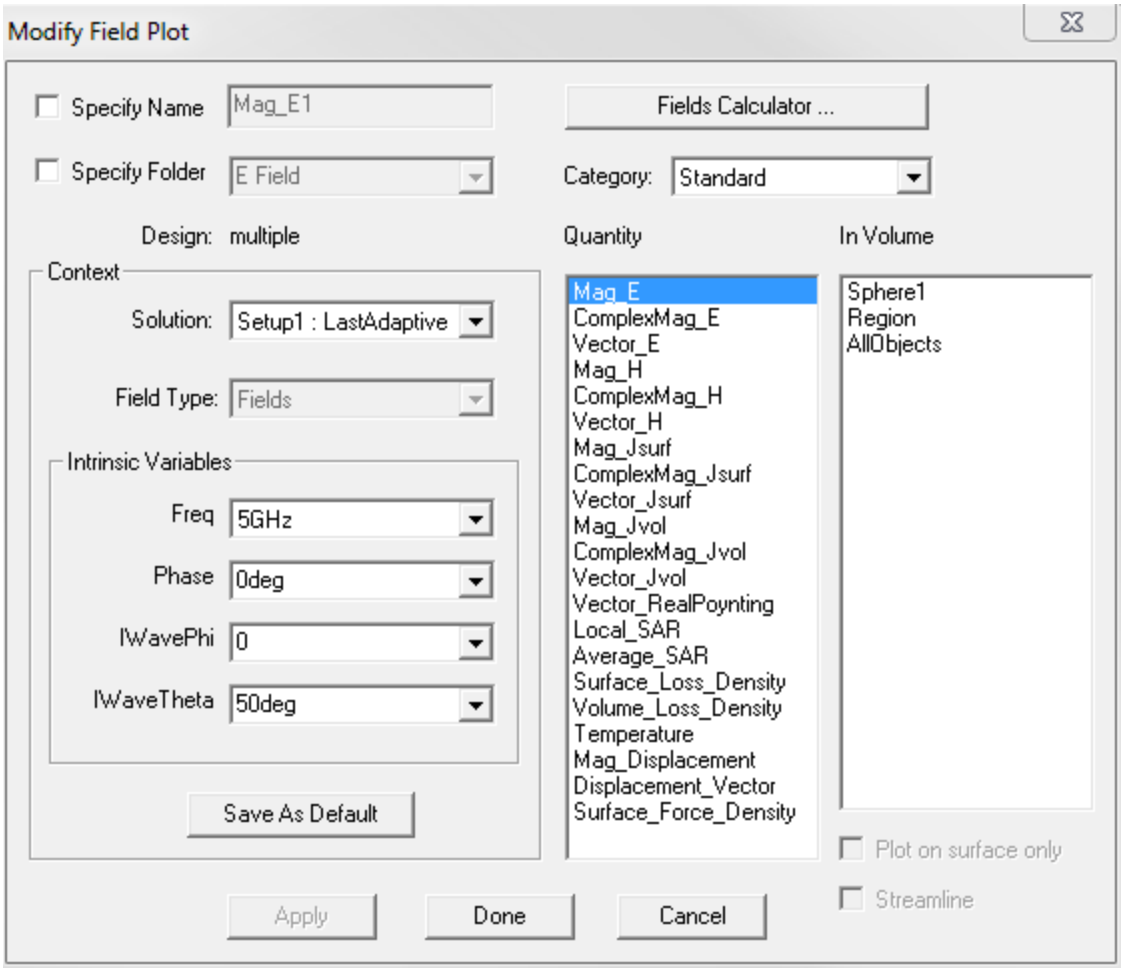
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following figure shows the scattered E-field plot on the surface of the dielectric sphere (radius = 30mm, relative permittivity = 10, and solution frequency = 5 GHz) at incident angle of theta = 40 degrees.



This E-field plot is defined for IWaveTheta = 50 deg and IWavePhi = 0 deg.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



> **Note:**
>
> Although the incident field sweep defines multiple plane waves, only one scale factor is applied on all the plane waves. This scale factor can be edited on the **Edit post process sources** panel.

Both the overlay field panel and the Reporter provides you options to select the desired incident plane wave.

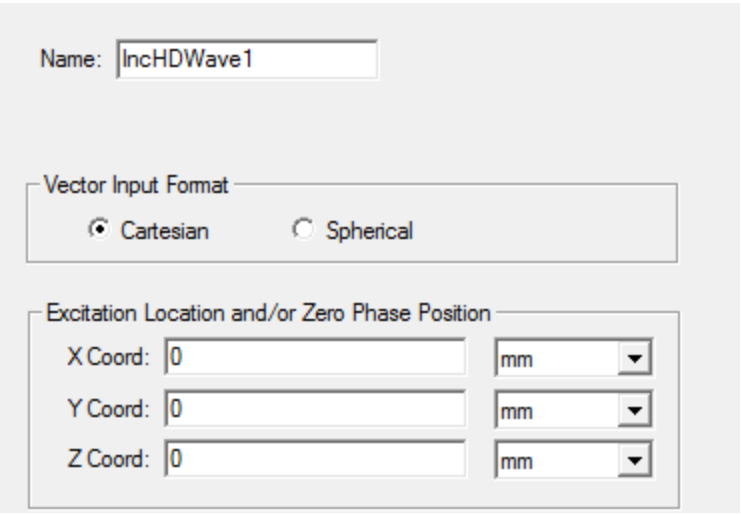
Hertzian-Dipole Wave

An incident Hertzian-Dipole wave can be specified as either an Electric dipole or a Magnetic dipole. The Electric dipole simulates the field of an elementary short dipole antenna placed at the origin. The Magnetic dipole is useful for EMC/EMI applications. Specify a Hertzian dipole as follows:

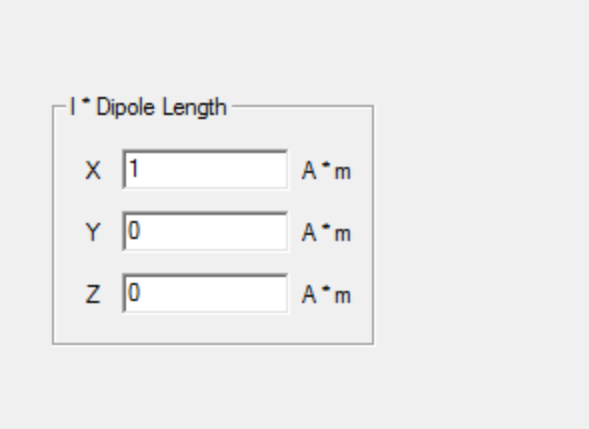
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cartesian Coordinates

1. Click **HFSS>Excitations>Assign>Incident Wave>Hertzian-Dipole Wave**.



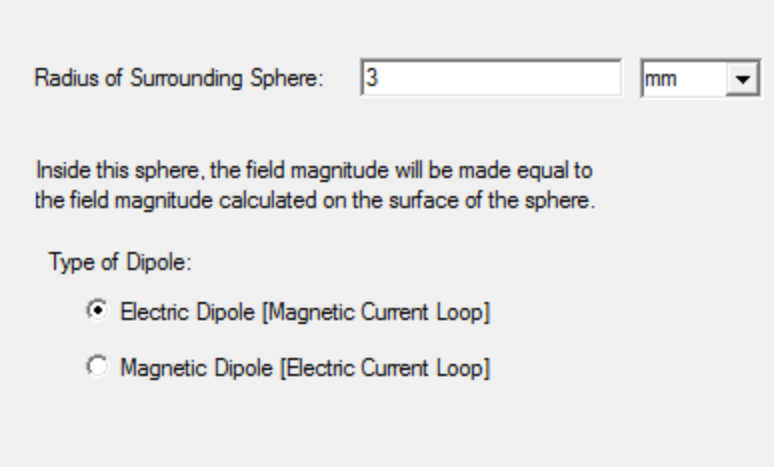
2. Select **Cartesian** and click **Next**, the subsequent dialog lets you define the dipole length.



3. Define the types of dipole and the radius in the **Hertzian-Dipole Wave Options** dialog box. The radius of Surrounding Sphere should be small enough so that a Hertzian dipole is placed into the computational domain when scattered field formulation is used. If you place a Hertzian dipole out of the computational domain, the Surrounding sphere can

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

have any radius.



Incident Cylindrical Wave

An incident Cylindrical wave is a wave that simulates the far field of an infinite line current placed at the origin.

1. Click **HFSS>Excitations>Assign>Incident Wave>Cylindrical Wave**.
2. Select the **Vector Input Format** as **Cartesian** coordinates.
3. Enter the X-, Y-, and Z-coordinates of the **Excitation Location and/or Zero Phase Position** (the origin for the incident wave).
4. Click **Next**.
5. If you selected **Cartesian**, the **Incident Wave Source: Cartesian Vector Setup** page appears. Enter the X-, Y-, and Z-components for **the I Vector** in the **X**, **Y**, and **Z** boxes. **I** is the current amplitude (peak value). Units are Amps (A).

   A single incident wave will be defined.

> **Note:**
>
> Only a single incident wave angle can be defined for periodic structures which are defined with primary and secondary (Lattice pair) boundaries

6. Click **Next**. the **Incident Wave Source: Cylindrical Wave Options** page appears.
7. Select the **Radius of Surrounding Cylinder**. Inside this cylinder, the field magnitude will be made equal to the field magnitude calculated on the surface of the cylinder. To restore the default (10 mm), click **Use Defaults**.
8. Click **Finish**.The incident wave you defined is added to the **Excitations** list in the **Project**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Gaussian Beam Wave

An incident Gaussian Beam wave propagates in one direction and is of Gaussian distribution in the directions perpendicular to its direction of propagation. The steps for assigning a Gaussian Beam Wave are the same as those for assigning Plane Wave. The only difference is that in this case **Gaussian Beam Wave Options** will appear as one of the tabs instead of **Plane Wave Options** on the **Incident Wave Source** dialog box. For more information see Plane Wave.

Linear Antenna Wave

An incident linear antenna wave is a wave that simulates the far field of a linear antenna placed at the origin. The steps for assigning a Linear Antenna Wave are the same as those for assigning a Hertzian Dipole wave. The only difference is that in this case **Linear Antenna Wave Options** will appear as one of the tabs instead of **Hertzian Dipole Wave Options** on the **Incident Wave Source** dialog box. For more information see Hertzian Dipole Wave.

Far Field Wave

A far field wave originates at a distance several wavelengths from the computational domain. Far field values are defined on the surface of a unit sphere.

When you use a Far Field link, the origin of the global coordinate system of the source project should be in the phase center of the antenna.

- Far Field Wave Options
- External Data File for Far Field Wave
- Setup Link for Far Field or Near Field Wave
- Example That Uses Far Field Links
- Near/Far Field Polarization Definition Table for HFSS, SBR+ and Hybrid Designs

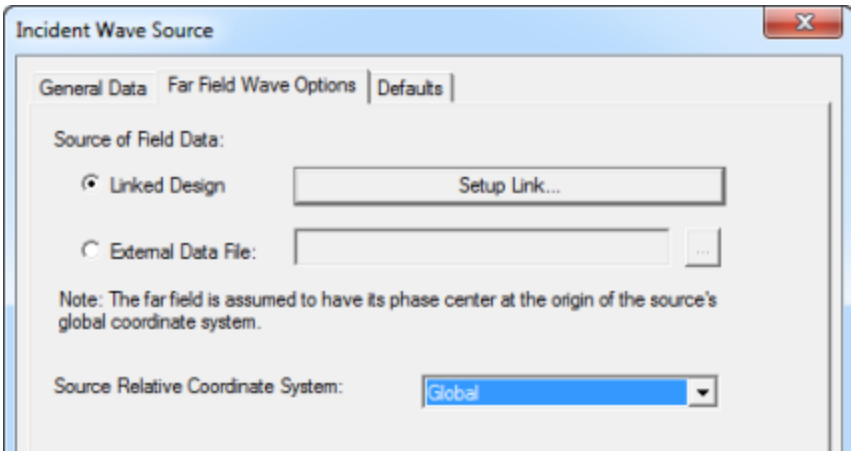
Far Field Wave Options

This section describes the panel **Far Field Wave Options**. Bring up this panel on the **Incident Wave Source** dialog box as follows.

First click **HFSS>Excitations>Assign>Linked Field>Far Field Wave** and then, click **Next** on the **Far Field Wave Options** tab.

Click **Setup Link** on the **Incident Wave Source** dialog to browse for a source project for specifying the linked design.

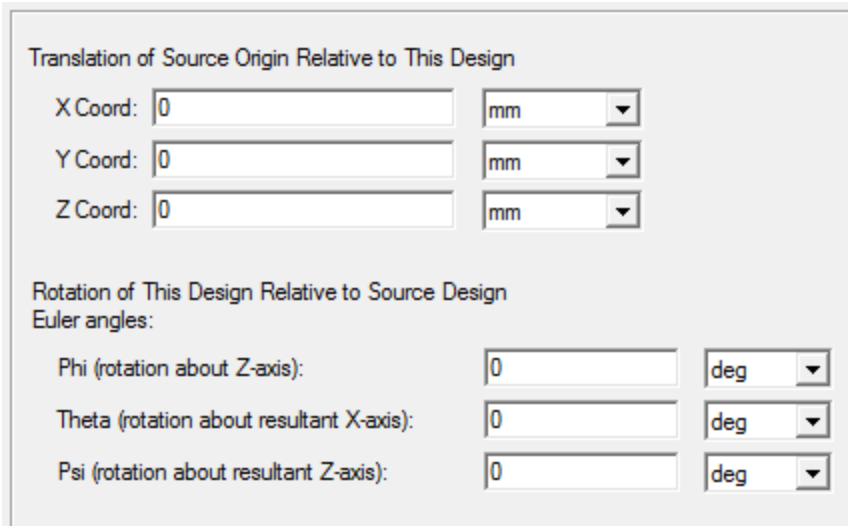
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Use a local coordinate system defined in the target design to represent the source origin and rotation relative to target design. All the legacy origin and Euler angles will be translated into local coordinate system.

Legacy Translation of Source Origin Relative to This Design

For legacy projects if the coordinate system you are using in the source design is different from that in the target design, you originally defined the relationship between those coordinate systems as a translation and a rotation. The translation is the offset between the **origins** of the two coordinate systems, and the rotation was defined through the use of **Euler angles.**



The X-, Y-, and Z-coordinates of the source location and/or Zero Phase Position (the origin for the incident wave) represents the translation of the source design's origin with respect to the target design's origin. For instance, if the source design's origin is located in the target design co-

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ordinate system at (-2, -2, 1), then the translation between the two coordinate systems is (-2, -2, 1).

The Euler angles represent the angles through which you can rotate the target design to achieve the desired rotation angles.

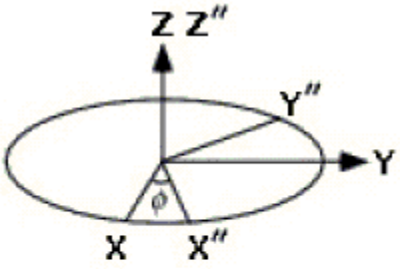
All the legacy origin and Euler angles will be translated into local coordinate systems.

> **Note:**
>
> A figure that represents the angles and the rotation of a design is at the end of the section.

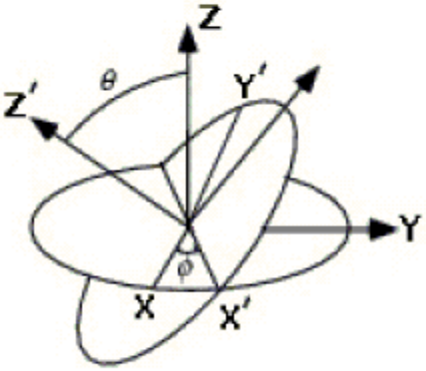
See Computing Antenna Parameters for a description of how to obtain antenna parameters during post processing.

X, Y, Z Source Coordinate System

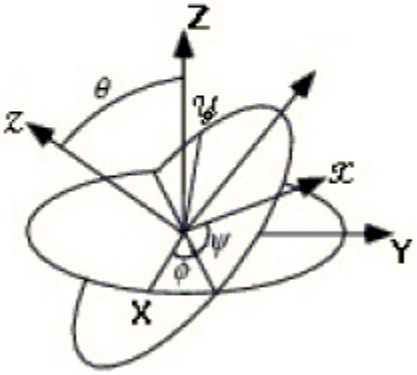
X", Y", Z" Source Coordinate System
After Rotation by ϕ [phi]



X', Y', Z' Source Coordinate System
After Rotation by θ [theta]



X, Y, Z Source Coordinate System
After Rotation by ψ [psi]



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

External Data File for Far Field Wave

You can define a Far Field **Incident Wave Source** as a plain text data file with a .ffd suffix. The fields can be independent or dependent on the frequency.

> **Note:**
>
> The far field data file must be a plain text file. Wordpad includes invisible characters that HFSS does not support. The code tolerates white space separators anywhere (blank, tab, newline, carriage return). Between double values in same logical grouping, it tolerates comma or semicolon separators.

Examples of ffd format files

- Frequency Independent Far Fields
- Frequency Dependent Far Fields

Related Topics

Setup Link for a Far Field or Near Field Wave

Frequency Independent Far Fields

In this case, the file format is simple. No keywords are used. The first two lines contain doubles representing start, stop, and num points for the sweeps of theta and phi. The angle units are degrees. In order to have a unique coordinate system theta and phi should be in the range of 0 < theta < 180 and 0 < phi < 360.

```
ThetaStart ThetaStop ThetaNumPoints
PhiStart PhiStop PhiNumPoints
```

Then, the subsequent lines give the complex components of electric far field in the theta and phi directions. By definition, far field is the scattering amplitude of the electric field at the region that is at a large distance from the antenna. Typically the distance must be much greater than 2 D * D / wavelength where D is the largest dimension of radiator or scatterer. The unit of far field is Volt as it is the product of distance and electric field. For these lines, theta will be held constant while phi is swept through all values, then theta will increment to the next value, etc. So the lines will correspond to (th1, ph1), (th1, ph2), (th1, ph3),…,(th1, phN), (th2, ph1), (th2, ph2), etc… Note that this format enforces a strict uniform grid of theta and phi samples.

```
E_theta_real E_theta_imag E_phi_real E_phi_imag
E_theta_real E_theta_imag E_phi_real E_phi_imag
...
```

See the following example of a data file:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
0,180,3
0,360,5
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
```

> **Note:**
>
> Instead of commas you can use double-space or semicolon separators.

Frequency Dependent Far Fields

For **frequency-dependent** far field links, the data is supplied in blocks. The syntax for a frequency dependent far field uses the following format.

thetaStart, *thetaStop*, *count* (specified as *degrees, degrees, integer*, for example, `0,180,2`

phiStart, phiStop, count (specified as *degrees, degrees, integer*), for example, `0,360,5`

frequencies *count* (specified as *string*, *integer*), for example, `frequencies 3`

frequency *freq* (specified as *string*, *double*), for example, `frequency 3.0e9`

Then for each point,

real F(theta), imag F(theta), real F(phi), imag F(phi), that is, 4 doubles, for example, `0.0,0.0,0.0,1.0`

Then the rest of the points

Then the rest of the frequencies

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the infinite sphere setup is not Theta-Phi (CS definition in the infinite sphere dialog) then theta is in reverse order.

Data File Example

See the sample data file below.

```
0,180,2
0,360,5
frequencies 3
frequency 3.0e9
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
0.0,0.0,0.0,1.0
frequency 6.0e9
0.0,0.0,0.0,2.0
0.0,0.0,0.0,2.0
0.0,0.0,0.0,2.0
0.0,0.0,0.0,2.0
0.0,0.0,0.0,2.0
0.0,0.0,0.0,2.0
0.0,0.0,0.0,2.0
0.0,0.0,0.0,2.0
0.0,0.0,0.0,2.0
0.0,0.0,0.0,2.0
frequency 9.0e9
0.0,0.0,0.0,3.0
0.0,0.0,0.0,3.0
0.0,0.0,0.0,3.0
0.0,0.0,0.0,3.0
0.0,0.0,0.0,3.0
```

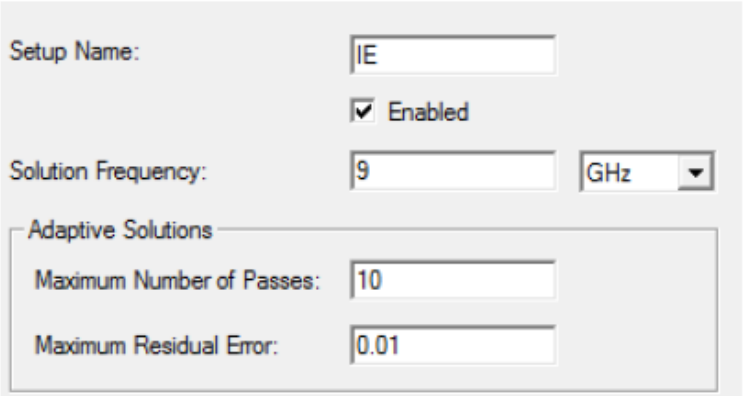

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

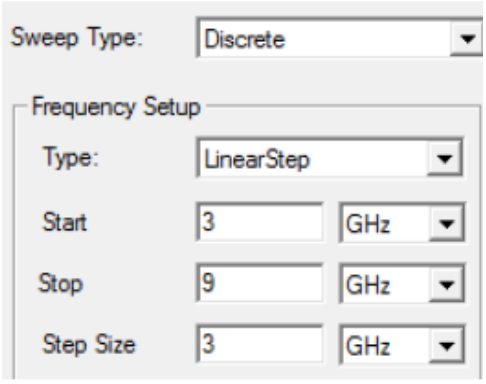
```
0.0,0.0,0.0,3.0
0.0,0.0,0.0,3.0
0.0,0.0,0.0,3.0
0.0,0.0,0.0,3.0
0.0,0.0,0.0,3.0
```

The syntax and their values are as follows:

- 0, 180, and 2 are the values of ThetaStart, ThetaStop, and ThetaNumPoints in the first line.
- The values 0, 360, and 5 do the same thing for Phi.
- NumFrequencies represents the number of blocks defined in the data file. In this example, 3 frequency blocks are defined.
- FrequencyValue represents the value of frequency for which you supply the far field data. In the Data File Example, FrequencyValue is as follows:
- 3.0e9 Hz for the first block;
- 6.0e9 Hz for the second block;
- 9.0e9 Hz for the third block.

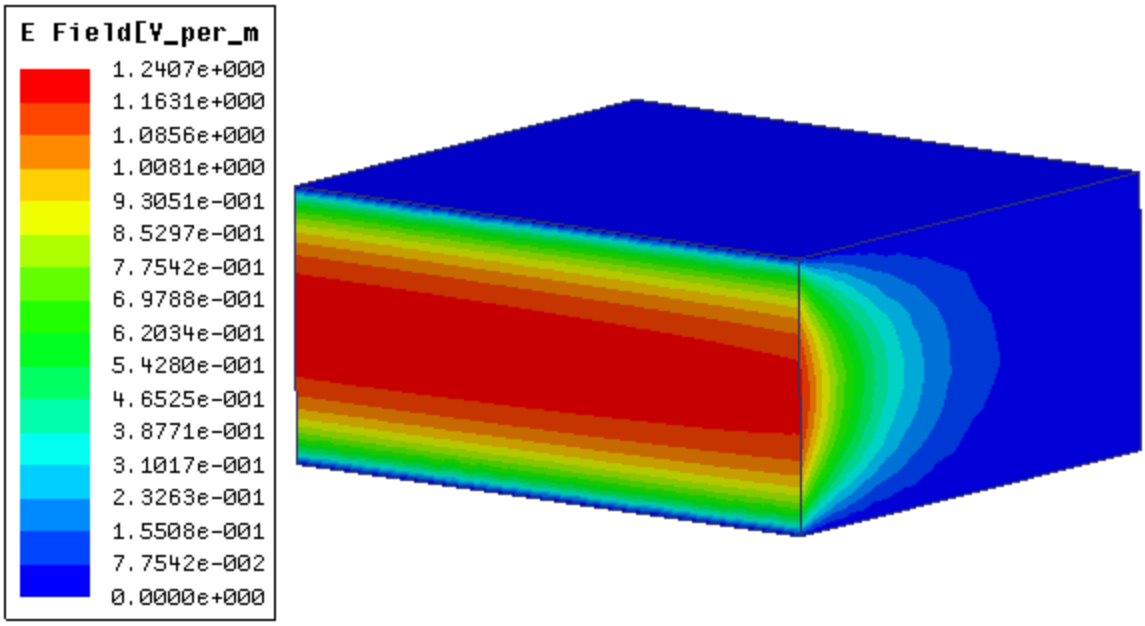
HFSS supports only the discrete sweep type for Far Field (and Near Field) data links. The frequencies defined in the sweep must be consistent with the frequencies specified in the data file. If not, an error message will pop up. So make sure that all the frequencies that you supply in the data file are consistent with those in the **Frequency Setup** section of the **Edit Frequency Sweep** dialog box. For example, based on the **Frequency Setup** shown below in the target project HFSS will expect to find data for 3GHz, 6 GHz, and 9 GHz in the data file. Also, the **Solution Frequency** in the **Solution Setup** should match one of the frequencies in the data file and the sweep if the project type is HFSS in the target design. Also, the frequency blocks can appear in any order in the data file.





Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

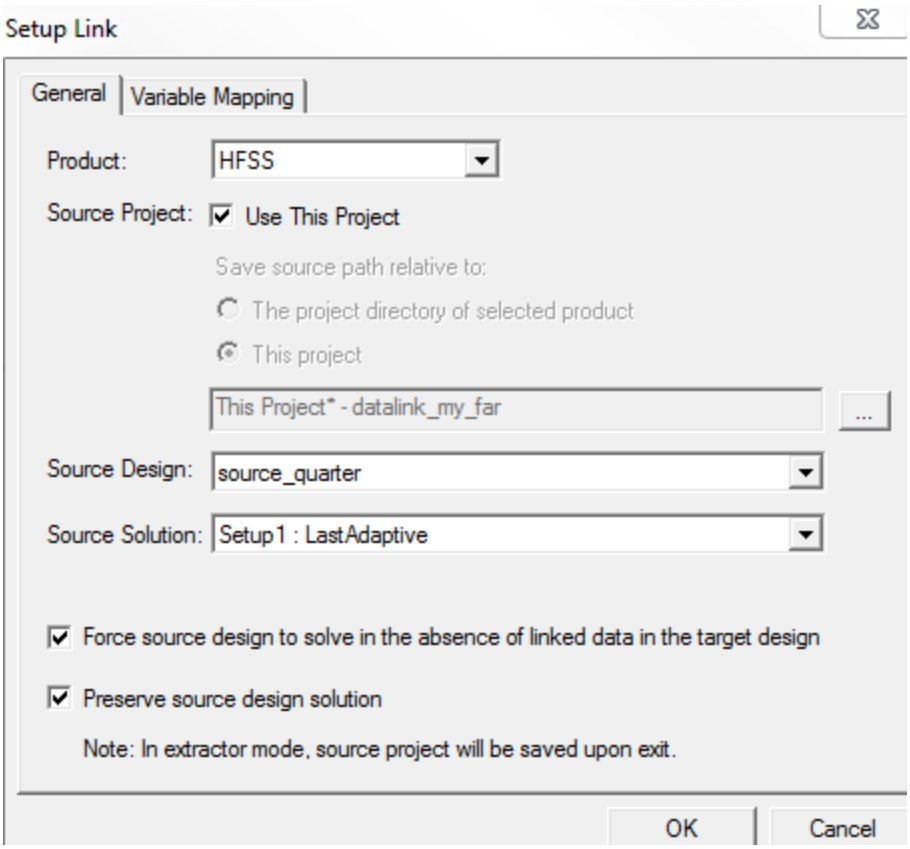
The sample frequency dependent data file was used to apply far field on an open ended WR-90 waveguide. The frequency sweep shown was used to simulate it. The results for the E-field at 9 GHz are shown below.



Setup Link for Far Field or Near Field Wave

You can access the **Setup Link** dialog from the Far Field Wave options dialog or Near Field Wave options or from an SBR+ design linked to a Driven Modal antenna.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



1. Select the **Edit Link** radio button to activate the options in the panel.
2. Select the type of the source project (e.g., HFSS, HFSS 3D Layout, SIwave, Maxwell) from the **Product** drop-down menu.

   > **Note:**
   >
   > An **HFSS-IE** project can be used as target or source. **Maxwell** does not appear in the drop-down menu because it is not applicable for far fields.

3. Browse for the source project or check **Use This Project**. If you check the latter option, you must select a source design from the current project.

   > **Note:**
   >
   > The origin of the global coordinate system of the source project should be in the phase center of the antenna, when you use a Far Field link.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Specify the **Source Solution**. The first option in the Source Solution field is the default.

   > **Note:**
   >
   > The solution in the source design must provide data for the target design's adaptive frequency as well as its sweeps. That is, the adaptive frequency for the target design must be included in the sweep in the source design. If necessary, you can open the source design and add an appropriate frequency point to an existing sweep.

5. Use the last two check boxes at the bottom of the panel as needed.

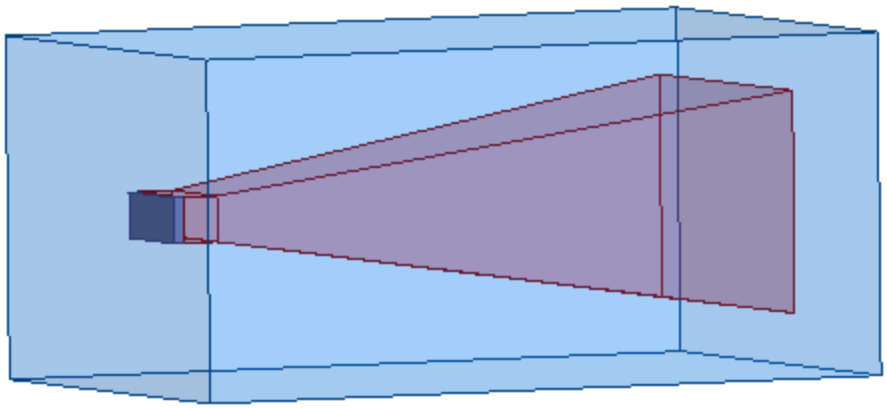
   > **Note:**
   >
   > **Extractor mode** means that the software is opened during the link solely for the purpose of solving.

6. Under the **Variable Mapping** tab, you can set the desired variable values in the source design. If the source and target designs contain same named variables, you can choose to Map Variable By Name. In this case, same named variables are mapped automatically.

Example That Uses Far Field Links

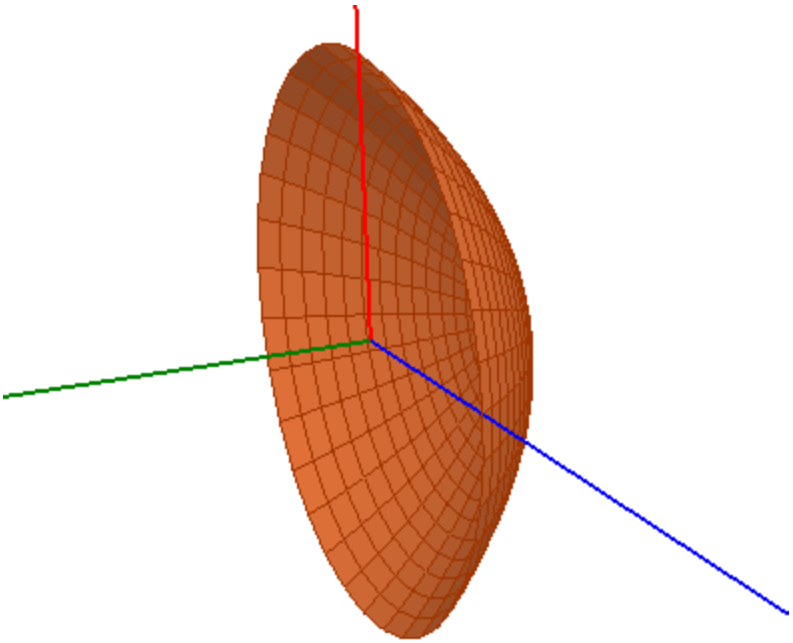
This section describes linking an HFSS project to an HFSS-IE project while using the common feature of assigning Far Field Wave excitation.

The horn antenna enclosed in an air box in the figure below represents the source project designed in HFSS.

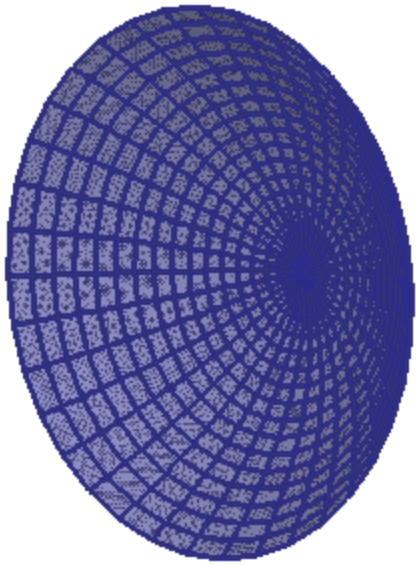


The dish/reflector in the following figure is the target project designed in HFSS-IE.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

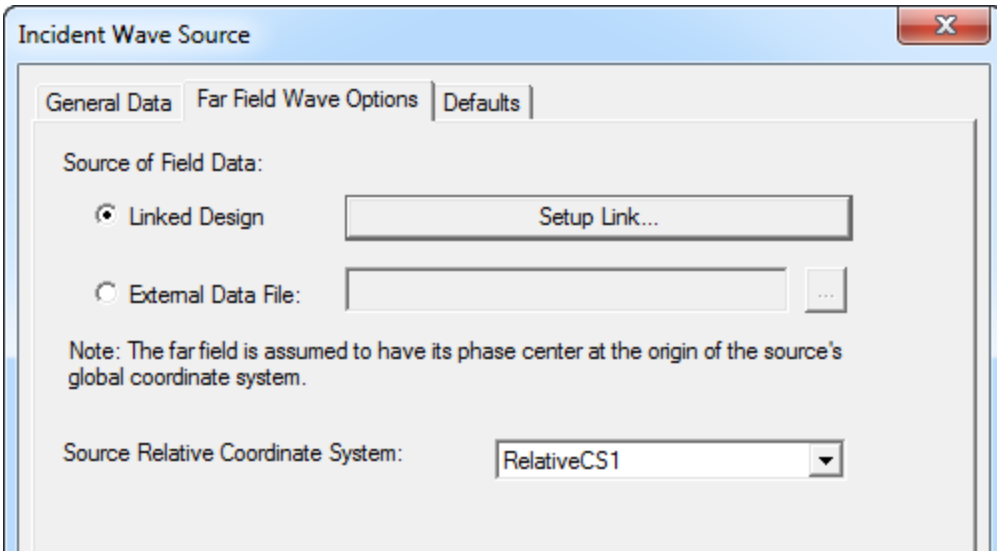


A perfect E-boundary is assigned on the reflector.

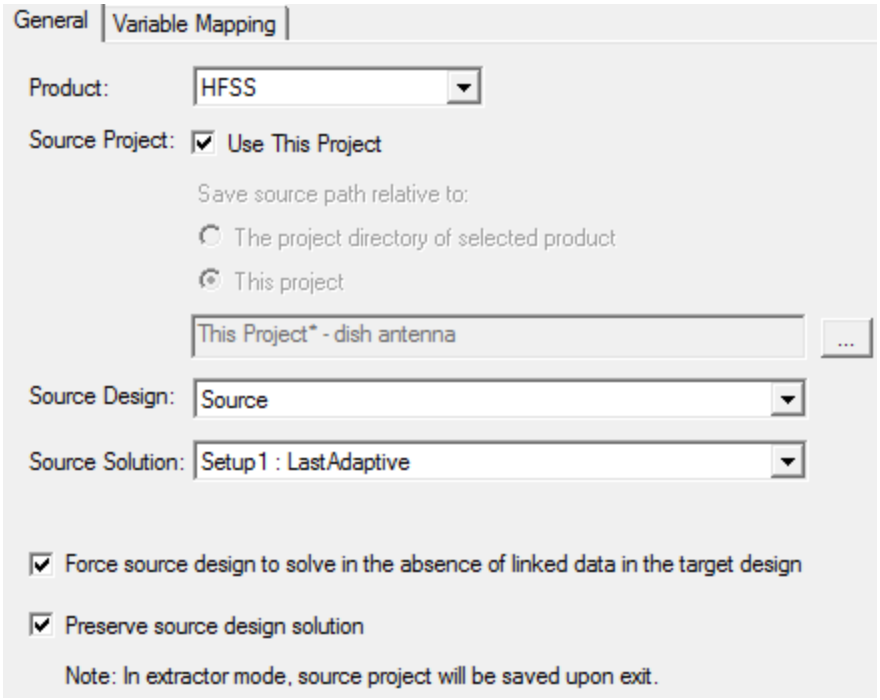


The **Far Field Wave Options** panel has the following settings.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

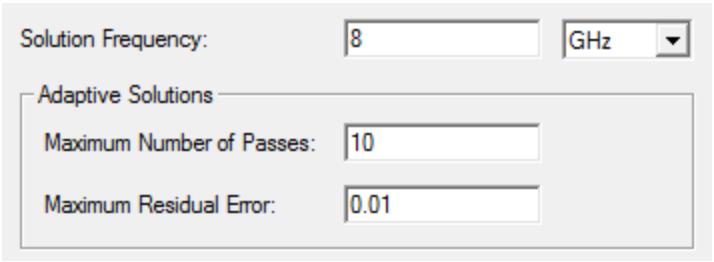


From the **Setup Link** option, the fields are set as shown in the figure to link the target to the source project (the horn antenna).

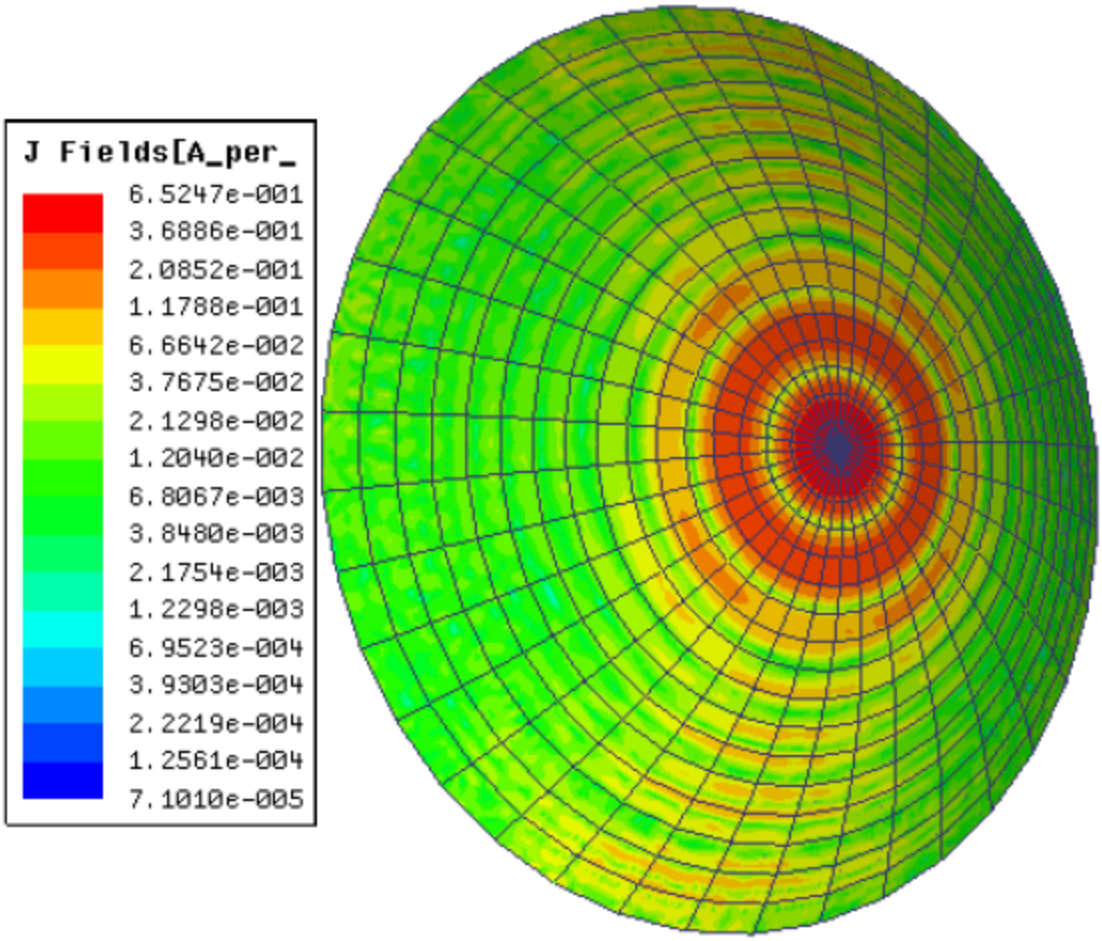


A sample solution setup for this project is shown below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The resulting J Fields (magnetic fields) after solving the project are shown below.



Near Field Wave

A Near Field wave source is close enough to the design for near field effects to occur, typically within one wave length. Objects in the near field wave source project should NOT be too close to

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the radiation surface where a Near Field Wave pings in. Near field waves tend to have both evanescent and propagating content.

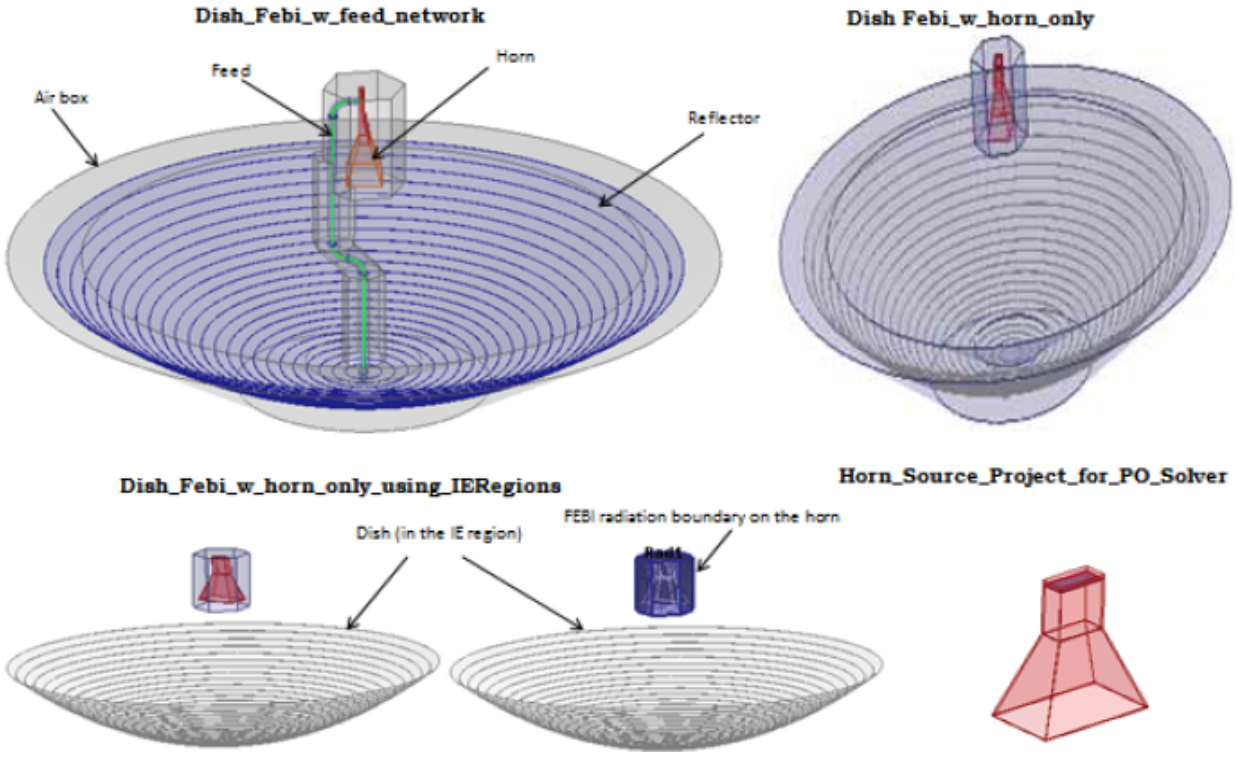
- Assign Near Field Wave
- Example that Uses Near Field Links
- Access Near Field External Data File
- Ansys Near Field Data File
- Generate Near Field Data File
- Near/Far Field Polarization Definition Table for HFSS, SBR+ and Hybrid Designs
- *Technical Notes*: Ansys Near Field Description Format Specification.

> **Note:**
>
> HFSS can also use dynamic links to other simulation tools to create one seamless workflow. The source designs can be created in HFSS, HFSS 3D Layout, SIwave or Maxwell3D. Refer to Using Field Solutions from Other Simulations.

Assign Near Field Wave

This section describes the **Near Field Wave Options** panel on the **Incident Wave Source** dialog box and how to assign near field waves. The following designs illustrate different ways of using near field sourcing to run a range of analyses on a dish antenna fed by a circular horn antenna.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

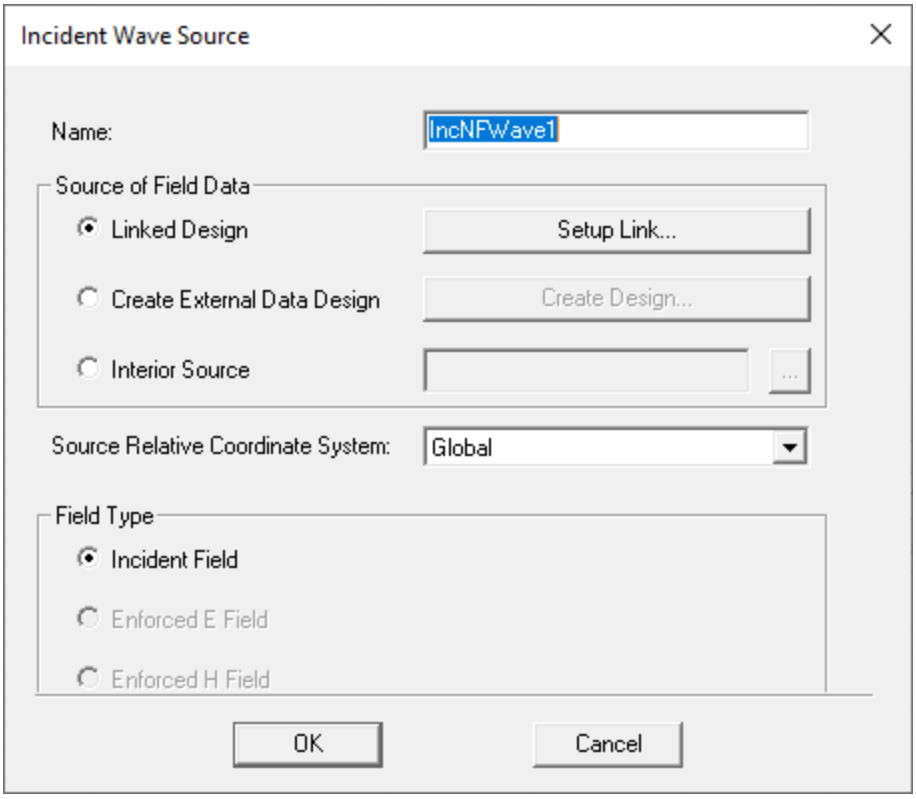


First we will bring up the **Near Field Wave Options** panel and then, assign Near Field Wave.

1. Click **HFSS>Excitations>Assign>Linked Field>Near Field...**.

   The **Incident Wave Source** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You assign the **Excitation** name or accept the default.

You can select Linked Design, Create External Data Design, or Interior Source. If you are using a linked design go to step 4.

If you are using an external data file to use to **Create External Data Design...**, go to Access Near Field External Data Fileand follow the instructions.The dialog supports the choices of E field only and H field only if you select external data.

Do to the nature of the data link, both of these approaches inevitably lose the source design mesh information. Although it might be a reasonable approximation when mesh is complex, in other cases you may want to directly define mesh and near field field, especially when data link source is an external near field design and when source geometry is composed of near field primitives with relatively simple mesh.

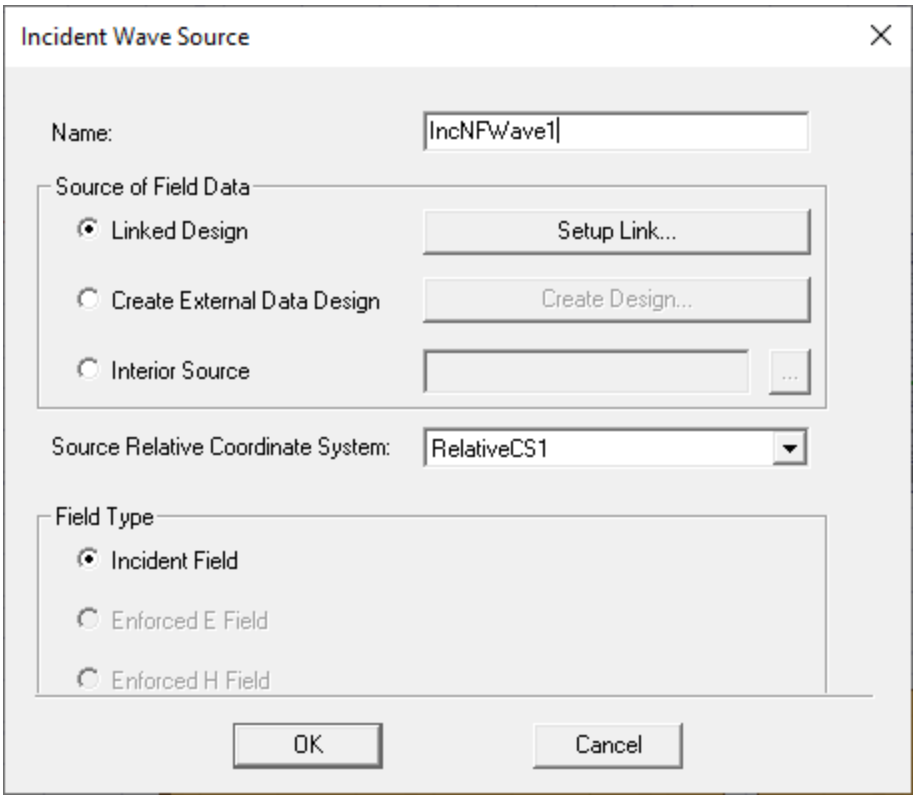
HFSS includes an option to add this external near field link as "interior" source. When you select Interior Source you select a data file (.and/.nfd) and HFSS then automates a series of setups, such as near field geometries, mesh operation setup and solution setup, similar to what current Linked Design does. This interior source has the same effect as if its equivalent external near field design is linked to that target design. This interior source also allow relative coordinate system definition. For an example see Example with Interior Source for Near Field Link.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Next** to access the **Near Field Wave Options** panel.

> **Note:**
>
> For array problems, the **Active** option includes the effects of the Near Field in the simulation. To disable the effects, uncheck the **Active** option.

3. Determine whether you want to use a linked design, an external data file, or an interior source. If you are using a linked design go to step 4. If you are using an external data file to use to **Create External Data Design...**, go to Access Near Field External Data Fileand follow the instructions.



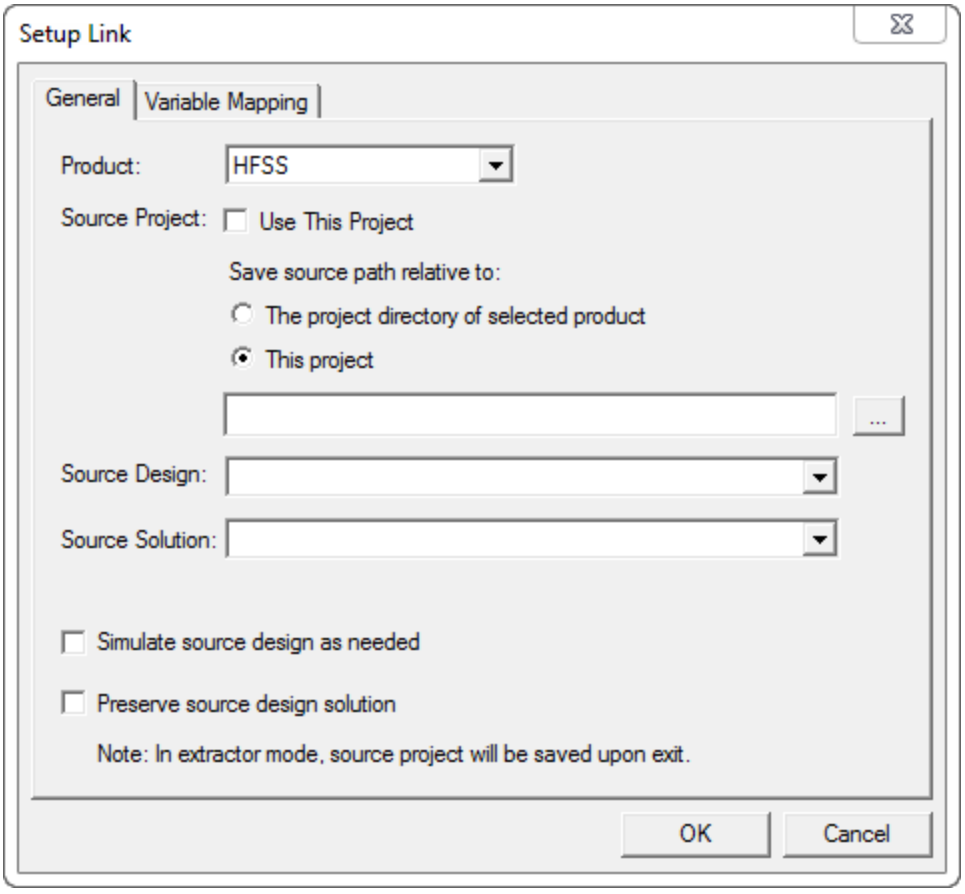
4. Click **Setup Link** on the **Incident Wave Source** dialog to specify the linked design.

> **Note:**
>
> It is through the **Setup Link** dialog box that you will link the source design to your target design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. On the **Setup Link** dialog box select the radio button **Edit Link**.

   The fields in the dialog box become active and editable as shown below.



6. Select the product in which you created your source design from the **Product**drop-down menu.

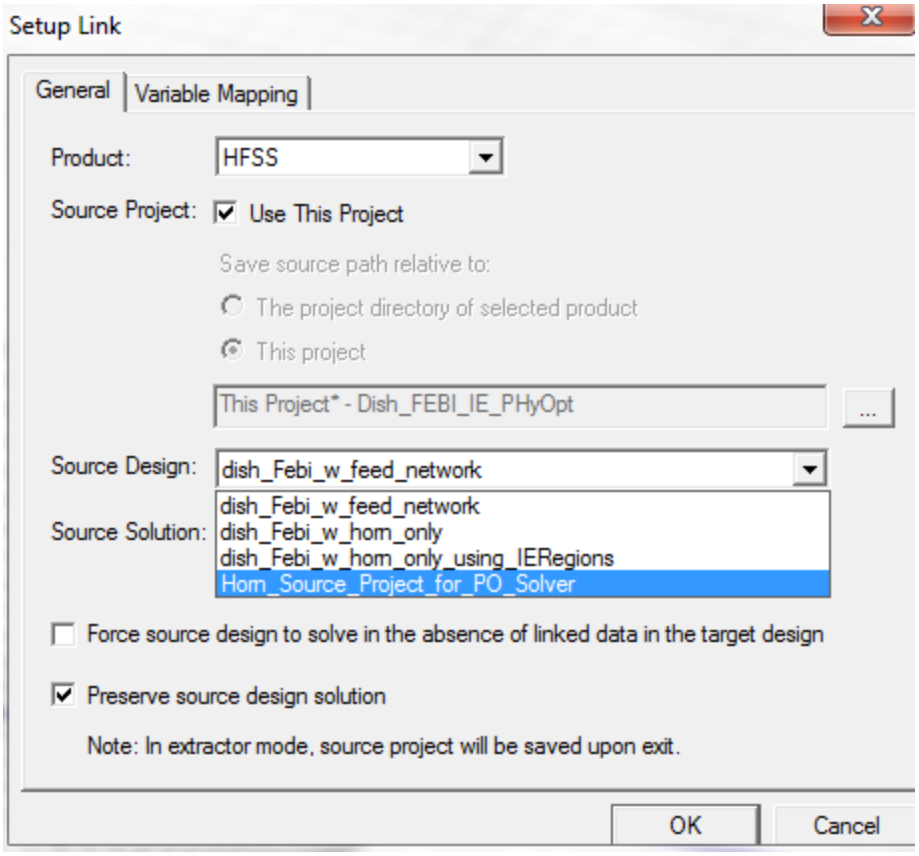
   > **Note:**
   >
   > Only HFSS, Maxwell, and SIwave appear in this list.

7. Specify the project that has the source design in one of the following ways:
   a. Check **Use This Project** to pick a design from a current working project.
      b. Click the ellipsis and browse your computer to pick the design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The option **The project directory of selected product** points to the default project directory location relative to the product that you selected from the drop-down menu. This is the same project directory that you set on the **General Options** dialog under **Tools> Options.** This option is useful especially, when you link across different products. The option **This project** points to the directory relative to where your current working project is located.



8. Pick the source design from the **Source Design** drop-down menu which lists all the designs in the specified project.
9. Pick the setup type from the **Source Solution** pull-down menu (e.g., Last Adaptive, Sweep).

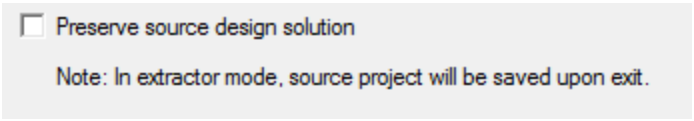
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. Select the last two check boxes as needed and click **OK**.

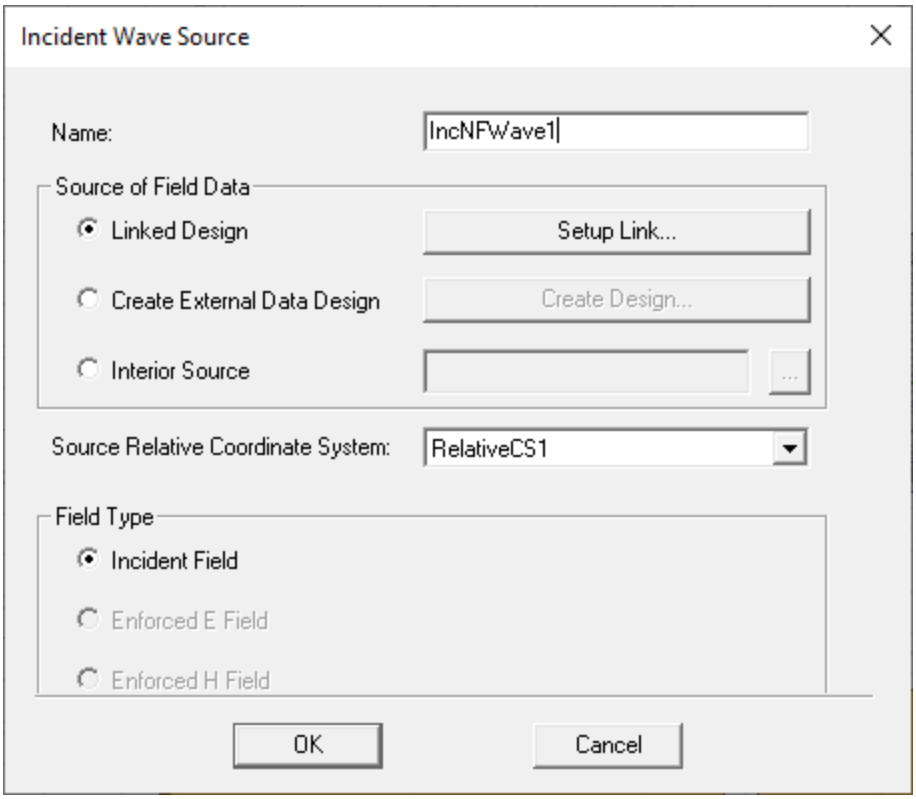
> **Note:**
>
> Go to Simulate source design as needed, for more information about that option.

Preserve source design solution

This is the last option on the **Setup Link** dialog box. When the source design is closed or it is not included in the project manager window containing the target design, the source design will not be saved. However, if you select this option, which is useful when you are using Maxwell or SIwave the results of the source design is saved.



11. Close the Setup Link dialog, and if necessary, specify the Source Relative Coordinate System on the **Near Field Wave Options** panel of the **IncidentWave Source** dialog box.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

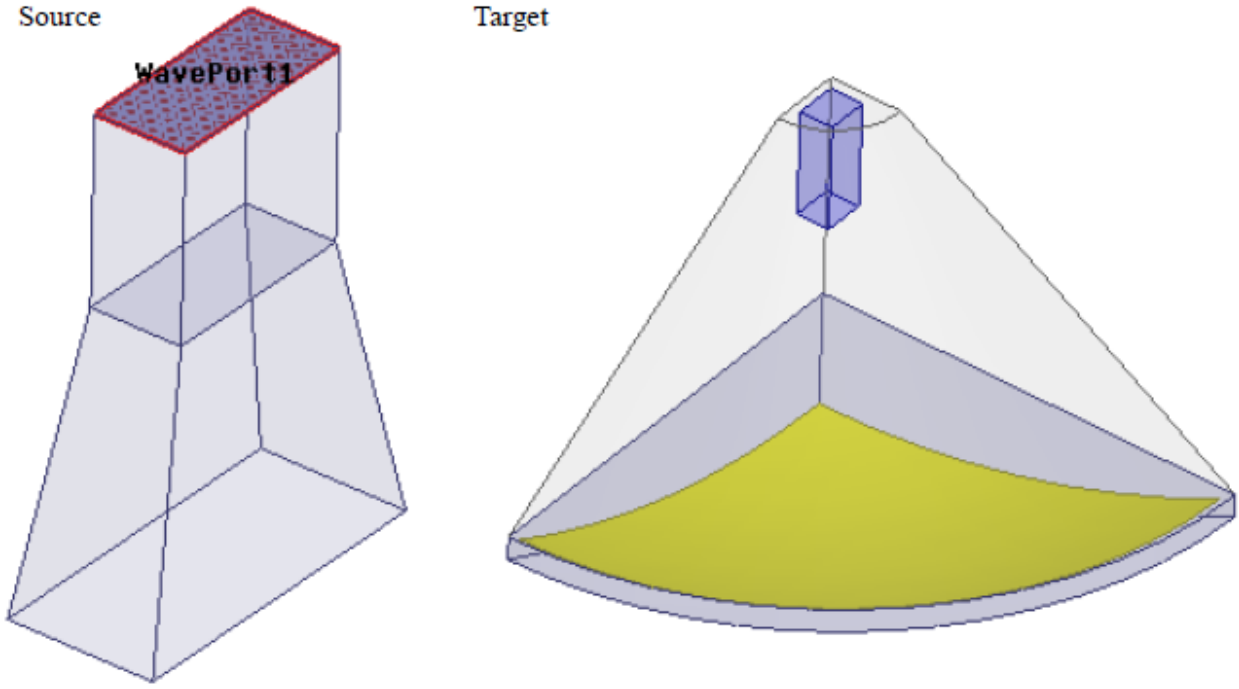
12. Click **OK** to close the dialog. The **Near Field** wave source point and direction is highlighted in the modeler window, and the wave appears in the **Excitations** list in the **Project**.

Simulate source design as needed

Select this option if you modify the source design and you want an updated source design solution to be linked to the target design. If you select this option, make sure that you invalidate the cached link data in the target. To do this, right-click **Analysis** and select the option **Clear Linked Data** from the short-cut menu.

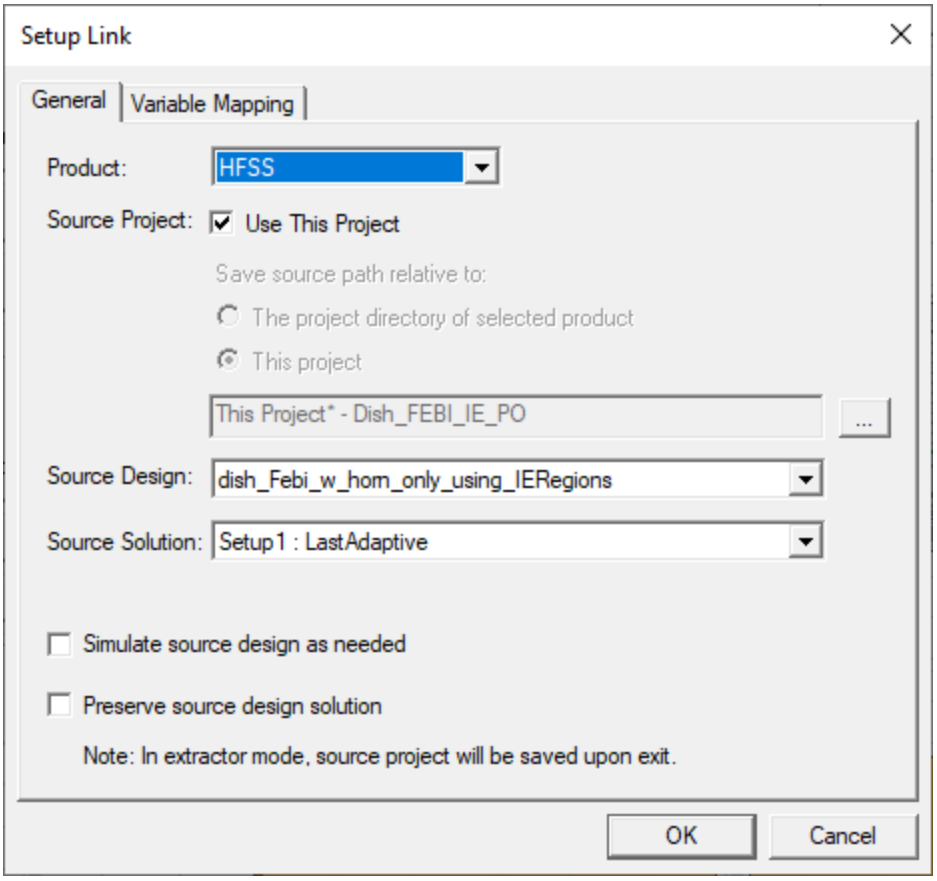
Example that Uses Near Field Links

The following figure shows a horn antenna that will be dynamically linked to a target disk where the source will be included inside of the target.



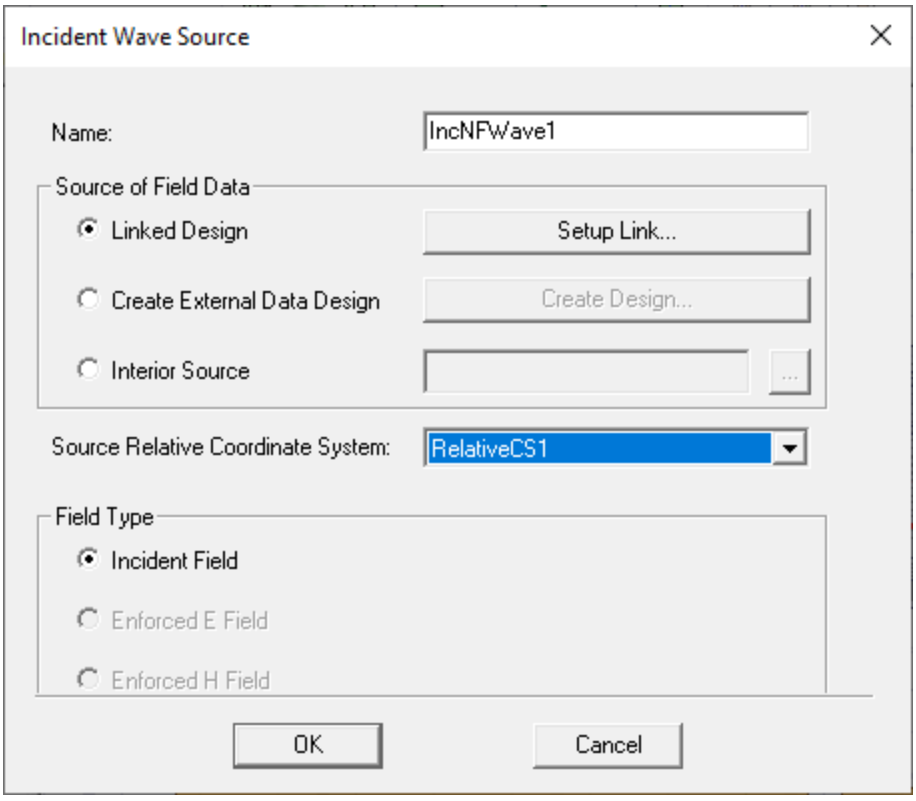
The Near Field Setup Link settings are as shown in the following figure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

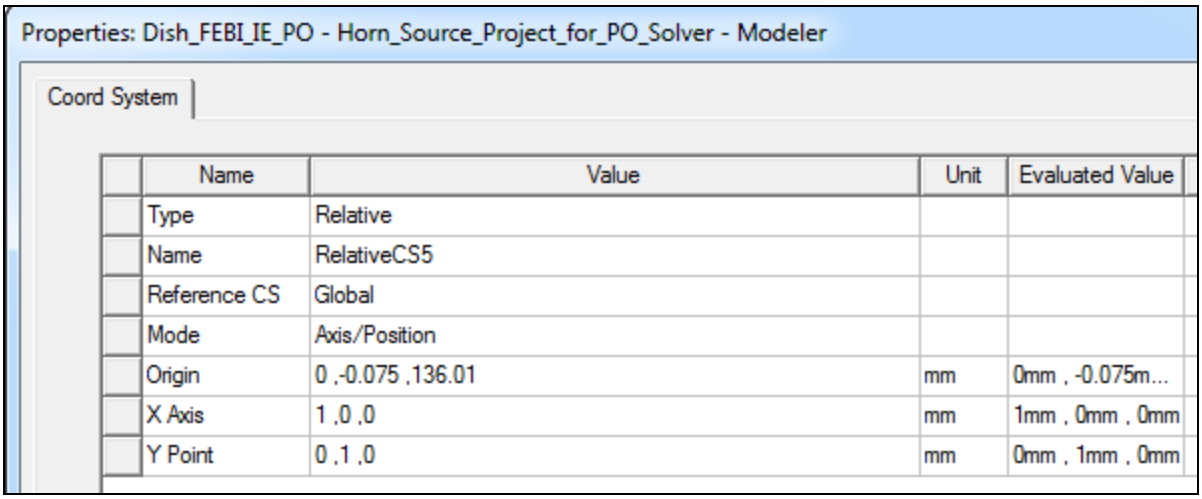


In the example of the horn antenna linking to a target dish, the co-ordinate systems of the source and the target coincide.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



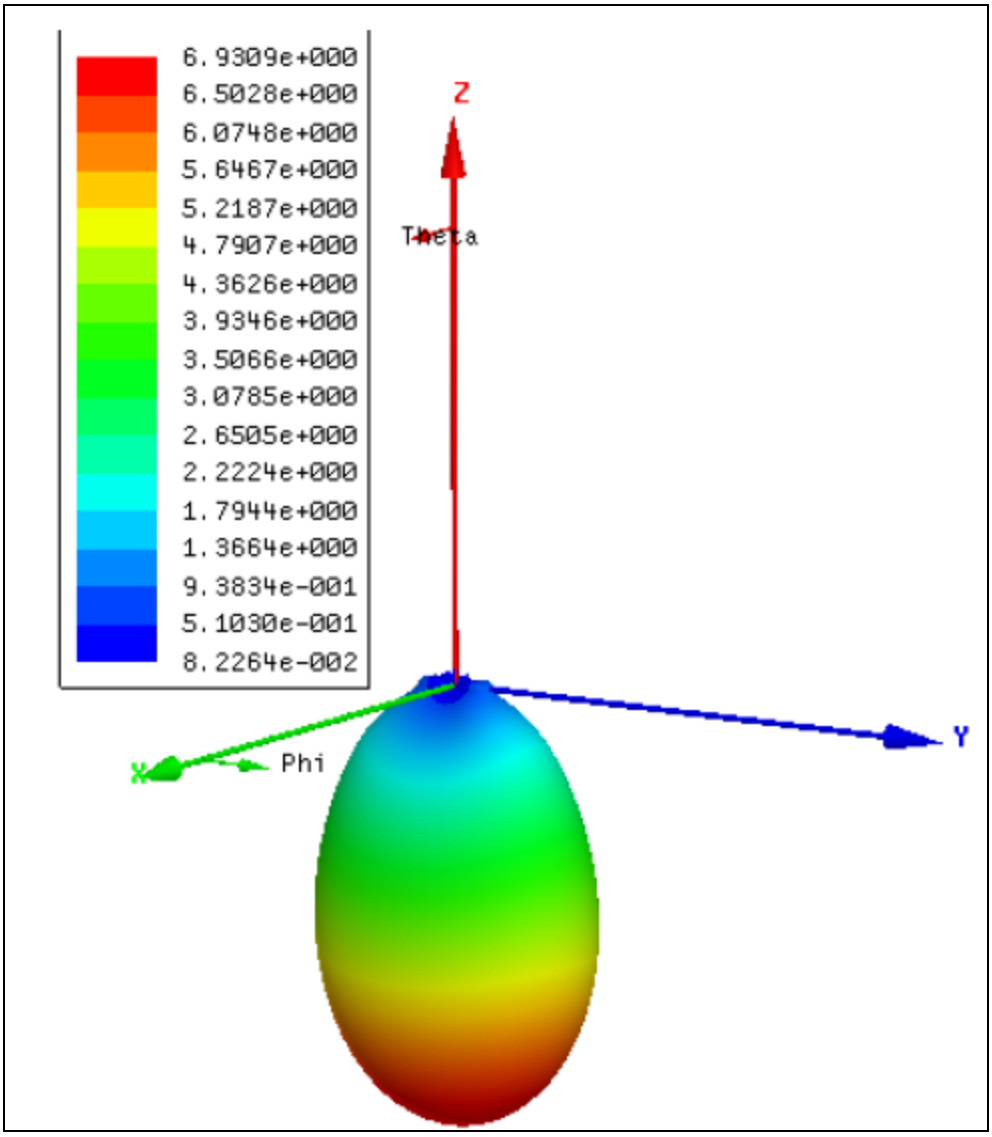
You can use the **Source Relative Coordinate System menu** to select the source co-ordinate system. For instance if you want to move the source and its co-ordinate system closer to the dish by 15 mm you can edit the Relative CS properties for the origin.



Name	Value	Unit	Evaluated Value
Type	Relative		
Name	RelativeCS5		
Reference CS	Global		
Mode	Axis/Position		
Origin	0 ,-0.075 ,136.01	mm	0mm , -0.075m...
X Axis	1 ,0 ,0	mm	1mm , 0mm , 0mm
Y Point	0 ,1 ,0	mm	0mm , 1mm , 0mm

The 3D polar plot with the values of directivity for the source project (the standalone horn antenna without the dish) is shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

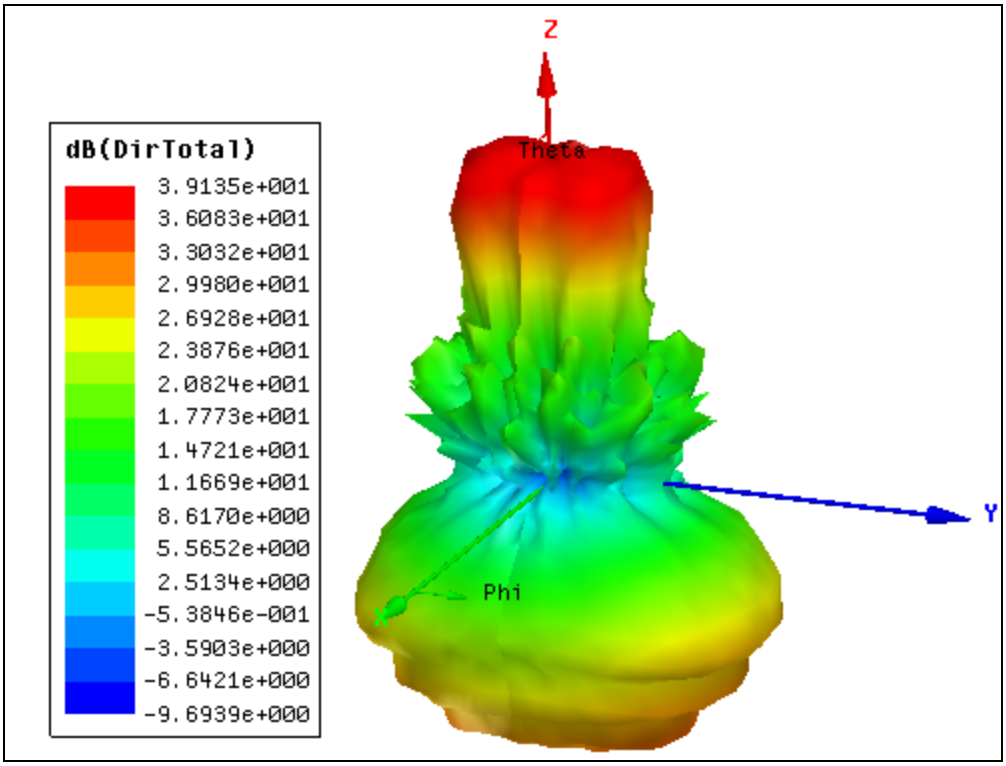


Then, when we apply the fields to the Target project with the dish, we get the values of the directivity shown in the following figure.

Note:

The standalone horn antenna without the dish radiates in the negative Z direction while the target project which includes the dish antenna, radiates in the positive Z-direction. The 3D polar plot with the values of directivity after the application of the Near Field link is shown in the following figure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Example with Interior Source for Near Field Links

HFSS includes an option to add this external near field link as "interior" source. When you select Interior Source you select a data file (.and/.nfd) and HFSS then automates a series of setups, such as near field geometries, mesh operation setup and solution setup, similar to what current Linked Design does. This interior source has the same effect as if its equivalent external near field design is linked to that target design. This interior source also allow relative coordinate system definition.

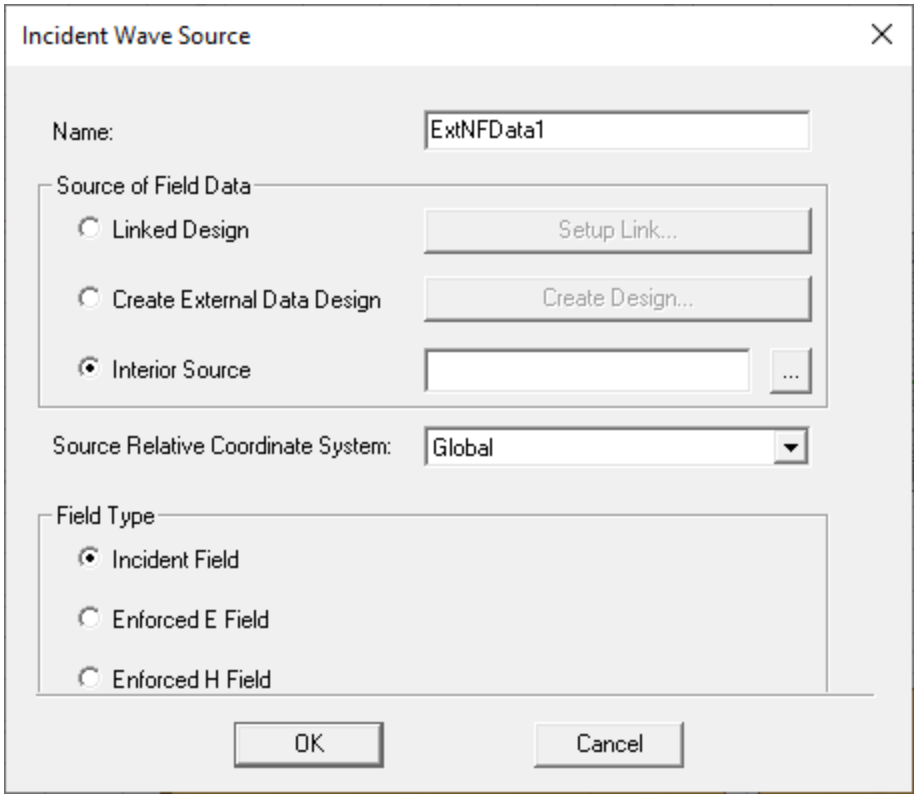
First we will bring up the **Near Field Wave Options** panel and then, assign Near Field Wave.

1. Click **HFSS>Excitations>Assign>Linked Field>Near Field...**.

   The **Incident Wave Source** dialog box appears. Select Interior Source.

The Interior Source settings are as shown in the following figure.

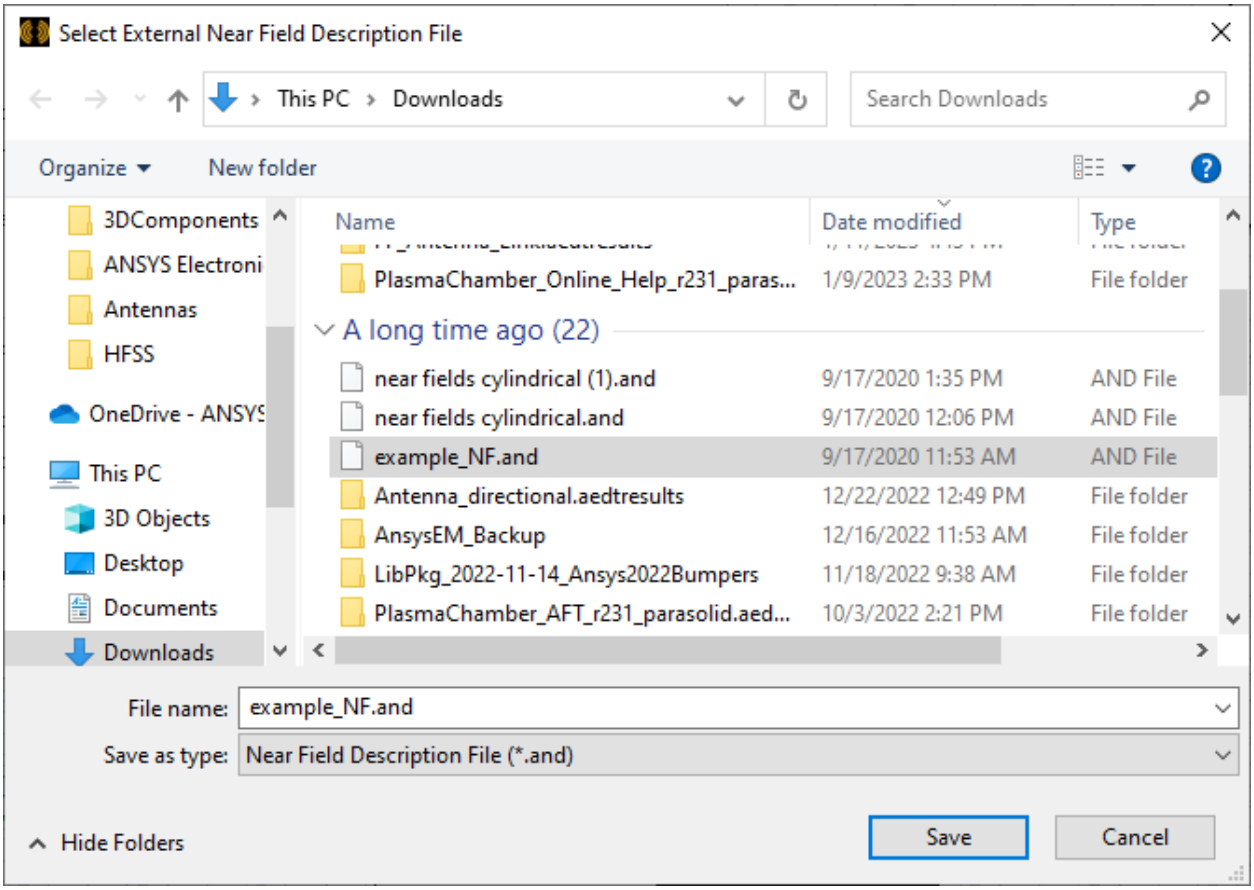
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Click **...** to open a browser to select the .and file with the near field data.

   The **Select External Near Field Description File** dialog box appears.

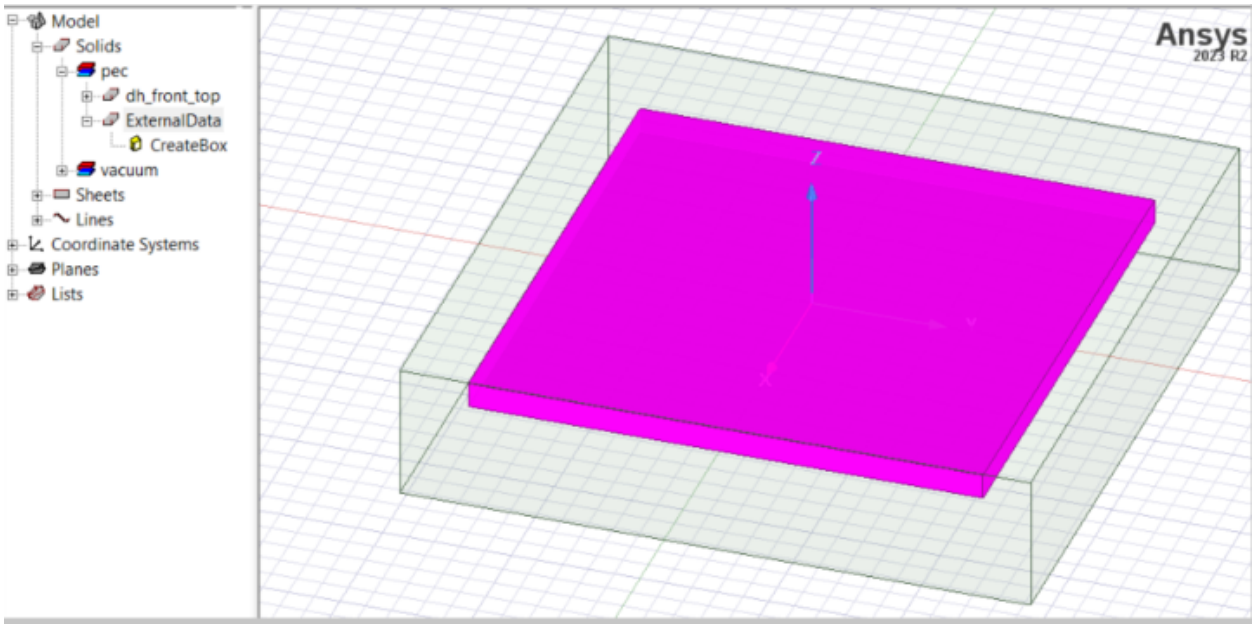
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



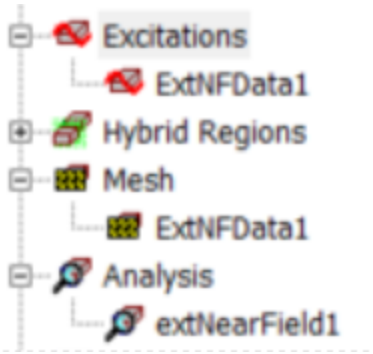
3. Select a previously prepared Near field description file (*.and) and Save.

   HFSS then creates the near field geometry as specified in the .and file. It also creates a corresponding mesh setup and solution setup if design doesn't have them.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



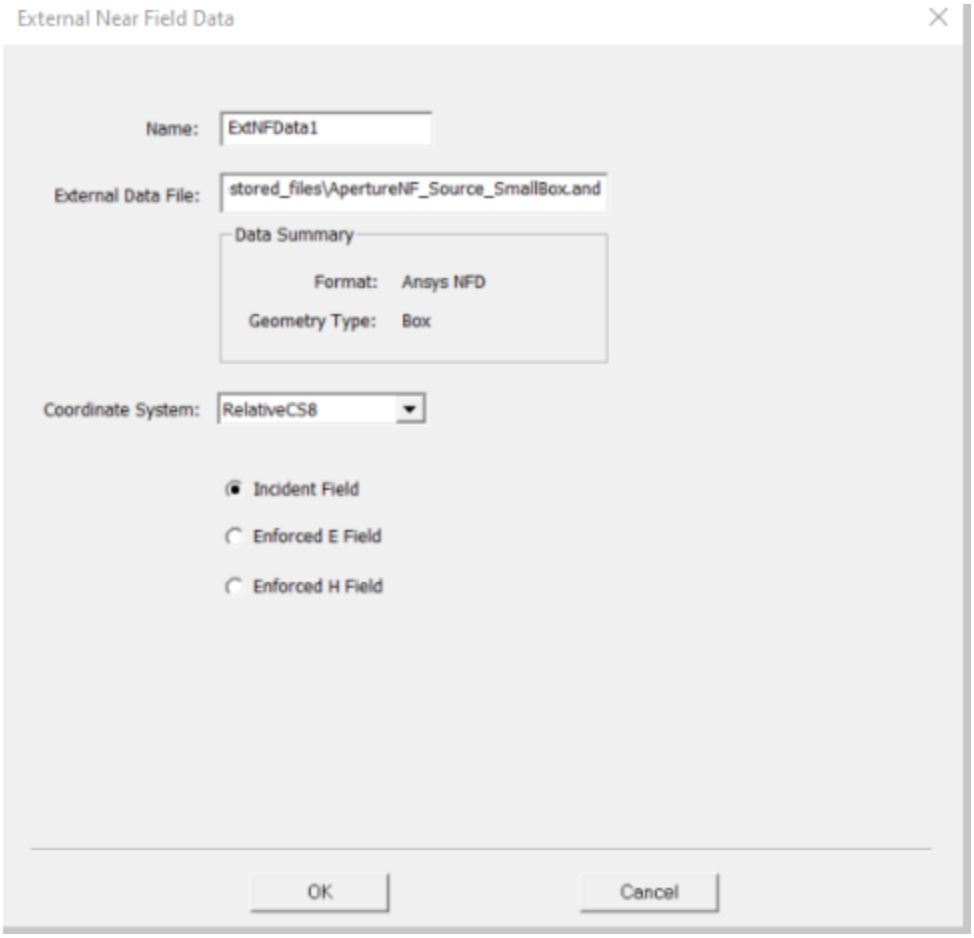
The Project tree shows the Linked Field External Near Field Data, the new mesh, and a setup for the near field interior source.



Editing the Interior Near Field Source

You can edit this interior near field source by either double clicking the excitation or through the right-click menu. A different configuration dialog appears, where you can change the data file, the coordinate system, and the field type.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Access Near Field External Data File

In this section we will show how to access a source of near field which is imposed on a target design through a data file. Instead of using a **Linked Design** as a source for a near field you can also use data files to impose the near field upon your target project.

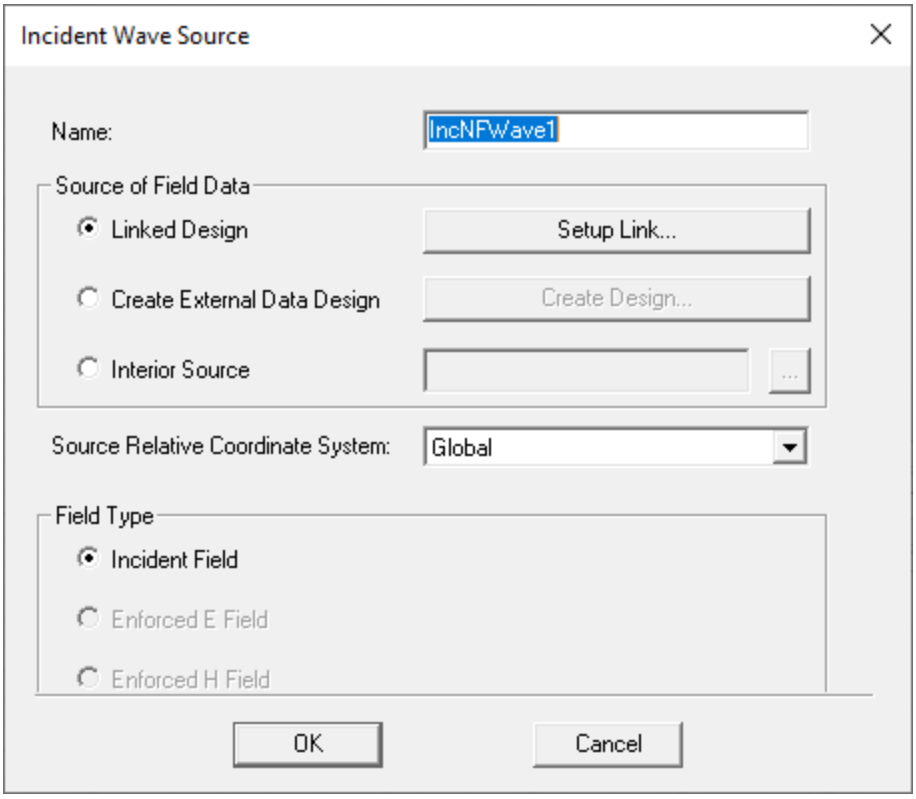
> **Note:**
>
> For more information about Ansys Near Field Data File (**.and*) format see the Ansys Near Field Data File section.

Presupposing that you have prepared the *.and file and associated near field data files:

1. Go to **HFSS**>**Excitations**>**Assign**>**Linked Field**>**Near Field Wave...**.

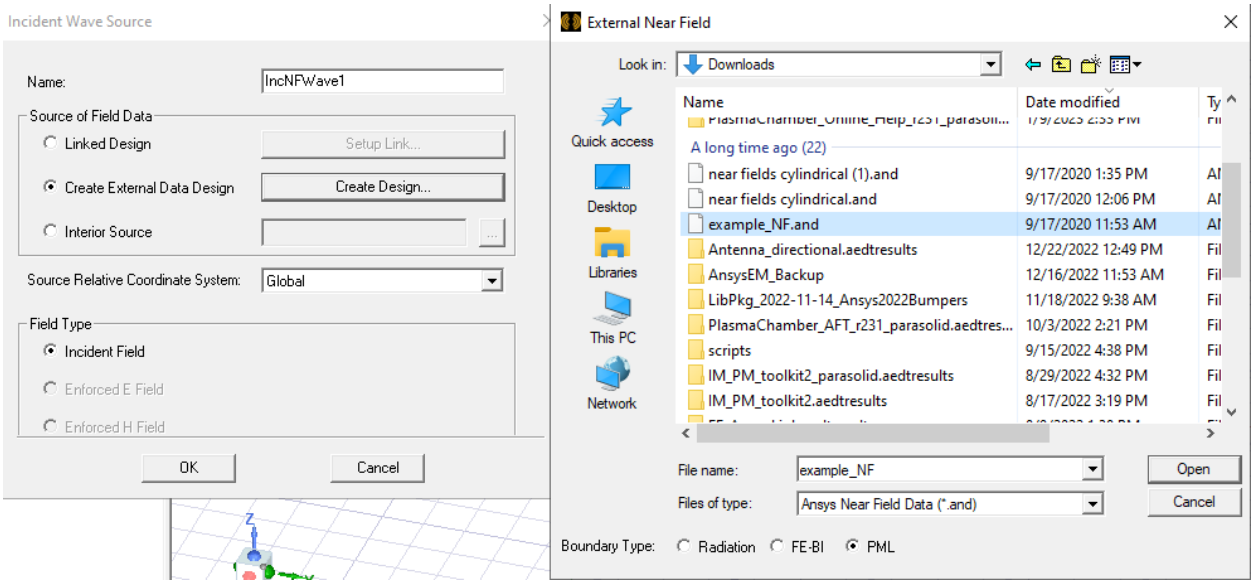
   The **Incident Wave Source**:dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



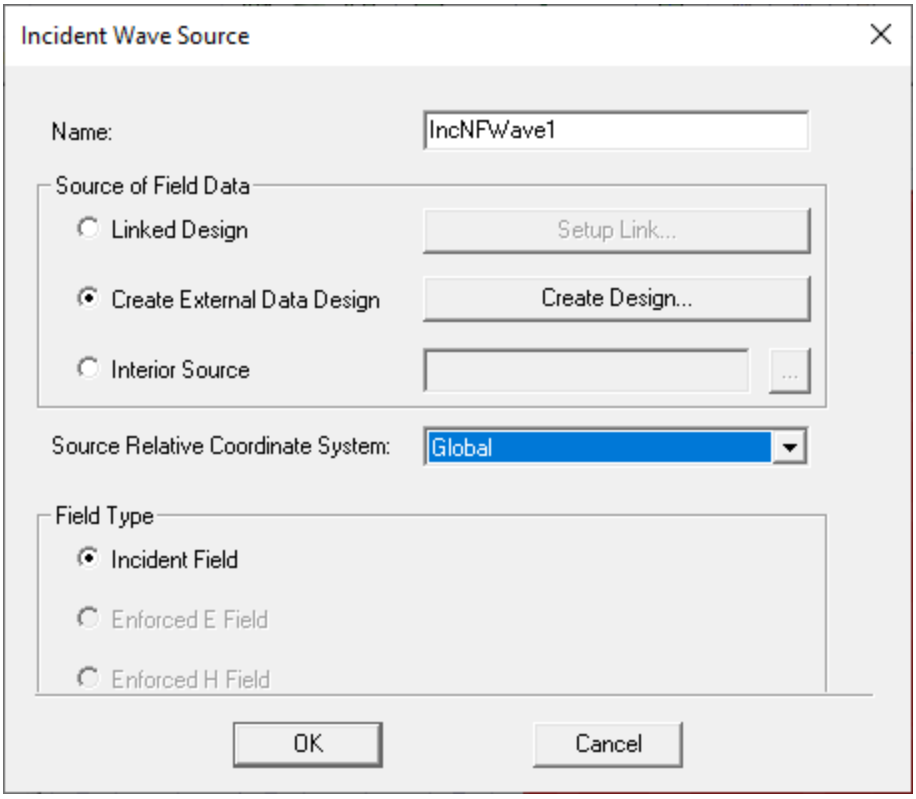
3. Select the **Create External Data Design** radio button to enable the **Create Design...** button.
4. Click **Create Design...** to browse your computer to access a previously created **.and* file, which describes the name and location of one or more the near field data files (*.nfd).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

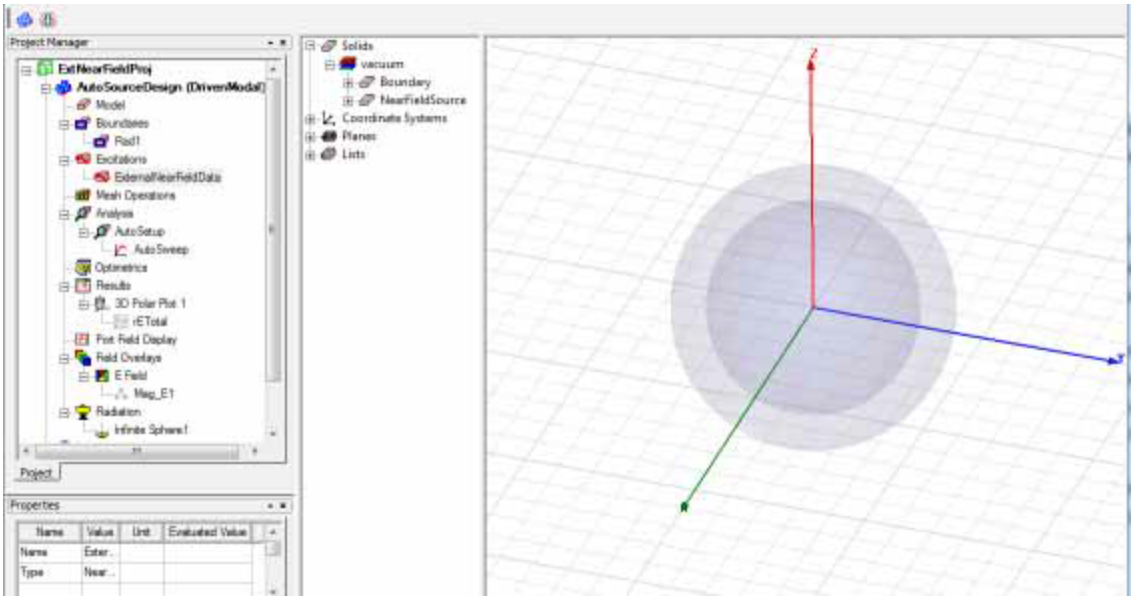


4. Select the Boundary Type for the auto-generated design, as Radiation, FE-BI or PML (the default).
5. If necessary, specify the Source Relative Coordinate System on the **Near Field Wave Options** panel of the **Incident Wave Source** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Upon a successful source design generation, the datalink setup for the near field incident wave source will be automatically set to the generated source design.

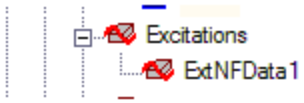


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This auto-design generation is neither undo-able nor scriptable. This design becomes the source design for external near field. This workflow allows you to view the external field data via HFSS.

Automatically generated design components include:

- **Geometry**: the external geometry and a region.
- **Boundaries**: your choice of Radiation, FE-BI or PML (the default).
- **Excitations**: an External Near Field Data Excitation. This is an excitation type for user defined external near field.

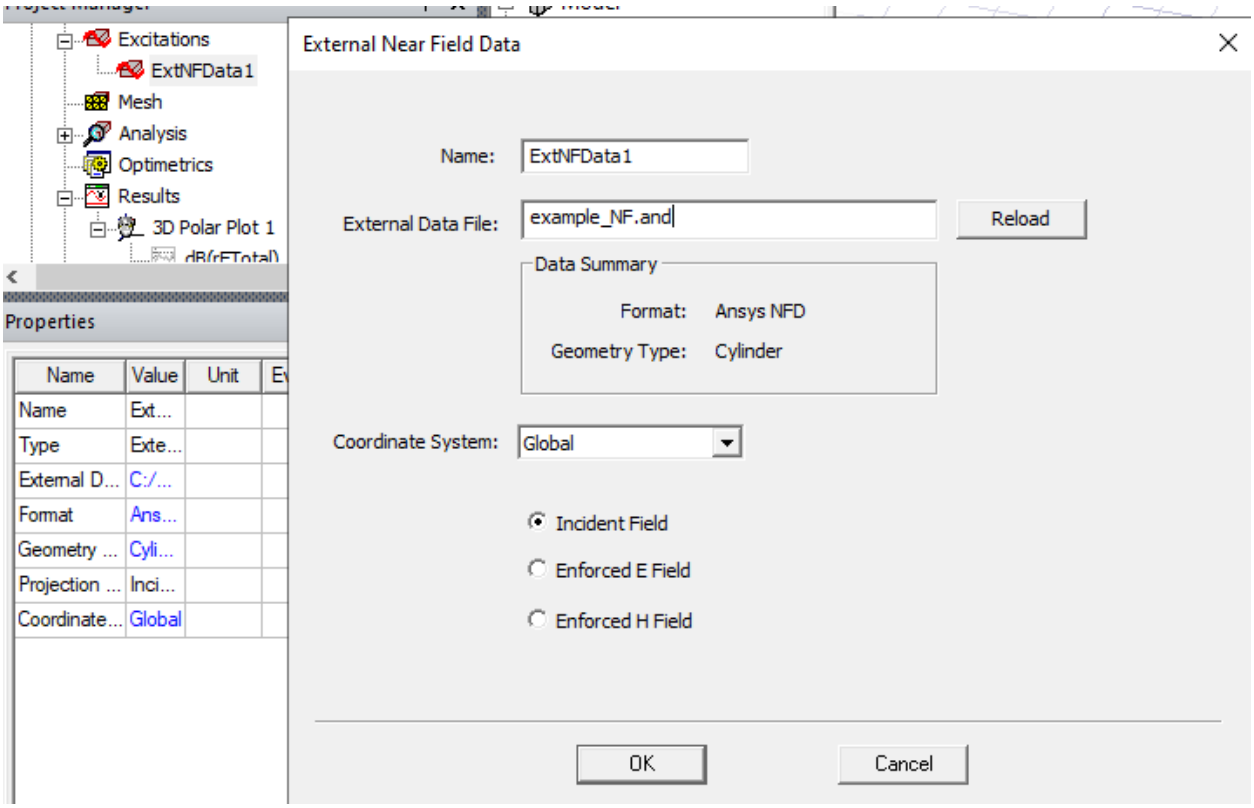


The padding ratio is the same as open region.

- For plane case (where "plane" is the geometry specified in the .AND file), it is an open region.
- For all other cases (for other geometries specified in the .AND file), it is auto-open.

  Accessing the **External Near Field Data Excitation** dialog can only be done through the Auto Design under Excitations in Project tree. Other than the Name, no other properties this Excitation are editable through this dialog. It exists to provide you with information about the AND File that was used to create this Auto-Design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Analysis**:

  A default solve setup is created with the adaptive frequency set to the highest frequency in the external data

  A discrete frequency sweep is created if multiple frequencies are present in the External Data.

6. When you run the Analysis of your target design, it uses this newly created source design for the Near Fields.

Near/Far Field Polarization Definition Table for HFSS, SBR+ and Hybrid Designs

This table summarizes which coordinate system is used to report Near and Far field polarization components (XYZ and Phi/Theta) in HFSS design with only FEM solver, SBR+ design, and HFSS design with Hybrid SBR+ region.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Near Field and Far Field setup options and design type	CS used to report XYZ polarization	CS used to report Theta/Phi polarization
SBR+ design type		
NF rectangle	Global CS	Global CS
NF sphere	Global CS	Global CS
Far Field sphere	Relative FF CS	Relative FF CS
HFSS (FEM) design type		
NF rectangle	Global CS	Global CS
NF sphere	Relative NF sphere CS	Relative NF sphere CS
Far Field sphere	Relative FF CS	Relative FF CS
HFSS (FEM) design type with Hybrid SBR+ region		
NF rectangle	Global CS	Global CS
NF sphere	Global CS	Global CS
Far Field sphere	Relative FF CS	Relative FF CS

Ansys Near Field Data File

The data file that you select when you click **Create External Design** of the **Incident Wave Source** dialog box is in the **.and* format. The **.and* file describes the location and contents of one or more Ansys Near Field Data Files which are in the **.nfd* format. If no units are specified for spherical, cylindrical, or Cartesian coordinates, the **.ndf* files use SI units of radians and meters respectively. You can append units, such as mm or deg, as needed for each coordinate value.

An example of a **.and* file is shown below, followed by examples of .nfd files with Cartesian and Cylindrical coordinate systems. The File format specification is provided in *Technical Notes*: Ansys Near Field Description Format Specification.

Sample **.and* File Sphere Geometry

```
$begin 'NearFieldHeader'
  type = 'nfd'
  geometry = 'sphere'
  center = '0mm, 0mm, 0mm'
  radius = '20mm'
  fsweep='1GHz, 10GHz'
$end 'NearFieldHeader'

$begin 'NearFieldData'
  FreqData("1GHz", "exportfields-1GHz.nfd")
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  FreqData("2GHz", "exportfields-2GHz.nfd")
  FreqData("3GHz", "exportfields-3GHz.nfd")
  FreqData("4GHz", "exportfields-4GHz.nfd")
  FreqData("5GHz", "exportfields-5GHz.nfd")
  FreqData("6GHz", "exportfields-6GHz.nfd")
  FreqData("7GHz", "exportfields-7GHz.nfd")
  FreqData("8GHz", "exportfields-8GHz.nfd")
  FreqData("9GHz", "exportfields-9GHz.nfd")
  FreqData("10GHz", "exportfields-10GHz.nfd")
$end 'NearFieldData'
```

This example is a simple text file and saved with the *.and* extension. It is clear from the sample that the near field data was generated on a sphere centered at the origin and radius = 20mm for frequencies 1 GHz through 10 GHz. For the sample **.and* file 10 **.nfd* files are specified so 10 **.nfd* files were generated. Examples of such Ansys Near Field Data Files (*exportfields-1GHz.nfd*) are shown below.

Users can also use external data in the field format of J and M, J only and M only.

Sample *.and File, Cylinder Geometry

The following example shows a **.and* file using a Cylindrical Coordinate System

```
$begin 'NearFieldHeader'
  type='nfd'
  fcoords='cylindrical'
  geometry='cylinder'
  center='0mm,0mm,0mm'
  axis='0,0,1'
  radius='20mm'
  height='258mm'
  fsweep='1GHz'
$end 'NearFieldHeader'
$begin 'NearFieldData'
FreqData("1GHz", "fields_cylinder.nfd")
$end 'NearFieldData'
```

Sample NFD File in Cartesian Coordinate System

The sample below shows the near field data that the *.nfd file contains. The file uniquely specifies the position of several points (700 in this case) and the real and imaginary values of the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

E field and H field vectors at these points along the x, y, and z axes in the Cartesian Coordinate system.

```
#Index, X, Y, Z, Ex(real, imag), Ey(real, imag), Ez(real, imag), Hx
(real, imag), Hy(real, imag), Hz(real, imag)
Frequencies 5
Frequency 1.000000e+009
1, 0, 0, 0.02, 0.0938, -0.1140, -0.0136, -0.0132, -0.6038,
-0.4006, 0.0004, -0.0004, -0.0006, -0.0008, -0.0037, 0.0038
....... ....... ....... ....... ....... .......
....... ....... ....... ....... ....... .......
700, 2.1e-018, -1.2e-018, -0.02, 0.0839, -0.1080, -0.0125,
-0.012, -0.0213, 0.1423, -0.0002, 0.0002, 0.0003, 0.0009,
-0.003, 0.0032

Frequency 5.000000e+009
....... ....... ....... ....... ....... .......
....... ....... ....... ....... ....... .......
```

Sample NFD File in Spherical Coordinate System

The sample below shows the near field data that the *.nfd file contains. The file uniquely specifies the position of several points (700 in this case) and the real and imaginary values of the E field and H field vectors at these points along the r, theta, and phi directions in the Spherical Coordinate system.

```
#Index, Theta, Phi, Er(real, imag), Etheta(real, imag), Ephi(real,
imag), Hr(real, imag), Htheta(real, imag), Hphi(real, imag)
Frequencies 5
Frequency 1.000000e+009

1, 0.000000e+000, 0.000000e+000, -1.835144e-001, -1.822055e-001, -
2.648018e-002, -2.096681e-001, -1.281892e-001, -5.361222e-002,
1.279651e-004, -1.016122e-004, 6.289402e-004, -6.598072e-004, -
1.281892e-001, -5.361222e-002
....... ....... ....... ....... ....... .......
....... ....... ....... ....... ....... .......

700, 3.141593e+000, 5.759587e+000, -1.282983e-001, -1.141968e-001, -
2.207641e-001, -9.119582e-002, -2.669357e-001, -1.063323e-001,
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
6.827586e-005, -9.142062e-005, 1.030250e-003, -1.406250e-003, -
2.669357e-001, -1.063323e-001

Frequency 5.000000e+009
....... ....... ....... ....... ....... .......
....... ....... ....... ....... ....... .......
....... ....... ....... ....... ....... .......
```

Sample NFD File in Cylindrical Coordinate System

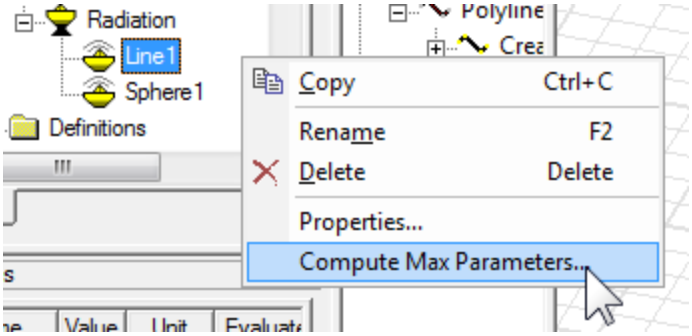
```
#Index, Rho, Phi, Z, Erho(real, imag), Ephi(real, imag), Ez(real,
imag), Hrho(real, imag), Hphi(real, imag), Hz(real, imag)
Frequencies 1
Frequency 1.000000000000000e+009
1, 2.0e+01mm, 0.0000000000000000e+000, -1.29e+2mm, 1,0, 1,0, 1,0,
1,0, 1,0, 1,0
2, 2.0e+01mm, 0.7853981633974483e+000, -1.29e+2mm, 1,0, 1,0, 1,0,
1,0, 1,0, 1,0
3, 2.0e+01mm, 1.5707963267948966e+000, -1.29e+2mm, 1,0, 1,0, 1,0,
1,0, 1,0, 1,0
4, 2.0e+01mm, 2.3561944901923449e+000, -1.29e+2mm, 1,0, 1,0, 1,0,
1,0, 1,0, 1,0
5, 2.0e+01mm, 3.1415926535897931e+000, -1.29e+2mm, 1,0, 1,0, 1,0,
1,0, 1,0, 1,0
6, 2.0e+01mm, 3.9269908169872415e+000, -1.29e+2mm, 1,0, 1,0, 1,0,
1,0, 1,0, 1,0
7, 2.0e+01mm, 4.7123889803846897e+000, -1.29e+2mm, 1,0, 1,0, 1,0,
1,0, 1,0, 1,0
8, 2.0e+01mm, 5.4977871437821382e+000, -1.29e+2mm, 1,0, 1,0, 1,0,
1,0, 1,0, 1,0
9, 1.0e+01mm, 0.0000000000000000e+000, -1.29e+2mm, 1,0, 1,0, 1,0,
1,0, 1,0, 1,0
10, 1.0e+01mm, 0.7853981633974483e+000, -1.29e+2mm, 1,0, 1,0, 1,0,
1,0, 1,0, 1,0
11, 1.0e+01mm, 1.5707963267948966e+000, -1.29e+2mm, 1,0, 1,0, 1,0,
1,0, 1,0, 1,0
...
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

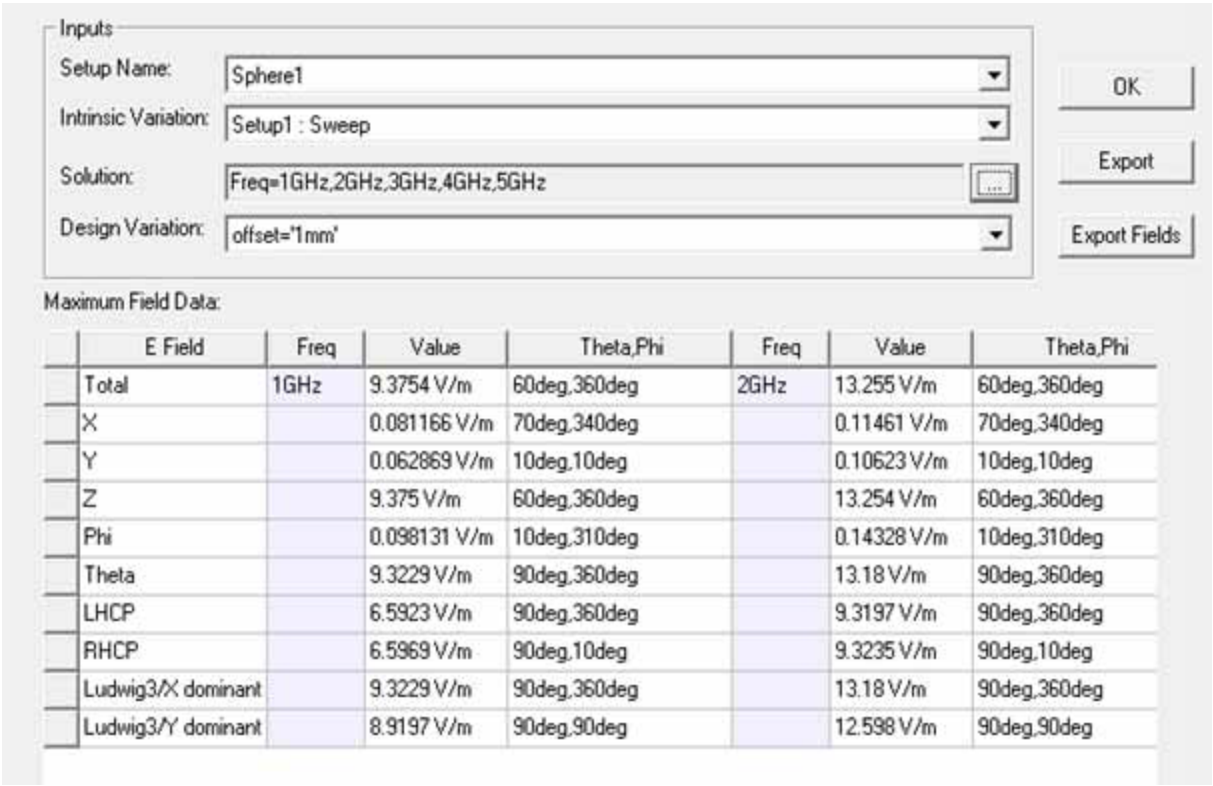
Generate Near Field Data File

Generate the ***.nfd*** file from your existing data or from solved HFSS design with near fields using the following steps.

1. Right-click the previously created near-field **Sphere or Line1** under **Radiation** as shown below.



2. Select **Compute Max Parameters**.
3. The **Max Field Data** dialog box appears for the selected Line or Sphere as shown below.



E Field	Freq	Value	Theta,Phi	Freq	Value	Theta,Phi
Total	1GHz	9.3754 V/m	60deg,360deg	2GHz	13.255 V/m	60deg,360deg
X		0.081166 V/m	70deg,340deg		0.11461 V/m	70deg,340deg
Y		0.062869 V/m	10deg,10deg		0.10623 V/m	10deg,10deg
Z		9.375 V/m	60deg,360deg		13.254 V/m	60deg,360deg
Phi		0.098131 V/m	10deg,310deg		0.14328 V/m	10deg,310deg
Theta		9.3229 V/m	90deg,360deg		13.18 V/m	90deg,360deg
LHCP		6.5923 V/m	90deg,360deg		9.3197 V/m	90deg,360deg
RHCP		6.5969 V/m	90deg,10deg		9.3235 V/m	90deg,10deg
Ludwig3/X dominant		9.3229 V/m	90deg,360deg		13.18 V/m	90deg,360deg
Ludwig3/Y dominant		8.9197 V/m	90deg,90deg		12.598 V/m	90deg,90deg

4. Click **Export Fields** and save the file in the *.nfd* format.

You can use an *.and file to refer to one or more *.nfd files to describe near fields.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using Field Solutions from Other Simulations

HFSS can use dynamic links to other simulation tools to create one seamless workflow. The source designs can be created in HFSS, HFSS 3D Layout, SIwave or Maxwell3D. Some example uses for dynamic links include:

- a detailed and optimized design of a cell phone radiating in a large environment (HFSS to HFSS).
- a Driven Modal (HFSS to HFSS SBR+).
- a complicated printed circuit board in an HFSS 3D Layout project file causing EMC/EMI problems in and around its housing (SIwave to HFSS).
- an electromechanical component causing EMC/EMI problems in a vehicle (Maxwell3D to HFSS).

In all cases, radiated fields from the source project are imposed as an incident wave in the target project. These radiated fields can be far fields and near fields.

> **Note:**
>
> It may be necessary to create the environment variable **`SIWAVE_INSTALL_DIR`** before executing the target application (e.g., SIwave is launched from HFSS and not separately).

> **Note:**
>
> From the target project, radiation boundaries must be assigned to specify where the fields from the source project enter the target project.

> **Note:**
>
> To avoid singular far field integral calculations, do not use identically sized and shaped surfaces in the target and source projects. Use surfaces that are smaller or larger.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using a Dynamic Link Between HFSS and HFSS 3D Layout (Example)

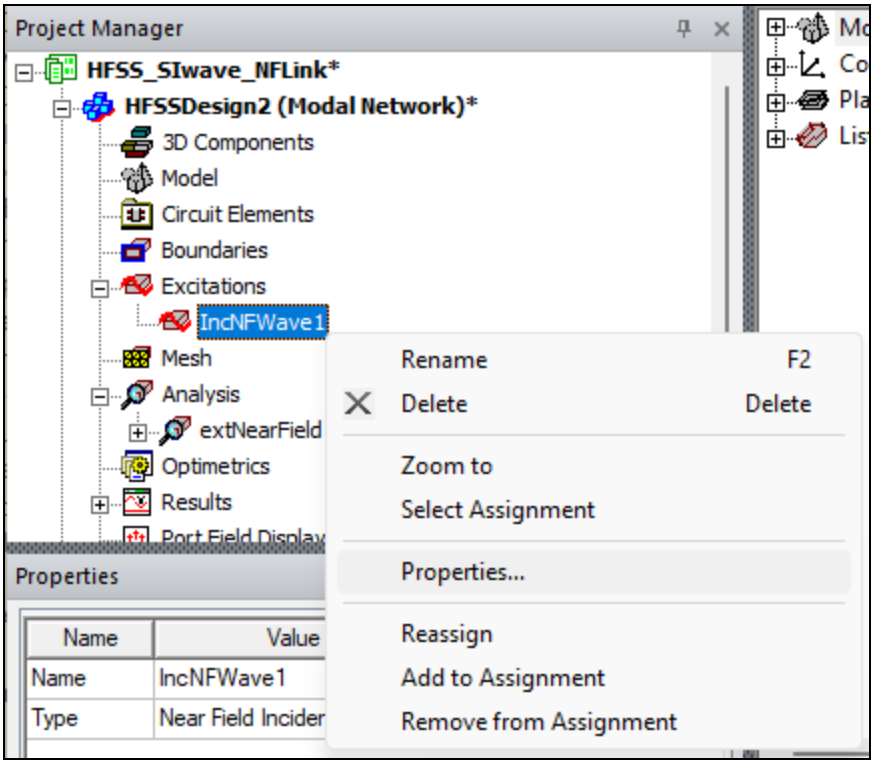
> **Note:**
>
> To execute the following instructions, the user will need an Ansys Electronics Enterprise license or an Ansys Electronics Premium SIwave Plus license.

Add a dynamic link between HFSS and HFSS 3D Layout to utilize HFSS 3D Layout's built-in **SIwave Near Field** solver. Among other benefits, integrating HFSS and HFSS 3D Layout allows bidirectional data and eliminates manual steps in the process, saving time and improving accuracy.

Complete the following steps to create a dynamic link between HFSS and HFSS 3D Layout.

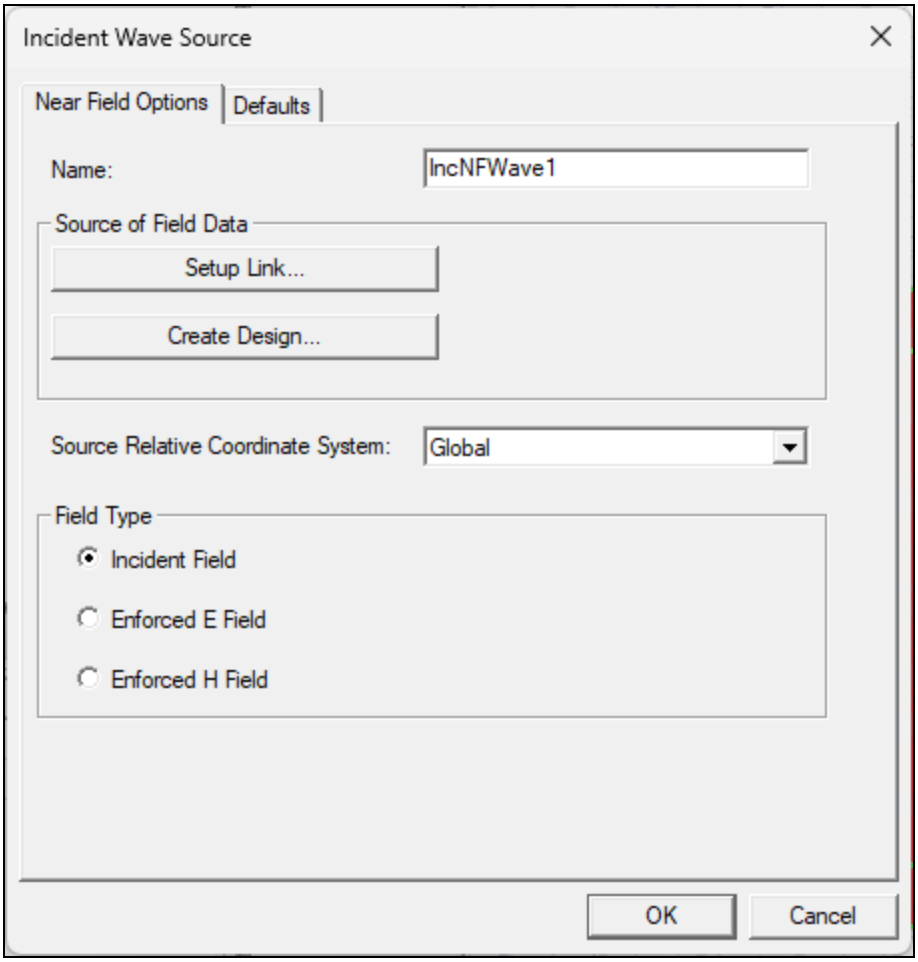
1. From the appropriate target HFSS file, create an Incident Plane Wave excitation.
2. From the **Project Manager** window, expand the **Project Tree** > ***active design folder*** (e.g., **HFSSDesign2**) > **Excitations**. Then do one of the following to open the **Incident Wave Source** window.
    - Double-click the excitation (e.g., **IncNFWave1**).
    - Right-click the excitation and select **Properties**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



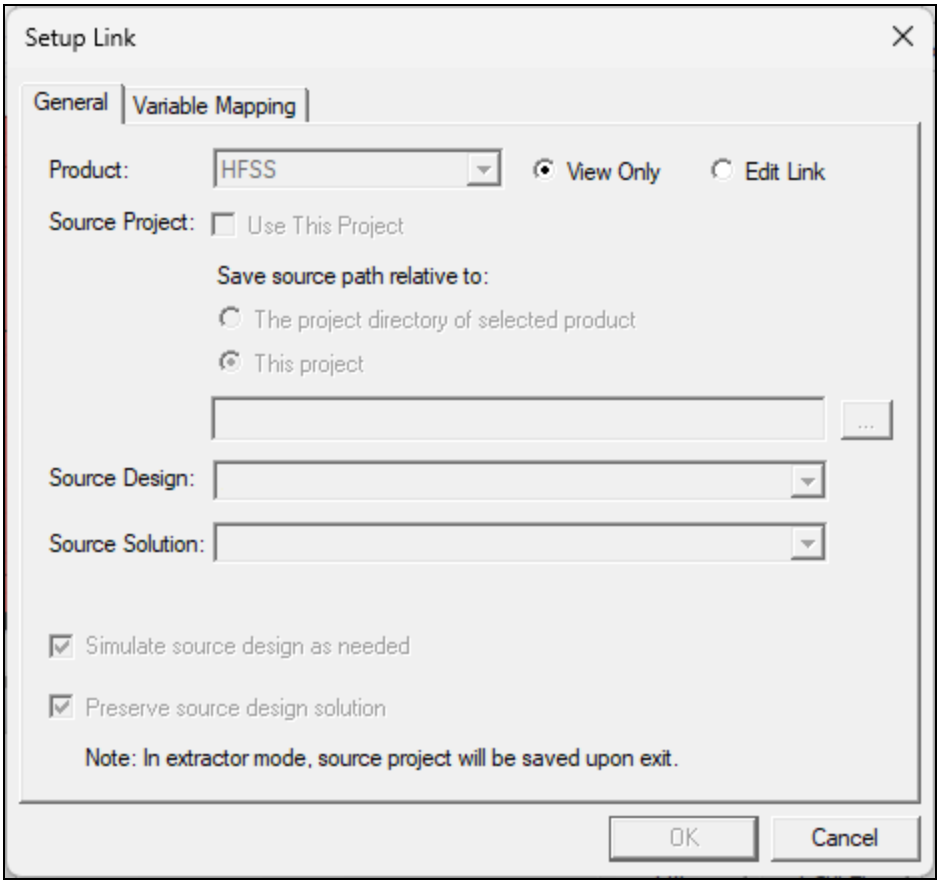
3. From the **Incident Wave Source** window, enter a **Name** for the excitation or accept the default (e.g., **IncNFWave1**).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



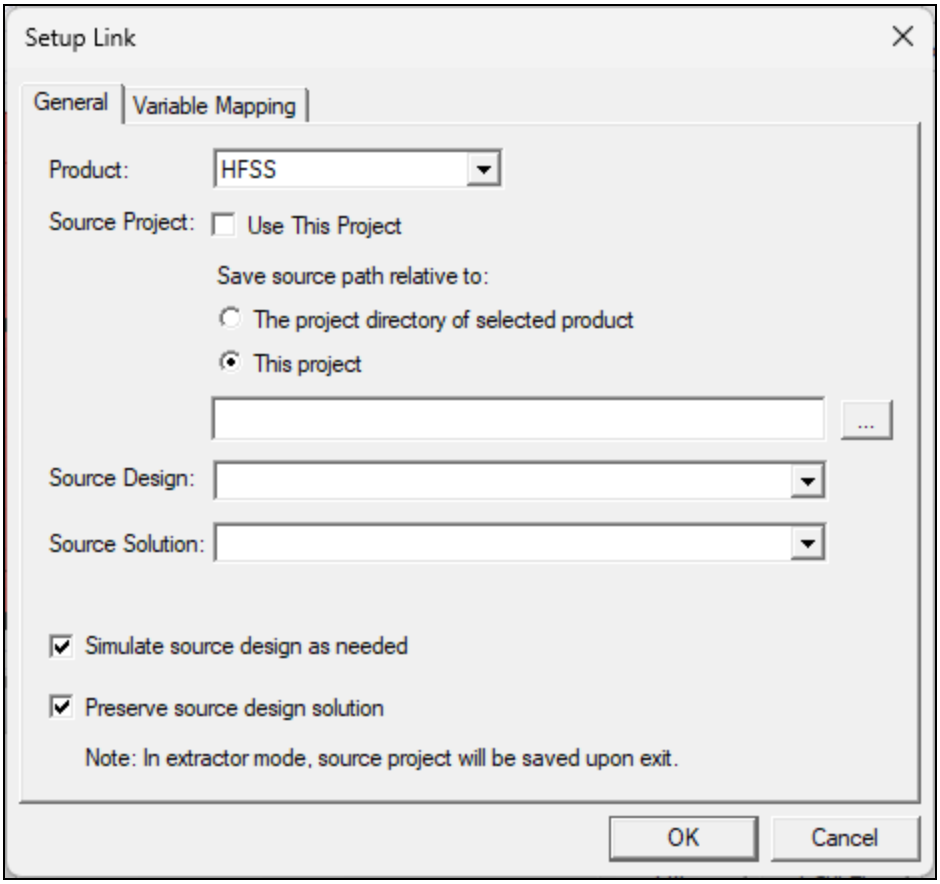
4. From the **Source of Field Data** area, click **Setup Link** to open the **Setup Link** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



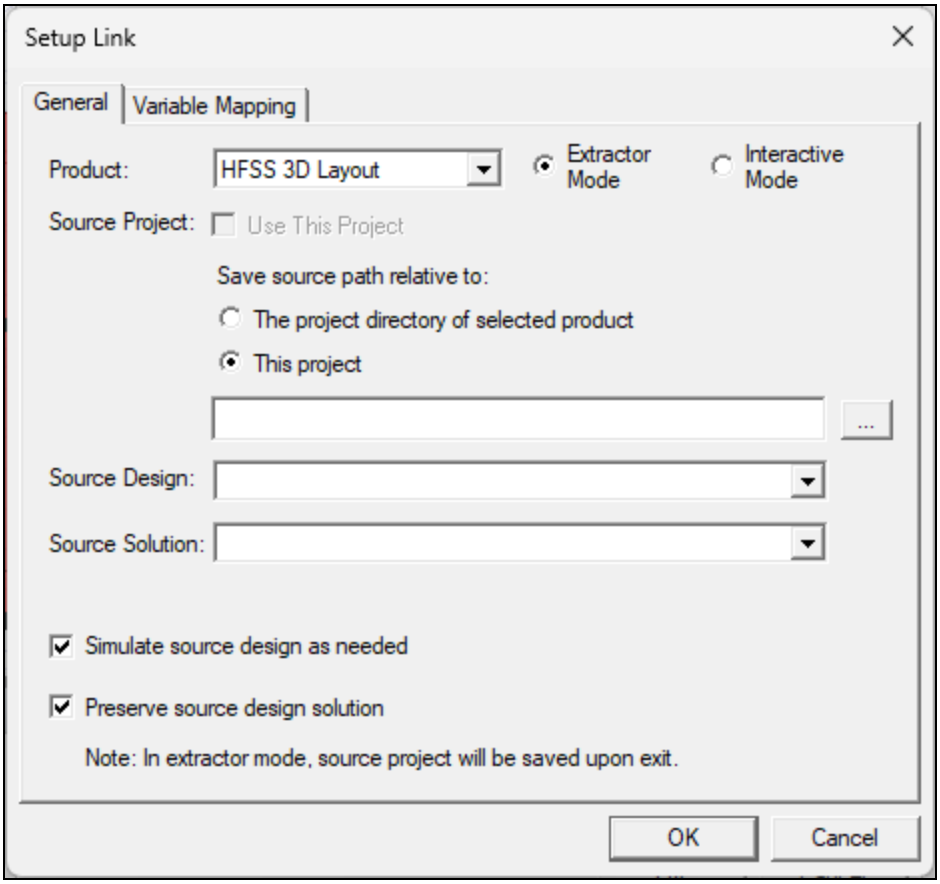
5. From the **Setup Link** window, click **Edit Link** to activate the window and make its contents editable.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



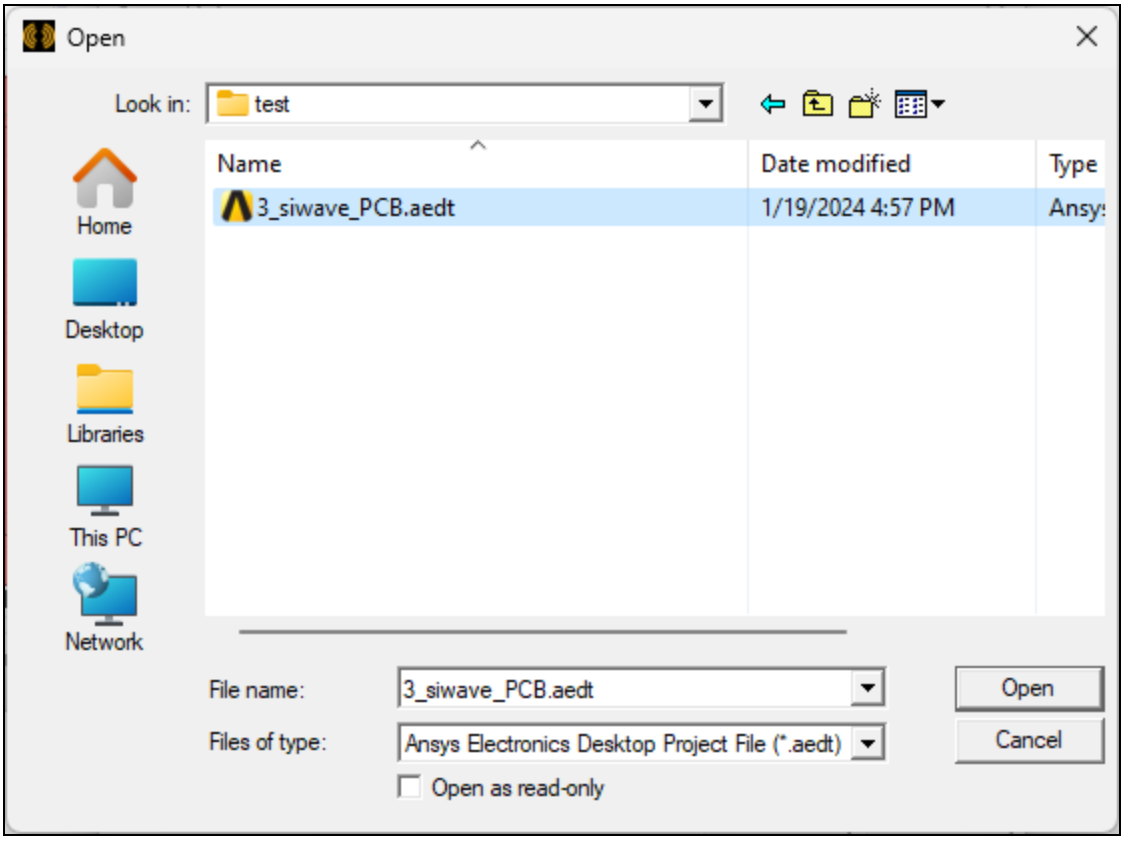
6. Select **HFSS 3D Layout** from the **Product** drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



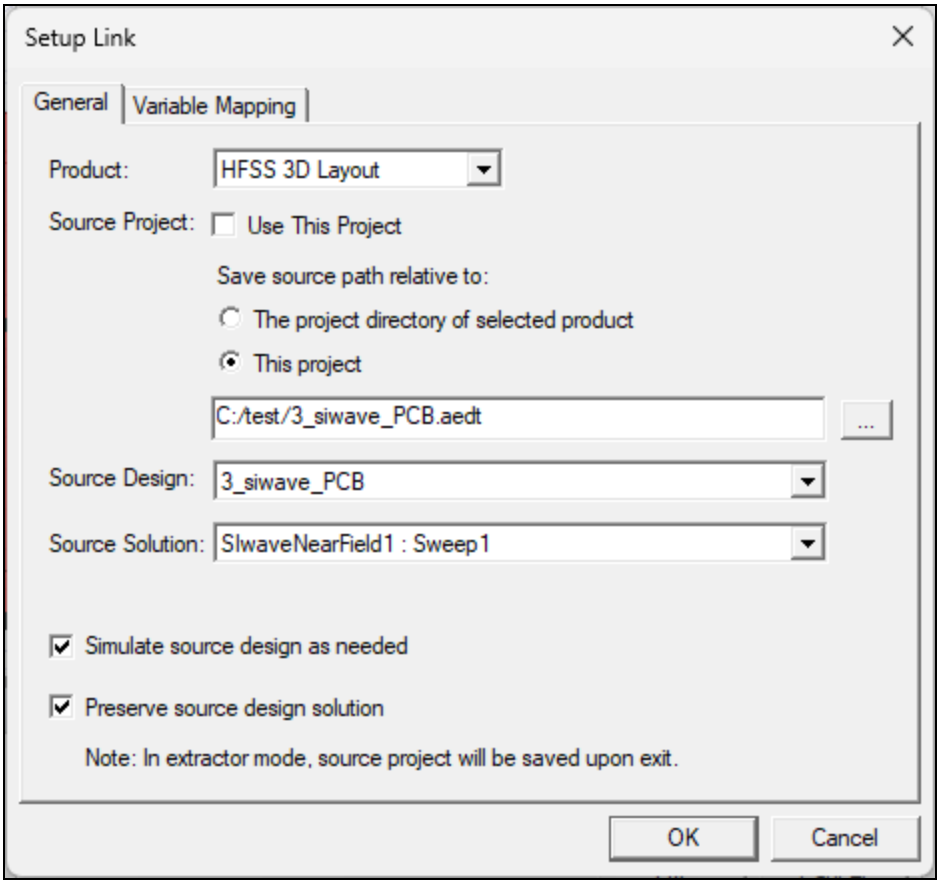
a. Click the **ellipsis** adjacent to the empty field to open an explorer window. Then navigate to the appropriate project file. Either double-click the file or select it and click **Open**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Setup Link** window will populate with the path to the source file, the available designs, and the available solutions. If the user has not already opened the source file, it will open in **Electronics Desktop** and appear in the **Project Manager** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. If necessary, select a more appropriate **Source Design** and/or **Source Solution** from the drop-down menu(s).
8. Check or uncheck the boxes adjacent to the following options:
   - **Simulate source design as needed** — if the source design has been modified, select this option to update the source design solution linked to the target design.

     > **Note:**
     >
     > Ensure the cached link data in the target design is invalidated before proceeding (i.e., from the **Project Manager** window, right-click **Analysis** and select **Clear Linked Data**.

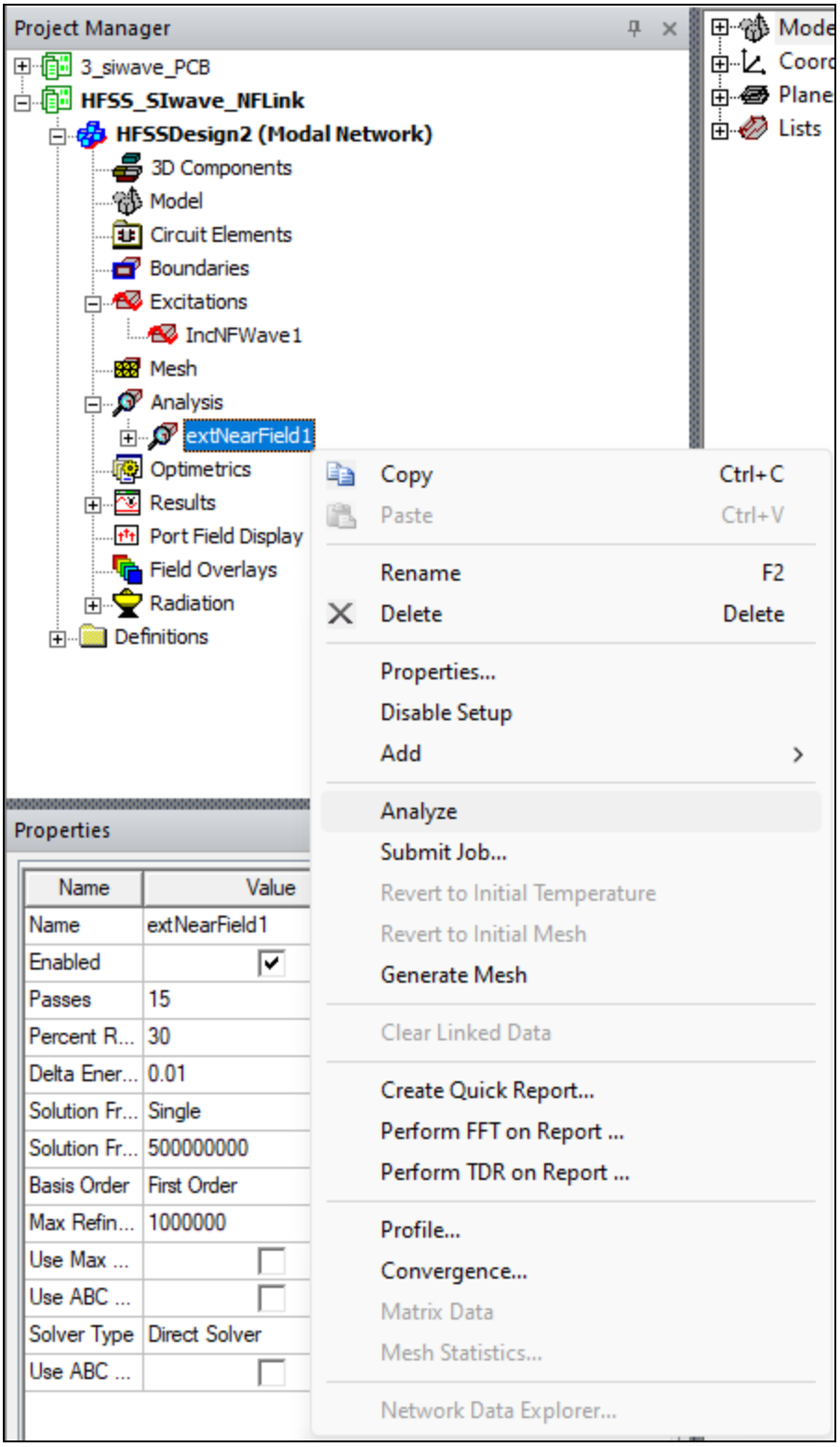
   - **Preserve source design solution** — if the source design is closed, it is **not** included in the **Project Manager** window containing the target design and the source design will **not** be saved. Select this option to save the design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> **Preserve source design solution** is most useful if the source design is an HFSS 3D Layout, Maxwell, or SIwave file.

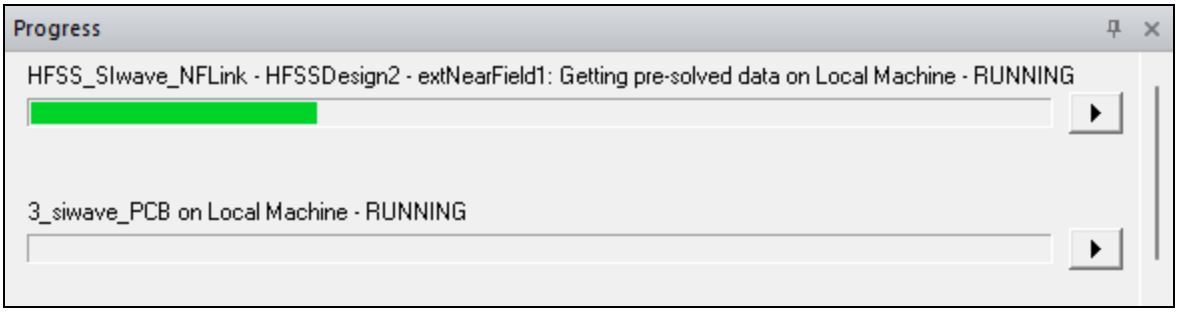
9. Click **OK** to close the **Setup Link** window and establish the link.
10. Click **OK** to close the **Incident Wave Source** window.
11. From the **Project Manager** window, expand **Analysis**. Then right-click the near field solution (e.g., **extNearField1**) and select **Analyze** to begin analysis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

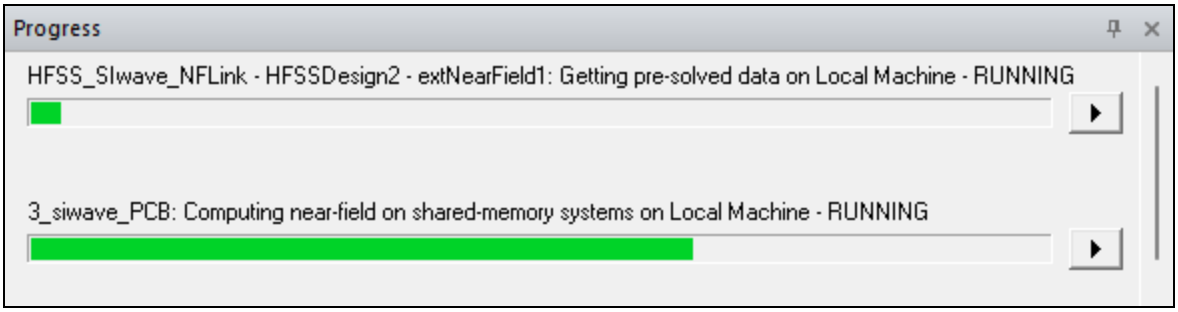


As the analysis proceeds, view its status in the **Progress** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



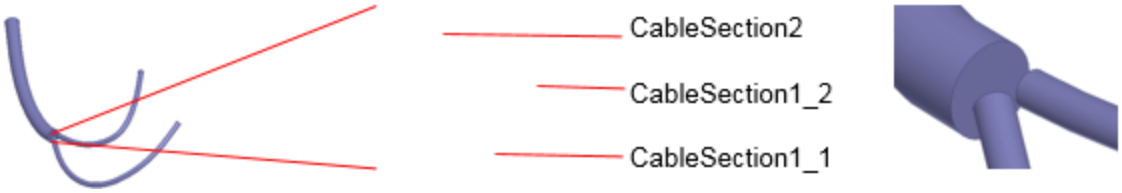
The analysis will utilize HFSS 3D Layout's **SIwave Near Field** solver to complete the simulation.



Cable Network

A Cable Network is linked field excitation used as part an overall cable solution implemented as dynamic data links between HFSS, 2D Extractor, and Circuit.

- Each cable harness in HFSS is modeled as a single external field source based on quasi-static simulation of each cable cross-section in 2D Extractor and an analysis of the cable network in Circuit. The magnitude and distribution of the fields along each cable section is determined by the voltages and currents at the ends of each section, and then transmission line model is applied to propagate these fields along the cable length.

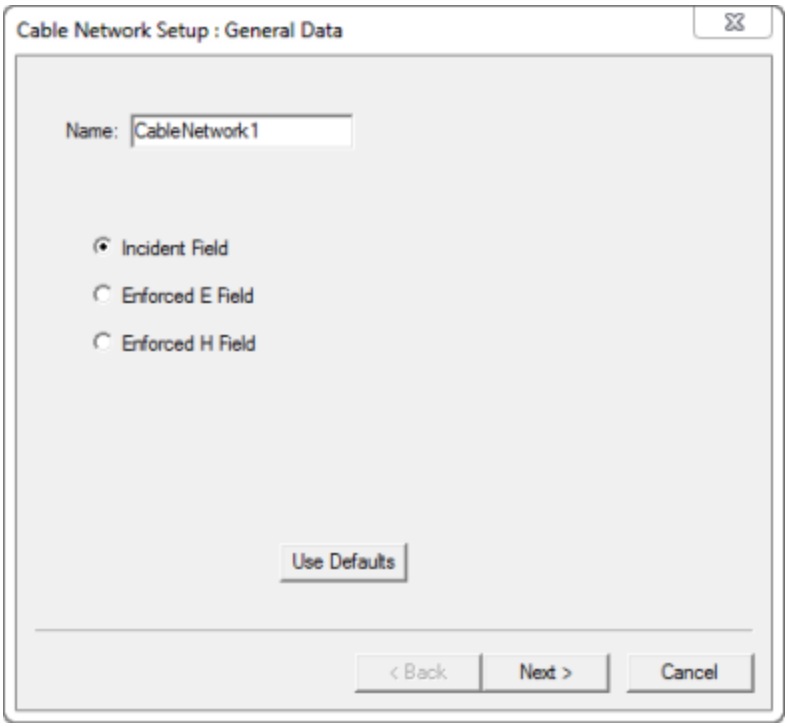


This HFSS design is a target of a new “Cable Field” coupled field datalink. The source of the datalink is the Circuit design that models the cable network. Each Cable Network will

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

be treated as a single source and can be scaled at Edit Sources.

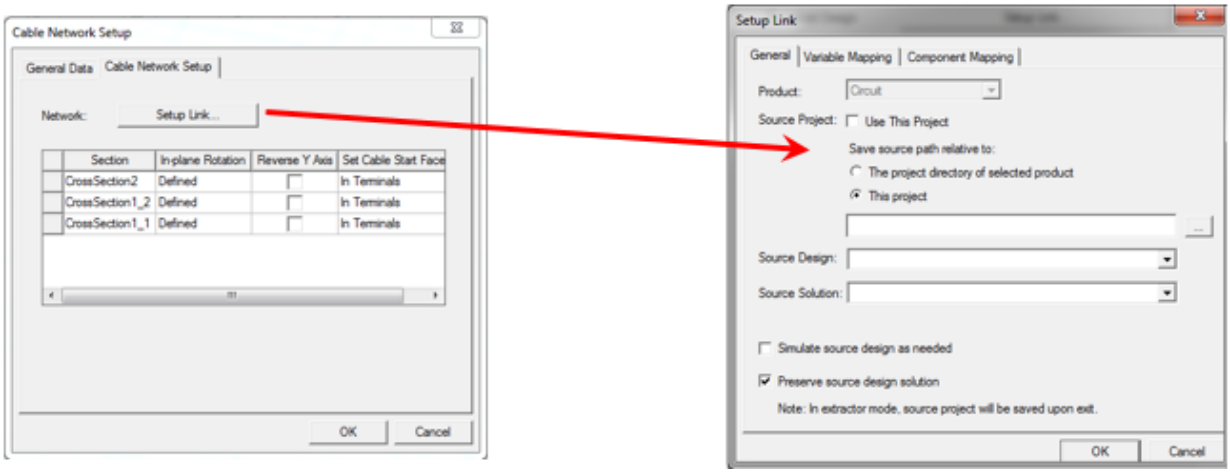
To setup a Cable Network datalink, in the HFSS Modeler window, you select all 3D cable cables that are part of a network. Then click **HFSS>Excitations>Assign>Linked Field>Cable Network...** or right-click **Excitations>Assign>Linked Field>Cable Network...** This opens the **Cable Network Setup** dialog:



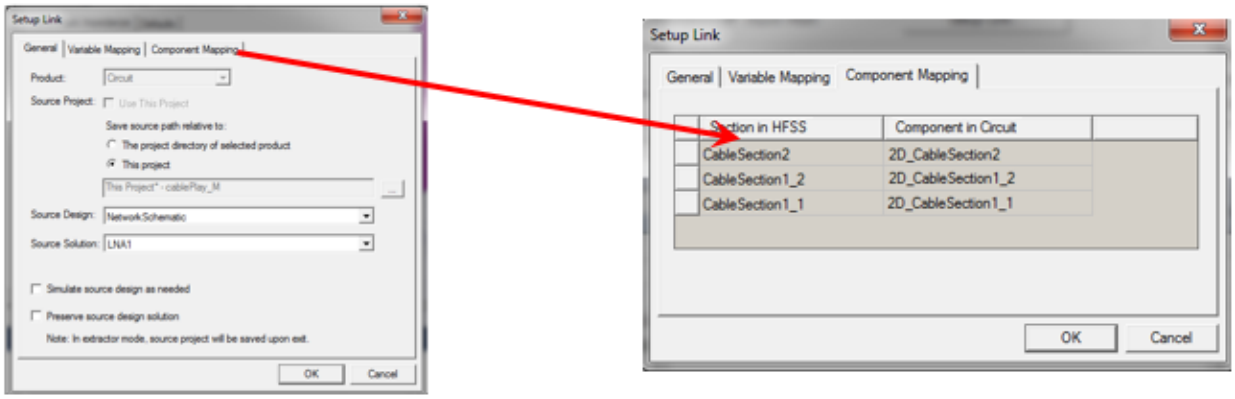
You have a field for the name, and selections for Incident Field, Enforced E Field, and Enforced H Field.

- To specify the Circuit design, user can go to the second page and click the button “Setup Link…”. This displays the **Setup Link** dialog.

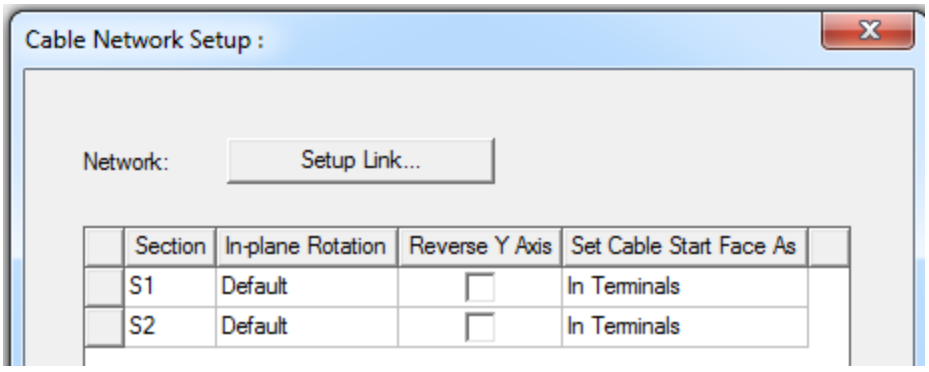
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- After specifying the source design, you switch to the **Component Mapping** tab to set up the mapping between each cable section in HFSS and a circuit component.



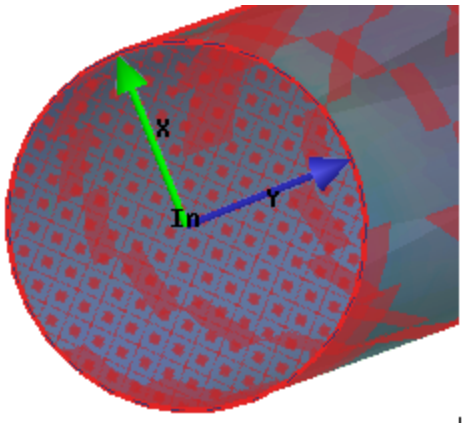
Click OK to exit the **Setup Link** dialog and complete the rest of the link setup in the **Cable Network Source** dialog.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

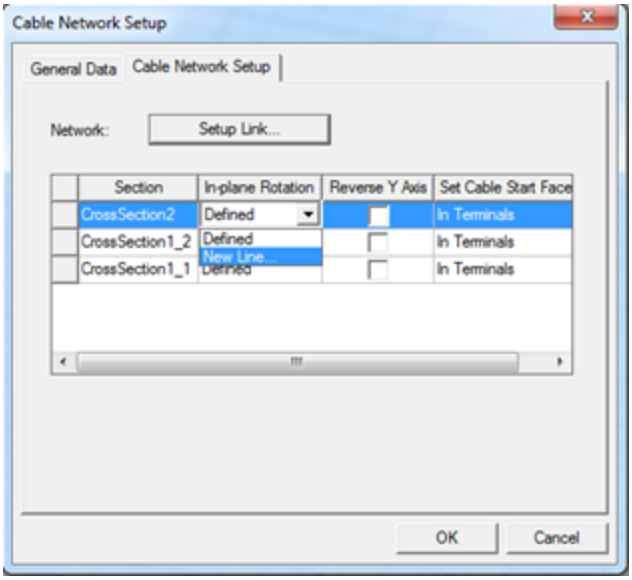
For each cable section, you need to specify:

1. The In-plane rotation of the HFSS 3D section to the 2D Extractor cross-section by a direction line (which will be the local X axis of the cable cross-section face). The default is zero in-plane rotation, which is represented by a direction line originated at the starting point of the polyline and ended at the seam of the cable.



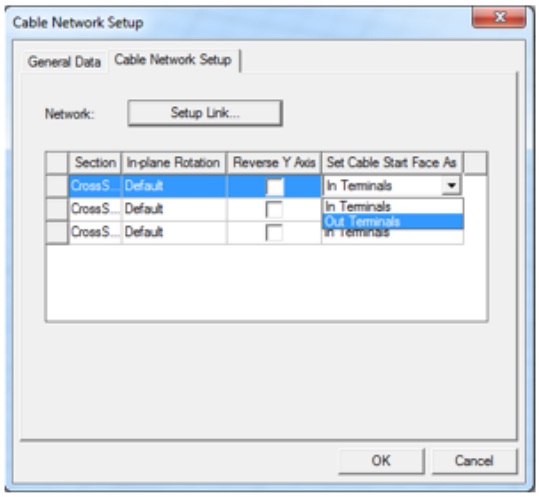
   You can create new line or reverse the Y axis of the start cable cross-section face.

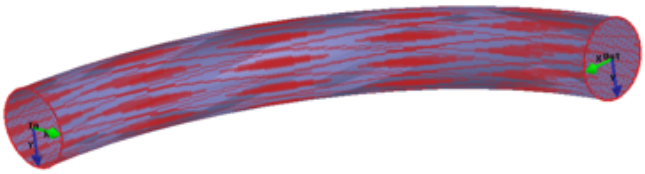
2. The X and Y axis on both start and end faces are shown, which is useful when you want to align multiple cable orientations.



3. You map pins in Circuit to wires at ends of the cable in HFSS by specifying the starting end of the cable as “In Terminals” or “Out Terminals”. By default, the starting end of the cable is selected as the “In Terminals”. The “In” and “Out” faces are highlighted in the View window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





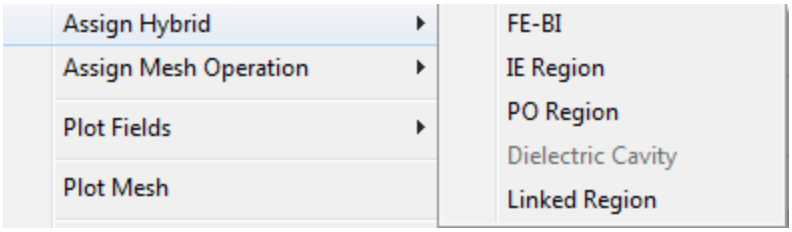
You can override the default selection by toggling the menu at the “Set Cable Start Face As” column.

Floating Cable/Linked Region

A Floating cable is defined as a cable object floating outside of FEM solution region. In order to define a floating cable, you need to define a linked region. A Linked region is used only for one-way link. We assume the source is inside the linked region, and the linked region only radiates to the field but does not have field scattered back to the region.

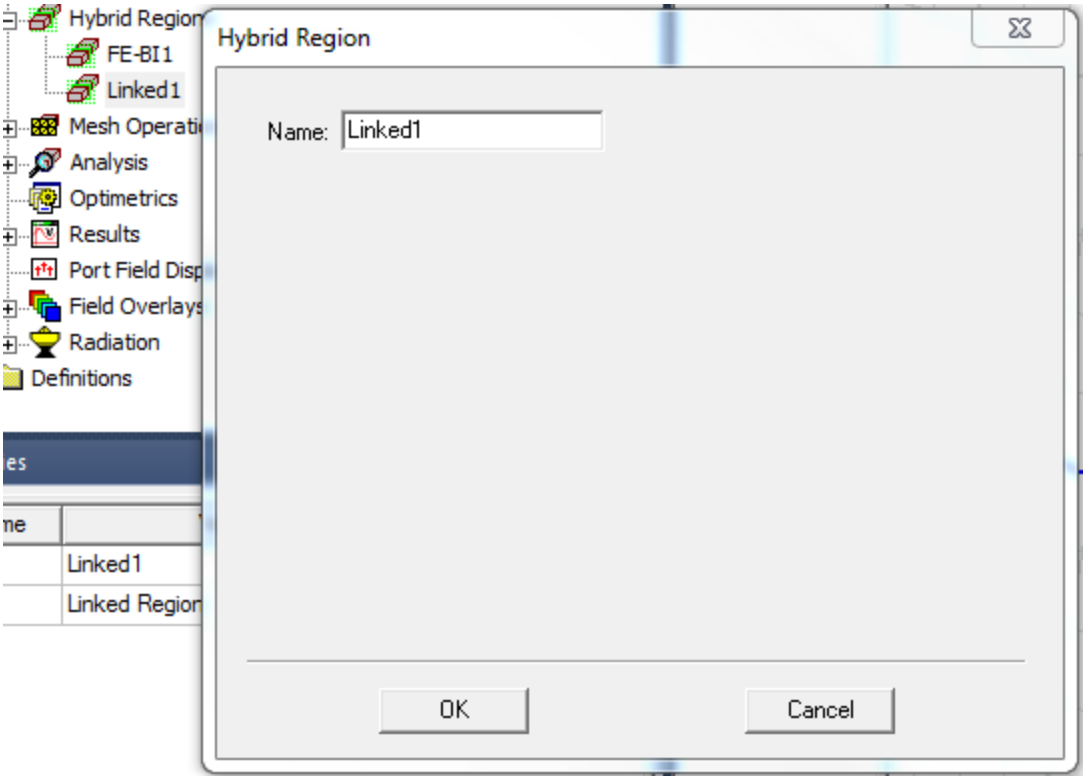
You can assign near/far field as well as a cable network to a linked region.

The Assign Linked Region menu item is in Assign Hybrid.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Linked region appears under “Hybrid Region” folder in project tree.



Cable Network Link Validation

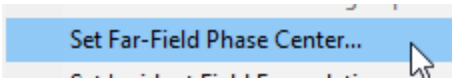
1. The number of 2D Extractor components in circuit should be equal to number of HFSS cable object
2. For each cable object, the 2D cross-section center should match the center of the start face in local coordinate system.
3. For each cable object, the 2D cross-section perimeter should match the perimeter of the start face.
4. The cross-section should be perpendicular to the cable path in HFSS.

Link from Driven Modal or Terminal Antenna to SBR+ Design

HFSS lets you create a one-way link from a Driven Modal design or a Driven Terminal design with a single terminal to an SBR+ design. The source can be a finite array design, including designs with blockages. For finite array design, the phase center is configured using the physical cells, and automatically transformed to the virtual cells. The source design **Excitations** menu includes a **Set Far Field Phase Center** command that lets you specify the phase center for

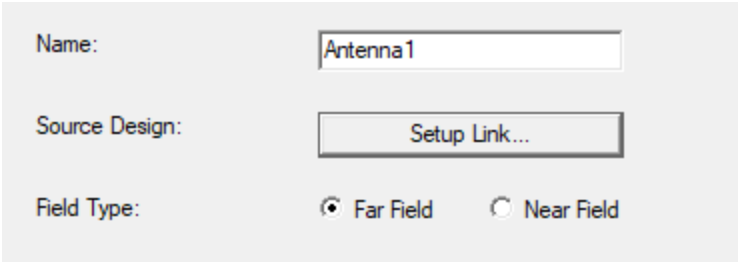
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

each lumped port and wave port it contains. You assign the phase center to a coordinate system. The far-field phase center determines the point from which SBR+ rays are launched when the linked source is configured as far-field. It has no effect for linked sources configured as near-field. It can affect the Export Element Pattern from Antenna Parameters dialog.



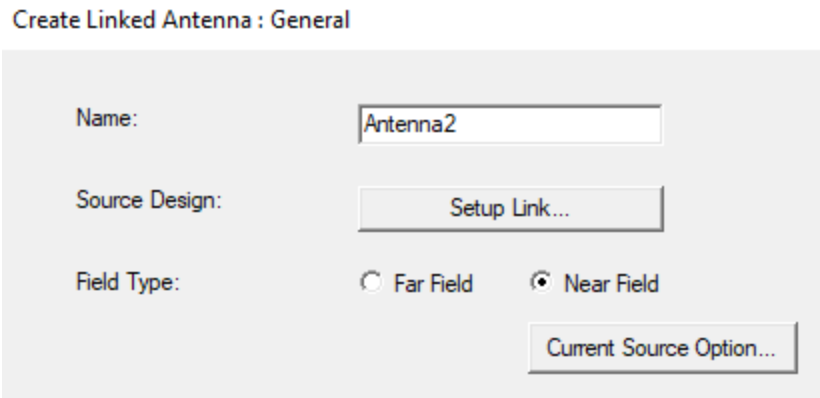
The target SBR+ design includes the associated command **3D Component>Create Antenna>Link to Source Design**.

From the target design, you can select either Far-Field or Near-Field as the linked field style.

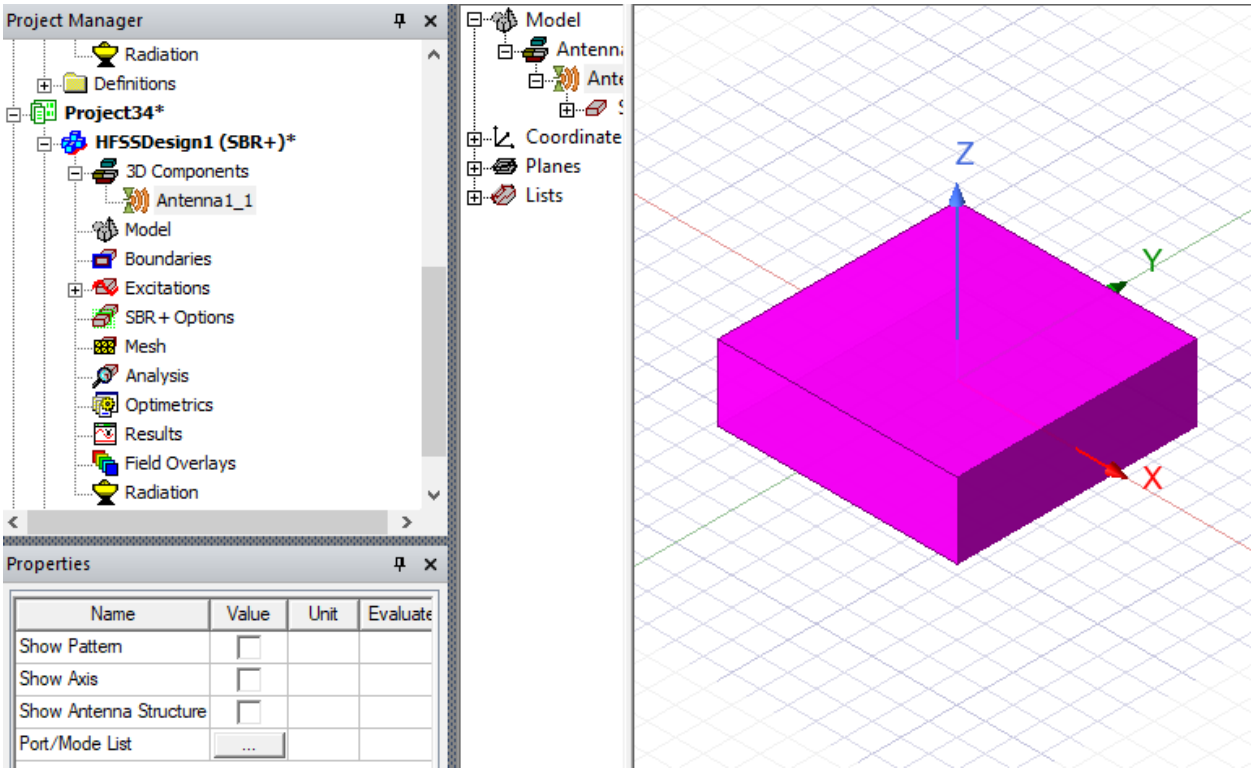


If you select Near Field, the **Create Linked Antenna** dialog includes a button for **Current Source Option...** allowing you to specify a customized power fraction. Use Global is the default. Enabling Thin Sources lets you specify a Power fraction should be greater than 0.1 and less than 1. Solutions are invalidated if you change current source conformance, use Thin Sources, Power Fraction, or change the Global option.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



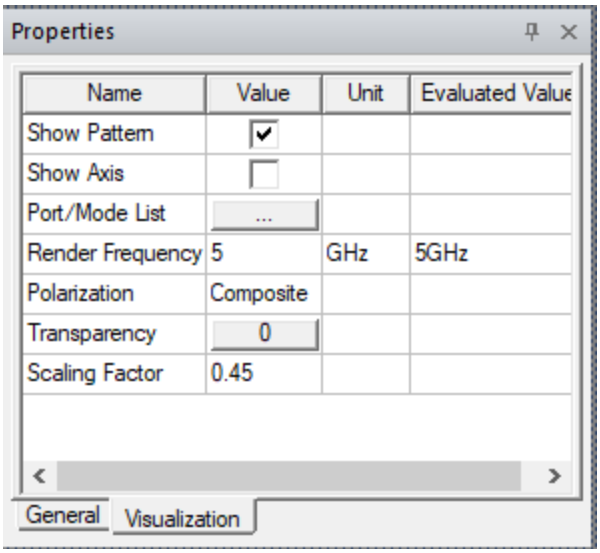
In the target design, the antenna link brings over the source design model and includes visualization options. The selected geometry is shown in its shaded view in the SBR+ target design with its designated color and transparency..



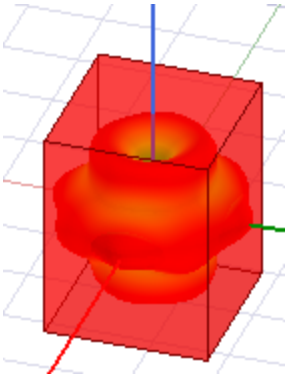
The linked source design appears in the target design under 3D Components. The source design ports appear in the target design under the excitation folder. You have the option when linking to a source project to Link as Composite Port, to simplify and speed up source projects with many excitations. However, you cannot edit those imported ports because they are part of the linked antenna component. You can turn on Field visualization of the source design by

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

checking **Show Pattern** on the **Visualization** tab of the 3D component property in the target design.



With **Show Pattern** checked, the source pattern is visible from the linked 3D Component bounding box in the target SBR+ design.



Prerequisites

- Set the phase center in the source design. The source design requires the presence of at least one wave port or lumped port to activate the **Set Far-Field Phase Center...** command.
- The source design must be solved before you can **Show Pattern** for the 3D Component Instance in the target SBR+ design.
- For simulation, the target design SBR+ solution frequency range must be in the range of source design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To support directivity and gain in the SBR+ design with the link, you must use an interpolating sweep with the 3D Fields Save Option selected in the linked design, either **Save Fields (At Basis Freqs)** or **Save radiated fields only**. This provides interpolatable S-parameters from the source design for the SBR+ target design.
- To model EM scattering effects of antennas in SBR+ simulations, you can select geometries from the source design to represent blockages. These can be model, or non-model objects. For this feature the SBR+ solver is passed a CAD representation of the antenna and it ignores this CAD representation when launching the rays from the antenna source but includes it when tracing the reflected rays from the platform.

Limitations

When the source design is in Composite-Driven Modal solution type or Composite-Driver Terminal with a single terminal:

- You can only choose one single phase center as the global phase center for all ports.
- Composite excitation becomes an incident wave in the target design. Therefore, we naturally should not have the port/mode option anymore.
- For Terminal designs, a single terminal is permitted.

When you select **Show Pattern**, the pattern data is cached for future use. The cached data is available until the design is closed. If you update the source design, **Clear Linked Data** is available for you to clear the cached data. If you clear the Linked data, field pattern is re-computed when you select **Show Pattern** again.

To model blockage structures, the source design can be Driven Modal or Driven terminal or a finite array. If you use a finite array, the **Model Blockage** tab on the **Setup Link** dialog shows up if the array has model / non-model blockage objects. The GUI operations for array blockages are same as non-array projects. See steps 4 to 7 below.

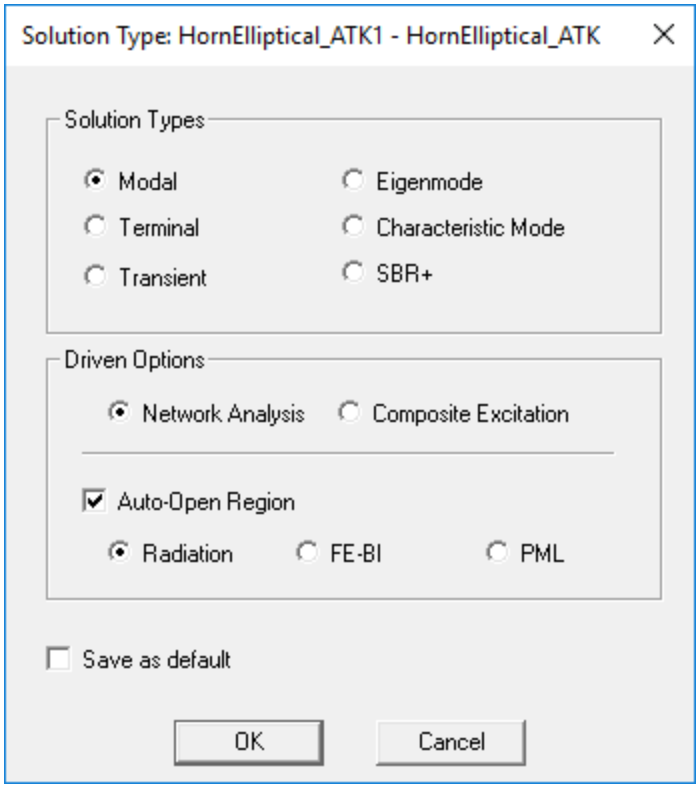
Creating a Link from Driven Modal or Driven Terminal Source to SBR+ Target

To create a link from a Driven Model Source or Driven Terminal Source to an SBR+ target:

1. Set up a driven Modal solution or Driven Terminal as the source.

   You can choose any **Modal** options, although if you choose **Composite Excitation**, you must choose a single phase center for all ports. You can use **Auto-Assign Region** options if convenient.

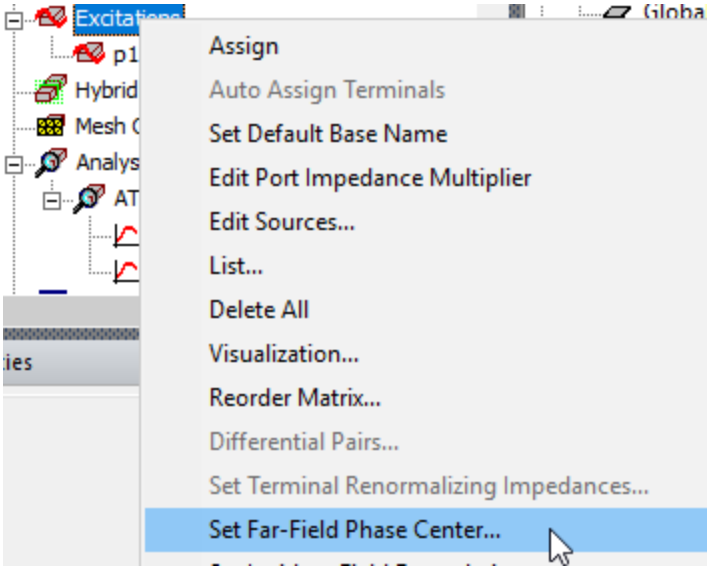
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



All ports in the source design must be either wave ports or lump ports.

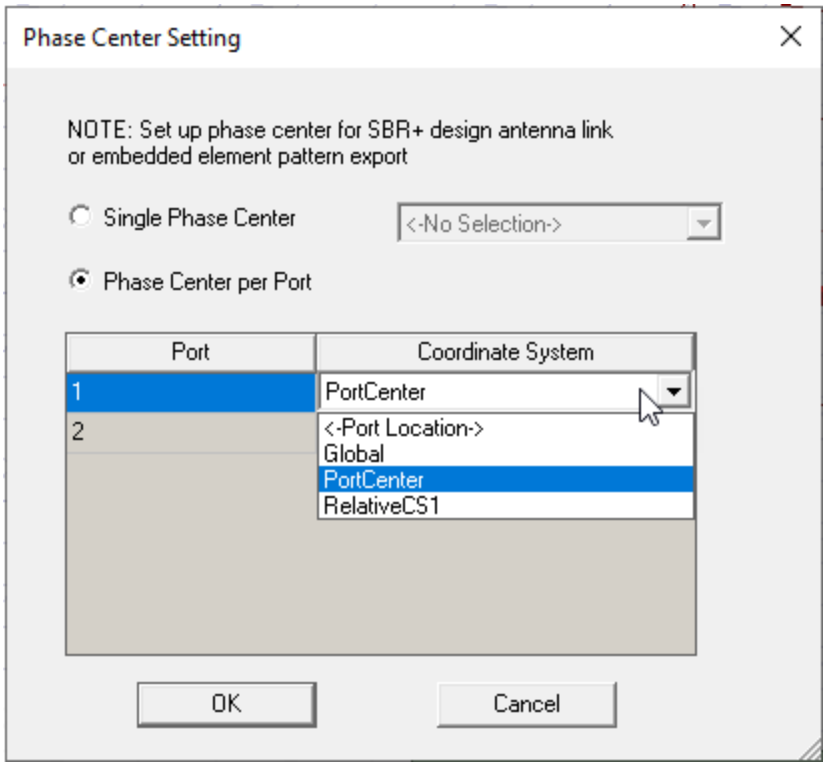
2. Set the Phase Center in the Source design. The phase center(s) determine the point(s) from which SBR+ rays are launched when the linked source is configured as far-field. It is important for accurate results to position the phase center as near as possible to the physical phase center of the antenna for the selected port, especially if the source is placed nearby to SBR+ scattering structures. Otherwise, the accuracy of ray fields illuminating nearby structures will be compromised, leading to inaccurate scattered fields and final results. Either right-click **Excitations** in the Project tree and select **Set Far Field Phase Center...** or click **HFSS>Excitations>Set Far Field Phase Center...**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



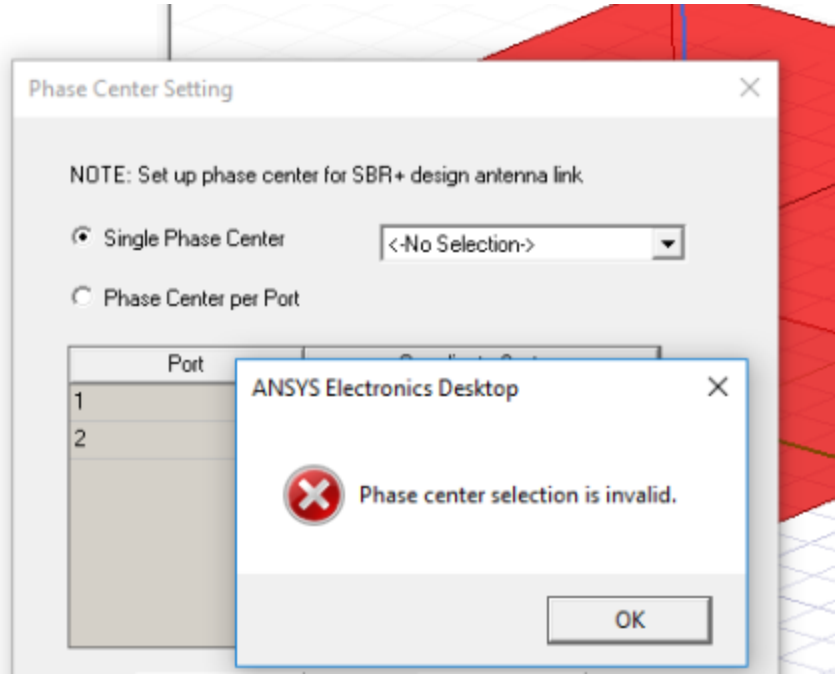
The **Phase Center Setting** dialog appears, with **Phase Center per Port** selected as default with the mode <- Port Location->. By using **Phase center per Port**, you have more fine-grained control over each phase center. This feature provides a convenient option to assign phase center at port location (port geometrical center), which is a common selection for Phase Center per Port mode. If you have created additional Coordinate Systems, these appear in the drop-down menu as options for the phase center.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



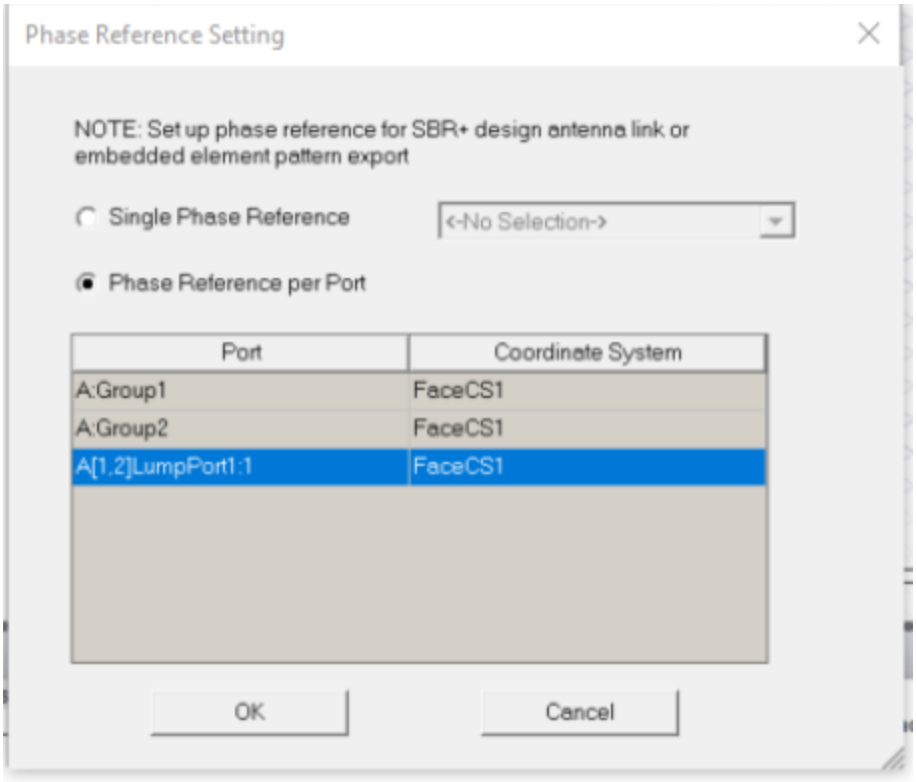
Single Phase Center is to assign the global option to all the ports (one port or multiple ports with the same phase center option). You also have the option to select Link as Composite Port when you setup up the link from HFSS to SBR+ design. Phase Center Per Port lets you specify different phase centers for different ports. Validation checks to ensure that each port is assigned a valid phase center. You must not leave any ports with <No Selection>.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

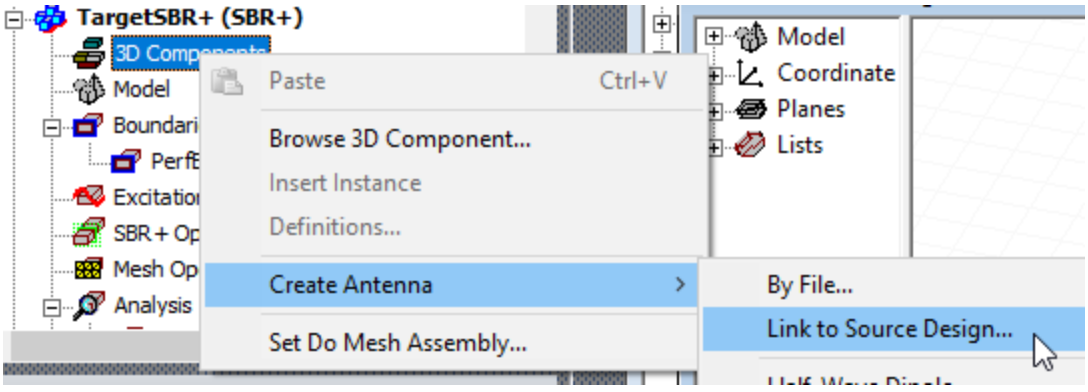


In the case of a design using the Beta feature for subarrays, the **Phase Reference Setting** dialog shows composite groups if you want to specify unique phase reference for each group. In this case, the “Port Location” selection is not available, and you must select an actual reference coordinate system.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



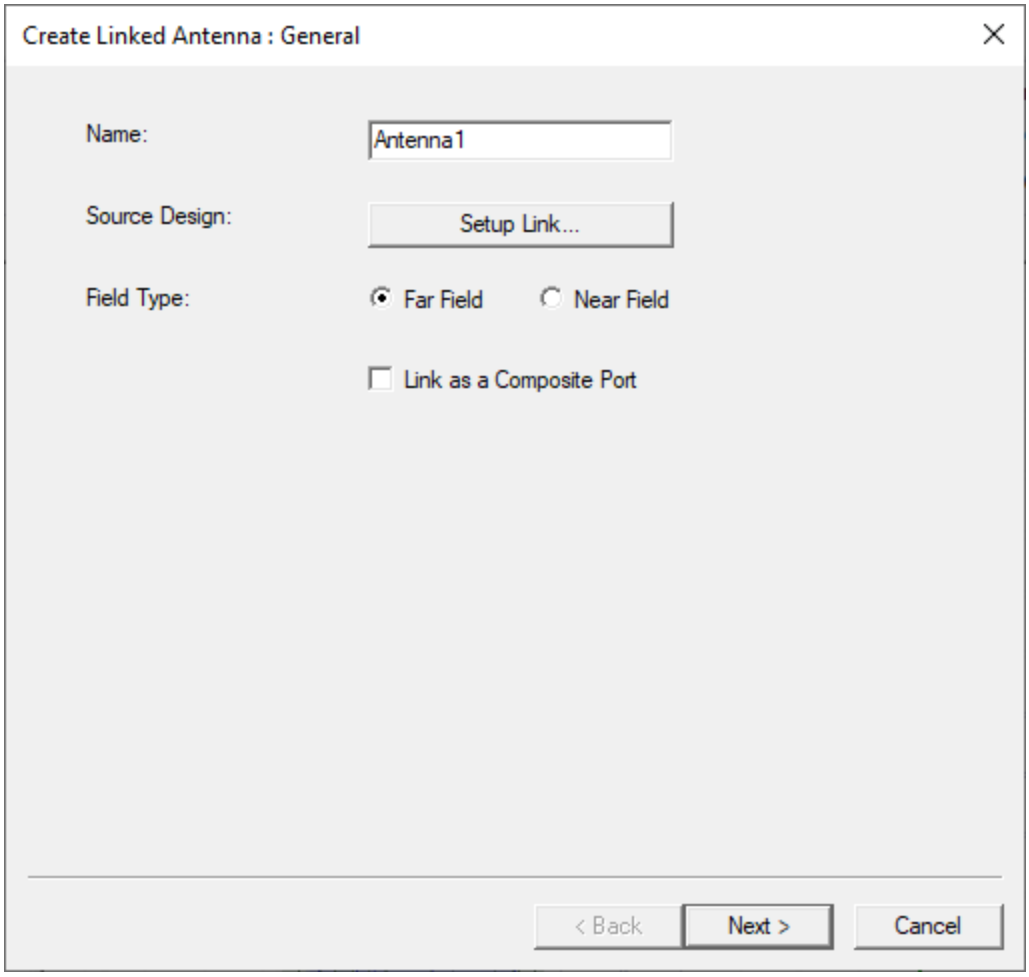
3. When you have completed the source design setting (phase centers, solve setups, etc) and solved it, create an SBR+ design as the target design.
4. Right-click **3D Components** in the Project tree and select **Create Antenna>Link to Source Design...**



You can also right-click **Excitations** in the Project tree to select **Create Antenna>Link to Source Design...**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Create Linked Antenna** dialog appears.



Select **Far Field** or **Near Field** as appropriate.

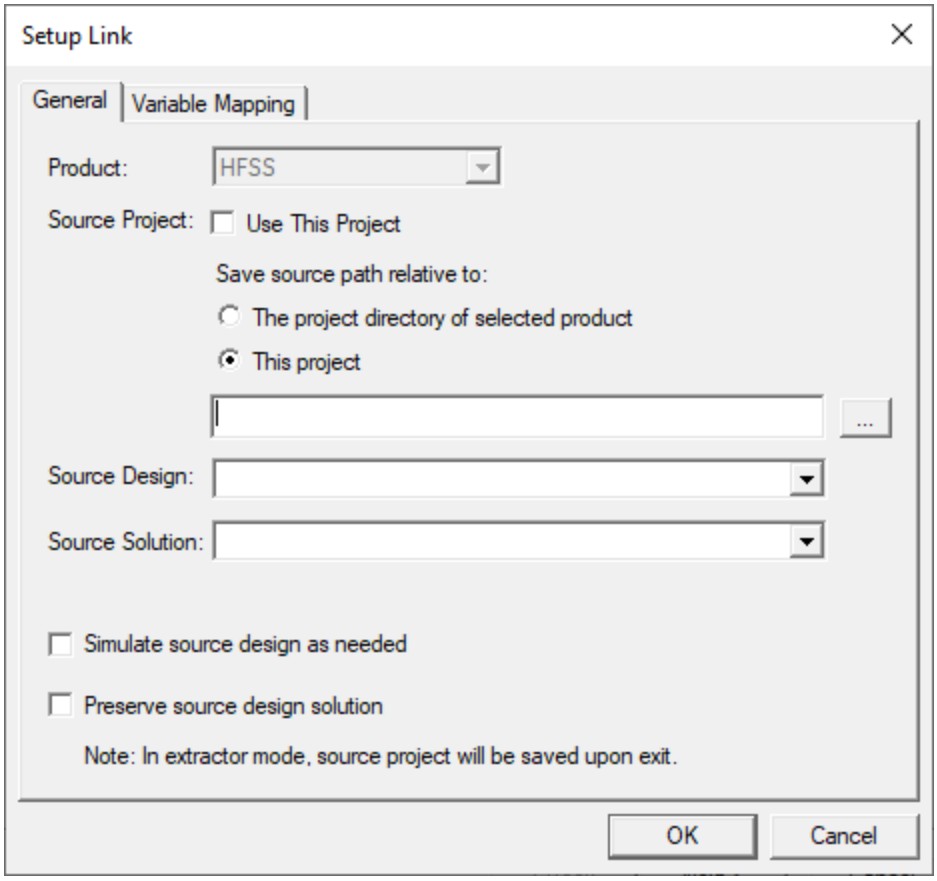
The **Link as Composite Port** option addresses the situation where an imported design contains many ports. By default, the model window displays the same number of virtual ports in the SBR+ target design. However, if the source design contains many virtual ports, they will look cluttered and require significant computational resources to solve. You create a single composite port in the SBR+ target design to represent all the ports in the HFSS source design. During computation, the SBR+ target design sends the information about a single

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

composite port to the solver, rather than information about all the ports. This saves a significant amount of resources. For further discussion, see Link as Composite Port.

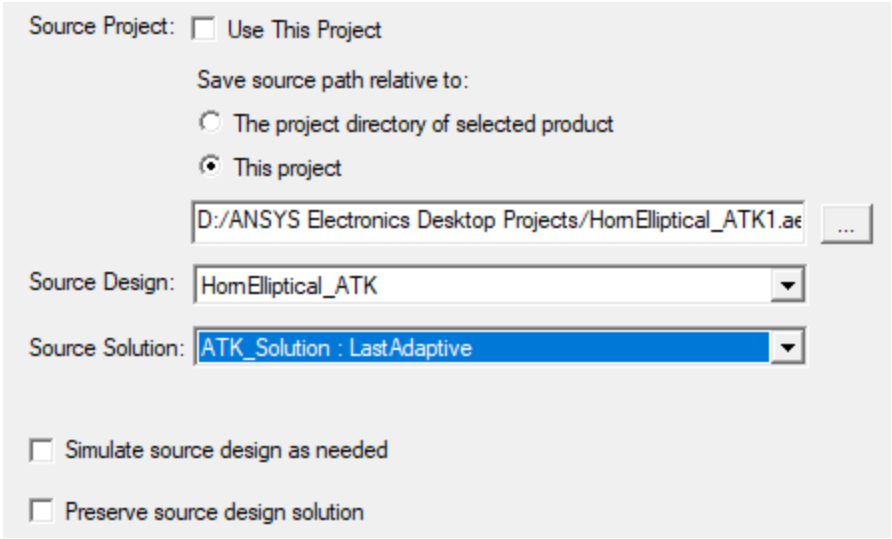
5. Click **Setup Link...**

The **Setup Link** dialog appears.



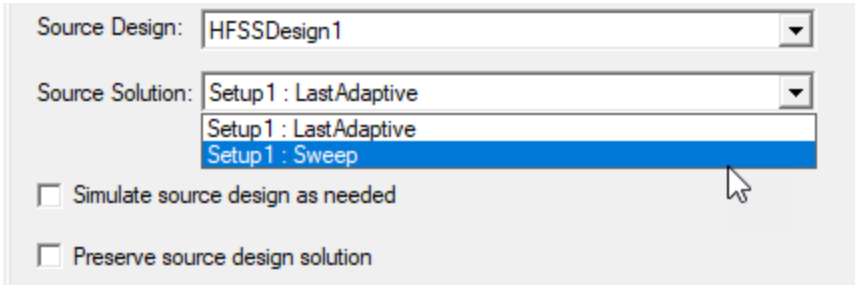
If the source design is in a different project, you can click the ellipsis button **[...]** to open a file browser to select the project of interest. When you select the project and OK the browser, the fields are filled in.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the source design is in the same project, you can check **Use this Project** and select from available Source Designs.

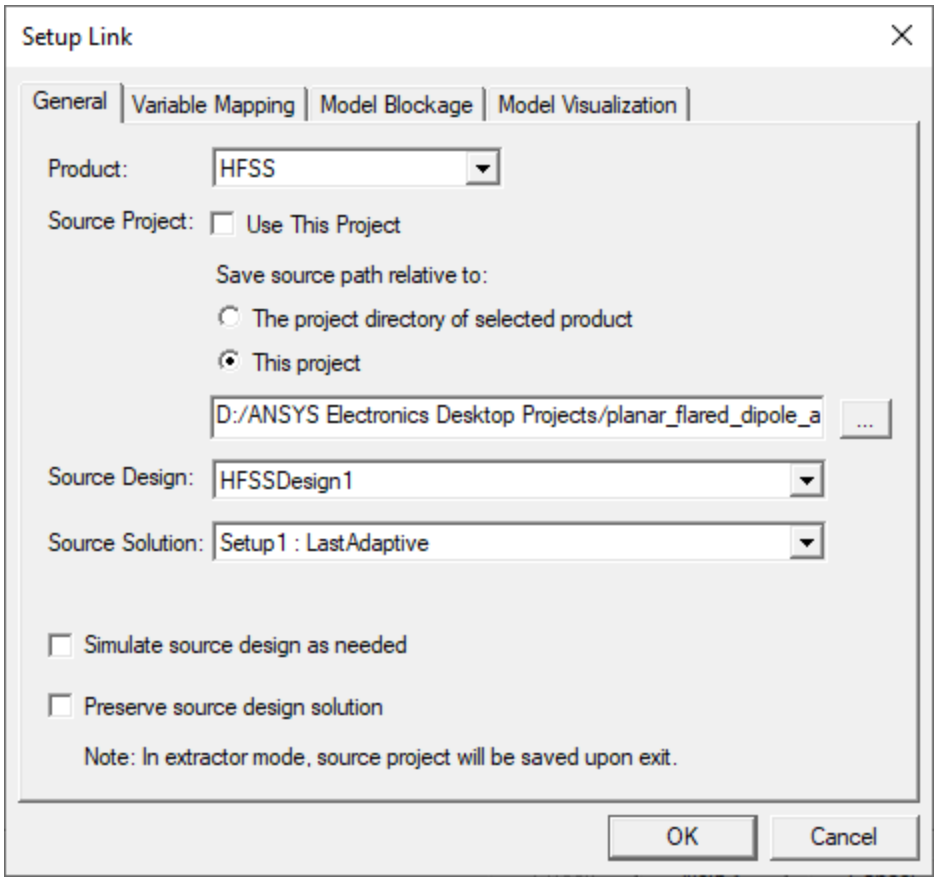
If you have defined an interpolating frequency sweep in the source design, you can select it on the drop-down for the source solution. (This feature supports directivity and gain in the SBR+ design with the link. (See "Requirements for Creating a Gain Report for SBR+ Design with Driven Modal or Terminal Source" on page 19-190). It provides the S-Parametersfrom the source to the target. In the source sweep setup, you must have selected a 3D Fields Save Option.)



For this release Model blockages can handled as a separate step outside creation of the linked design, by selecting geometry from the SBR+ region to be designated as “Model Blockage” associated with a particular source antenna, as described in . Note there is no longer any restriction on what objects and/or materials can be designated as “model blockage” geometry. For either solution type, SBR+ or hybrid HFSS-SBR+, this selection is from the available SBR+ region geometry objects. Once you specify antenna blockage geometries for each source antenna, we can then solve the combined target SBR+ design. For details see Design Flow for SBR+ Solution Type.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

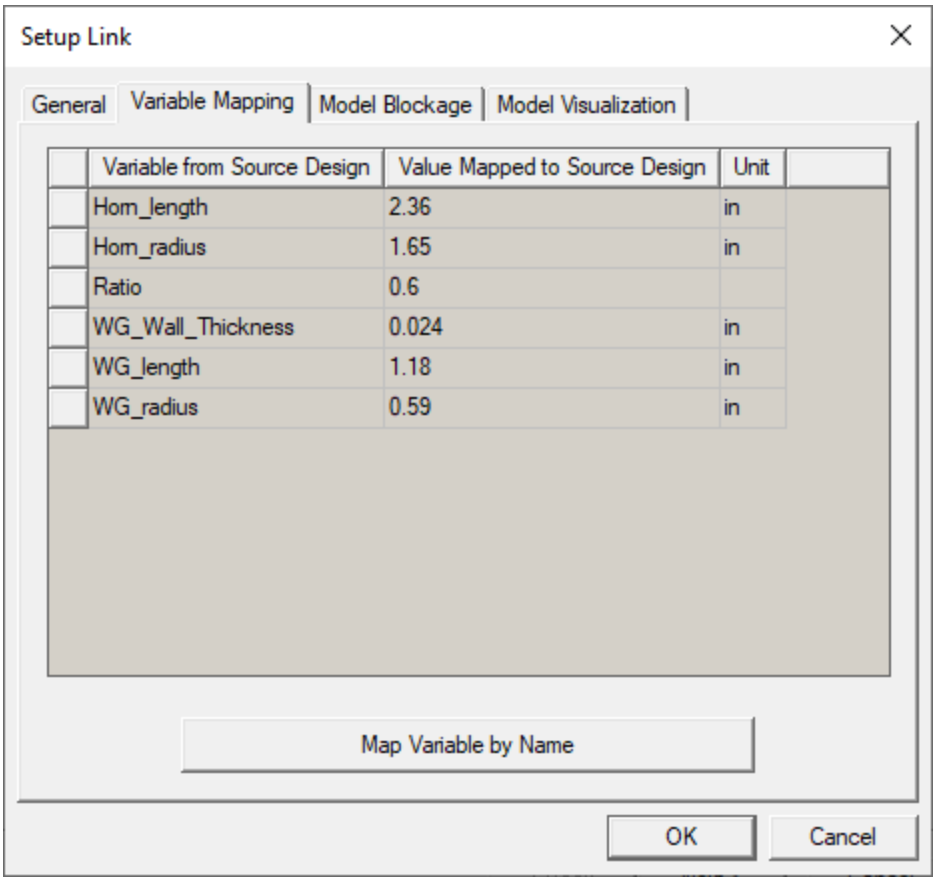
If you use the older method for designating blockages, once you select the source design, additional tabs for **Model Blockage** (if model or non-model blockages exist in the source) and **Model Visualization** appear.



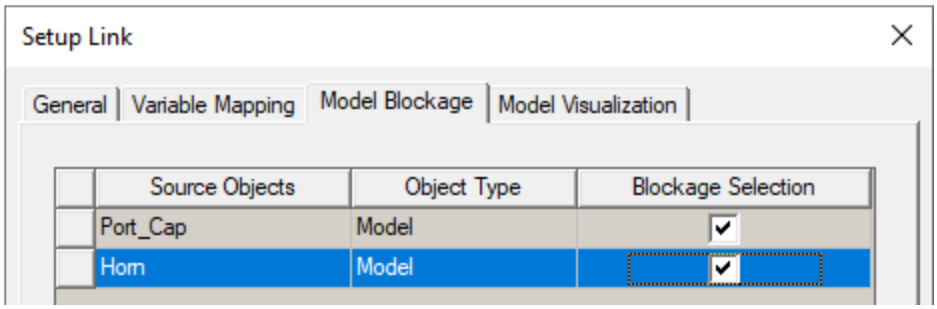
The **Visualization** tab lets you specify how the linked model is displayed in the SBR+ target design.

6. The **Variable Mapping** tab lists variables and lets you define mappings between the driven modal source design and the SBR+ target design, if necessary.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

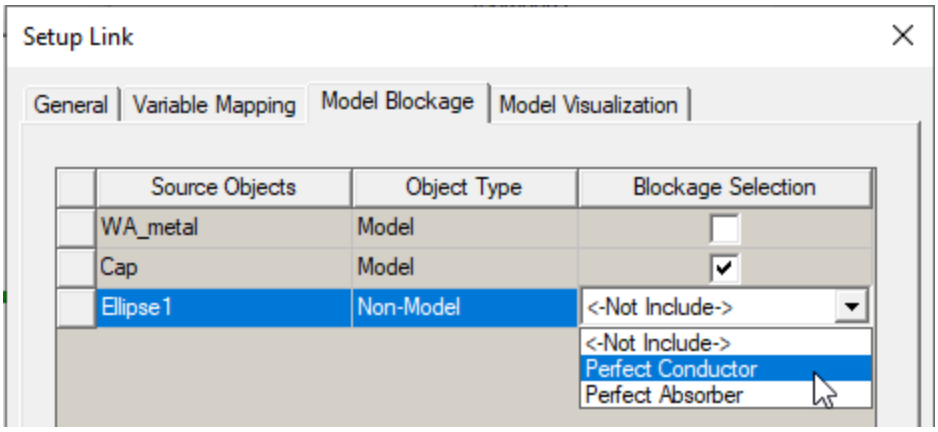


7. The **Model Blockage** tab lets you select geometries from source design to represent the blockage structure. Non-solve inside, and non-model objects are available for selection.



For Non-model objects, you can select <Not Include> Perfect Conductor, or Perfect Absorber.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

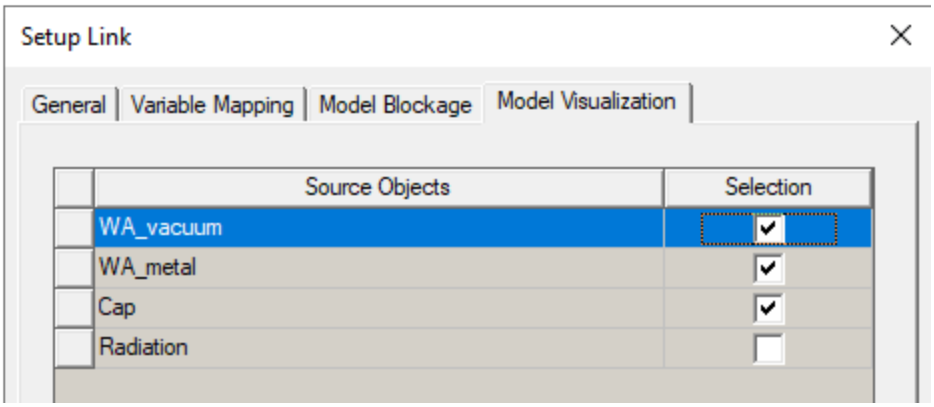


In many cases, it is sufficient to represent SBR+ sources entirely in terms of their far-field or near-field radiation characteristics, and there is no need to configure Model Blockage. However, in some cases, the antenna structure can generate significant secondary scattering after rays have first reflected off the main scattering geometry that the source illuminates. An example would be a feed antenna blocking part of the main beam of a reflector antenna, where the reflector surface is simulated using SBR+. Model and non-model structures selected under Blockage Selection will have special behavior relative to SBR+ ray-tracing. These surfaces will be invisible to SBR+ rays when they are first launched from the source. However, after the first bounce, they will interact with SBR+ rays just like any other surfaces in the scattering geometry (with some further qualification, described shortly). For example, to capture reduced reflector antenna gain due to blockage by the feed horn, you may select the elements representing the body of the horn under Blockage Selection. In some cases, the antenna structure may be too detailed to effectively interact with SBR+ rays, recognizing that ray-tracing is a high-frequency (short wavelength) approximation. In those situations, one may instead create a simplified blocking structure, such as a rectangular or circular plate, to stand in for the physical antenna structure. To do this, create the simplified blocking structure as one or more non-model objects in the source design and then select them under Blockage Selection, configuring them as either Perfect Conductor or Perfect Absorber.

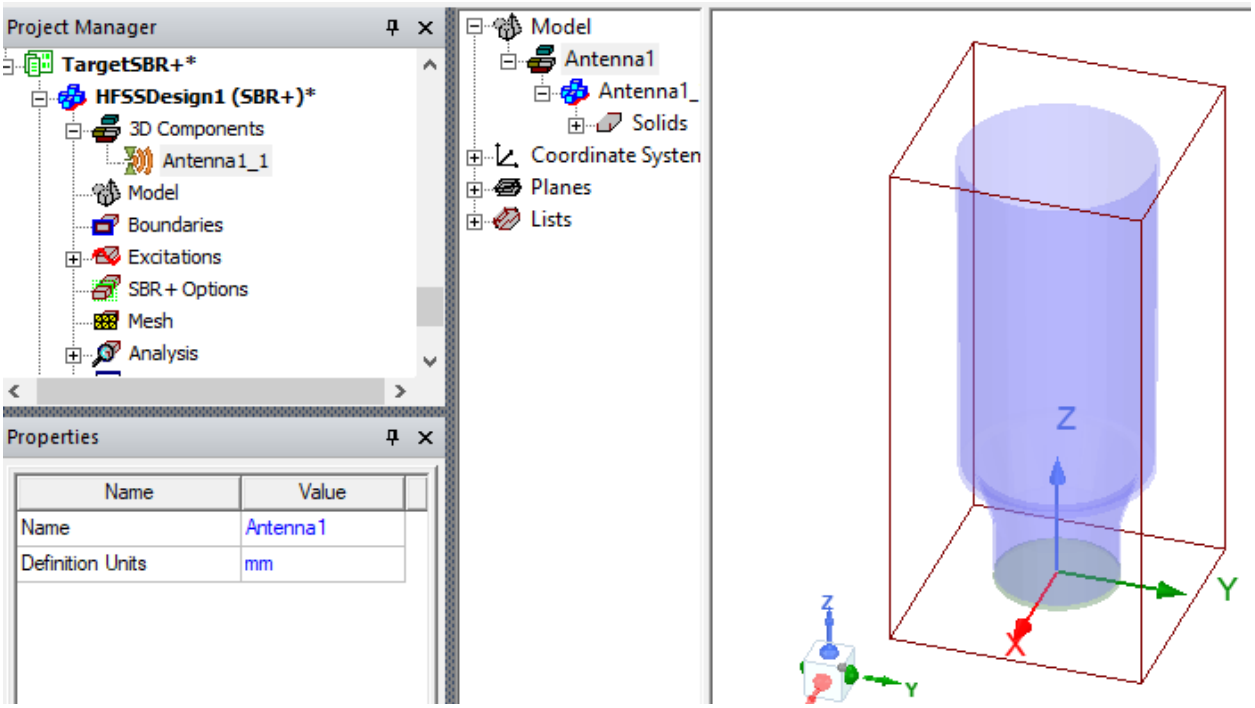
The behavior of model and non-model objects selected under Blockage Selection have some further qualifications in the context of SBR+ with multiple source antennas. Suppose there are two antennas A and B, and both have Model Blockage objects selected, designated as S_A and S_B, respectively. S_A will be invisible to first-bounce SBR+ rays launched from antenna A, and then become visible on subsequent bounces. However, S_B will always be visible to SBR+ rays launched from antenna A, even on the first bounce. A related point is that when rays hit S_A, they will generate scattered fields to every antenna accept to antenna A. Likewise, when rays hit S_B, they generate scattered fields to every antenna accept to antenna B.

8. The **Visualization** tab lets you specify how the linked model is displayed in the SBR+ target design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

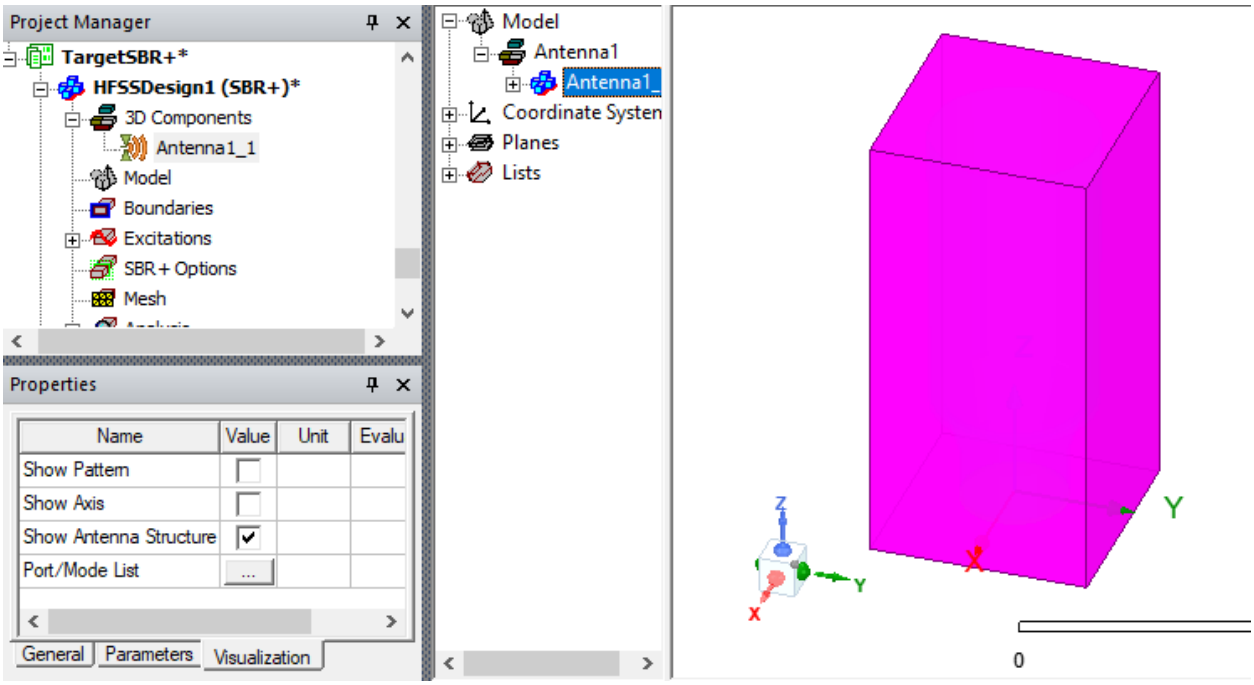


For example:

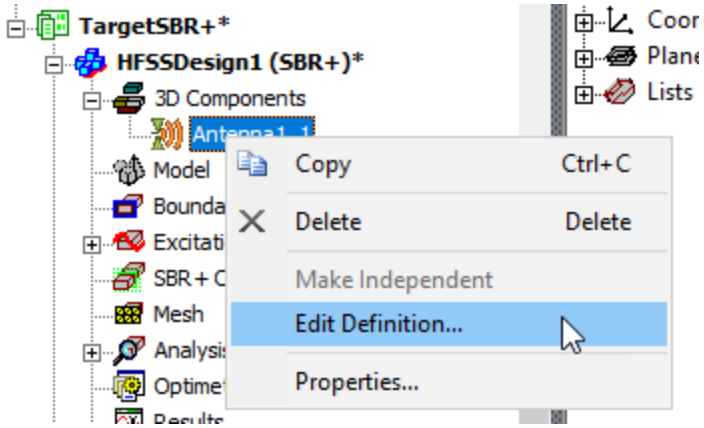


Selecting the linked 3D Component also provides a **Visualization** tab in the properties with additional control over how a linked project displays in the SBR+ target design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also gain access to the Setup Link **Visualization** tab by Editing the definition of the 3D Component.

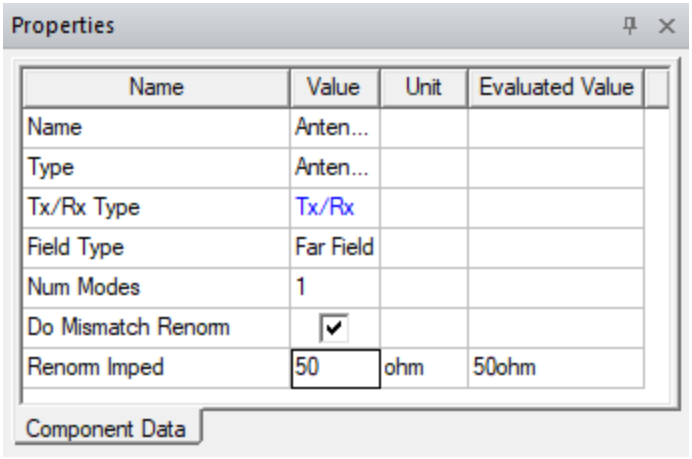


This opens the **Setup Link: General** window, where you can click the **Setup Link...** button.

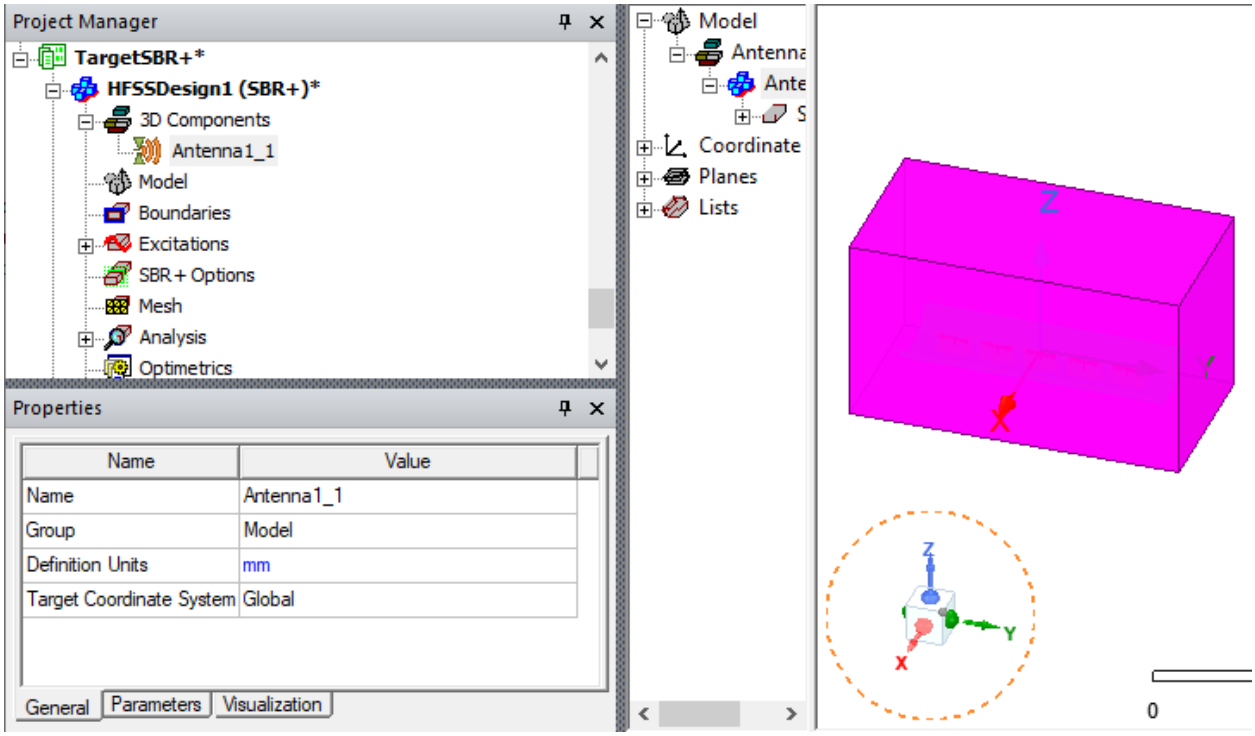
9. Click **OK** to finish importing a linked antenna to the target SBR+ Design. The target design displays the linked antenna as a 3D Model in the Project tree and in the Model window as a model based on how you specify the visualization.
10. Ports appear in the **Project** tree under **Excitations**. If you right click on the **Excitations** icon, you can view and edit Tx/Rx selection for all fields, including coupling between ports within the same antenna and self coupling. The properties for each excitation shows the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

current Tx/Rx type. If you select do **Mismatch Renorm**, an editable **Renorm Imped** property appears which lets you Renormalize linked ports.

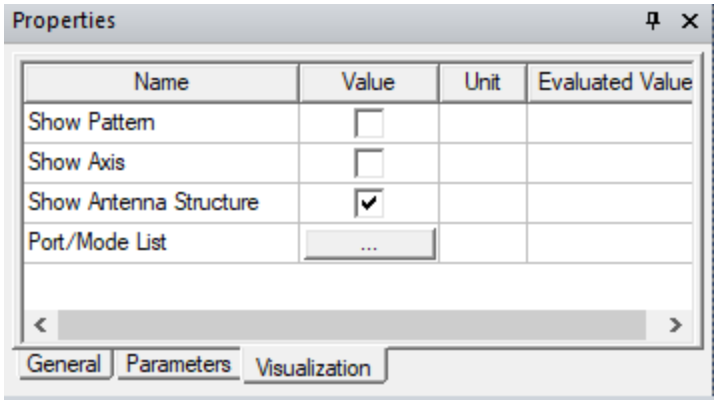


11. Select the 3D Component for the source antenna in the target **Project** tree to view the properties.

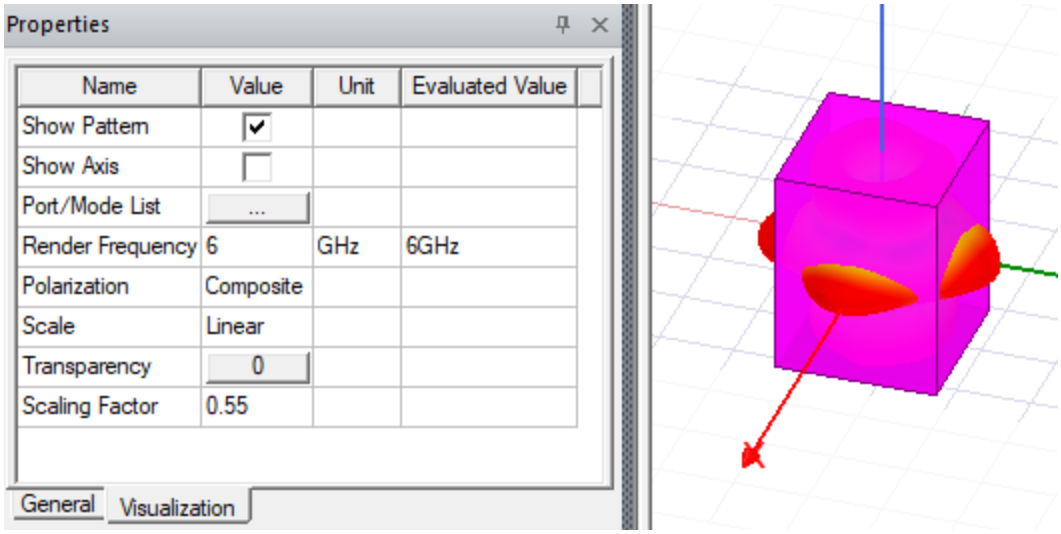


If you select the **Visualization** tab, you see the following properties.

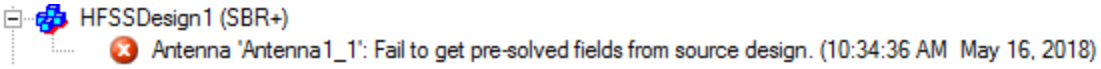
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you have previously solved the source design, and correctly assigned valid phase centers, you can check **Show Pattern**, and see additional display properties. The antenna pattern is computed and displayed in the bounding box in the Model window.



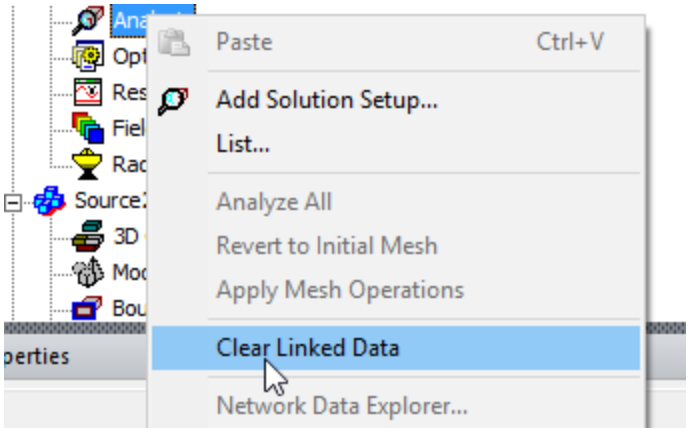
If the source design has not been solved, the **Message** window will display an error message.



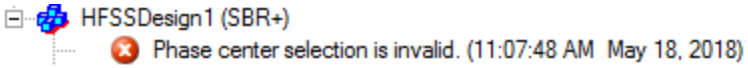
If you update the source design, the **Clear Linked Data** command becomes available for you to clear the cached data.

If you clear the linked data, the field pattern is re-computed when you check **Show Pattern** again.

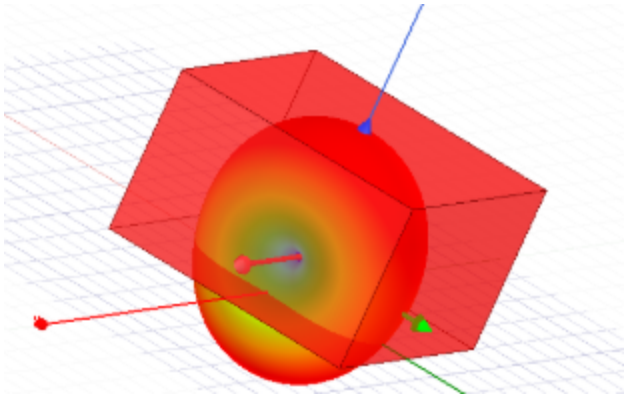
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you made changes to the source design that conflict with the link or phase center configuration, an error message is displayed.

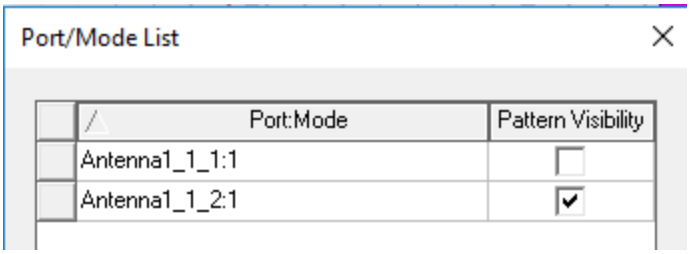


If you check the **Show Axis** property, the phase center axis is shown at the antenna pattern.

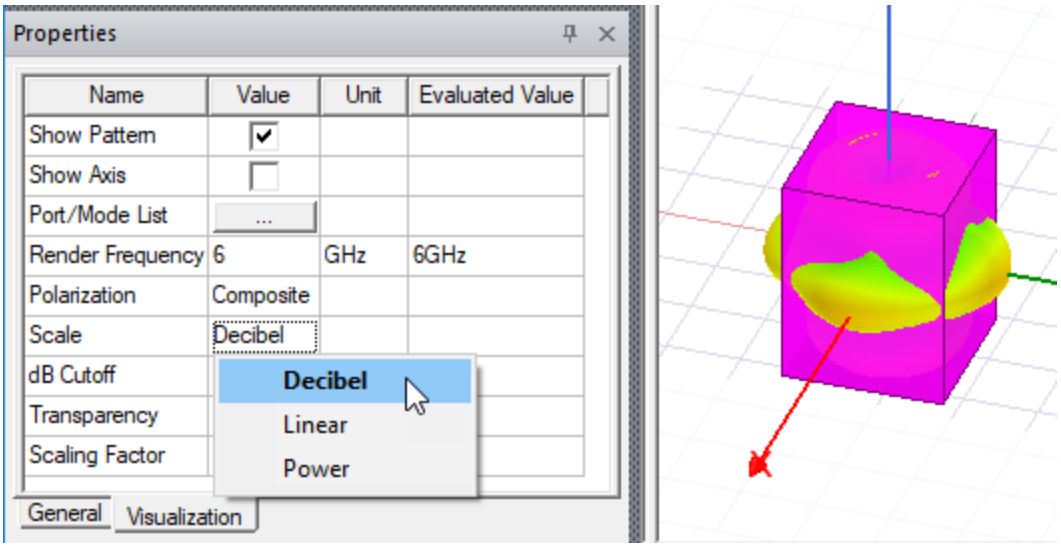


The **Port/Mode List** is available unless the source design is driven modal composite. If you click the ellipsis [...] button for **Port/Mode List** on the **Visualization** tab properties, a dialog displays that lets you select which ports to show:

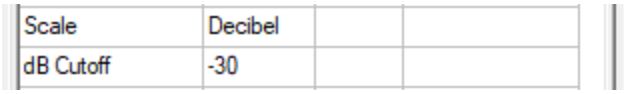
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Scale** property can be Decible, Linear, or Power.



If you select Decibel, an additional property for dB Cutoff appears.

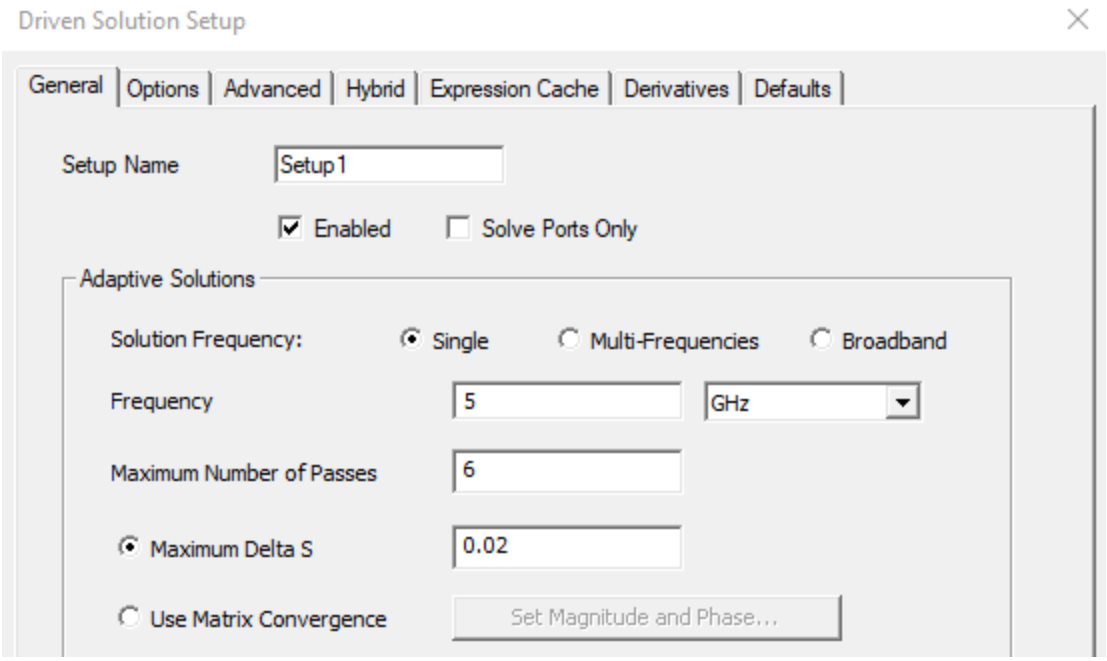


Simulation Setup for Driven Modal Antenna to SBR+ Design Link

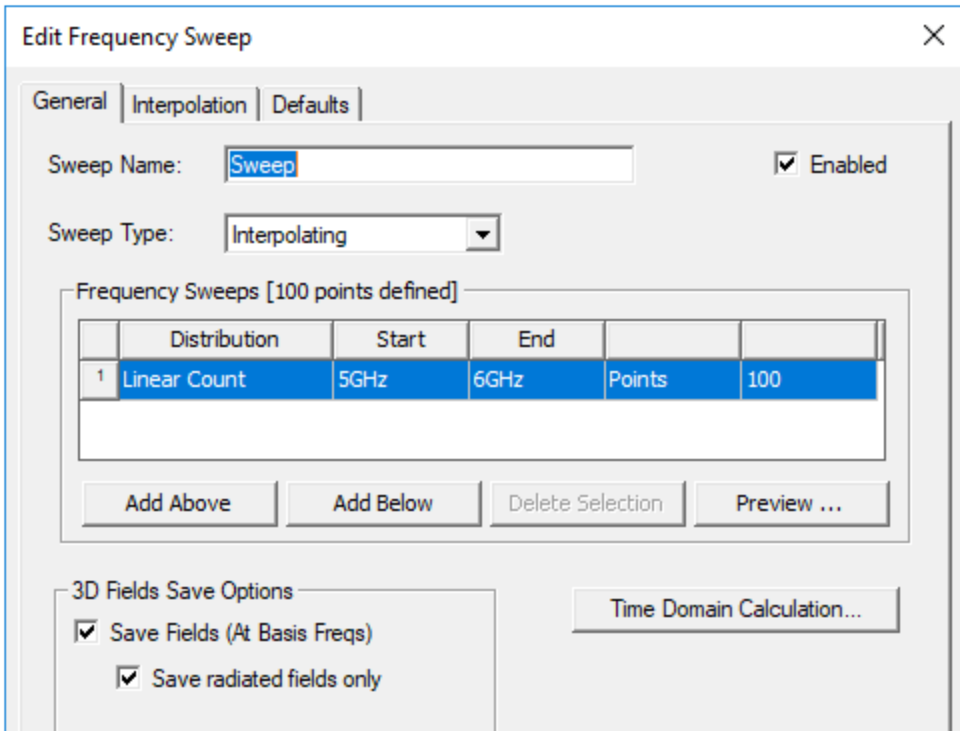
Once you have configured the SBR+ design with an Linked Antenna, you set up a model object and **Solution Setup**. The simulation frequencies in target design must fall in the range of the source design frequencies. Below is an example of source design setting and target design setting, using compatible frequencies. and Save Far fields so that Gain can be plotted.

Example Source Design Setup

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



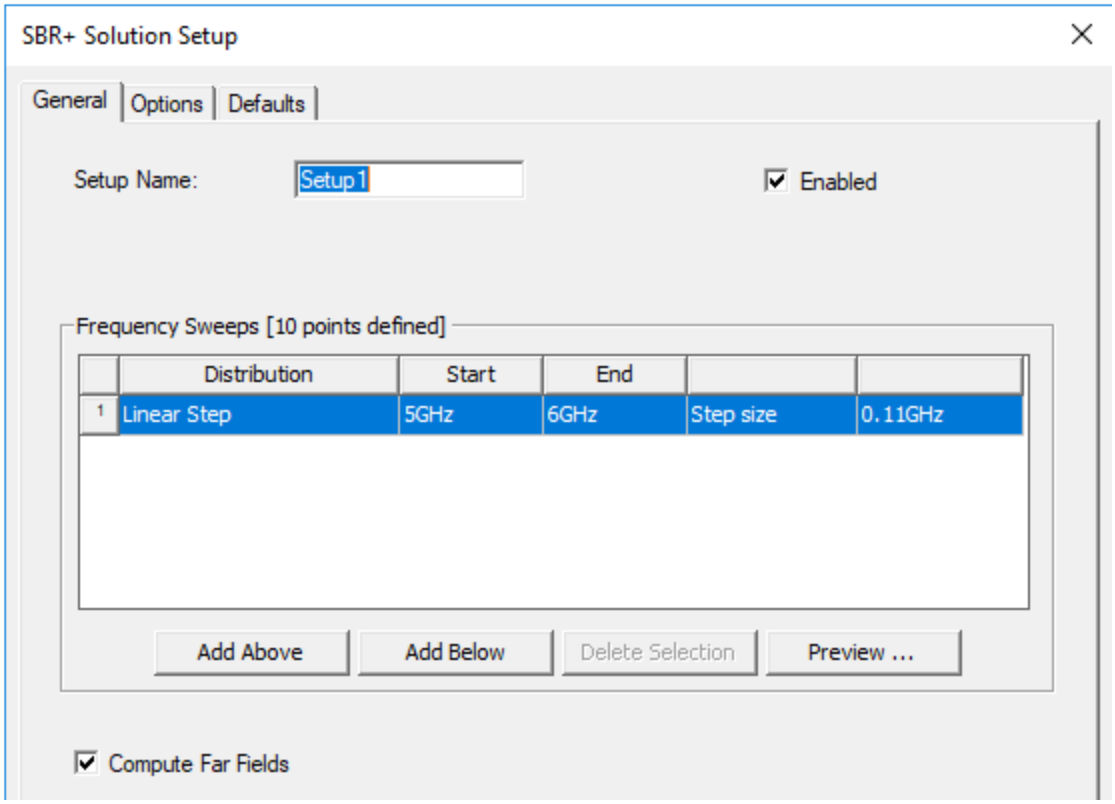
Source Design Frequency Sweep



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The sweep will be selectable in the Source link, and the **3D Fields Save Options** permit the Target SRB+ design to compute far fields and plot gain.

Example Target Design Solution Setup

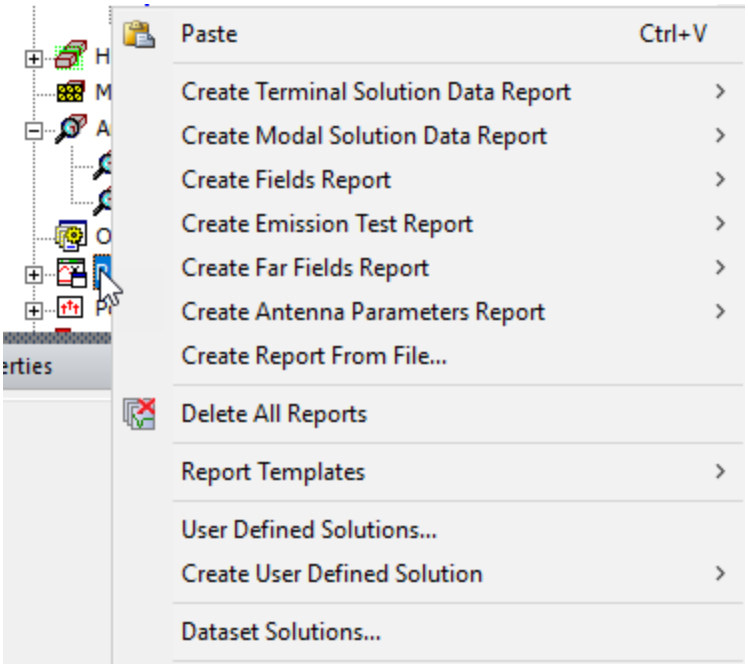


Compute Far fields is so that a Gain Report can be plotted based on the fields saved from an interpolating sweep in the linked source design..

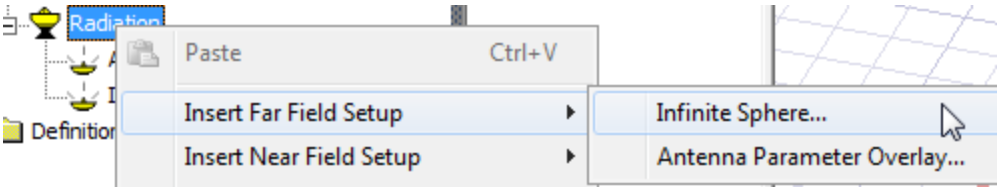
Viewing Results in the SBR+ Simulation

After you run an SBR+ simulation with linked Model antenna, you can create a variety of reports, including S Parameters, Far Fields, Gain, and Antenna Parameters.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You must create a Far Field Setup to access the Far Fields and Antenna Parameters reports.

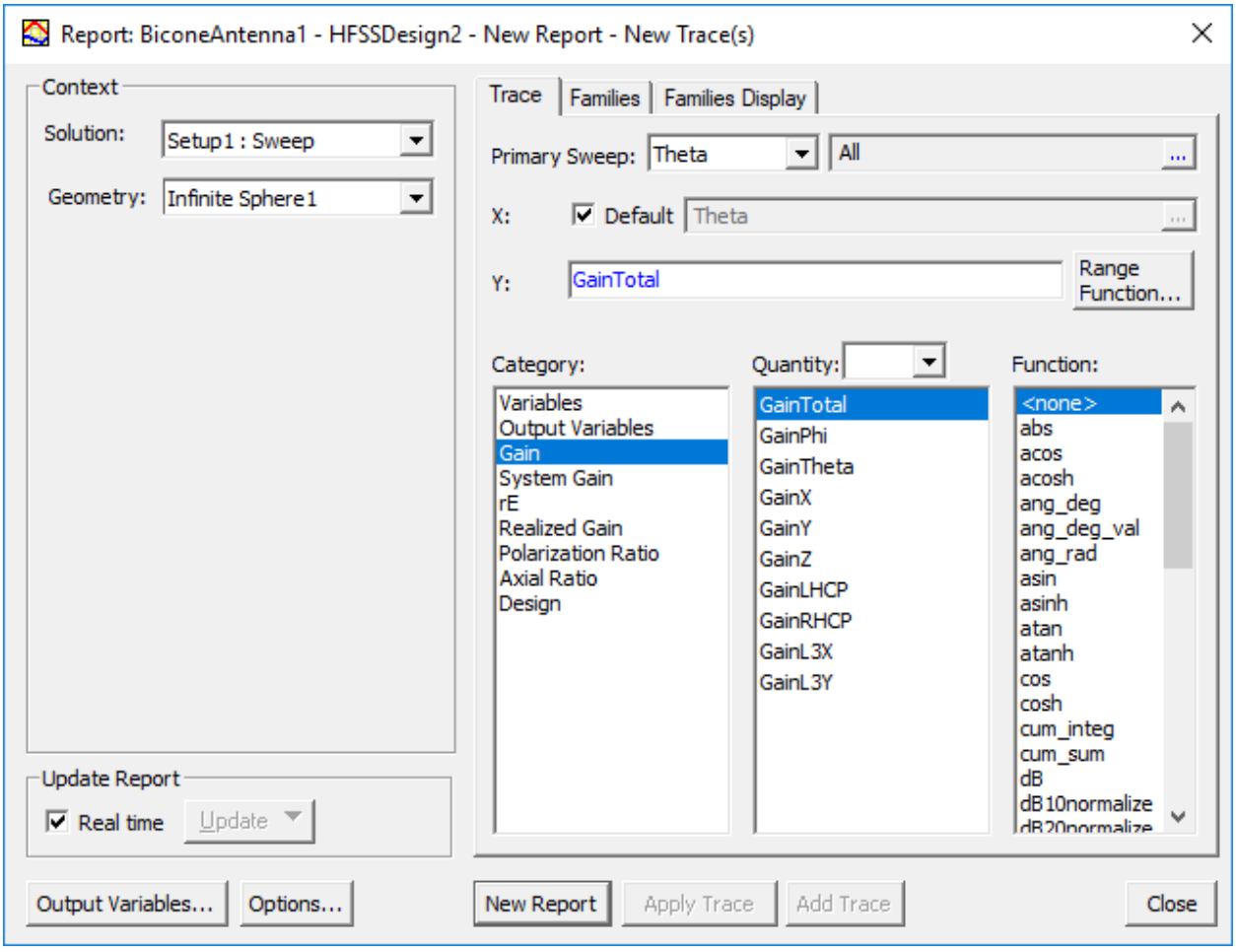


When a driven modal or driven terminal antenna is linked to an SBR+ target, an additional SBR+ power loss is considered for radiated power calculation. See *Technical Notes*: Radiated Power.

Requirements for Creating a Gain Report for SBR+ Design with Driven Modal or Terminal Source

If you have created an Infinite Sphere, solved an Interpolating Frequency sweep in the Source design and saved the 3D fields, and selected Compute Far Fields in the Source setup, you can create a Gain Report.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Select Tx/Rx Antennas for SBR+ Solutions

The SBR+ solution type allows you to create parametric antennas for a streamlined workflow in Ansys Electronics Desktop HFSS, which helps support ADAS and other similar application areas with large scene models. To allow for more efficient simulations, SBR+ Tx/Rx Antenna Selection lets you specify how each antenna is to be used in the simulation, as either Transmit, Receive, Transmit and Receive, or Off (Tx, Rx, Tx/Rx, or Off). This can dramatically reduce the number of antenna-to-antenna coupling pairs that must be simulated for a given scenario using SBR+ so that full NxN coupling analysis is not required. Even though SBR+ can handle these larger scene models more efficiently than some other approaches, if you select the specific Tx-Rx roles for each antenna you can obtain dramatically shorter runtimes because each transmitting antenna requires a separate ray trace. Near field Analysis for SBR+ Solutions is available if you specify Tx or Rx/Tx roles.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

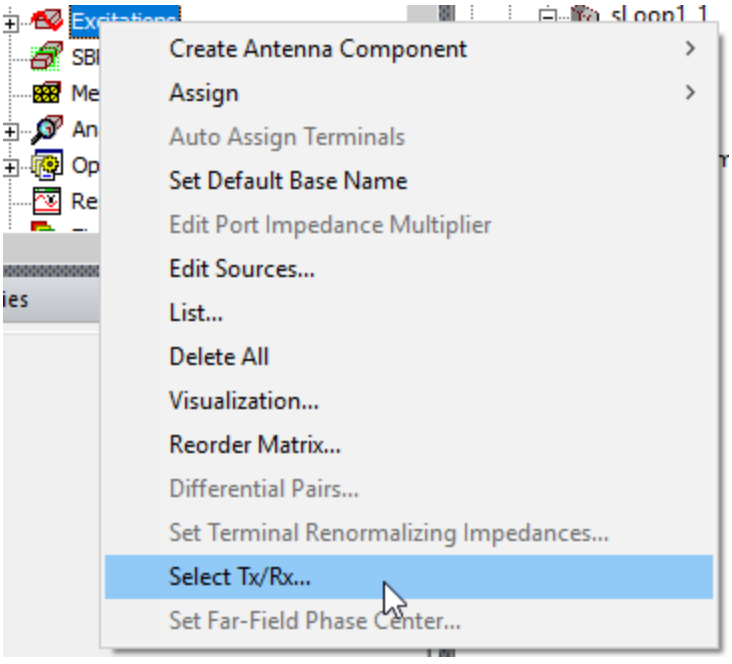
When antenna are assigned NxN full coupling among all sources as in all previous SBR+ region applications, HFSS SBR+ averages the values of cross coupling S matrix terms Smn and Snm to force them to be the same. When all antenna are assigned to TxRx, Smn and Snm might not be both available in the final solution. As a result, when the TxRx feature is assigned, such averaging is performed only when HFSS SBR+ detects that both Smn and Snm are present.

Prerequisites

- SBR+ Solution Type
- Parametric Antenna Components

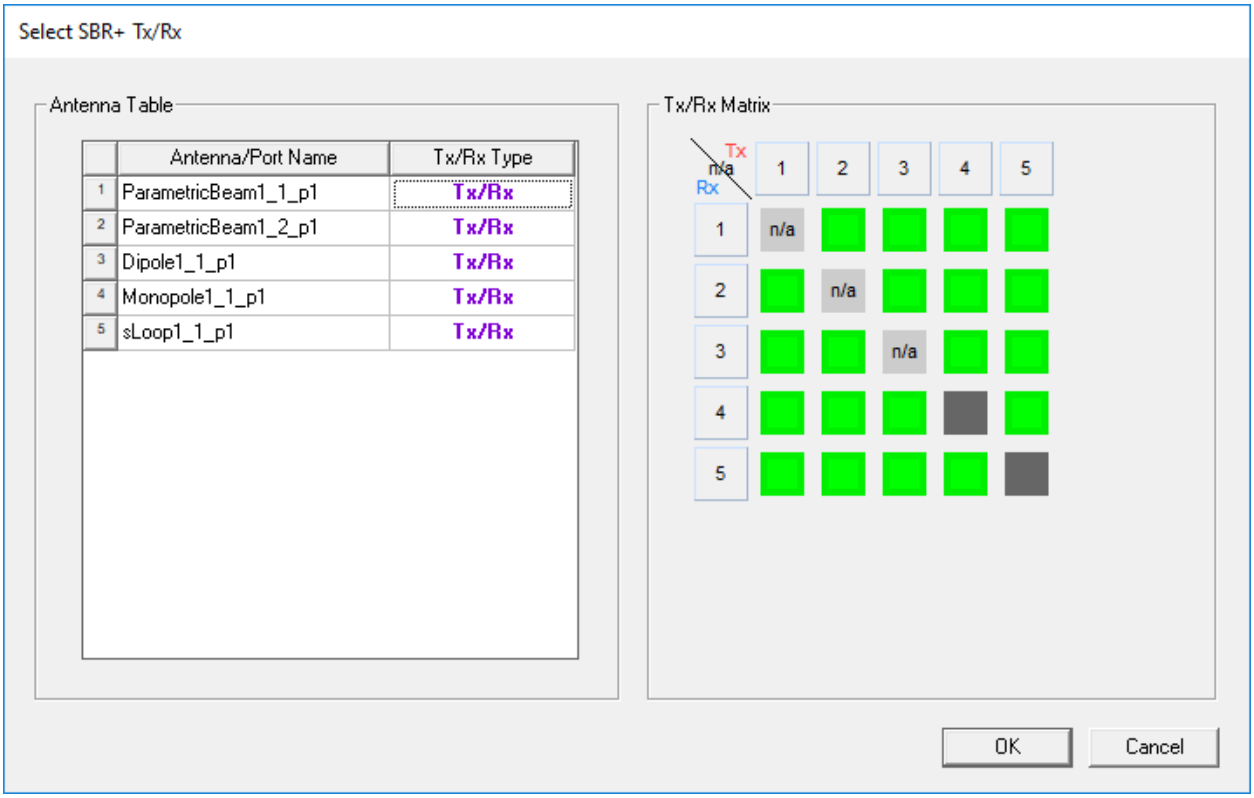
To Assign Transmit and Receive Assignments for Antennas in SBR+

1. Right-click the **Excitations** icon in the Project tree, and click, **Select Tx/Rx...** or click **HFSS>Excitations>Select Tx/Rx...**



This opens the **Select SBR+ Tx/Rx** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The left panel contains the list of all available antennas/ports available and their configured Tx/Rx Types, and the right panel contains the matrix (the S-param matrix) of active/inactive indicators for each transmit/receive (tx/rx) pair configured for the simulation. You can resize the dialog both horizontally and vertically to show more antennas in the list and more cells in the matrix. Scrollbars appear for both the table and the matrix if there are too many antennas/ports to fit in the dialog size.

Default types are set to “Tx/Rx” for all antennas/ports, with all Tx/Rx matrix cells set to active. This corresponds to a full NxN coupling configuration. If you want full NxN coupling for an SBR+ simulation there is no need to open or configure the Tx/Rx Selection settings.

Beta Feature HFSS Interactive Dialogues. If you enable this feature, you can edit the design outside the **Select Tx/Rx** dialog, and the dialog will update. The supported edits include:

Antenna Table Updates:

- New source is created: added to the last row of antenna table. Default Tx/Rx type is “Tx/Rx”.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

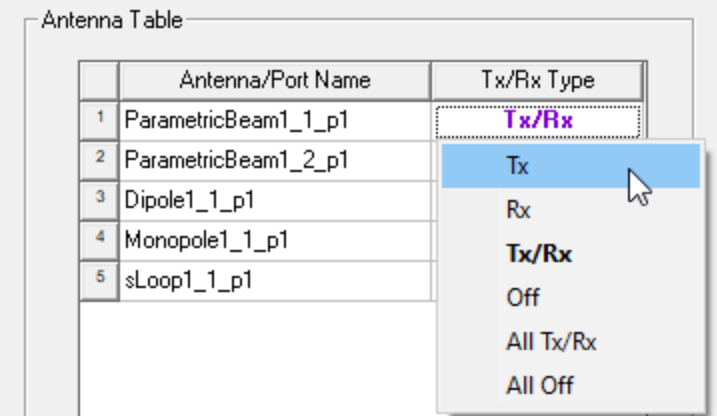
- Sources are deleted: remaining sources retain the original order on the table, with deleted sources removed.
- Source name changes: corresponding source name on the table updates to the new name.

Tx/Rx Matrix Update

- New source is created: a new row and column will be added to the matrix. Depending on the number of Tx and Rx channels in the matrix, the number of buttons added to the matrix will adjust itself to form the new matrix. Default button state is on, except the buttons that have the same source of Tx and Rx.
- Sources are deleted: corresponding Tx and Rx of those sources in the matrix are deleted.
- Source Tx/Rx type changes: row and column visibility updates based on the change of Tx/Rx matrix.

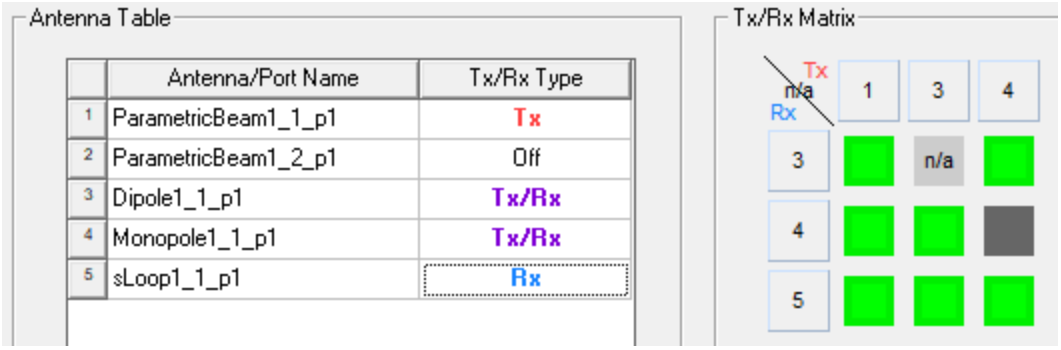
Note: The dialog supports undo and redo for creating, deleting and renaming elements. After undo/redo, added elements will recover to default Tx/Rx type and button state.

2. For the Antennas table, you can assign each antenna/port a type of **Tx**, **Rx**, **Tx/Rx**, or **Off**. This is done via a drop-down menu selection on the table cells under the **Tx/Rx Type** column. For convenience, the selection list includes **All Tx/Rx** and **All Off** selections.



The text color changes with the Tx/Rx Type selection as follows: Tx = red, Rx = blue, Tx/Rx = purple, Off = black.

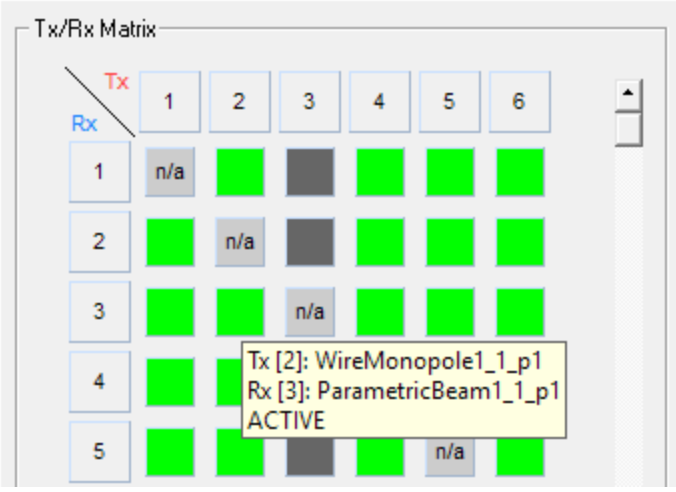
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Tx/Rx Type** also determines which rows and columns are displayed in the **Tx/Rx Matrix** panel on the right. Transmit antennas appear as columns across the top (notice the red Tx label), and receive antennas appear as rows down the left side (notice the blue Rx label), just as shown in the S-parameters matrix (available under Results>Matrix Data). If you set the type to **Tx**, an antenna/port will only show its **Tx** column in the matrix, and the **Rx** row has all its cells set to inactive and is not shown. Similarly if you set the type to **Rx**, the **Tx** column for that antenna/port in the matrix has all its cells set to inactive and hidden.

Antenna rows/columns are numbered to match those used in the table on the left. The order of both corresponds to the order of the antennas/ports in the output data, and may be different than the alphabetical order shown in the project tree under the **Excitations** folder.

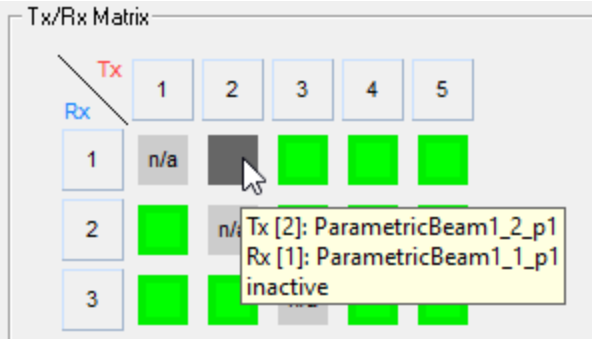
In the **Tx/Rx** matrix, tooltips are shown when the mouse is hovered over any cell in the matrix to show the corresponding antenna/port names and active/inactive status.



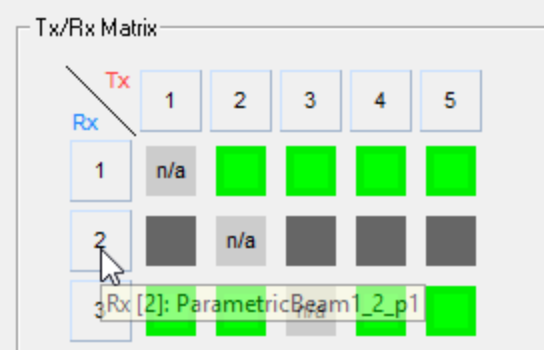
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A green indicator means this Tx/Rx combination is active and the SBR+ simulation compute results for the interactions between this antenna port pair; dark grey means it is inactive for the simulation. Any self-coupling cells are marked with "n/a" as not applicable for the SBR+ solution type and may not be modified, which includes the diagonal matrix cells as well as any self-coupling cells for multi-port antennas.

3. To toggle a Tx/Rx combination as active/inactive simply click on the corresponding cell in the matrix. The color will toggle between green (ACTIVE) and dark grey (inactive).



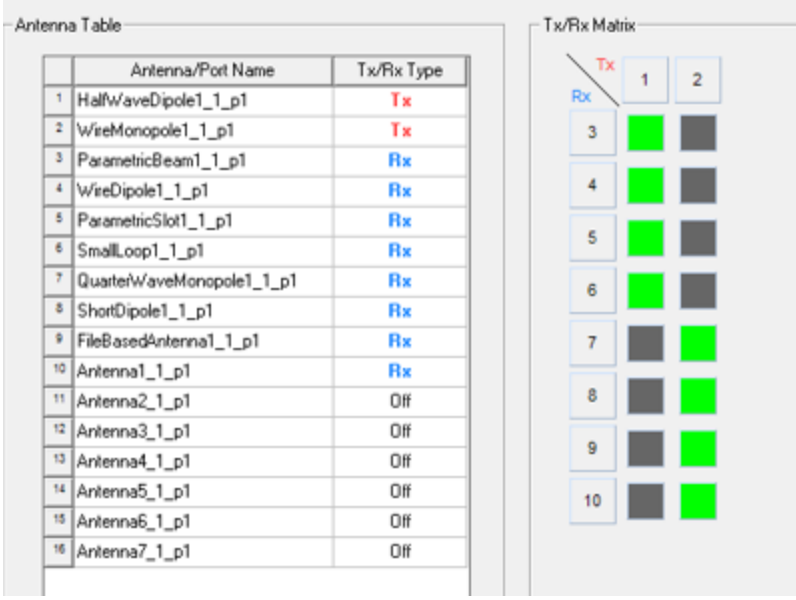
4. To toggle an entire Tx column or Rx row, click one of the numbered header cells. Clicking a Tx column has the effect of turning this Tx either on or off to all receivers, and likewise, clicking an Rx row has the effect of turning this Rx on or off to all transmitters.



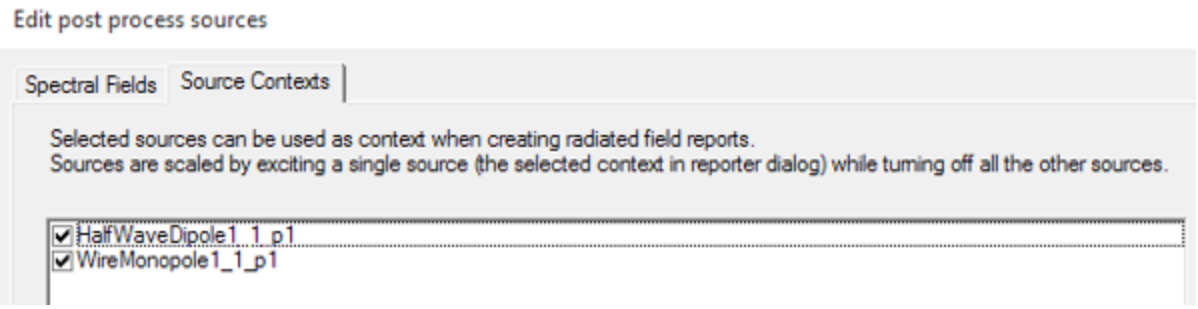
> **Note:**
>
> Turning an entire row or column off (making it inactive) achieves a similar end to specifying an antenna as either **Tx** only or **Rx** only.

The **Tx/Rx** roles you select in the SBR+ **Tx/Rx Selection** dialog can impact the post-processing sources available for the simulation. For example, suppose you configure an antenna/port to be **Rx** or **Off**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

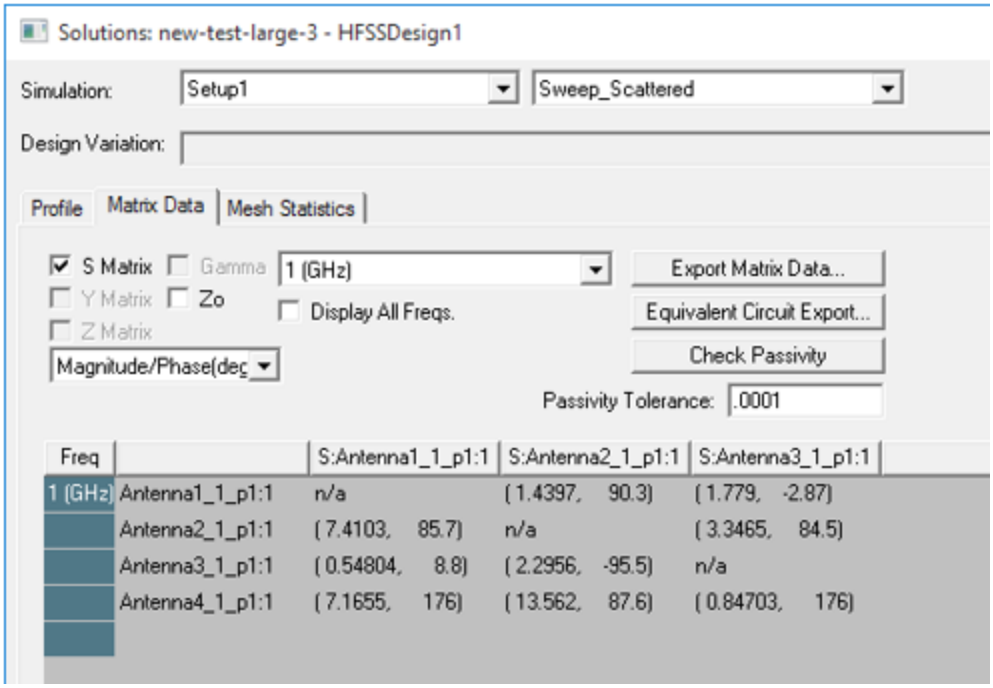


Then that item will not be available as a post-processing source context under Edit Sources. Only **Tx** or **Tx/Rx** antennas/ports will appear on the **Source Contexts** tab, since those are the only antennas/ports configured to produce radiating fields (ie, transmitting).

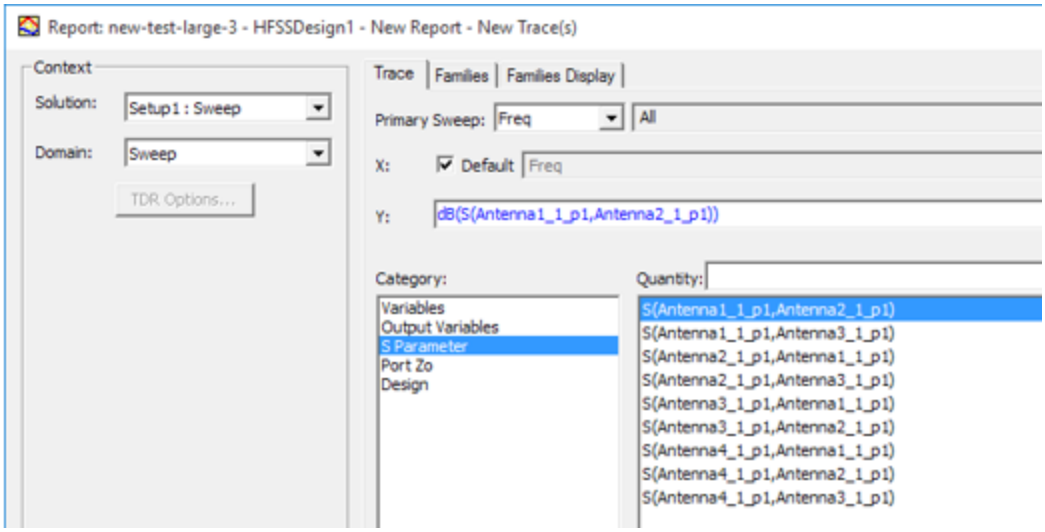


5. Upon completion of the simulation, the results in the S-parameter matrix data display will correspond directly to the Tx/Rx Matrix configuration, with simulation results only computed for ACTIVE (green) cells. All other inactive and "n/a" cells will show "n/a" results for those same entries in the S-parameter matrix since no SBR+ simulation data was computed for those coupling pairs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. In the **Create Report** dialog box, you see only the active Tx/Rx coupling pairs listed as the available report quantities, not the full list of NxN coupling matrix pairs (unless configured as full NxN coupling).



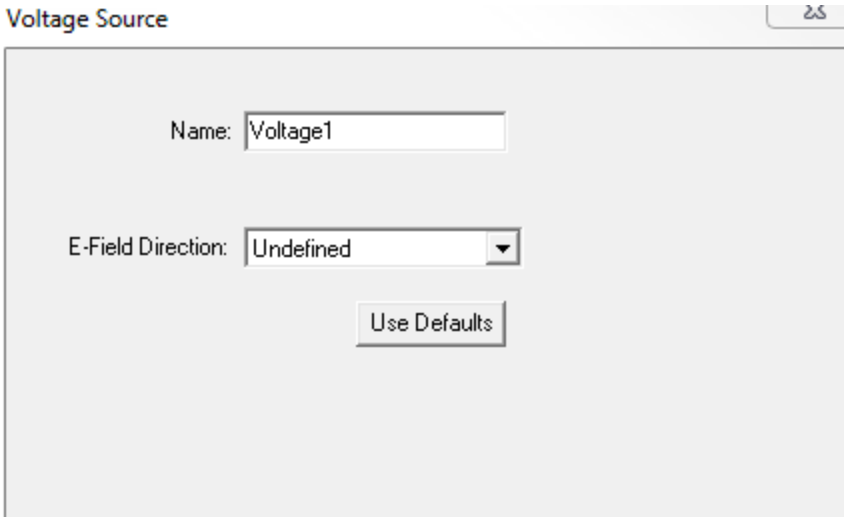
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assigning Voltage Sources

A voltage source in HFSS can be defined on a surface located anywhere in the 3D problem space. Typically the source is placed on a surface between two conductors such that a user defined total voltage is maintained between the conductors. Voltage source is used when the feed structure is very small compared to the wavelength and a constant electric field may be assumed across the feed points; in this case, HFSS assigns a constant electric field across the gap on which you specified the voltage.

> **Note:**
>
> For more information, see Voltage Source in **Technical Notes**.

1. Select the object face to which you want to assign the voltage source, and click **HFSS>Excitations>Assign>Voltage** to display the **Voltage Source** dialog box.



2. Specify the direction of the electric field by drawing a vector.

   When the source is selected, an arrow indicates the direction and a letter v indicates the type of source.

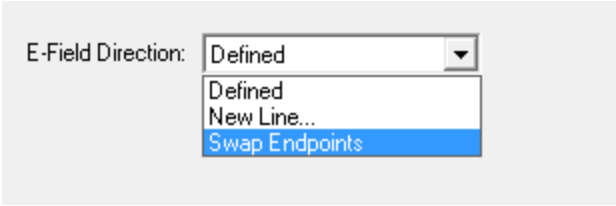
   > **Note:**
   >
   > For Transient Solution types, you also designate sources as Active and Passive.

Port sources are created and solved with unit magnitude and 0 degree phase. They can be scaled within the Edit Sources dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The **E-field** direction can be reversed from the **Voltage Source** dialog box.



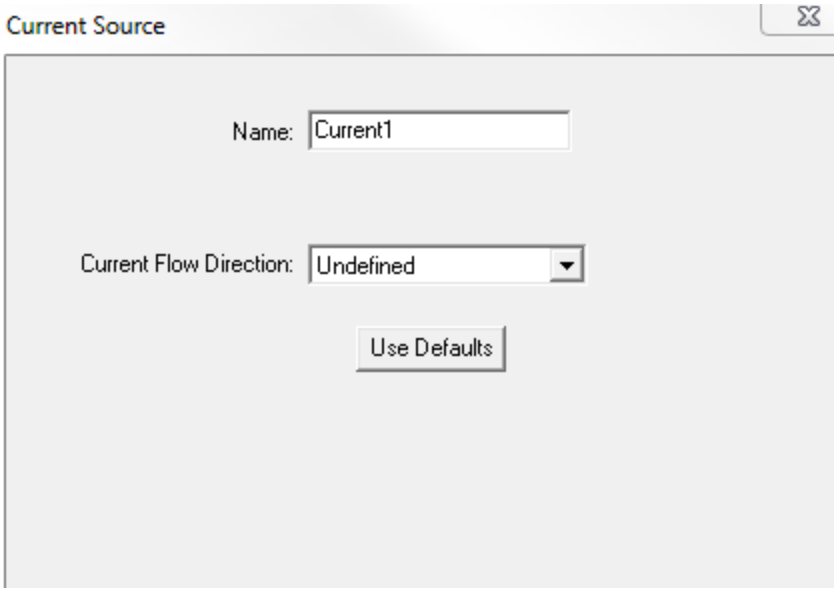
Assigning Current Sources

Assign a current source when you want to define the direction of the current flow through a surface. A current source is used when the feed structure is very small compared to the wavelength and the electric current on the surface is assumed to be constant across the feed points.

Note:

For more information, see Current Sources in **Technical Notes**.

1. Select the object face to which you want to assign the current source and click **HFSS>Excitations>Assign>Current** to display the **Current Source** dialog box.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Specify the current flow direction by drawing a vector:

   When the source is selected, an arrow indicates the direction and a letter i indicates the type of source.

3. For Transient Solution types., you also designate sources as Active and Passive.

Port sources are created and solved with unit magnitude and 0 degree phase. They can be scaled within the Edit Sources dialog.

Modifying Voltage or Current Sources

To change the name, value, or electric field direction of an assigned voltage/current source:

1. Double-click the source's icon under **Excitations** in the project tree.

   The **Voltage Source/Current Source** dialog box appears.

2. Edit the name or value of the source.
3. Select **Swap Endpoints** from the **E-Field Direction**/Current Flow Direction drop-down menu to reverse the direction of the e-field/current flow.

   The start and endpoints of the E-field/current flow line are switched; the line's direction is reversed.

Assigning Magnetic Bias Sources

When you create a ferrite material, define the net internal field that biases the ferrite by assigning a magnetic bias source. The bias field aligns the magnetic dipoles in the ferrite, producing a non-zero magnetic moment.

1. Select the 3D ferrite object to which you want to assign the magnetic bias source.
2. Click **HFSS>Excitations>Assign>Magnetic Bias**.
3. Specify whether the applied bias field is **Uniform** or **Non-uniform**.

   If a design already contains a magnetic bias field, you cannot assign another of a different type. If a single bias field exists in a design, you can edit the type.

4. If you selected the **Uniform** radio button, click **Next** and do the following:
   a. In the **Internal Bias** field, type the value of the ferrite in amperes/meters. You can assign a variable as this value.
   b. Enter the rotation of the permeability tensor with respect to the xyz-coordinate system in the **X Angle**, **Y Angle**, and **Z Angle** boxes. You can assign variables to these values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you selected **Non-uniform**, select the **Setup Link...** button to display the **Setup Link** dialog box. Under the **General** tab, do the following:

a. Select the radio button for **Extractor Mode** (the default) or **Interactive Mode.** Note that in **Extractor mode**, the source project will be saved upon exit. **Extractor mode** means that the software is opened during the link solely for the purpose of solving. Selecting **Interactive Mode** launches Maxwell.

(If you open the **Setup** dialog for a pre-existing Magnetic Bias source, the **General** tab shows radio buttons for **View Only** and **Edit Link**. With **View Only** selected (the default), all of the link settings are unavailable. Selecting **Edit Link** enables the fields and changes the radio buttons to **Extractor Mode** and **Interactive Mode**.)

b. Type the name of a Maxwell 3D Field Simulator project in the **Project File** box, or click the ellipsis [...] browse button display a file browser to select the project.

HFSS uses the Maxwell 3D project as the source of the non-uniform magnetostatic field information during solution generation. Linking invokes a Maxwell 3D window to provide the solution for the targeted HFSS project.

c. If there are multiple designs available for the project, you can select from the drop-down menu.
d. If there are multiple solutions available, you can select from the drop-down menu.

The "Default" solution is the product dependent solution of the first Setup.

That is the setup listed first in the source design's project tree (alphanumerical order). A product specific solution of this setup becomes the default solution. In most products, it is LastAdaptive. In a Transient solution type, it is "Transient."

e. Use the radio button to specify whether to save the source path relative to **The project directory of the source project** or **This project**.
f. Use the check box specify whether to **Simulate source design as needed**.
g. Use the check box to specify whether to **preserve the source design solution**. Note that in **Extractor mode**, the source project will be saved upon exit. **Extractor mode** means that the software is opened during the link solely for the purpose of solving.

The **Setup Link** dialog also contains a **Variable Mapping** tab.

It lists variables available within the Maxwell 3D Field Simulator and the value can (and often will) be a variable in the HFSS Setup. You can edit the Value fields by typing, and the Units fields by selecting from a drop-down menu. You can choose to Map Variable By Name. In this case, same named variables have their values mapped automatically. Different named variables are unaffected.

To accept the settings and close the **Setup Link** dialog box, click **OK**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **Finish to close the Magnetic Bias wizard**.

   The magnetic bias source is assigned to the selected object. If you have set up a link, HFSS invokes a Maxwell 3D window to provide the solution for the targeted HFSS project.

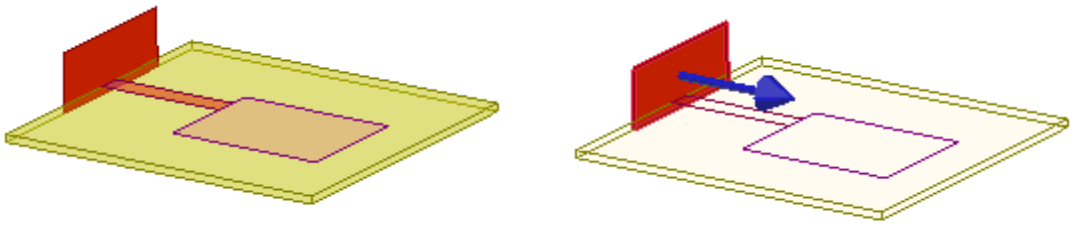
You can also access and edit the magnetic bias source information via the **Properties** dialog for the source. Magnetic bias sources always have the lowest priority compared to boundaries and other excitations in the solver view.

Applications for Deembedding

Deembedding helps to calculate the S-parameters near or on the plane of a discontinuity or when a long transmission line is attached to the port plane instead of explicitly modeling it in HFSS. For lossless ports, when you deembed them into or out of the model, there is change only in the phase of the S-parameters but not in the magnitude. For lossy ports, aside of the change in the phase of the S-parameters, there is only a slight change in the magnitude. Deembedding saves time and significantly reduces the simulation efforts.

Extract Input Impedance: Use Deembedding

This section describes how to extract the input impedance in the microstrip fed patch antenna model shown in the following figure. The port (highlighted in red ink) is placed at an appropriate distance away from the discontinuity. We cannot place the port near the discontinuity (the intersection of the trace and the patch antenna). However, if you want to measure the input impedance at the discontinuity, you can deembed the port as shown by the blue arrow in the figure.



For lossless ports, deembedding will not change the magnitude of the S-parameters. It will only change the phase.

Setup Link Window

Linked data can be a mesh, field results, or some other post-processing data that the source design generated. The possibilities depend on the source and target design and the solution types. The **Setup Link** window permits you to link the current project to another for

- Magnetic Bias source
- Near Field Wave source
- Far Field Wave source
- Initial Mesh source
- Linked Impedance Boundaries in HFSS

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can link HFSS projects. This link is controlled with a "Near Field" or "Far-Field" Incident Wave source that is grouped with the Excitations.

Modifying Excitations

To change the properties of an excitation, do one of the following:

- Double-click the excitation's icon under **Excitations** in the project tree.

  The excitation's properties window appears, in which you can modify its properties.

- Right-click the excitation in the project tree, and then click **Properties** on the shortcut menu.

  The excitation's dialog box appears, in which you can modify its properties.

- On the **HFSS** menu, click **List**.

  The **Design List** dialog box appears. Under the **Excitations** tab, you can modify the properties of one or more boundaries.

The following modifications to an excitation are possible:

- Change its properties.
- Delete it.
- Reassign it to another surface.
- Reprioritize it.
- Hide it from view.
- Show Nets for DC Continuity for 3D Conductors.
- Modify the impedance multiplier.
- Deembed the port.
- Set up Differential Pairs

For HFSS Transient solutions, render it Active or Passive.

> **Note:**
>
> The most common source of errors in simulations are related to defining ports. Examine your ports, their definitions, and inspect problem areas by using the **Zoom to** option. For more information, see Zoom to Selection.

To show or Hide Excitations:

See Setting Boundary and Excitation Visualization Options.

See Show Nets for DC Continuity for 3D Conductors.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Deleting Excitations

To delete *one excitation*:

1. Select the excitation you want to delete by clicking its icon in the project tree.
2. Click **Edit**>**Delete** .

   The excitation is removed from the design and the project tree.

   For terminal solutions, if you delete a port with terminals associated with it, deleting the port also removes the associated terminals.

To delete *all excitations*:

- Click **HFSS**>**Excitations**>**Delete All**.

You can also *delete one or more excitations* in the **Design List** dialog box:

1. Click **HFSS**>**List**.

   The **Design List** dialog box appears.

2. Under the **Excitations** tab, click the row of the excitation you want to delete.
3. Click **Delete**.

Reassigning Excitations

You can reassign an excitation to another surface. This is useful when you have modified objects with assigned excitations, invalidating the excitations. For example, if you unite two objects with assigned excitations, the second object's excitation will become invalid because united objects maintain the characteristics of the first object selected. In this case, you would need to reassign the excitation or delete it. You can use a **Reassign Excitation** dialog accessed through the [*solver*]**>Excitation>Reassign** menu, or right-click the Excitation of interest in the Project tree, and select from the shortcut menu to **Reassign**, **Add to Assignment**, or **Remove from Assignment**. You can also determine a current assignment by right-clicking on the excitation in the **Project** tree and by using the **Select Assignment** command on the short cut menu.

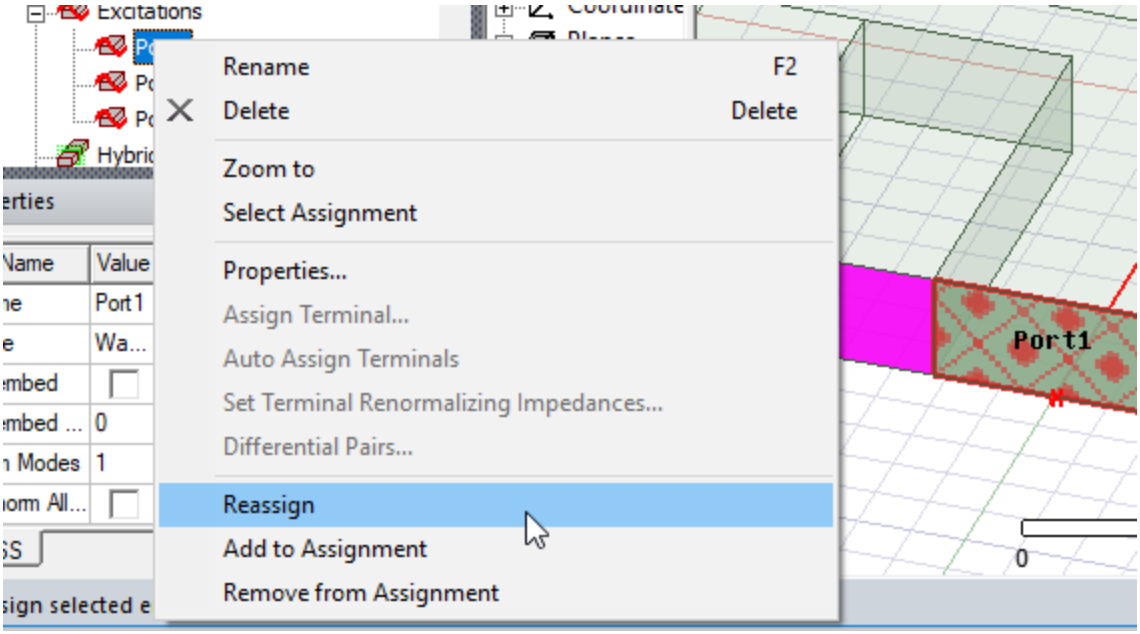
Prerequisites:

- The surface to which you want to **Reassign**, or **Add to Assignment** must be on the same plane as existing assignment. It does not have to be in contact to **Reassign**, but must be in contact to **Add to Assignment**.
- You must first select the surface that you want to reassign or add to an assignment.

To use the short-cut menu to reassign or add an excitation:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. With the target surface selected, right-click the Excitation of interest to invoke the menu.



2. Select **Reassign** to change the assignment to the selected surface, or **Add to Assignment** to include surface.

To use the shortcut menu to Remove a selected surface from an assignment:

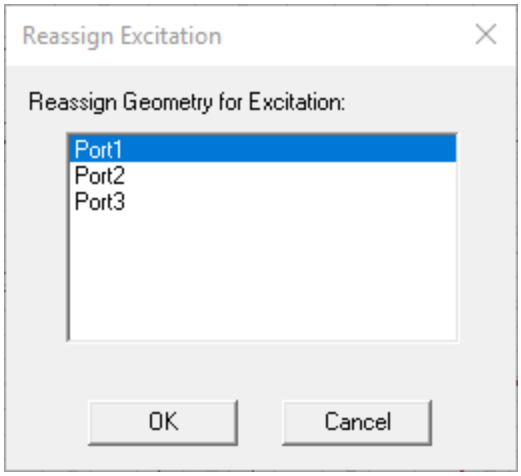
1. With the target surface selected, right-click the Excitation to invoke the menu:
2. Click **Remove from Assignment**.

To use the solver Menu and the **Reassign Excitation** dialog:

1. Select the object or object face to which you want to assign an existing excitation.
2. Click **HFSS>Excitations>Reassign**.

The **Reassign Excitation**window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Select an existing excitation from the list, and then click **OK**.

The excitation is reassigned to the object or object face.

> **Note:**
>
> When reassigning an excitation that includes vectors in its definition, HFSS attempts to preserve the vectors with the new assignment, but this is not always possible.

Setting the Impedance Multiplier

For designs with ports.

If one or more symmetry planes have been defined or if only a wedge of a structure is modeled, you must adjust the impedance multiplier or the computed impedances will not be for the full structure.

> **Note:**
>
> Changing the impedance multiplier invalidates solutions in projects where lumped ports are defined. In such projects, you need to re-solve the project after the change.

1. Click **HFSS>Excitations>Edit Impedance Multiplier**.

The **Port Impedance Multiplier** dialog box appears.

2. Type a value in the **Impedance Multiplier** box.

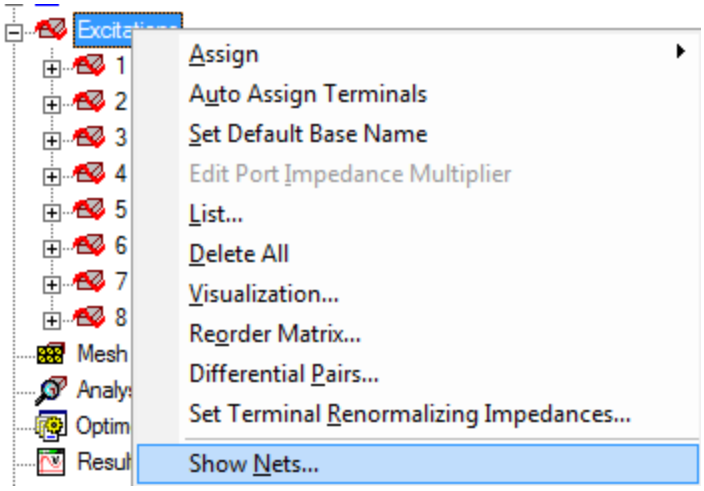
You can assign a variable as this value.

3. Click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Show Nets for DC Continuity for 3D Conductors

The **Show Nets** command appears in the **HFSS>Excitations** menu and by right-clicking on **Excitations** in the Project tree.



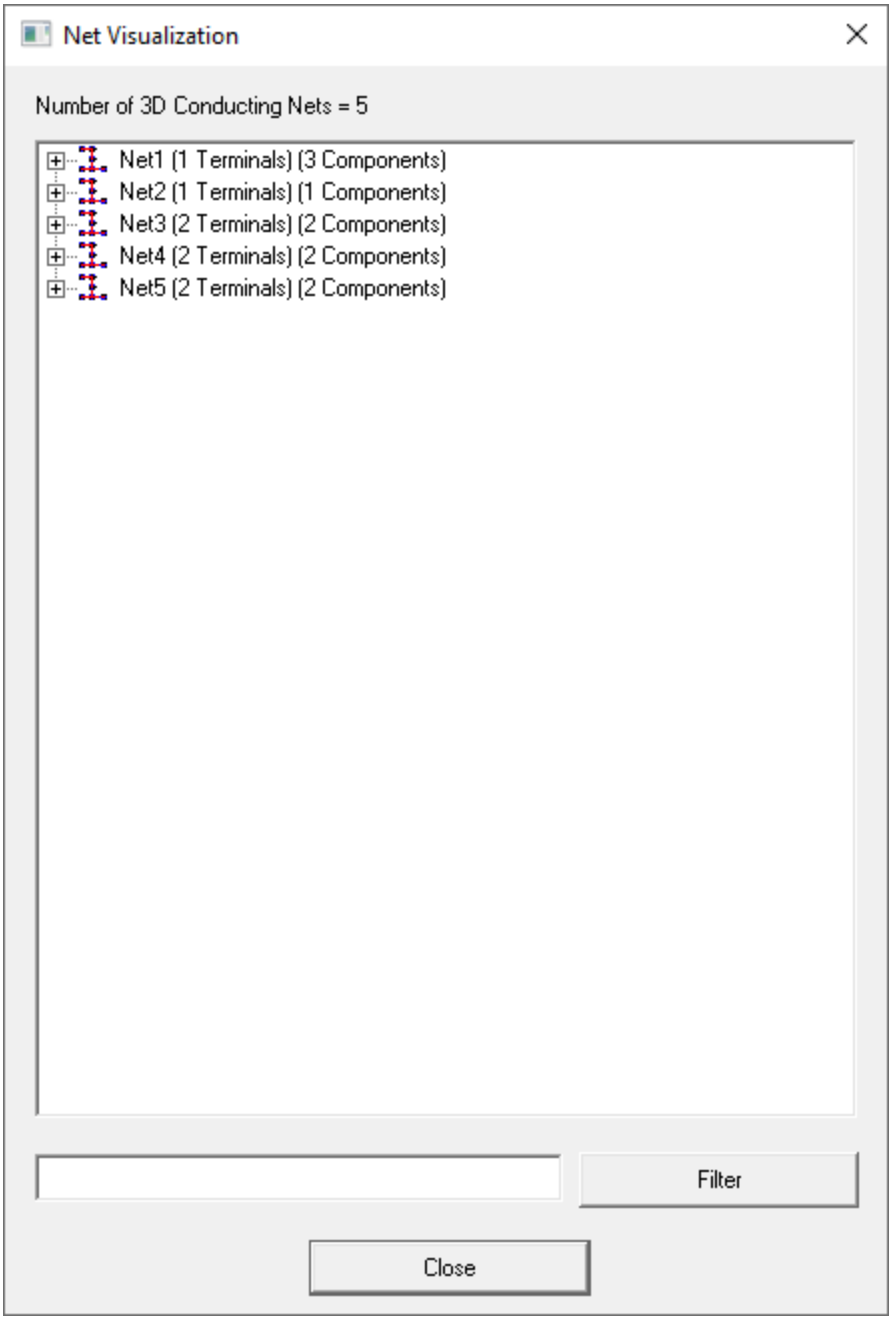
Selecting **Show Nets...** displays the **Net Visualization** dialog that lists the nets and associated terminals. You can use the dialog to select any net or terminal. You can continue to work in the Modeler window with the dialog open.

> **Note:**
>
> For a net to appear in the **Net Visualization** dialog box, the conductivity of the assigned material must be equal to or greater than 20,000 siemens/m, or the **Resistance** must be less than 3.142 ohms. Conductivity is set by material assignment (in the **3D Modeler** window or a boundary condition), and **Resistance** is set by boundary conditions (e.g., **Impedance** or **LumpedRLC**).

In the **Net Visualization** dialog, you can sort the nets in ascending or descending order, relative to the number of terminals. This helps you locate GND nets, which have no terminals.

Text at the top of list box provides the total number 3D Conducting nets. Naming of nets is automatic and follows the convention: "Net1 (0 Terminals)", "Net2 (1 Terminals)"…"Net<N> (<T> Terminals)". Net Names are not editable.

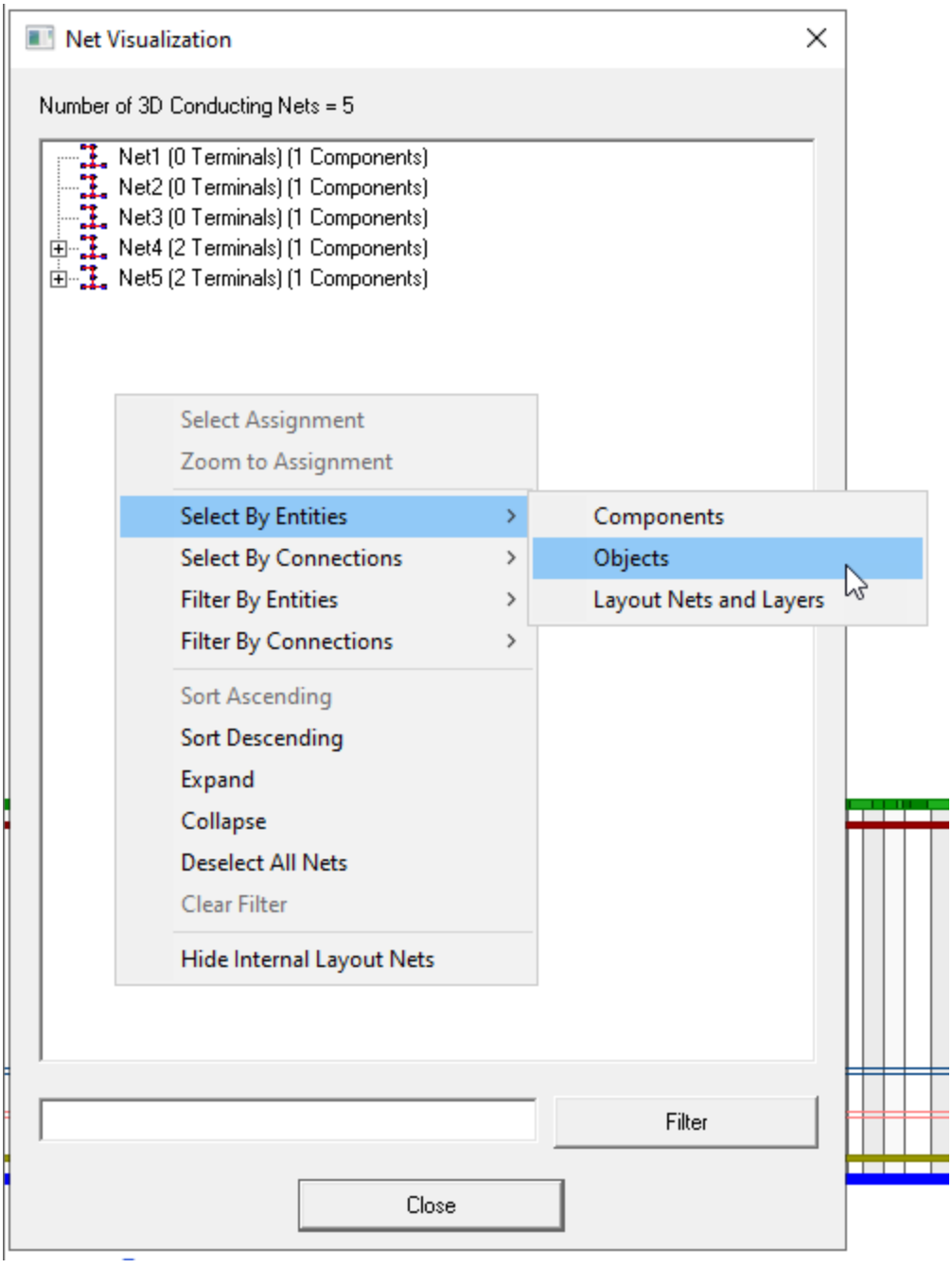
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For designs with many nets, you can enter text and wild cards in the field, and click the Filter button.

Right-click in the **Net Visualization** dialog to view the short-cut menu. The menu is context sensitive, depending on whether you click on a particular net object, or in the white area.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Select assignment**: If you right-click over selection in the tree, selects entities present in the selection. If the right click menu is invoked on the surrounding blank space, this menu item is disabled as there is no selection.
- **Zoom to assignment**: If you right click over a selection in the tree, zooms to the entities present in the selection. If you right click menu on the surrounding blank space, this menu

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

item is disabled as there is no selection.

- **Select by entities**: If you right invokes a dialog containing object, component, or net and layer names present in the selection if the right click menu is invoked over some selection in the tree. If the right click menu is invoked on the surrounding blank space, all object or component names present in the project are displayed. Selecting one or multiple of these object or component names selects all the nets which contain any of the selected entities.
- **Select by connections**: If you right click over some selection in the tree this produces a dialog containing object or component names present in the selection . If you right click menu on the surrounding blank space, a dialog with all object or component names present in the project displays. Selecting a pair of these object or component names selects all the nets which contain this pair of entities, signifying a direct or indirect connection between these entities.
- **Filter by entities**: A new dialog is invoked containing object, component, or net and layer names present in the selection if the right click menu is invoked over some selection in the tree. If the right click menu is invoked on the surrounding blank space, all object or component names present in the project are displayed. Selecting one or multiple of these object or component names filters all the nets which contain any of the selected entities.
- **Filter by connections**: A new dialog is invoked containing object or component names present in the selection if the right click menu is invoked over some selection in the tree. If the right click menu is invoked on the surrounding blank space, all object or component names present in the project are displayed. Selecting a pair of these object or component names filters all the nets which contain this pair of entities, signifying a direct or indirect connection between these entities.
- **Sort ascending**: Sorts the nets in ascending order of their names.
- **Sort descending**: Sorts the nets in descending order of their names.
- **Expand**: Expands the selection in the tree if the right click menu is invoked over some selection in the tree. If the right click menu is invoked on the surrounding blank space, menu item expands all the nets.
- **Collapse**: Collapses the selection in the tree if the right click menu is invoked over some selection in the tree. If the right click menu is invoked on the surrounding blank space, menu item expands all the nets.
- **Deselect All Nets**: Deselects all the entities present in the project. Clears the selection of all the previously selected entities.
- **Clear Filter**: Clears all the filtering applied on nets and shows all the nets.
- **Hide internal layout nets**: Hides all the nets which are internal to layout component

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

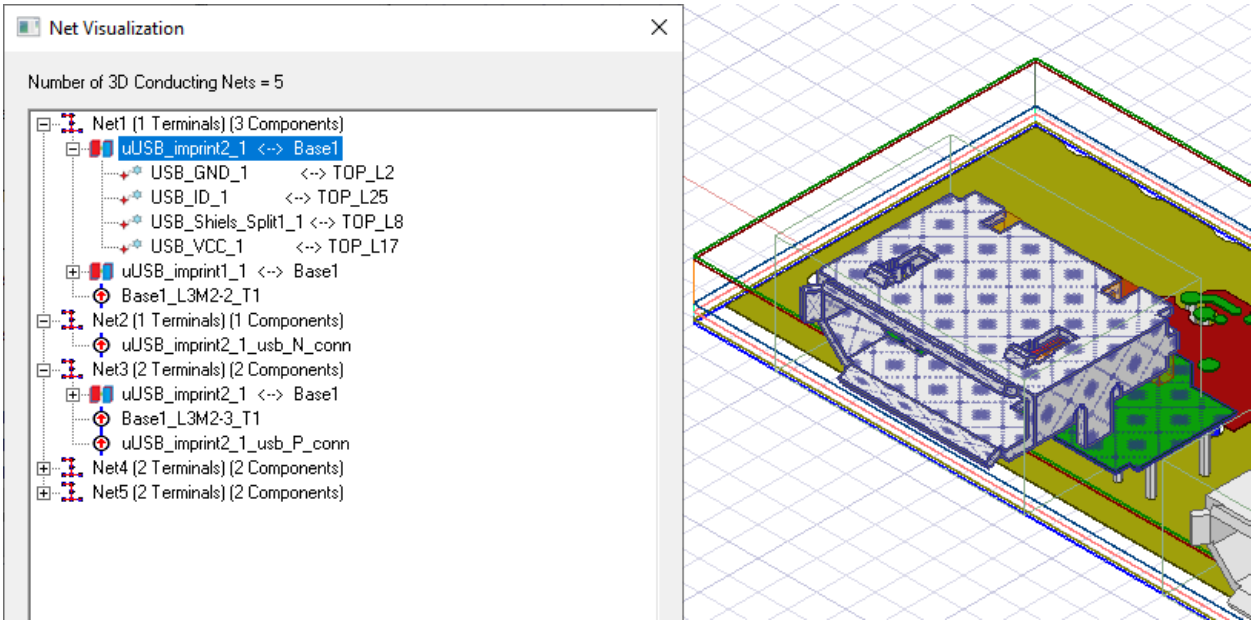
Along with the “filter by entities” and the “filter by connections” menu items, you can also filter nets using text input. The text box for this functionality is shown below:

Features of this text box are:

- It accepts one or more object or component names.
  - Multiple entries can be entered by separating them with comma operator (“,”).
  - Entity connections can be entered by separating entity names with dash operator (“-”).
  - Wild cards “*” and “&” can be applied. For example, LayoutComp1 and HfssComp2 can both be represented simultaneously as “*Comp*”.

  Similarly, LayoutComp1 and LayoutComp2 can be represented simultaneously as “LayoutComp&”. For example, (Box1, Box2, Comp1 – Comp2): This string applies filter based on if a net contains Box1, or Box2, or connection between Comp1 and Comp2.

Selected nets/components are highlighted in the Modeler window.

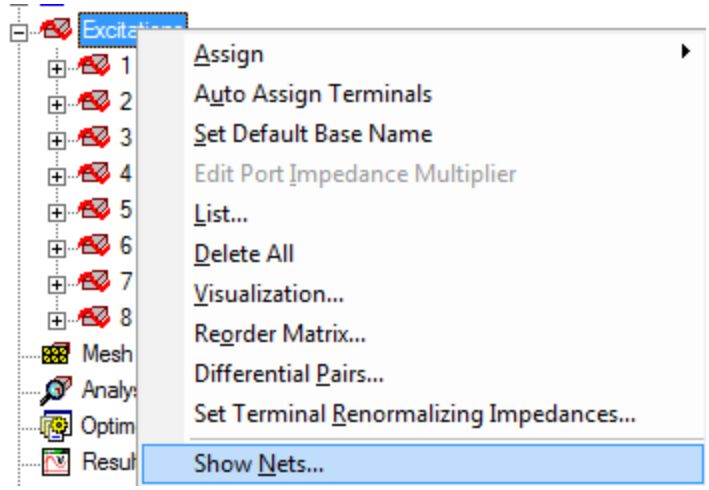


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Show Nets for Layout Components in HFSS 3D Geometries

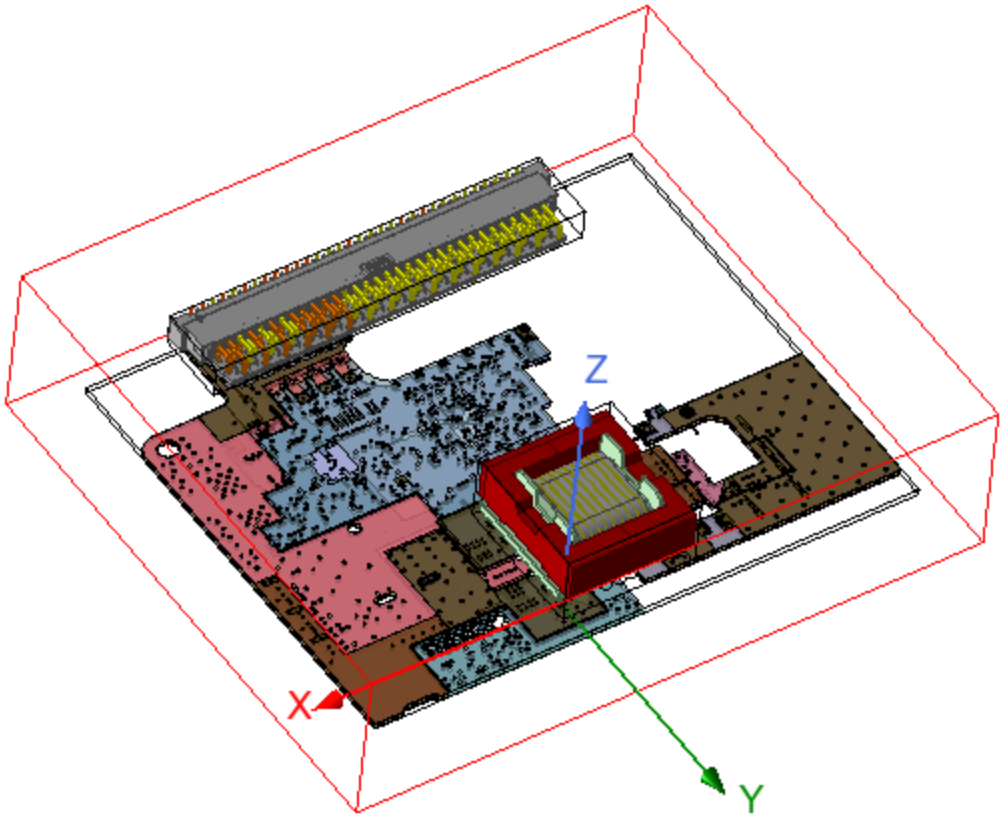
For Terminal Solution types, HFSS designs that include Layout Components, you can use the **Show Nets** command to visualize conducting nets and terminal associations formed by 3D geometries and layout components in a HFSS design.

The **Show Nets** command appears in the **HFSS** > **Excitations** menu and by right-clicking on **Excitations** in the Project tree.



Selecting **Show Nets...** displays the **Net Visualization** dialog box that lists the nets and associated terminals and includes any included Layout Components. You can use the dialog box to select any net or terminal. You can continue to work in the Modeler window with the dialog open. Consider the following example design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

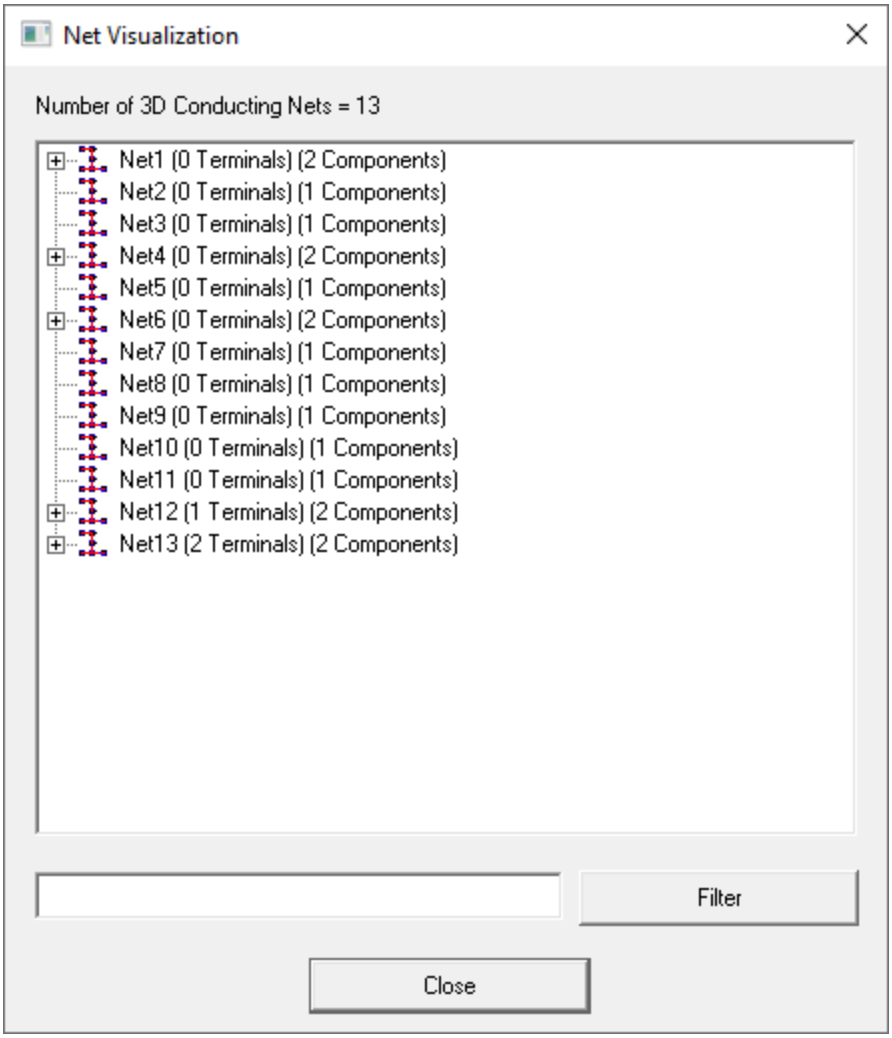
For a net to appear in the **Net Visualization** dialog box, the conductivity of the assigned material must be equal to or greater than 20,000 siemens/m, or the **Resistance** must be less than 3.142 ohms. Conductivity is set by material assignment (in the **3D Modeler** window or a boundary condition), and **Resistance** is set by boundary conditions (e.g., **Impedance** or **LumpedRLC**).

In the **Net Visualization** dialog box, you can sort the nets in ascending or descending order, relative to the number of terminals. This helps you locate GND nets, which have no terminals.

Text at the top of list box provides the total number 3D conducting nets. Naming of nets is automatic and follows the convention: "Net1 (0 Terminals)", "Net2 (1 Terminals)"…"Net<N> (<T> Terminals)". Net Names are not editable.

The **Net Visualization** dialog box is shown below.

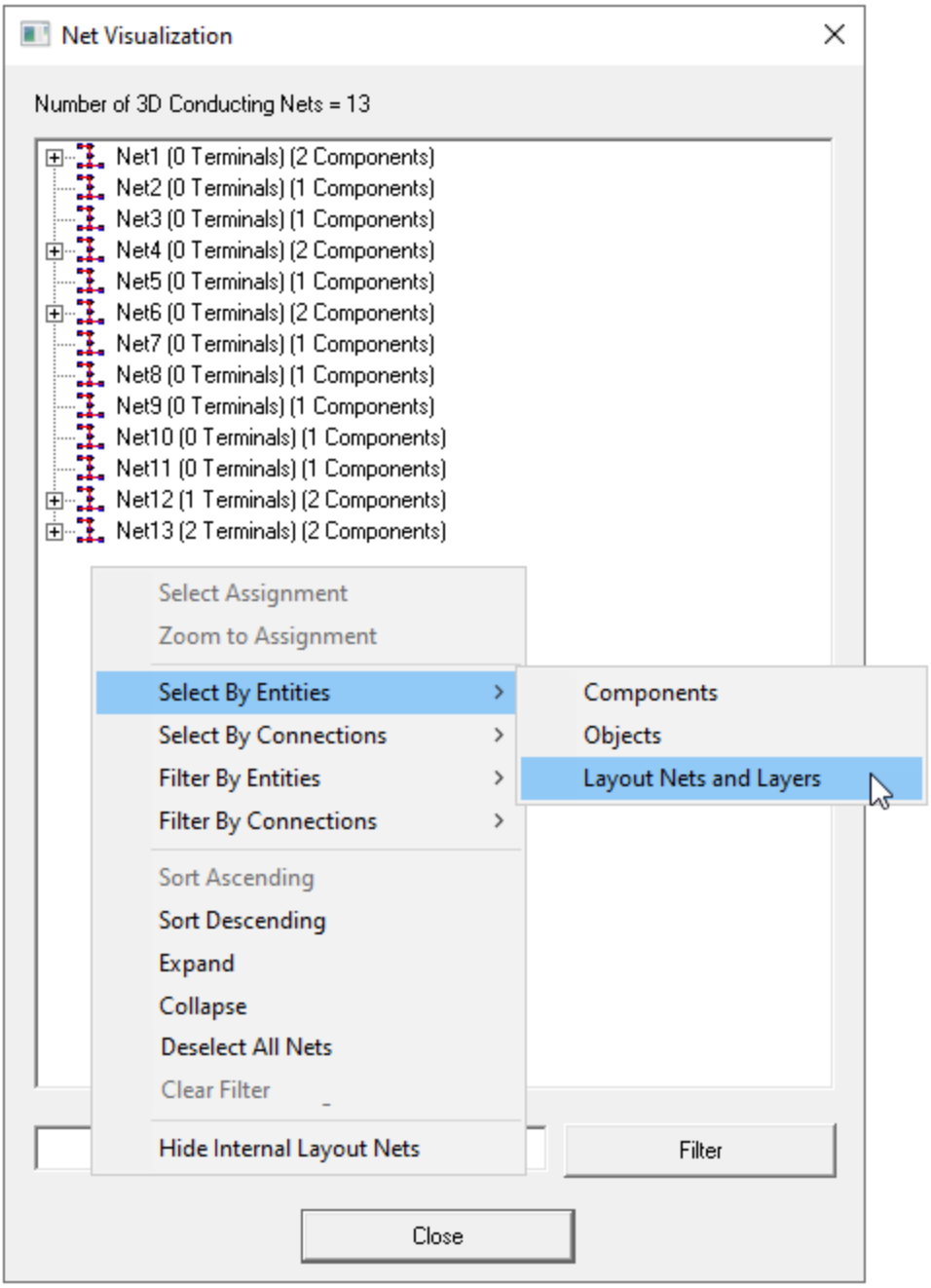
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For designs with many nets, you can enter text and wild cards in the field, and click the **Filter** button.

Right-click in the **Net Visualization** dialog box to view the short-cut menu. The menu is context sensitive, depending on whether you click on a particular net object, or in the white area.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Select Assignment**: If you right-click over selection in the tree, selects entities present in the selection. If the right-click menu is invoked on the surrounding blank space, this menu item is disabled as there is no selection.
- **Zoom to Assignment**: If you right-click over a selection in the tree, zooms to the entities present in the selection. If you right-click on the surrounding blank space, this menu item is disabled as there is no selection.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Select by Entities**: Invokes a dialog box containing object, component, or net and layer names present in the selection if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, all object or component names present in the project are displayed. Selecting one or multiple of these object or component names selects all the nets that contain any of the selected entities.
- **Select by Connections**: If you right-click over some selection in the tree, this produces a dialog box containing object or component names present in the selection. If you right-click in the surrounding blank space, a dialog box with all object or component names present in the project displays. Selecting a pair of these objects or component names selects all the nets that contain this pair of entities, signifying a direct or indirect connection between these entities.
- **Filter by Entities**: A new dialog box is invoked containing object, component, or net and layer names present in the selection if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, all object or component names present in the project are displayed. Selecting one or multiple of these objects or component names filters all the nets that contain any of the selected entities.
- **Filter by Connections**: A new dialog box is invoked containing object or component names present in the selection if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, all object or component names present in the project are displayed. Selecting a pair of these objects or component names filters all the nets that contain this pair of entities, signifying a direct or indirect connection between these entities.
- **Sort Ascending**: Sorts the nets in ascending order of their names.
- **Sort Descending**: Sorts the nets in descending order of their names.
- **Expand**: Expands the selection in the tree if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, it expands all the nets.
- **Collapse**: Collapses the selection in the tree if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, it expands all the nets.
- **Deselect All Nets**: Deselects all the entities present in the project. Clears the selection of all the previously selected entities.
- **Clear Filter**: Clears all the filtering applied on nets and shows all the nets.
- **Hide Internal Layout Nets**: Hides all the nets that are internal to layout component

Along with the “filter by entities” and the “filter by connections” menu items, you can also filter nets using text input. The text box for this functionality is shown below:

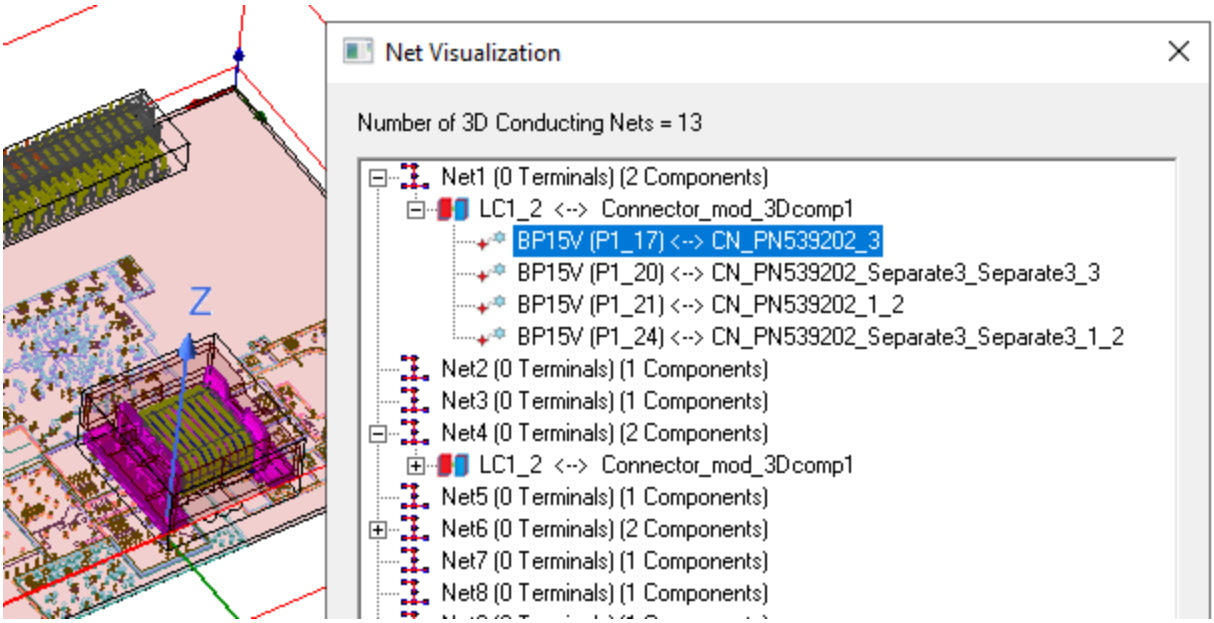
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Features of this text box are

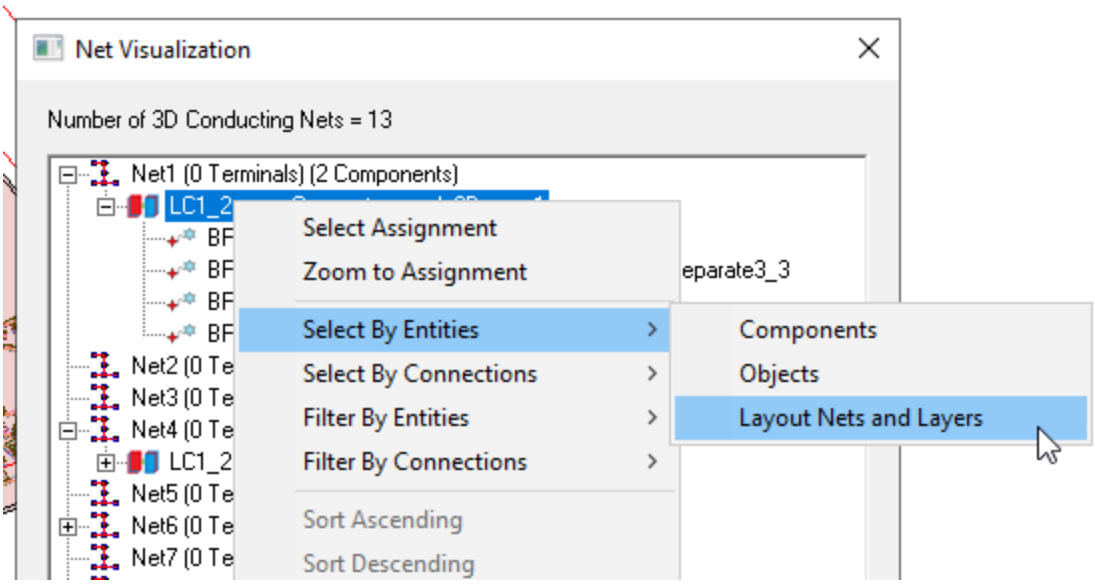
- **Select Assignment**: If you right-click over selection in the tree, selects entities present in the selection. If the right-click menu is invoked on the surrounding blank space, this menu item is disabled as there is no selection.
- **Zoom to Assignment**: If you right-click over a selection in the tree, zooms to the entities present in the selection. If you right-click on the surrounding blank space, this menu item is disabled as there is no selection.
- **Select by Entities**: Invokes a dialog box containing object, component, or net and layer names present in the selection if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, all object or component names present in the project are displayed. Selecting one or multiple of these object or component names selects all the nets that contain any of the selected entities.
- **Select by Connections**: If you right-click over some selection in the tree, this produces a dialog box containing object or component names present in the selection. If you right-click in the surrounding blank space, a dialog box with all object or component names present in the project displays. Selecting a pair of these objects or component names selects all the nets that contain this pair of entities, signifying a direct or indirect connection between these entities.
- **Filter by Entities**: A new dialog box is invoked containing object, component, or net and layer names present in the selection if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, all object or component names present in the project are displayed. Selecting one or multiple of these objects or component names filters all the nets that contain any of the selected entities.
- **Filter by Connections**: A new dialog box is invoked containing object or component names present in the selection if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, all object or component names present in the project are displayed. Selecting a pair of these objects or component names filters all the nets that contain this pair of entities, signifying a direct or indirect connection between these entities.
- **Sort Ascending**: Sorts the nets in ascending order of their names.
- **Sort Descending**: Sorts the nets in descending order of their names.
- **Expand**: Expands the selection in the tree if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, it expands all the nets.
- **Collapse**: Collapses the selection in the tree if the right-click menu is invoked over some selection in the tree. If the right-click menu is invoked on the surrounding blank space, it expands all the nets.
- **Deselect All Nets**: Deselects all the entities present in the project. Clears the selection of all the previously selected entities.
- **Clear Filter**: Clears all the filtering applied on nets and shows all the nets.
- **Hide Internal Layout Nets**: Hides all the nets that are internal to layout component

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting any component or net in the **Net Visualization** list highlights it. For example, the following figure shows a Layout Component selected in the expanded tree list and highlighted in the **Modeler** window.

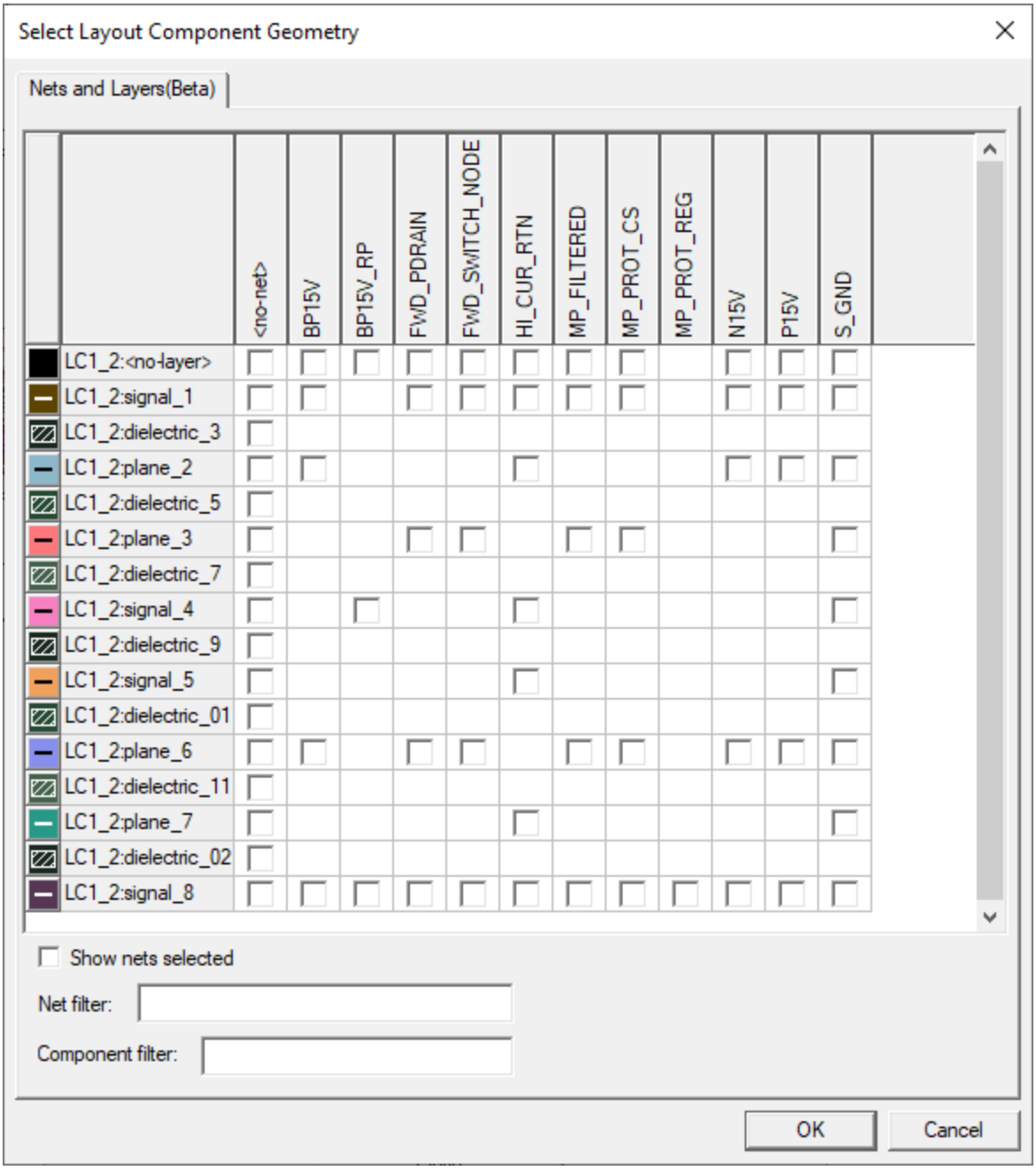


For example, right-click on the Layout Component name in the **Net Visualization** dialog box to display the menu and select **Select by Entities** > **Layout Nets and Layers**.



This displays the **Select Layout Component Geometry** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



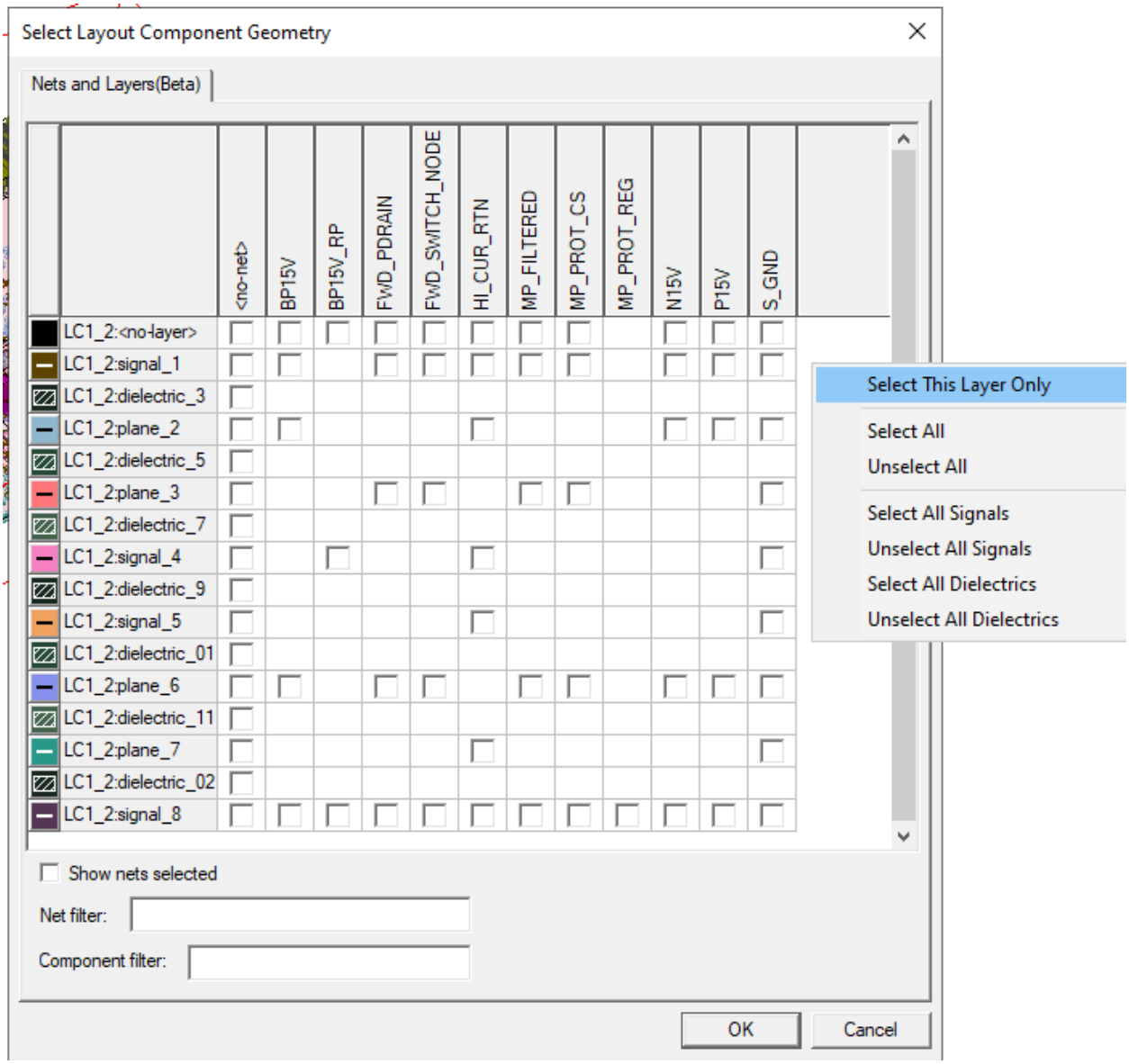
With the **Select Layout Component Geometry** dialog box, you can select objects from layout components' local nets and layers and search for nets in the overall design that contain these objects. This dialog displays all the layers and local nets of every layout component in the design. If there are objects in the layout component that belong to a specific layer and net, a checkbox will become available for you to select. Once you make selections and click **Apply**, you can go back to the parent **Net Visualization** dialog box and click **Select** button, which selects all the nets in the **Net Visualization** dialog that contain these objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Select Layout Component Geometry** dialog is modeless, that means you can work in the 3D Modeler while keeping this dialog box open. This way, you can use the modeler window to better find the layout component objects you want to search for.

By clicking the column headers, you can select or deselect checkmboxes in the entire column. The **Show nets selected** check box can be toggled to move all the selected nets to the left side of the table. Use the Net filter and Component filter to filter fields to designate specific nets and components. These filters support '*' and '?' wildcards and can delimit multiple filters with comma. The table is refreshed each time users update the filter text.

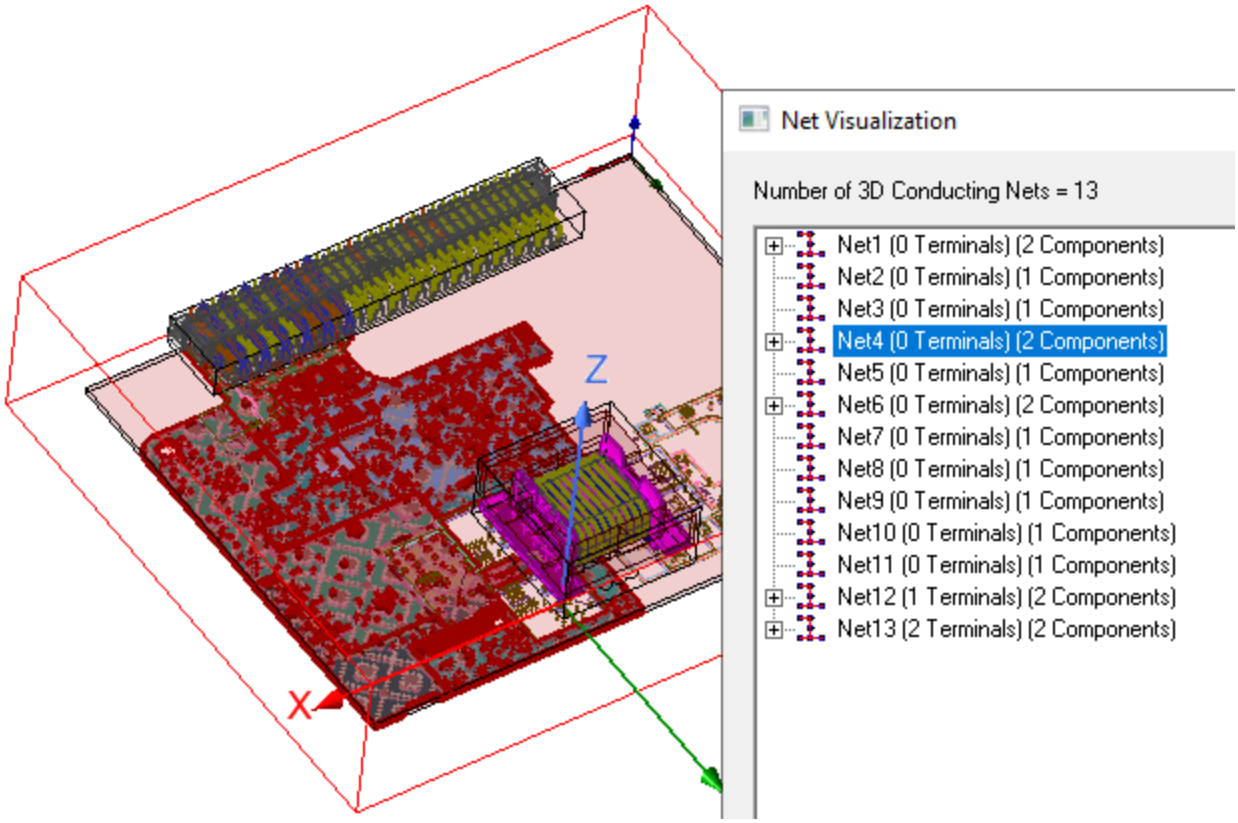
Additionally, the **Select Layout Component Geometry** dialog box has a right-click menu as shown below.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

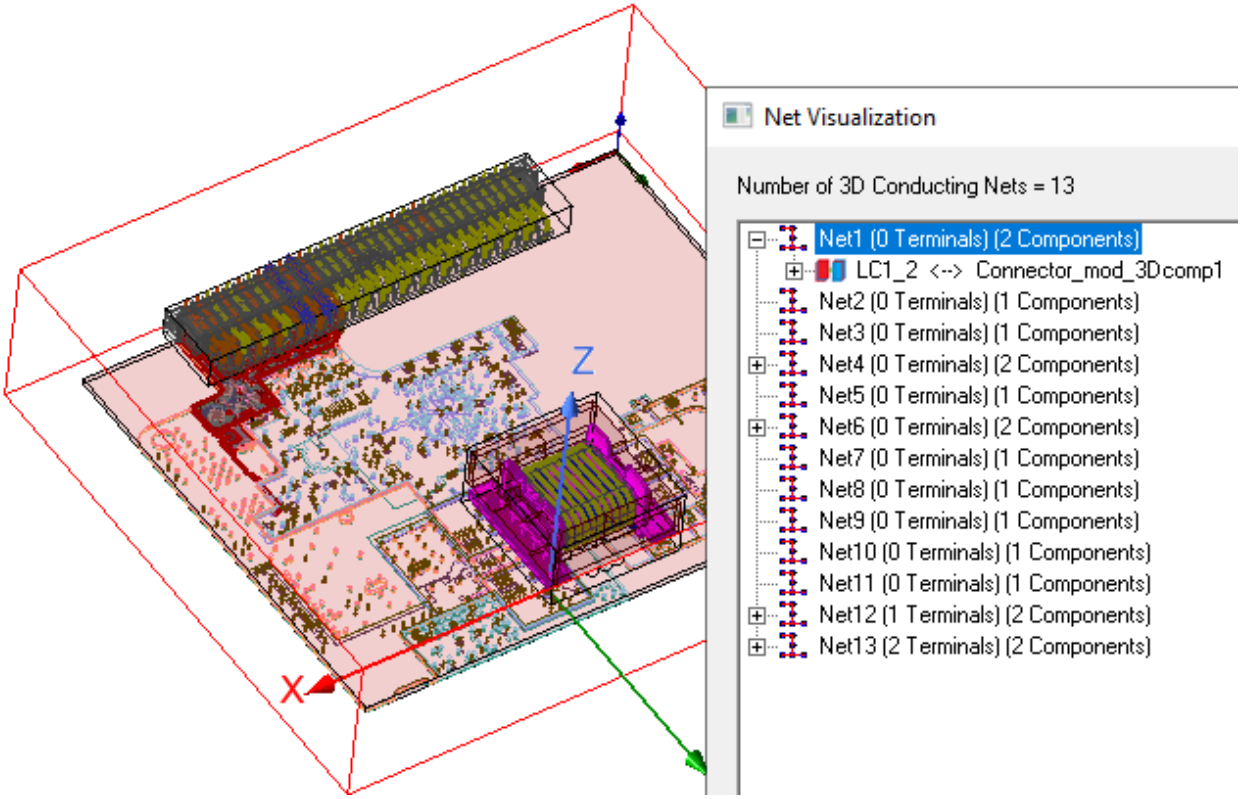
You can use the dialog to select any net or terminal. You can continue to work in the Modeler window with the dialog box open.

The **Net Visualization** dialog box identifies all nets and will include both 3D geometries and layout components with correct terminal associations. As shown in the figure below, any net you select in the Net Visualization list highlights the objects in the net in the modeler window. By default, 3D geometries are highlighted with blue texture, and objects from layout components will be highlighted with red texture. The rest of objects in the layout component not belonging to the selected net will be shown as wireframes.



The next figure shows a different selection from the list of 3D conducting nets to illustrate the changed highlighting.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



All the functionalities in the Net Visualization dialog that works for regular nets are also supported for nets containing layout component objects, such as **Zoom To**. Because layout components do not have any actual geometries in the modeler window, the **Select Assignment** will only select 3D geometries in each net.

Define an Integration Line

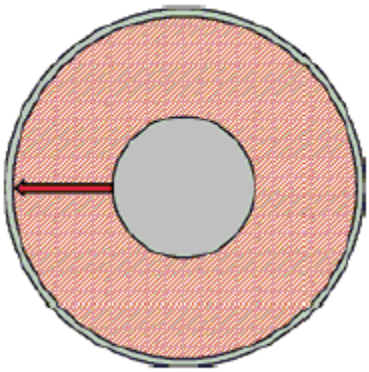
An integration line is a vector that can represent the following:

- A calibration line that specifies the direction of the excitation field pattern at a port. If you are analyzing more than one mode at a port, define a separate integration line for each mode; the orientation of the electric field differs from mode to mode.
- An line along which to integrate **E**.dl to compute a voltage for Zpv or Zvi impedance of a port. In this case, select two points at which the voltage differential is expected to be at a maximum. For example, on a microstrip port, place one point in the center of the microstrip, and the other directly underneath it on the ground plane. In a rectangular waveguide, place the two points in the center of the longer sides.

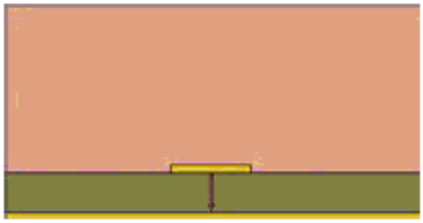
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

For more information, see Wave Port Dialog For Modal Solutions. For definitions of how HFSS defines these Zpv and Zvi values, see Calculating the PV Impedance, and Calculating the VI Impedance.



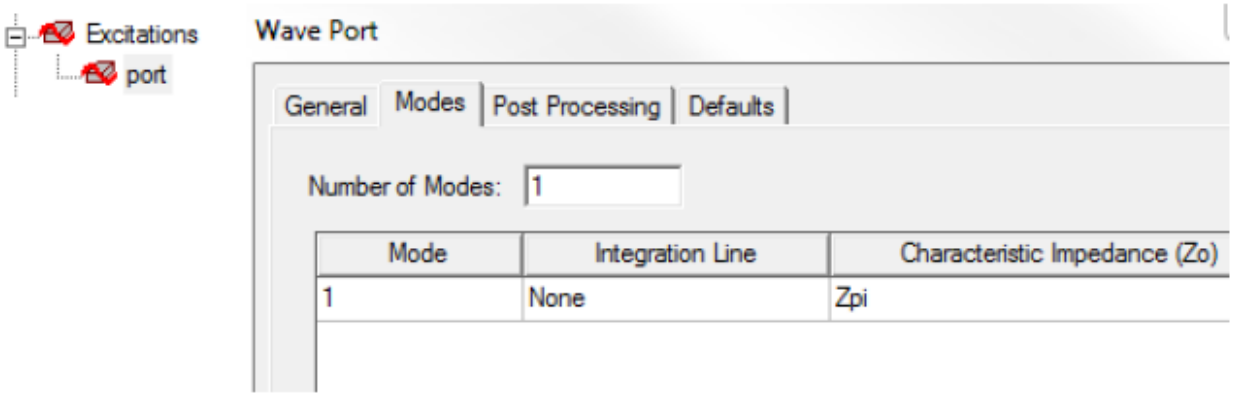
Coaxial cable wave port showing integration line (in red). Line is drawn between points of maximum potential difference. In this case, it is drawn in the space between the center conductor and the shield.



Microstrip wave port showing integration line (in red). Line is drawn between points of maximum potential difference. In this case, it is drawn in the space between the signal line conductor and the "ground" plane.

To define an integration line:

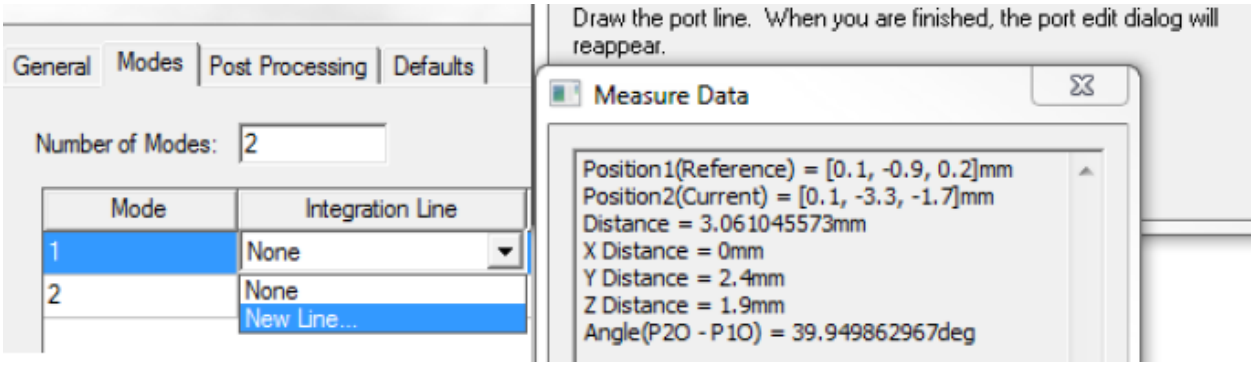
1. Double-click the port excitation from the project tree to bring up the **Wave Port** dialog box and click the **Modes** tab.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. From the **Integration Line** column, select **New Line**.

   The port dialog box disappears. If the **Show Measure** dialog option on the Modeler Options: Drawing tab is selected, the **Measure Data** dialog appears when you draw the vector.



3. Use the **Measure Data** dialog to locate the start and end points and draw the integration line.

> **Note:**
>
> The **Measure Data** dialog displays data for the face area, and the positions for the reference point (start point) and end point (end point) as you define them.

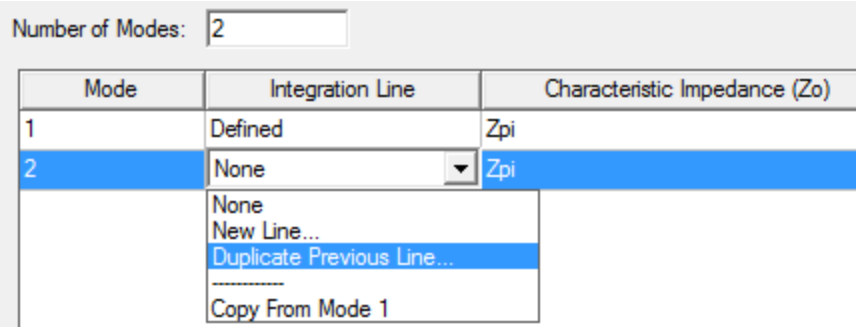
Duplicating Integration Lines

You can duplicate an integration line along a vector multiple times and assign to additional modes at the port. This section shows how to duplicate the integration line in the following figure.

1. In the **Wave Port** dialog box, click the **Modes** tab and draw the integration line 1 on the port.

2. Enter the number of modes and select **Duplicate Previous Line** from the Integration Line column.

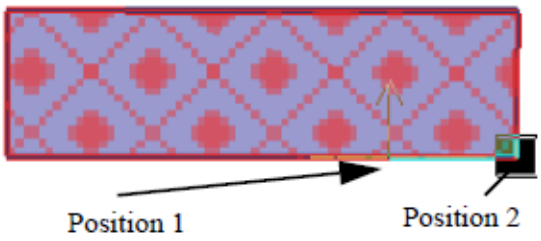
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

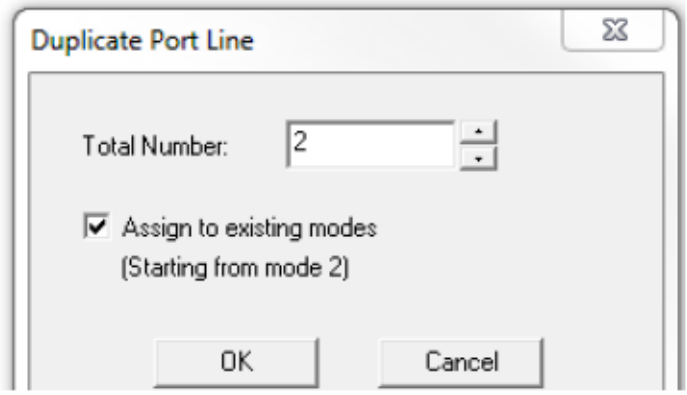


3. Click an arbitrary anchor point on the edge of the port face for position 1 and drag the cursor along the edge to position 2 as shown in the figures below.

   The **Integration Line 2** shifts from **Integration Line 1** by a distance equal to that between Position 1 and Position 2.

   > **Note:**
   >
   > Use the **Measure Data** dialog to set this distance.





4. Enter the total number of lines, including the original and duplicates, to make in the **Duplicate Port Line**.

   If you type a value that is greater than the number of assigned modes, the extra duplicates will appear as gray integration lines until they are assigned to a mode.

5. Optionally, select **Assign to existing modes**. The duplicates will be assigned to the modes defined for the port, beginning with the mode after the one with the line that was duplicated.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modifying Integration Lines

Modify an existing integration line under the **Modes** tab in the **Wave Port** or **Lumped Port** dialog boxes.

To *swap the coordinates* of an integration line's start point and endpoints:

- Select **Swap Endpoints** from the mode's **Integration Line** list.

The line's direction will be reversed.

To *copy* a previously defined Wave Port integration line's points:

- Select **Copy from Mode*n*** from the mode's **Integration Line** list.

The new integration line will have the same start and endpoints as the selected mode's integration line.

To *delete* a defined integration line for a mode:

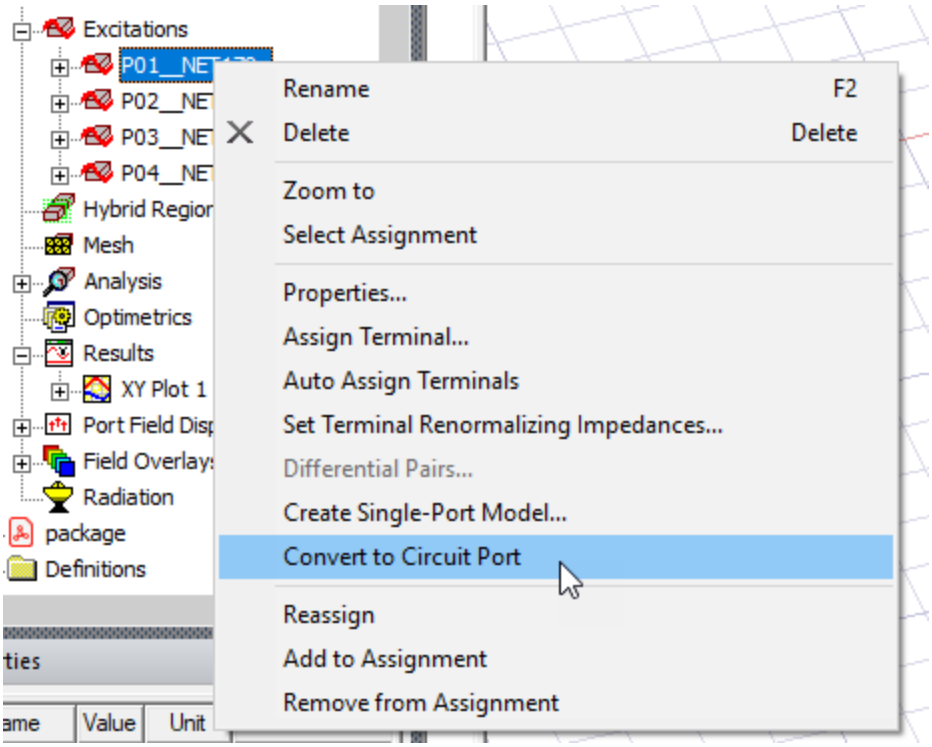
Select **None** from the mode's **Integration Line** list.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

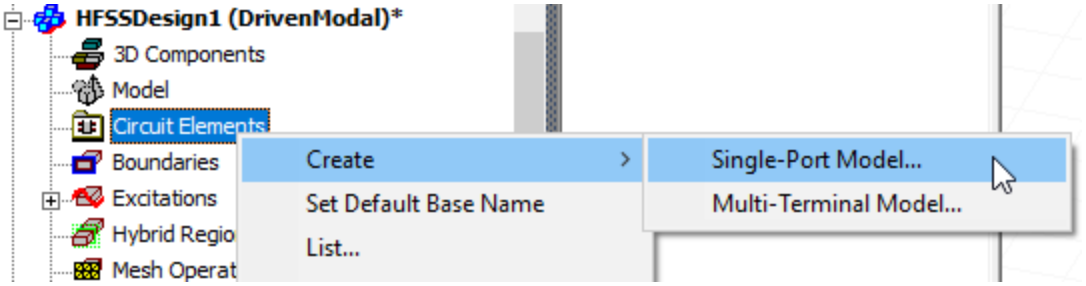
20 - Circuit Elements in HFSS

HFSS supports the creation of circuit elements using lumped ports or circuit ports in both driven modal and terminal solution types. Circuit ports are allowed with the DDM solver, including FEBI and IE Hybrid Regions. You can directly convert a selected lump port to a circuit port and from a circuit port to a lump port.

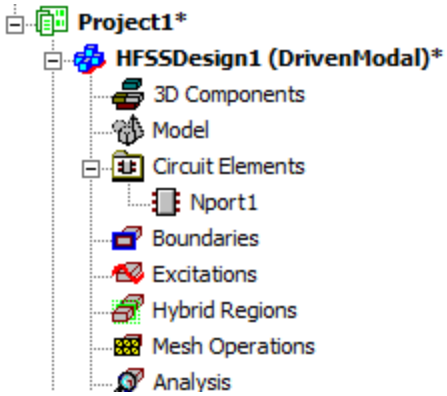


You can import N-port models a from a library of previously created Touchstone files for the S-matrix or link to projects from HFSS, Q3D, 2D Extractor, or SIwave. Within the approach of using touchstone files in HFSS the port inductance is present as added in series to the embedded touchstone file. In case the port inductance deembedding is intended then this needs to be done manually with the touchstone file as this will not be done within HFSS. You can create models as Parametric N-port models or use Nexsys Matlab models. You can also convert Circuit Elements back to lump ports in the HFSS project.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

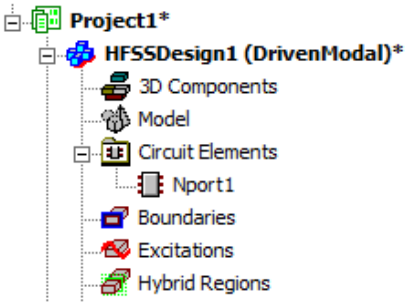


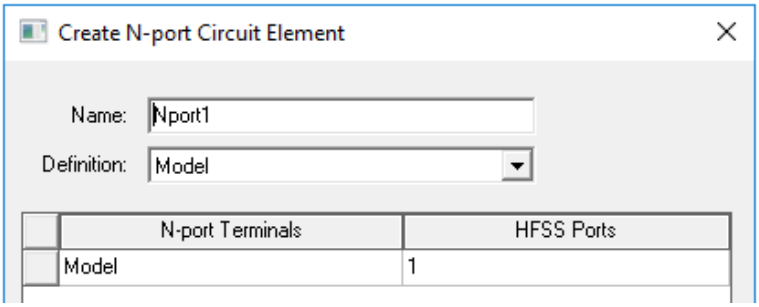
A folder named **Circuit Elements** appears in the **Project tree** where the circuit elements are added.



There are three cases to consider when you create these circuit elements:

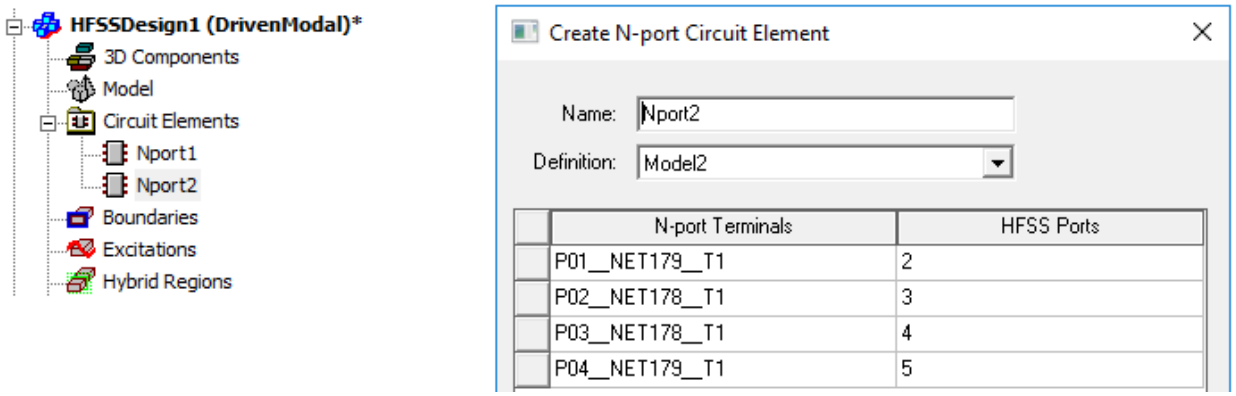
- Single-Port Model definition: A 1x1 or 2x2 S-matrix that requires you to select a single lumped port or circuit port to create the circuit element.



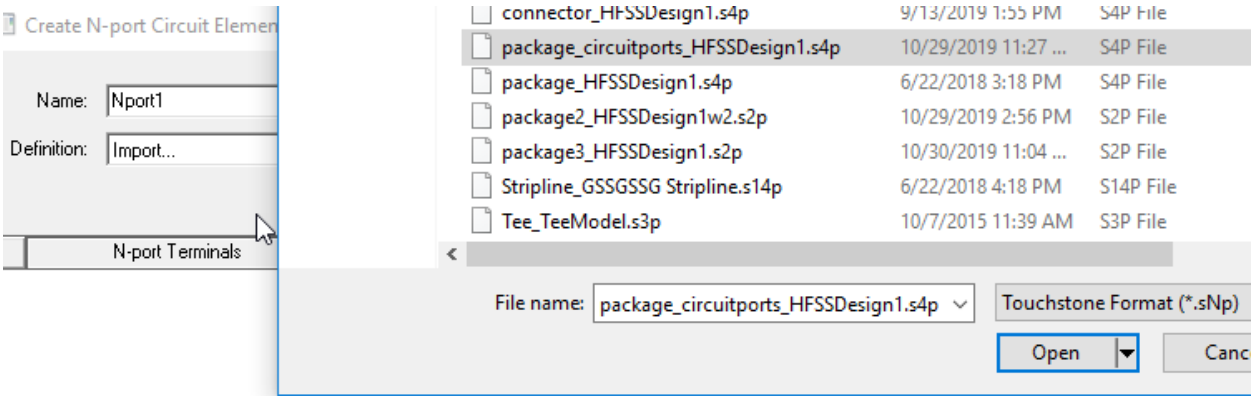


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Multi-Terminal Model definition: An S-matrix 2x2 or higher that would require you to select the corresponding number of lumped ports or circuit ports.

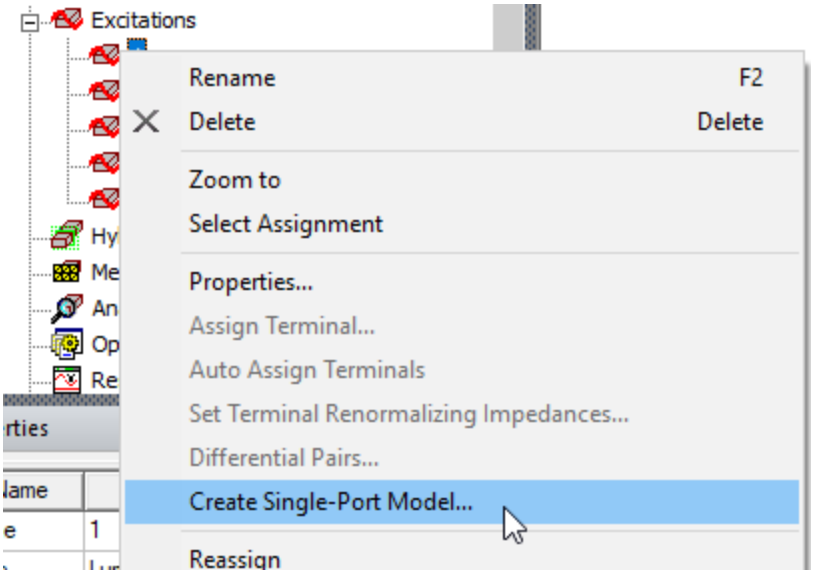


- Touchstone file import for circuit elements.

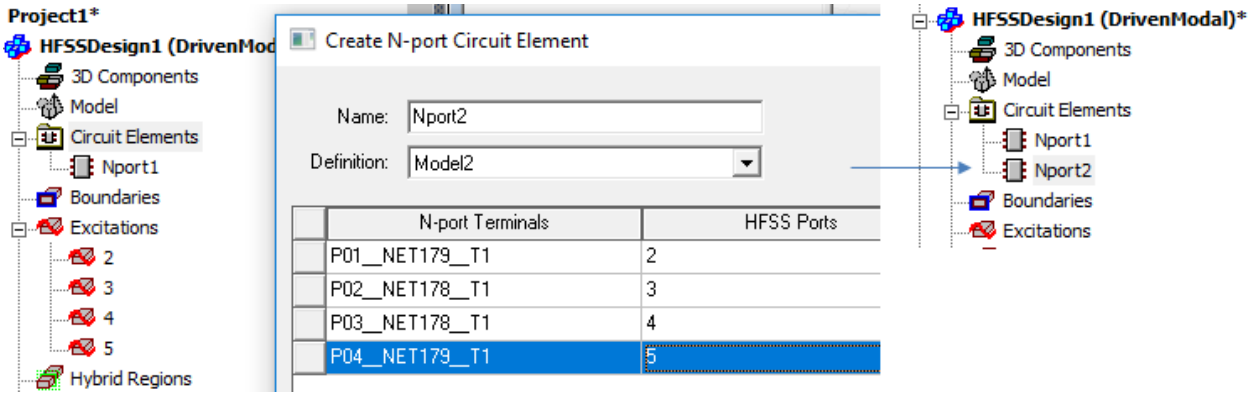


In the case of a Single-Port Model, you can also access the menu by right-clicking on a lump port in the **Project** tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

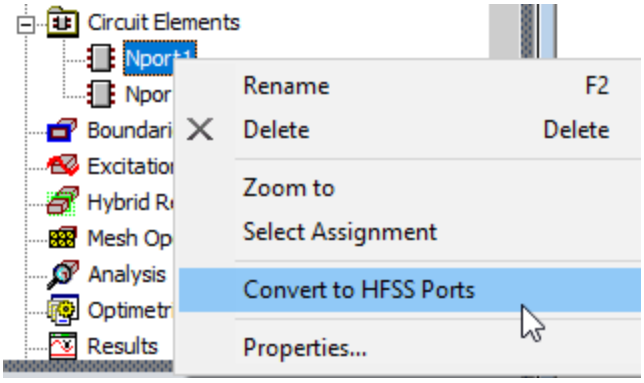


When you change a lump port to a circuit model, the selected port or ports are removed from the **Excitations** list as the models are displayed under the **Circuit Elements** list. The port post processing setting will not be effective once converted to circuit element.

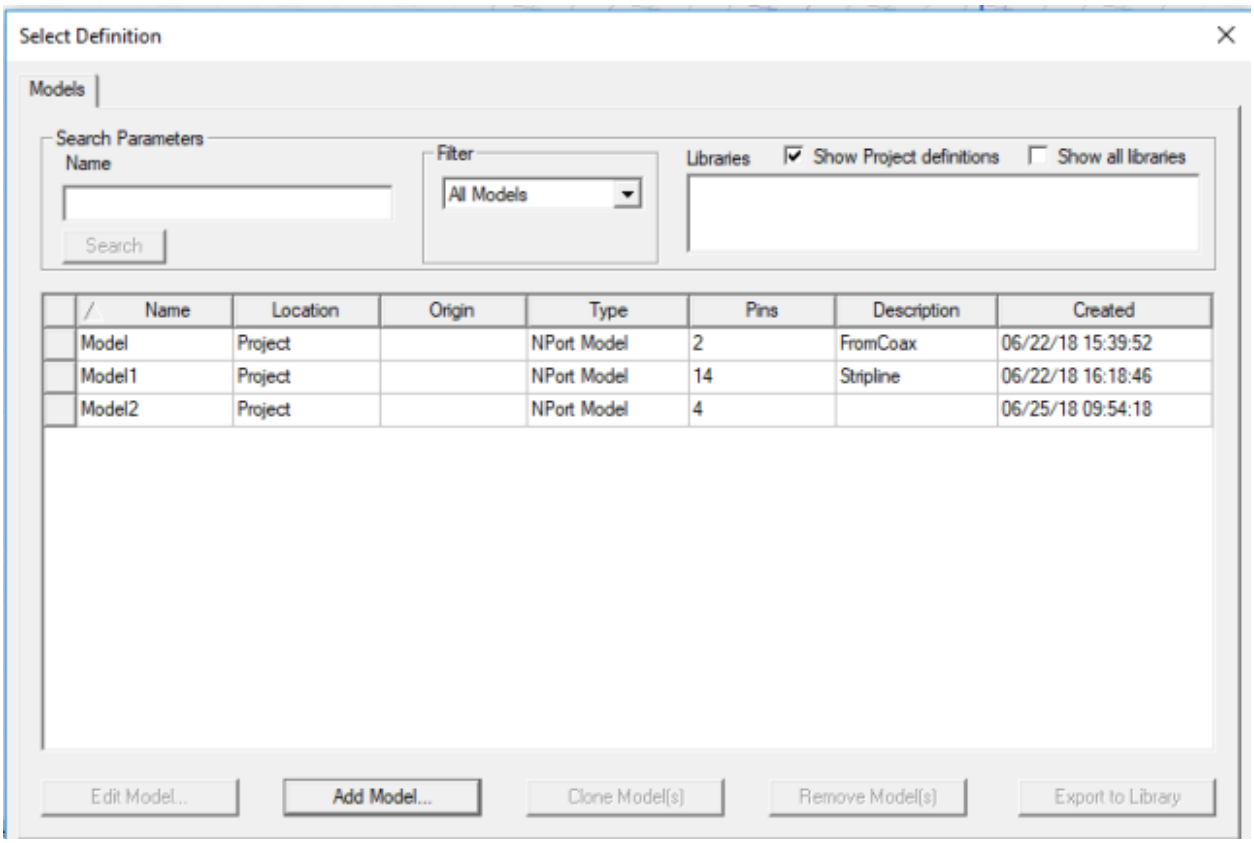


You can change a circuit element to an HFSS port.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also Create Models and Manage Circuit Model Libraries.



Prerequisites/Limitations

- Designs must be Driven Modal or Driven Terminal.
- Existing lump ports or circuit ports can be changed to circuit elements.
- Only passive S-parameter model is supported.
- Fast sweep is not supported.

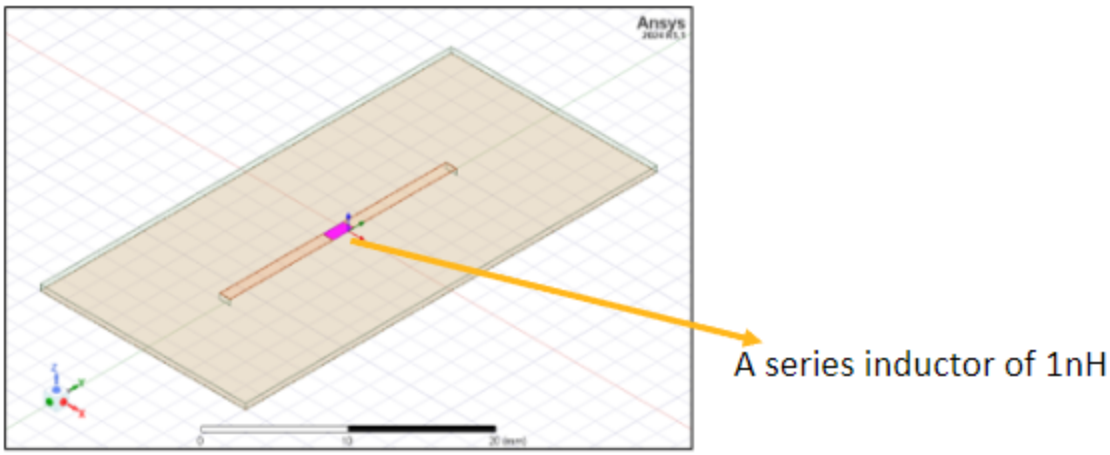
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Sensitivity analysis is not compatible with circuit elements. The solver detects unsupported use and issues a warning and returns zero for the derivatives.
- 3D Components created from these designs include the circuit elements. The Create 3D component dialog has a Circuit Elements tab for these designs.

Beta Feature: De-embedding for Circuit Elements

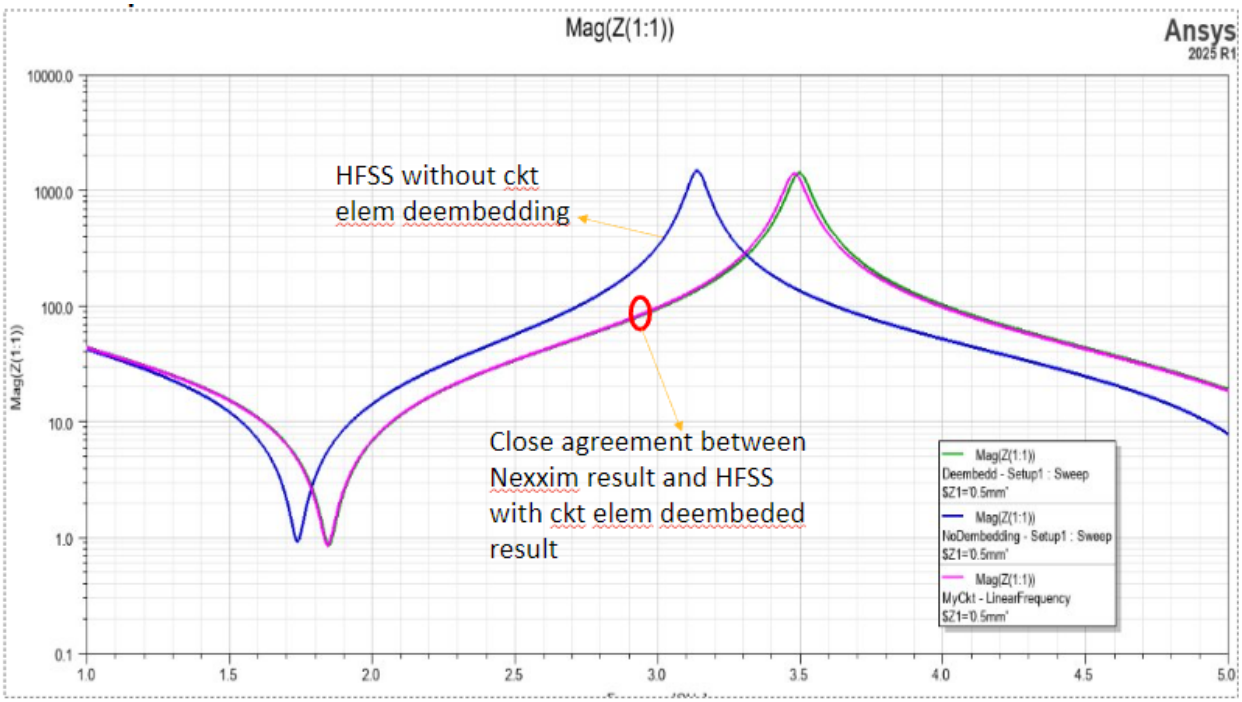
A Beta Feature allows de-embedding for Circuit Elements. For long and narrow circuit elements, the final S-parameters could show some difference. If you enable the HFSS Solver Beta option for HFSS Deembed circuit elements, all of the circuit elements in the geometry will be deembedded.

With the feature enabled, you can solve in HFSS and account for deembedding.



Compared to solving without deembedding for this kind of circuit, HFSS solve with deembedding is comparable to a Nexxim solve.

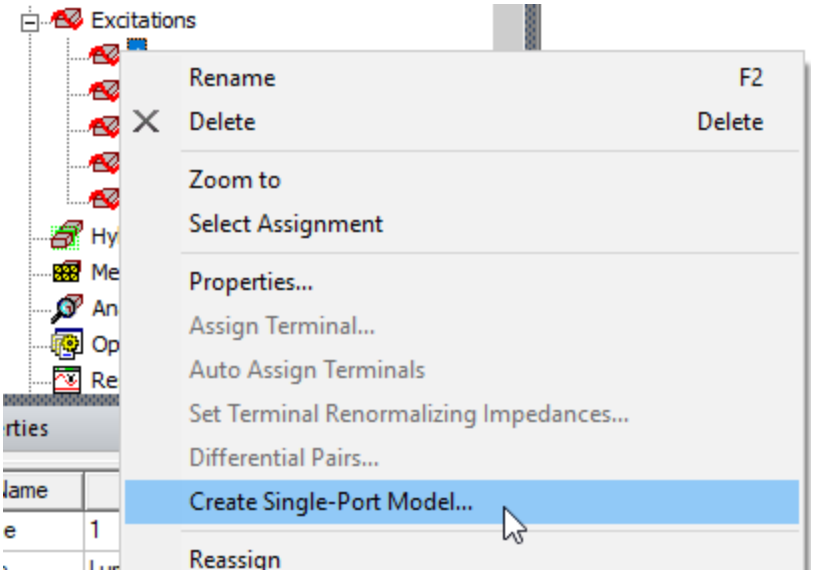
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Creating a Single Port Circuit Element in HFSS

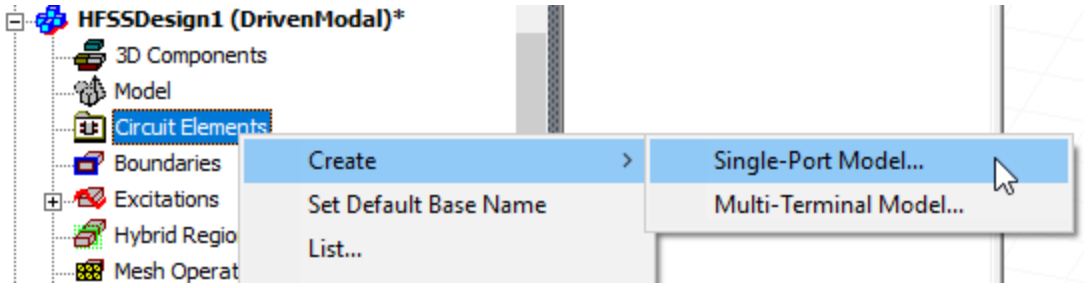
To create a single port Circuit Element:

1. Right click on a Lumped port in the **Project** tree and click **Create Single-Port Model**,

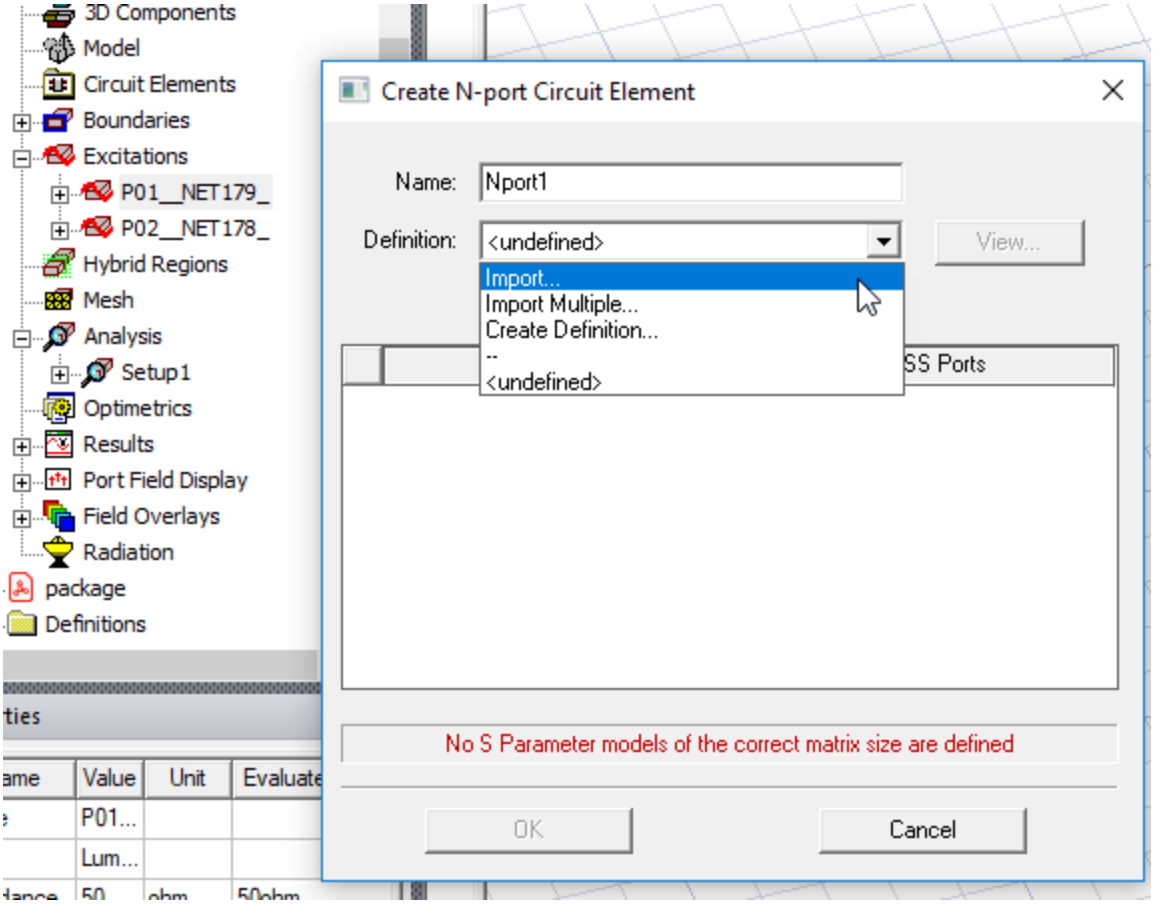


or right click on the **Circuit Elements** folder and select **Create>Single Port Model...**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Create N-port Circuit Element** dialog opens. If you selected the port menu, the dialog shows the HFSS Port assignment. If you selected from the Circuit Element menu, you can click on the HFSS Ports cell to select from available ports.



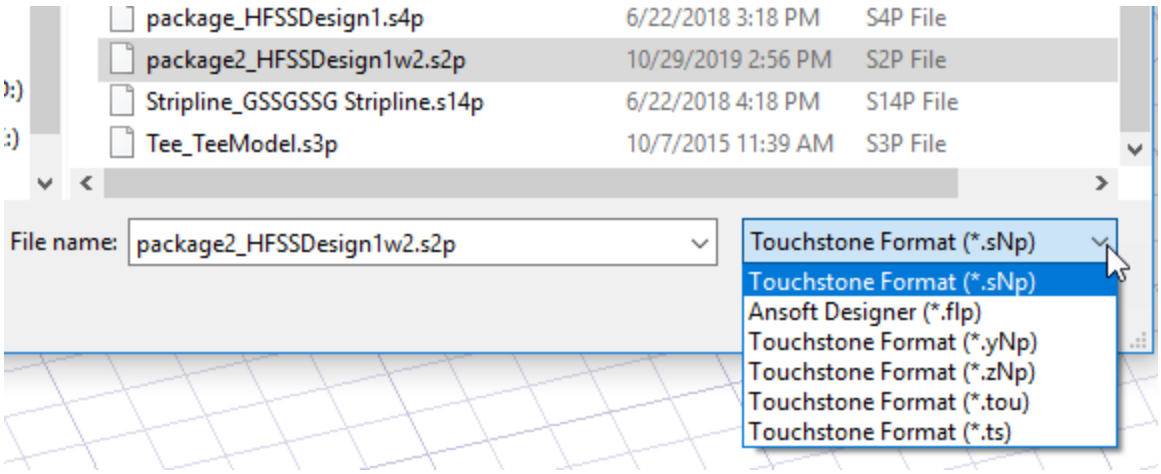
The Definition menu allows you to **Import...** for Touchstone information contained in a single file. The first example follows that choice.

Select "Import Multiple... Option" on page 20-10 if you want have access to multiple Touchstone files Indexed for selectable port definition (discussed below).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

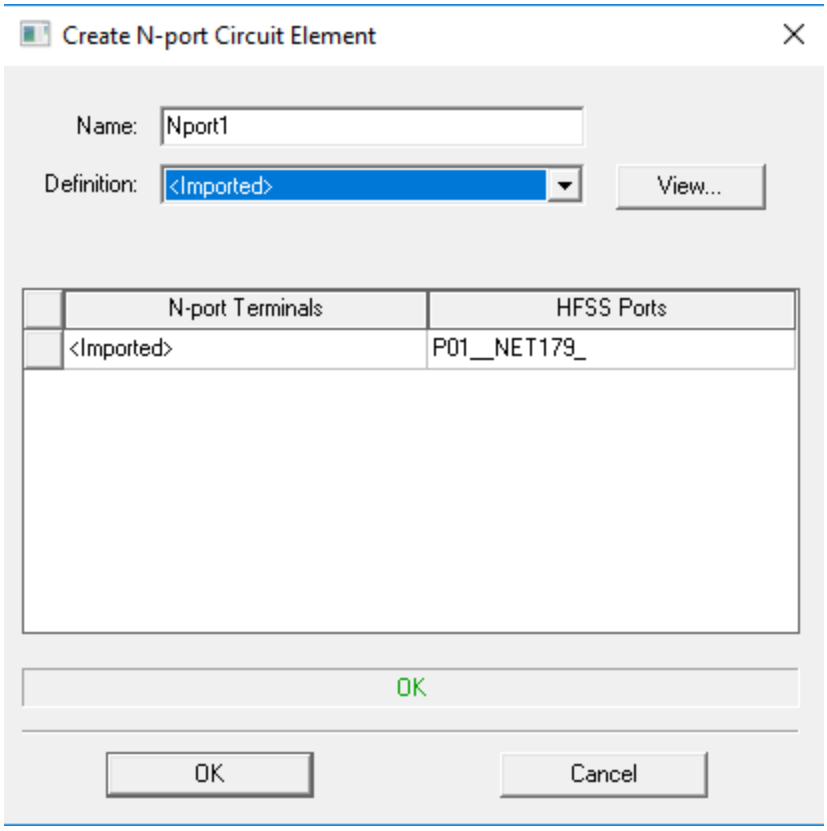
Select Create Definition, or, if you have previously accessed appropriate models, select from models listed. If you select **Create Definition...** you will see the **Select Definition** window where you can select and create circuit models and manage libraries.

If you have previously created Touchstone files of the correct matrix size you can use **Import...** to open a File browser to select one.

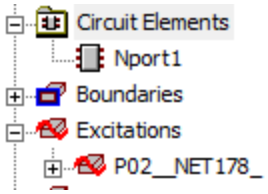


If the model has the corresponding Matrix size, the **Create N Port Circuit Element** dialog can assign the HFSS ports, and shows that the assignment is valid with a green OK. If a valid assignment cannot be made, the field has a red text describing the issue. With valid assignments, the **View** button ins enabled lets you open the port definition in Network Data Explorer. .

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



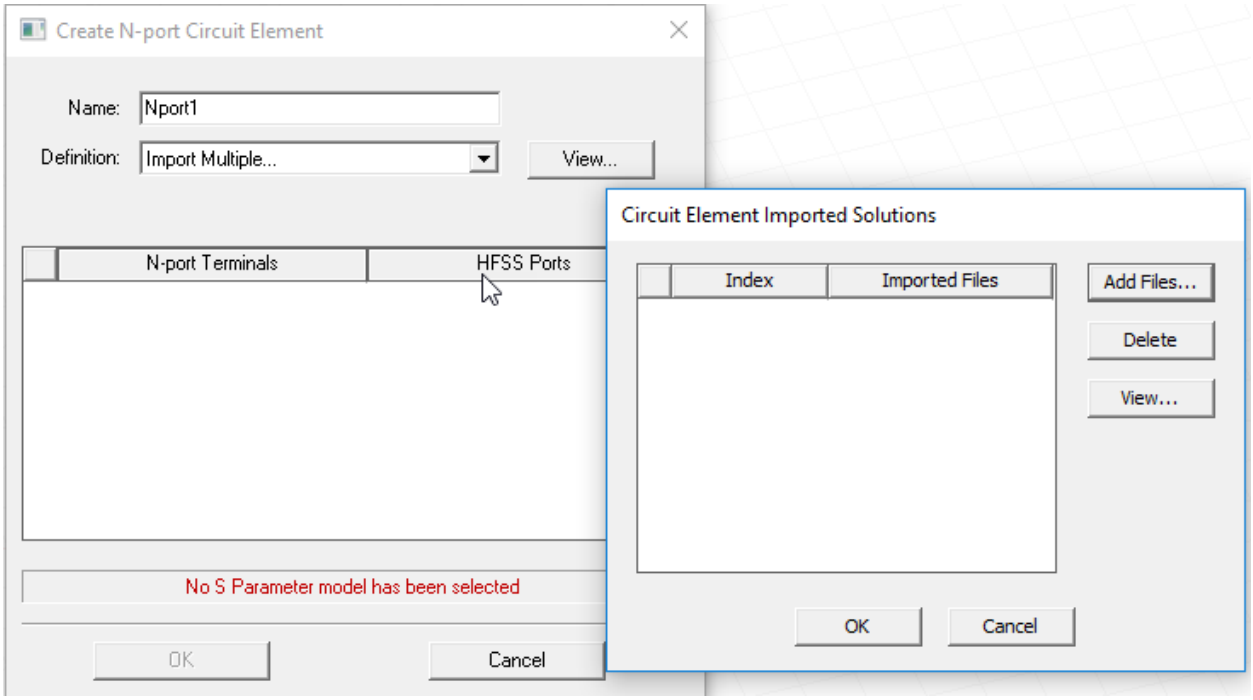
2. You can OK the dialog box, and see the **Circuit Element** listed in the **Project** tree.



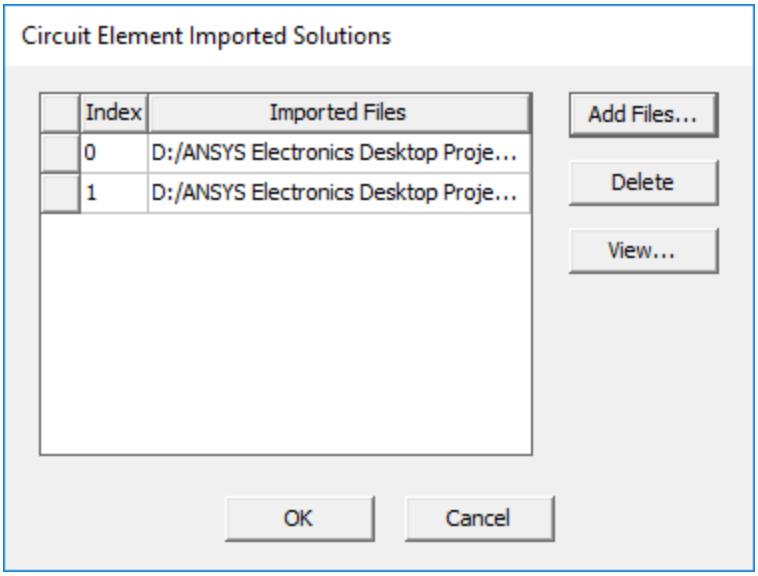
Import Multiple... Option

If you select **Import Multiple...** in the **Create N-port Circuit Element** dialog box, the **Circuit Element Imported Solutions** dialog opens.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

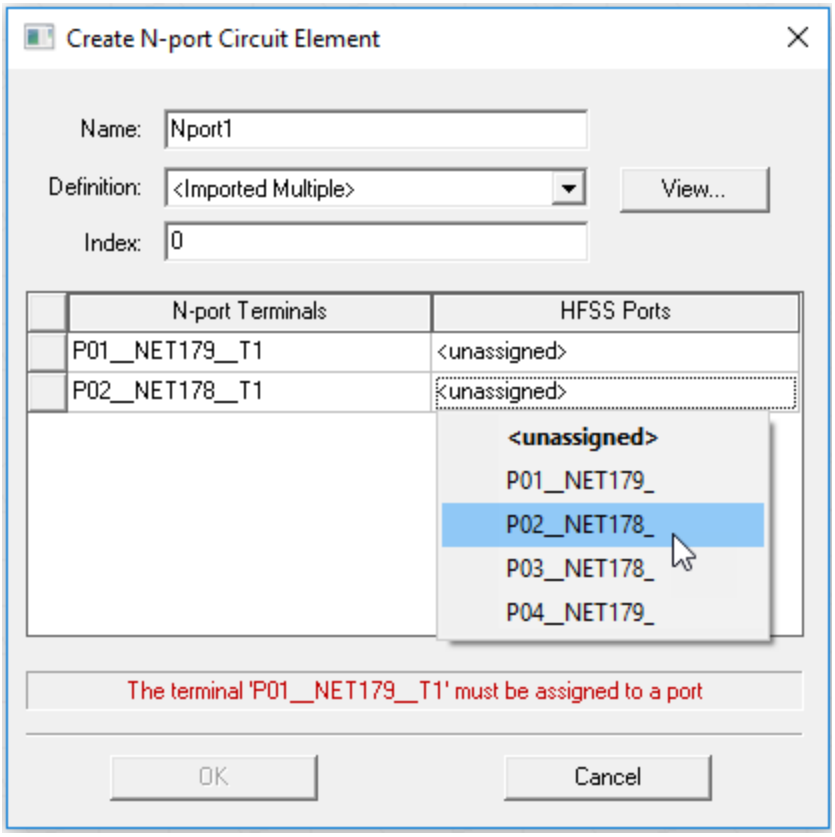


Click **Add Files...** to open a browser to add suitable Touchstone models. When you have added the models they are listed as Imported Files and you can OK the browser. The **View...** button opens the selected file in Network Data Explorer. **Note**: Each file you select has an **Index** assigned here that you can use to identify which file to use for the Circuit Element.



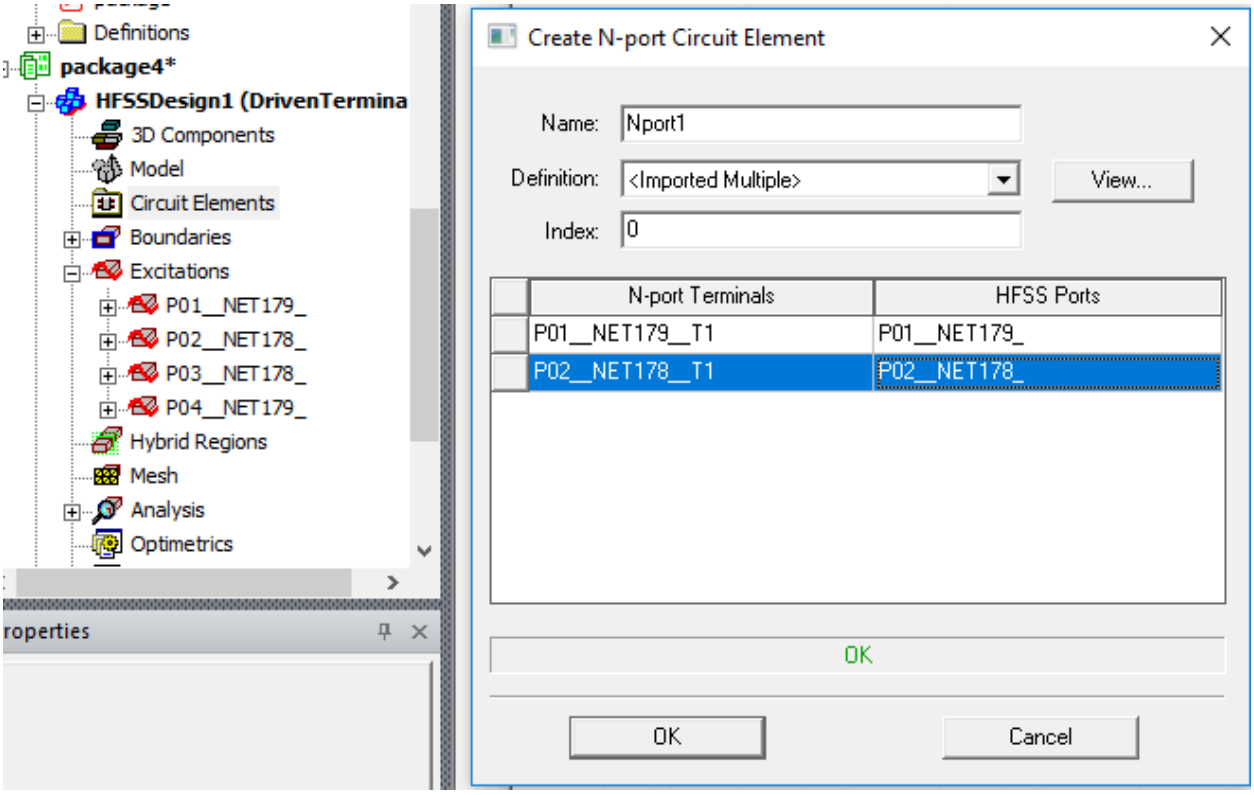
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OK to close the dialog box. You then make HFSS Port assignments in the **Create N-port Circuit Element** dialog box. The **View...** button allows you to use Network Data Explorer for the selected Touchstone file.



When you have made the assignments, the message field reports **OK** in green. The **View...** button allows you to use Network Data Explorer for the selected Touchstone file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

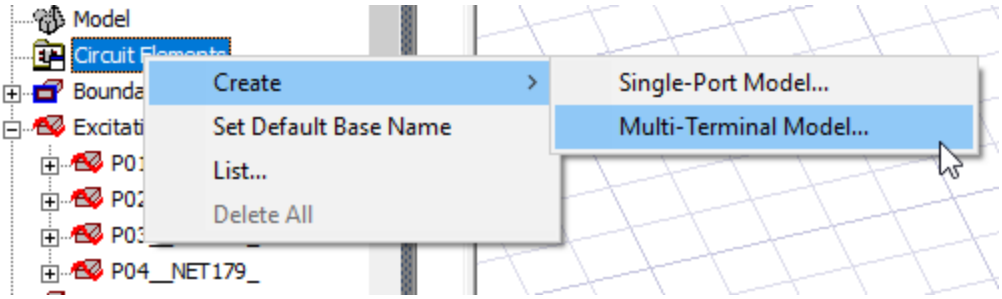


Notice that the **Index** value can control which touchstone file listed in the **Circuit Element Imported Solutions** dialog is used in the simulation by index. The Index value can be parameterized, that is, defined as a variable so to be used in a parametric study or exposed as a component parameter when included in a 3D component.

Creating a Multi-Terminal Circuit Element in HFSS

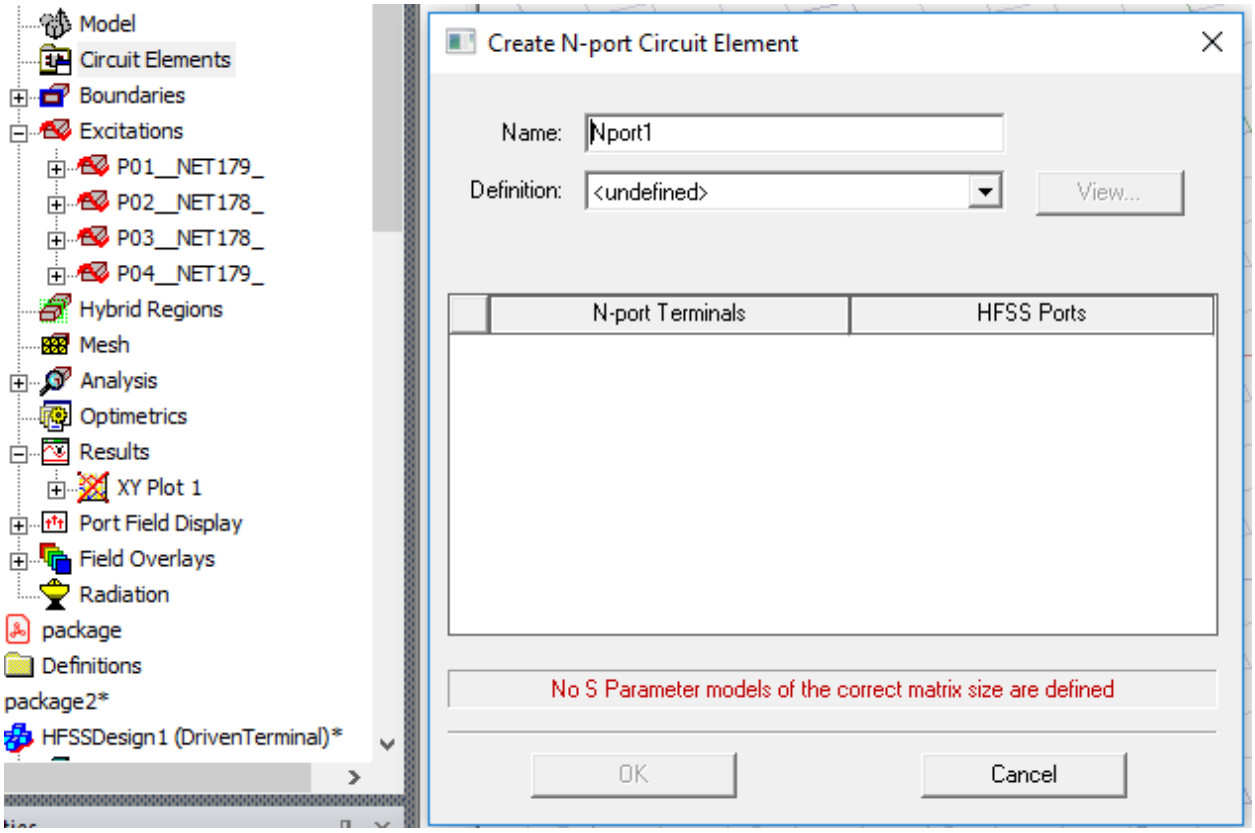
To create a Multi-Terminal model as a circuit element:

1. Right click on the **Circuit Elements** folder and select **Create>Multi-Terminal Model...**.



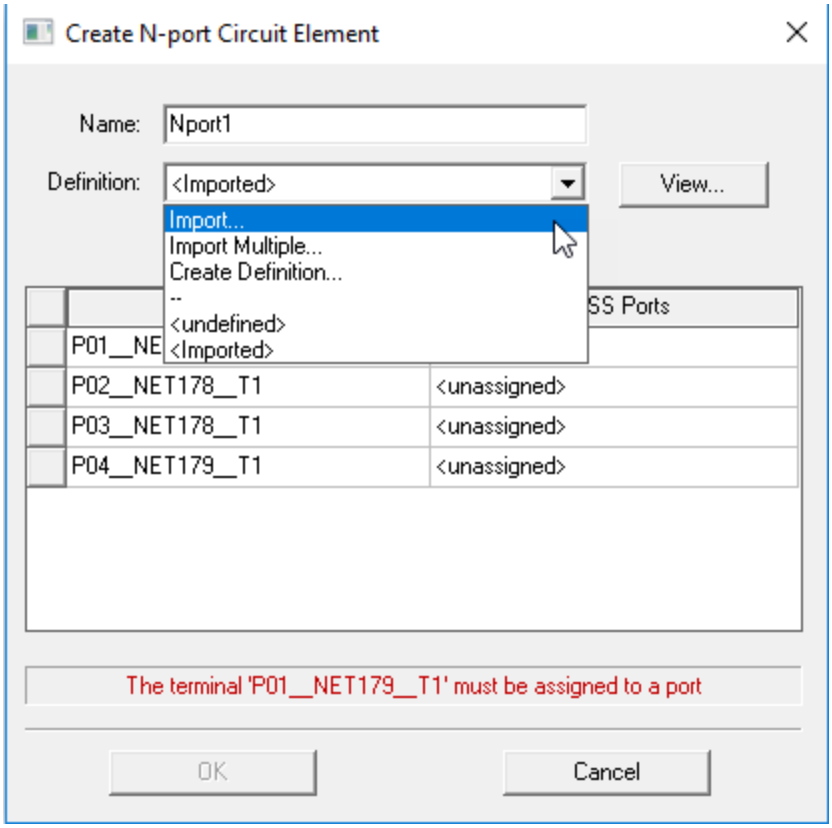
The **Create N-port Circuit Element** dialog opens.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



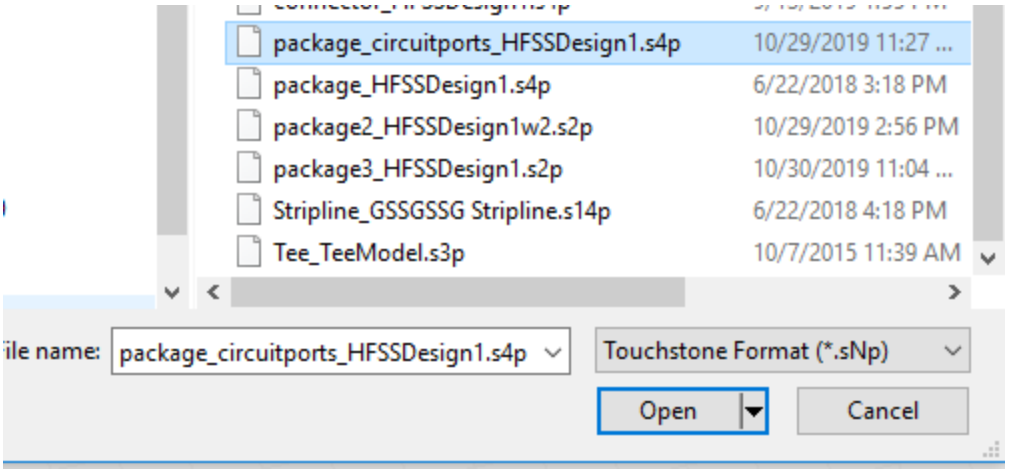
2. Select from the **Definition** pulldown to open a file browser to import a touchstone file containing models of the HFSS ports. The **View...** button allows you to use Network Data Explorer for the selected Touchstone file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



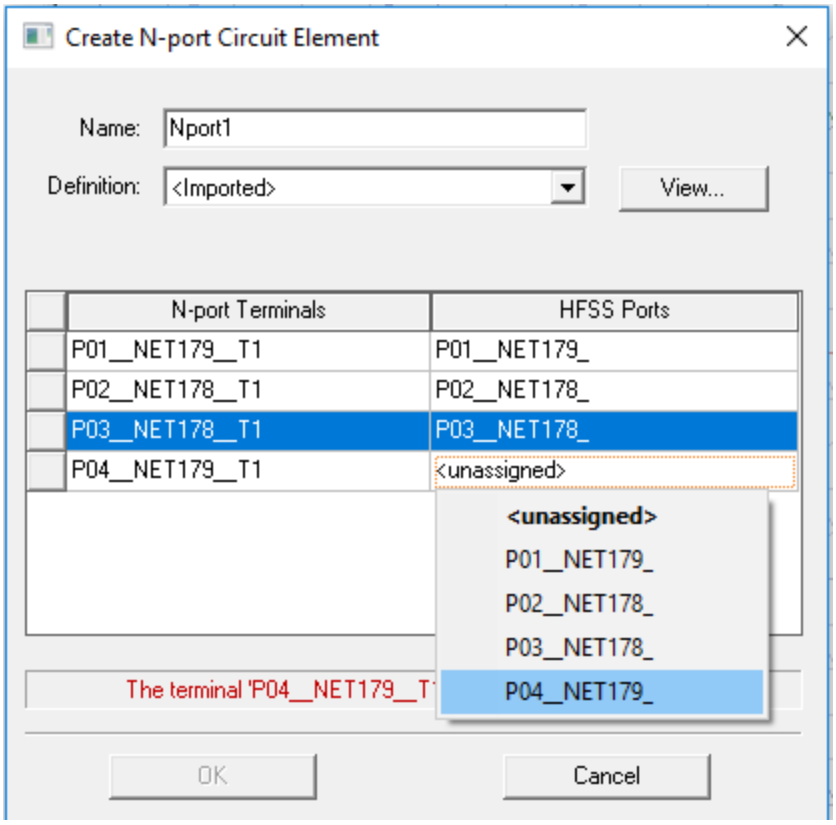
Select **Import...** to import a Touchstone file that contains solution information for all ports. This first example follows this choice. You could also use "Import Multiple... Option" on page 20-17 (discussed below) if you want have access to multiple Touchstone files Indexed for selectable port definition. You can use Create Definition to create and access libraries of Touchstone files.

3. Select an appropriate Touchstone file in the browser, and **OK**. You can select additional formats on the Touchstone dropdown menu.



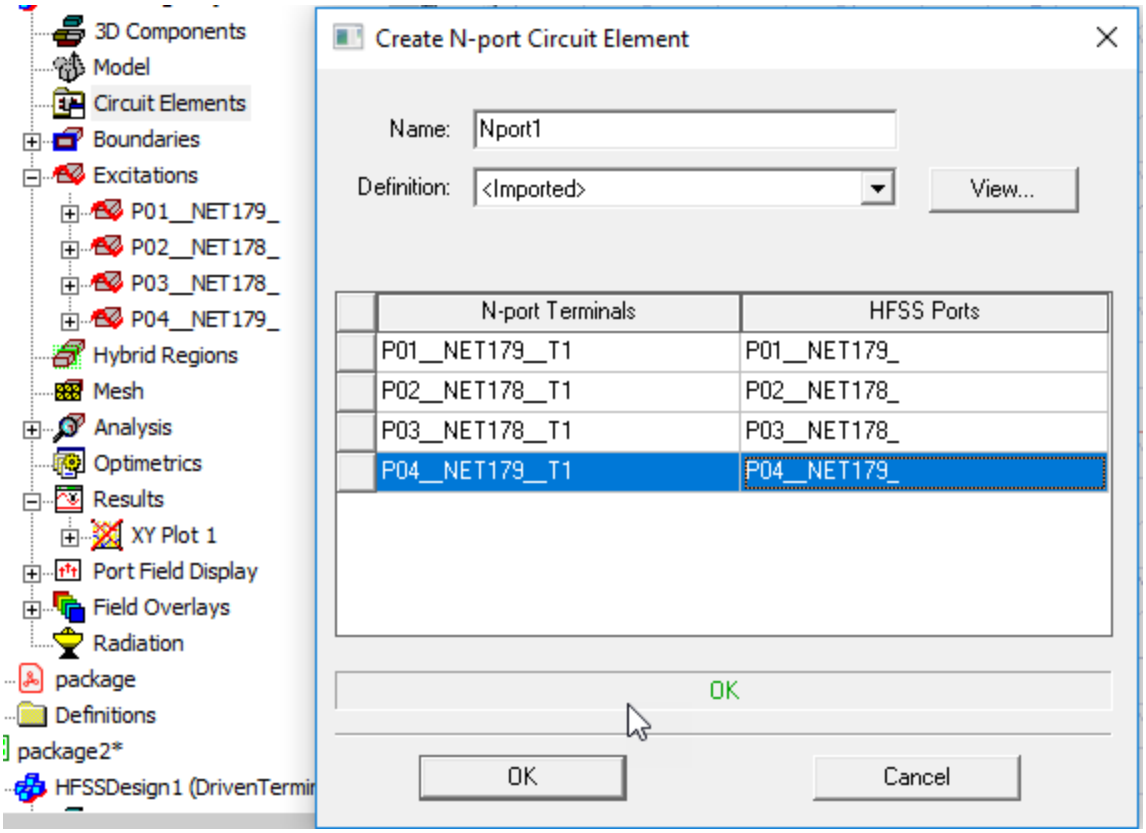
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Use the drop down menu for the HFSS Port rows to make terminal assignments. The **View...** button allows you to use Network Data Explorer for the selected Touchstone file.

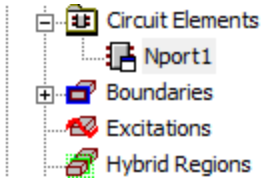


4. If you have selected a suitable model and assigned ports correctly, message field will show a green OK. The **View...** button allows you to use Network Data Explorer for the selected Touchstone file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



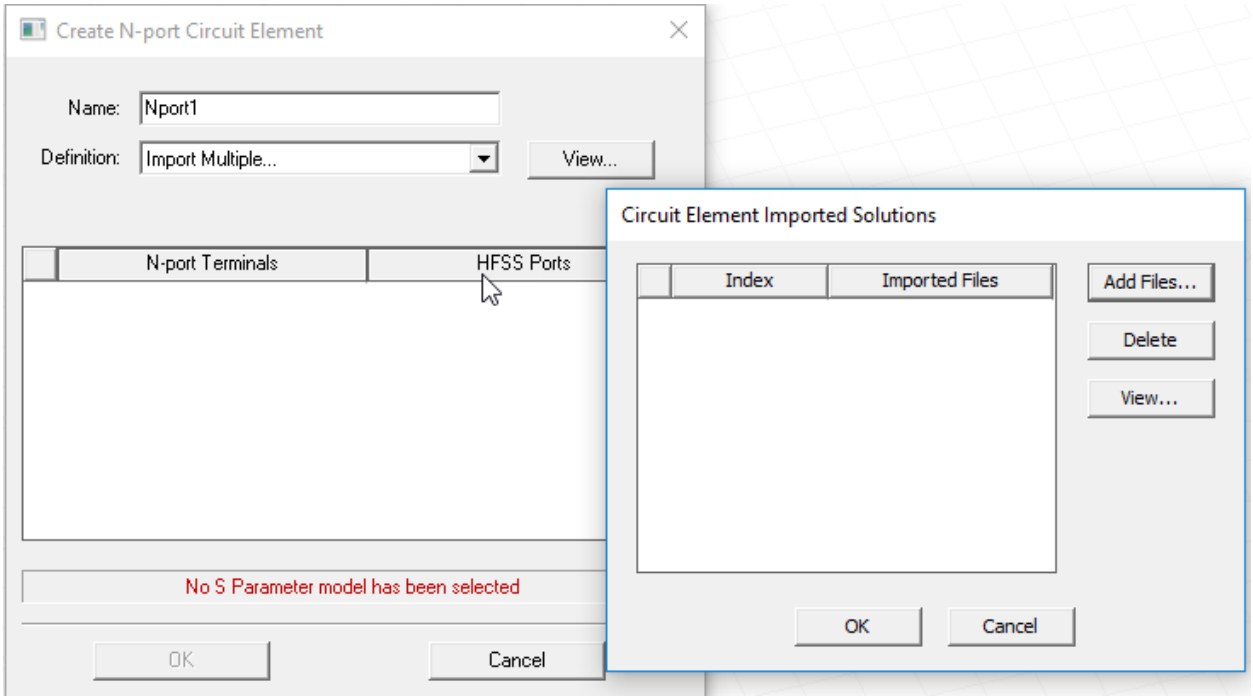
5. You can **OK** the dialog box, and see the **Circuit Element** listed in the **Project** tree and the Ports you selected are removed from the **Excitations** list.



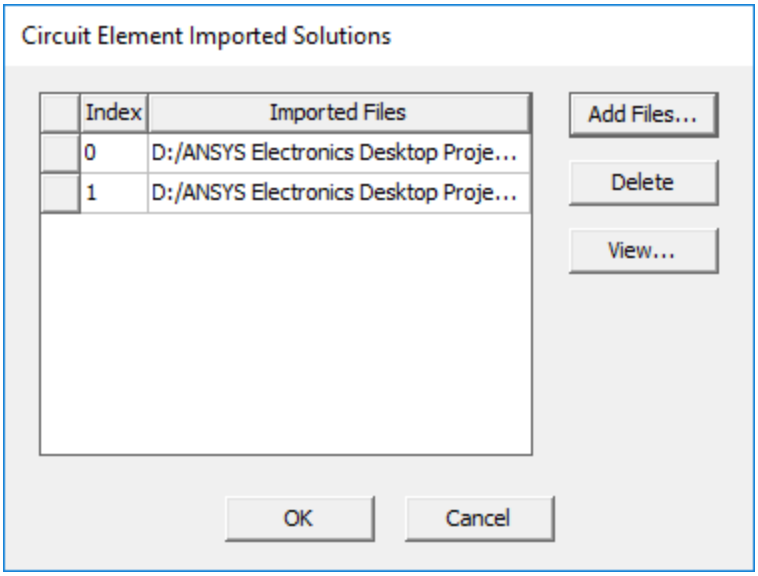
Import Multiple... Option

If you select **Import Multiple...** in the **Create N-port Circuit Element** dialog box, the **Circuit Element Imported Solutions** dialog opens.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

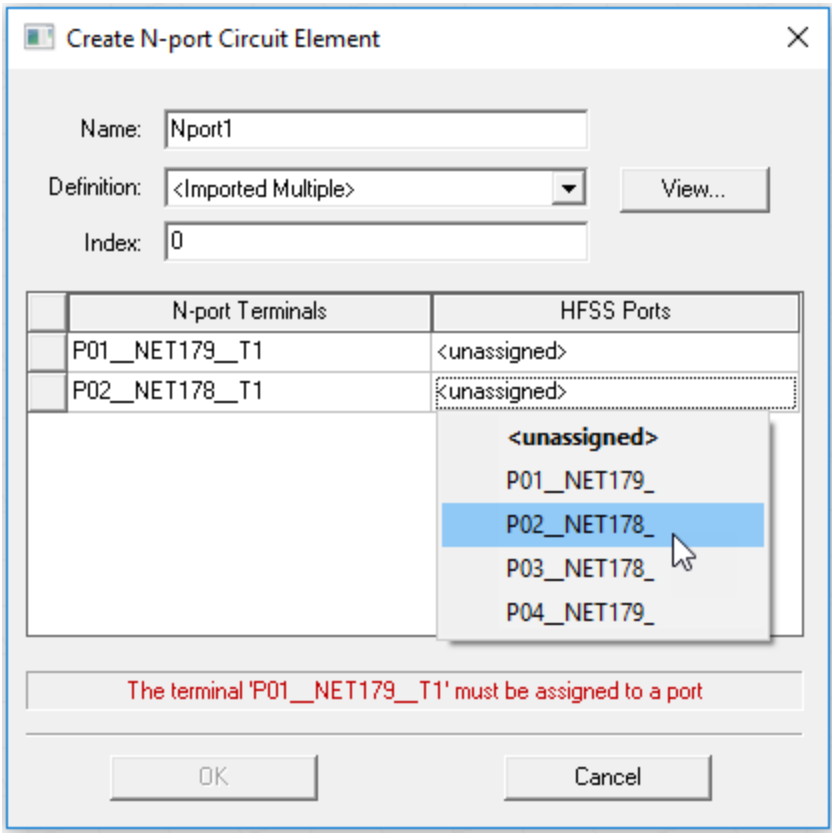


Click **Add Files...** to open a browser to add suitable Touchstone models. When you have added the models they are listed as Imported Files and you can OK the browser. The **View...** button opens the selected file in Network Data Explorer. **Note**: Each file you select has an **Index** assigned here that you can use to identify which file to use for the Circuit Element.



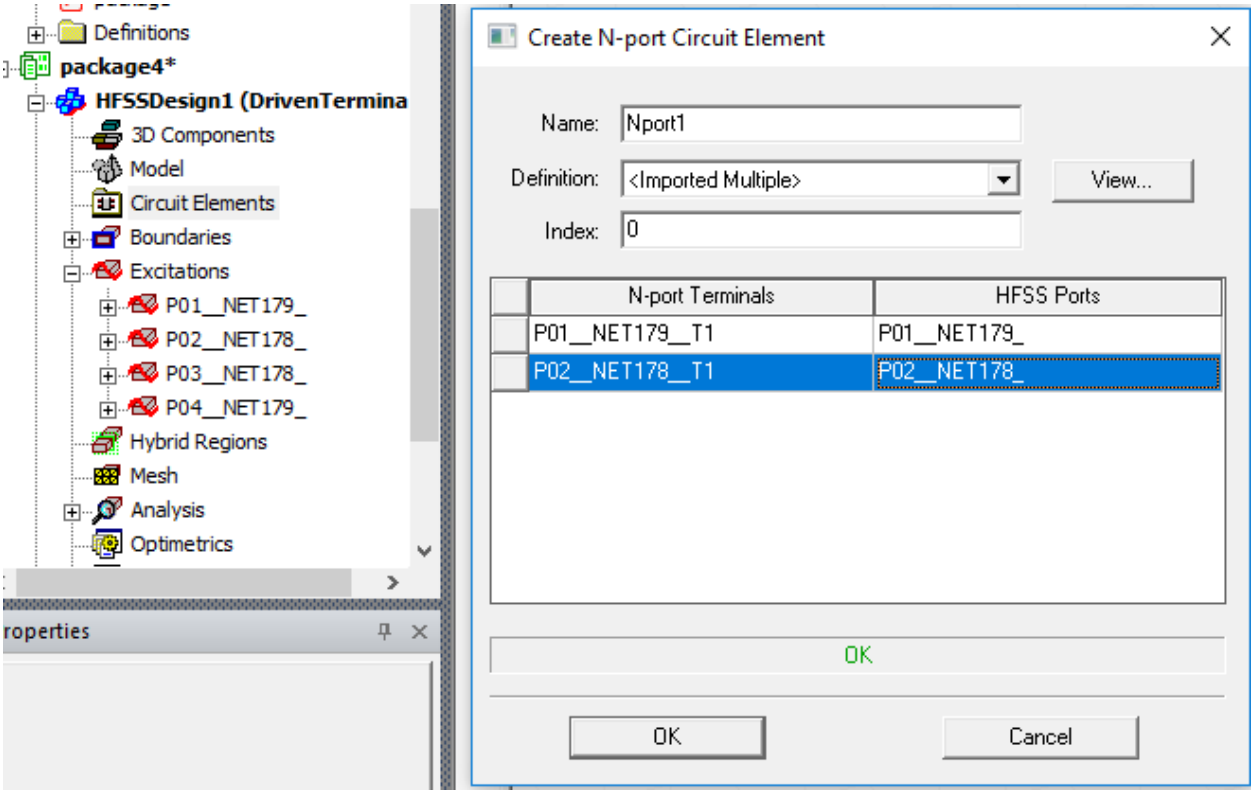
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OK to close the dialog box. You then make HFSS Port assignments in the **Create N-port Circuit Element** dialog.



When you have made the assignments, the message field reports **OK** in green.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

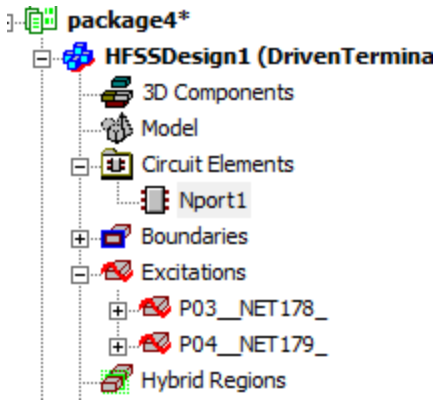


Notice that the **Index** value can control which touchstone file listed in the **Circuit Element Imported Solutions** dialog is used in the simulation by index. The Index value can be parameterized, that is, defined as a variable so to be used in a parametric study or exposed as a component parameter when included in a 3D component.

With valid assignments, the **View** button is enabled and lets you open the port definition in Network Data Explorer.

OK the dialog box. The N-port Terminal for which you have assigned HFSS Ports is created, and the corresponding Excitations are removed from the Project tree.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

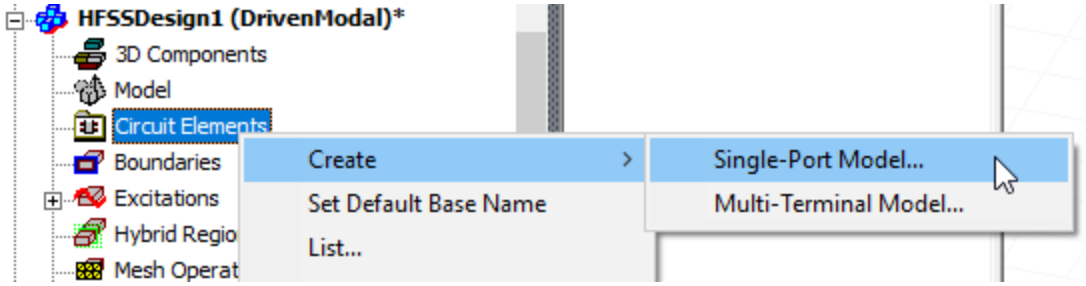


Import Touchstone Files for Circuit Elements in HFSS

To create a circuit element, you can import Touchstone files to create the definition for a Single Port Model or a Multi-Terminal Model:

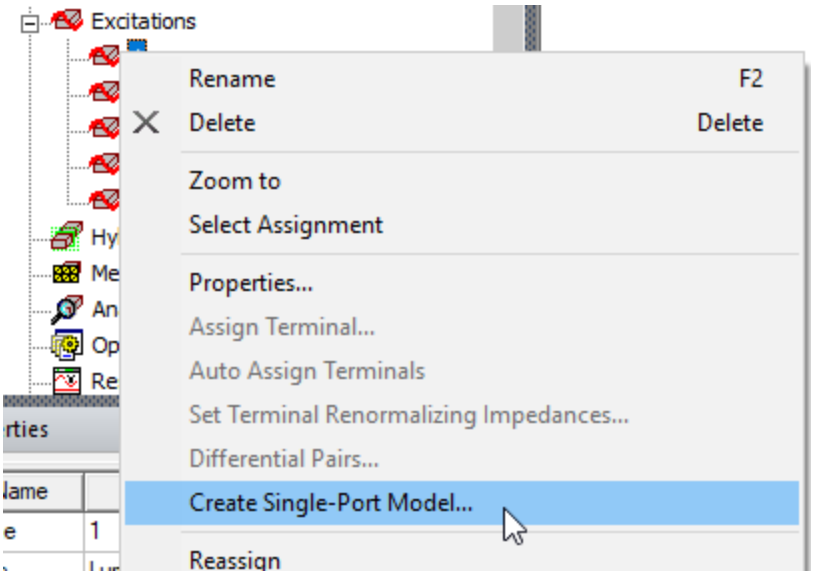
For a single Terminal Model:

1. Right click on a Lumped port in the **Project** tree and click **Create Single-Port Model**,

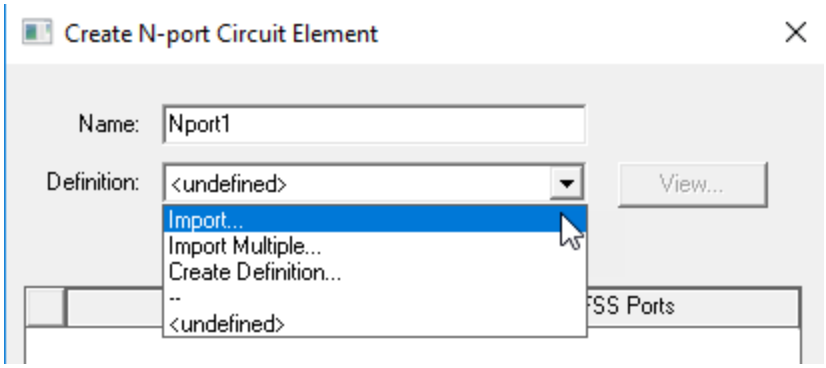


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or right click on the **Circuit Elements** folder and select **Create>Single Port Model...**.

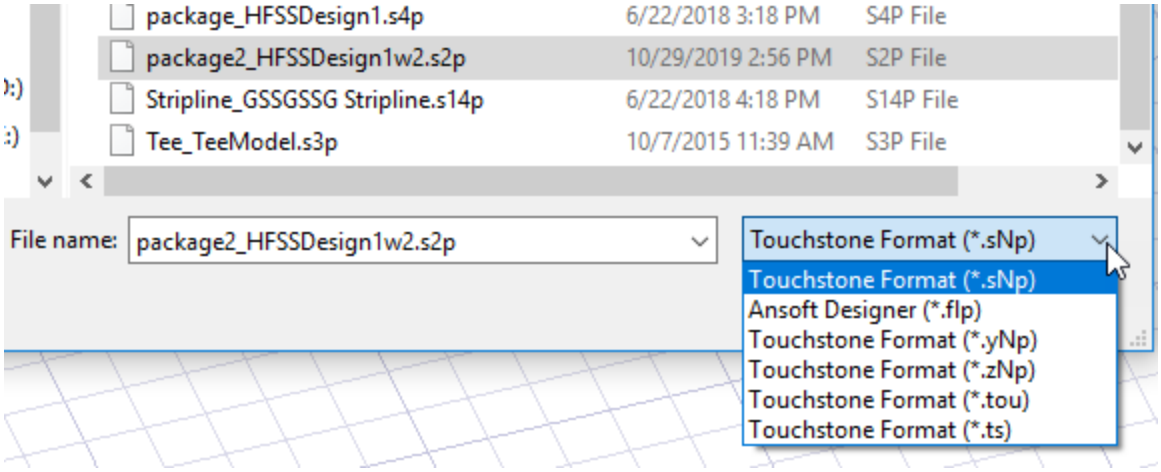


The **Create N-port Circuit Element** dialog opens. If you selected the port menu, the dialog shows the HFSS Port assignment. If you selected from the Circuit Element menu, you can click on the HFSS Ports cell to select from available ports.

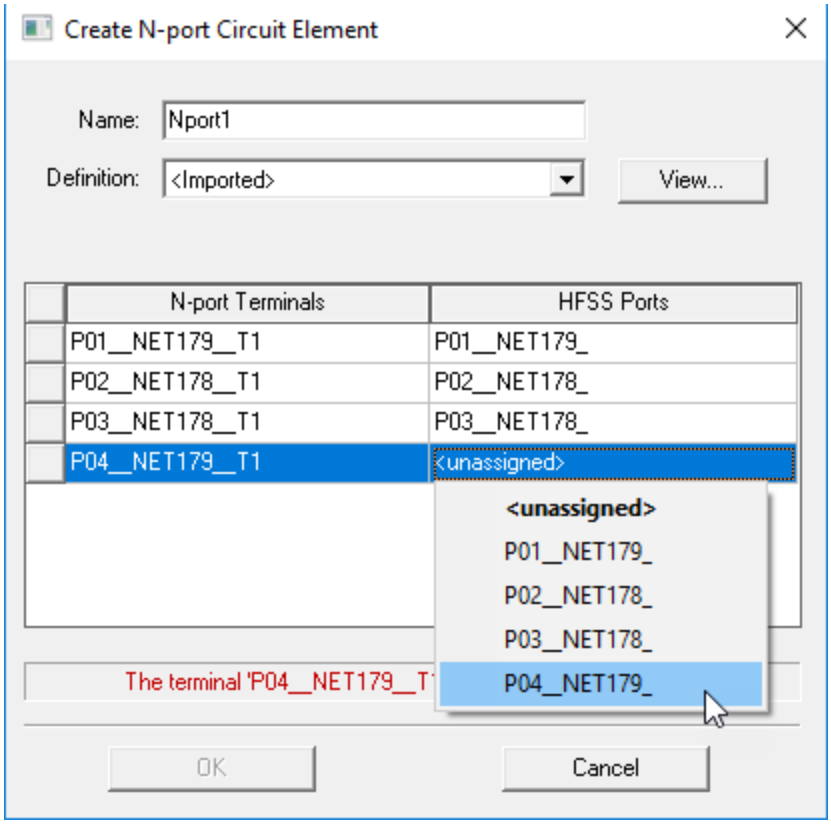


Selecting **Import...** automatically opens a browser window. You can select the Touchstone format and an appropriate file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

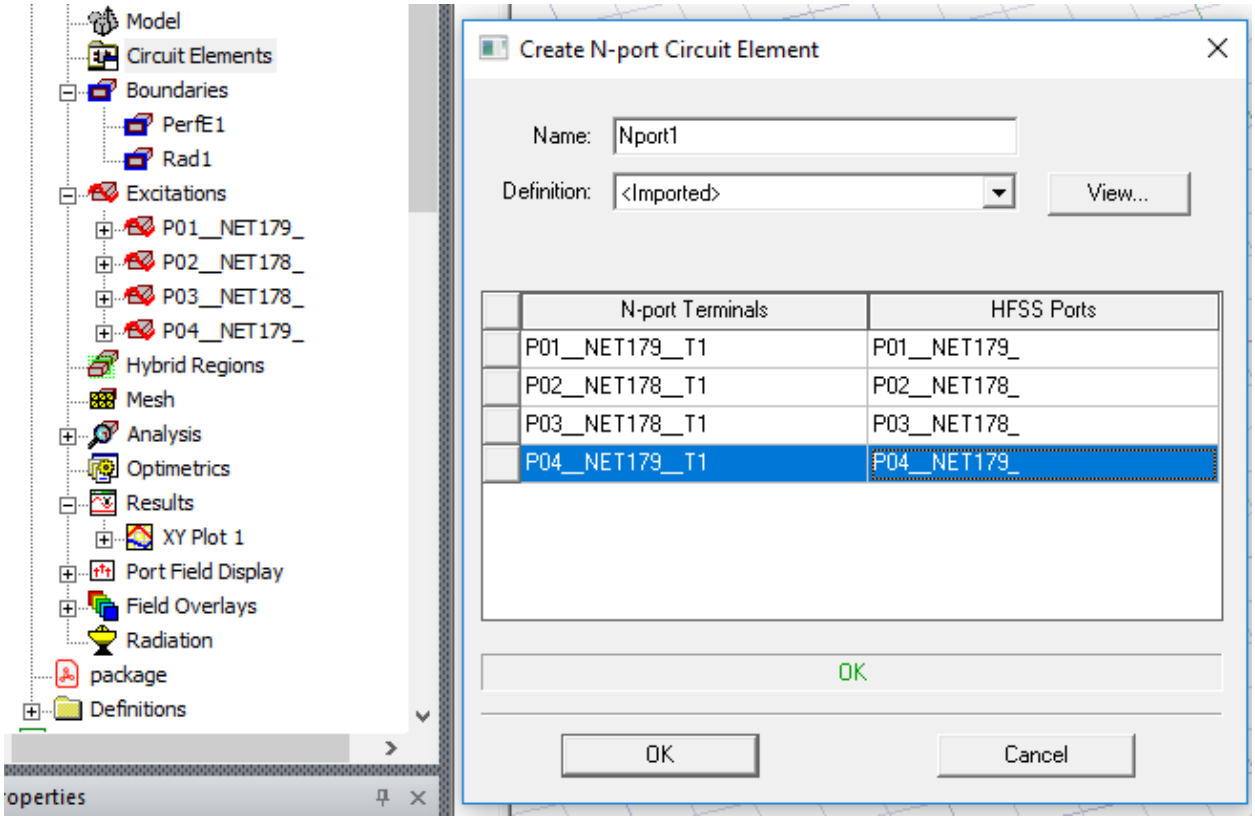


If the Touchstone file is consistent with the design and selection, the Create N-port Circuit Element dialog shows where you can make port assignments.

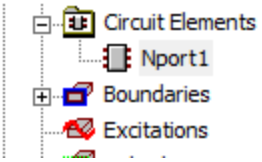


When you can competed Port/Terminal assignments, the Message field changes to a green OK.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

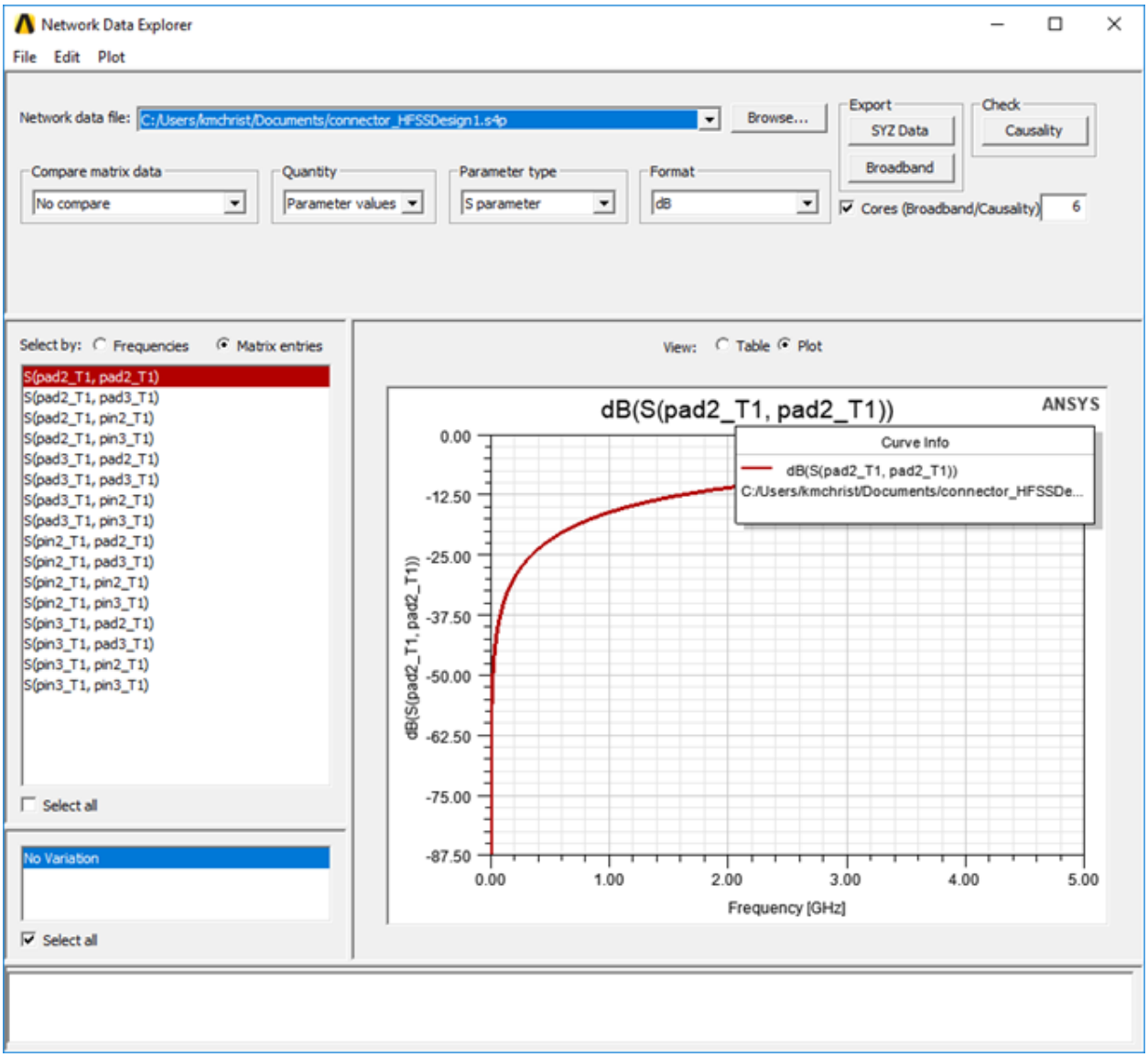


When click OK, the dialog closes, and the in Project tree, the Excitations are removed an Nport appears under the Circuit Elements.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. The **View...** button in the Create N Port dialog opens the circuit port in ndExplorer

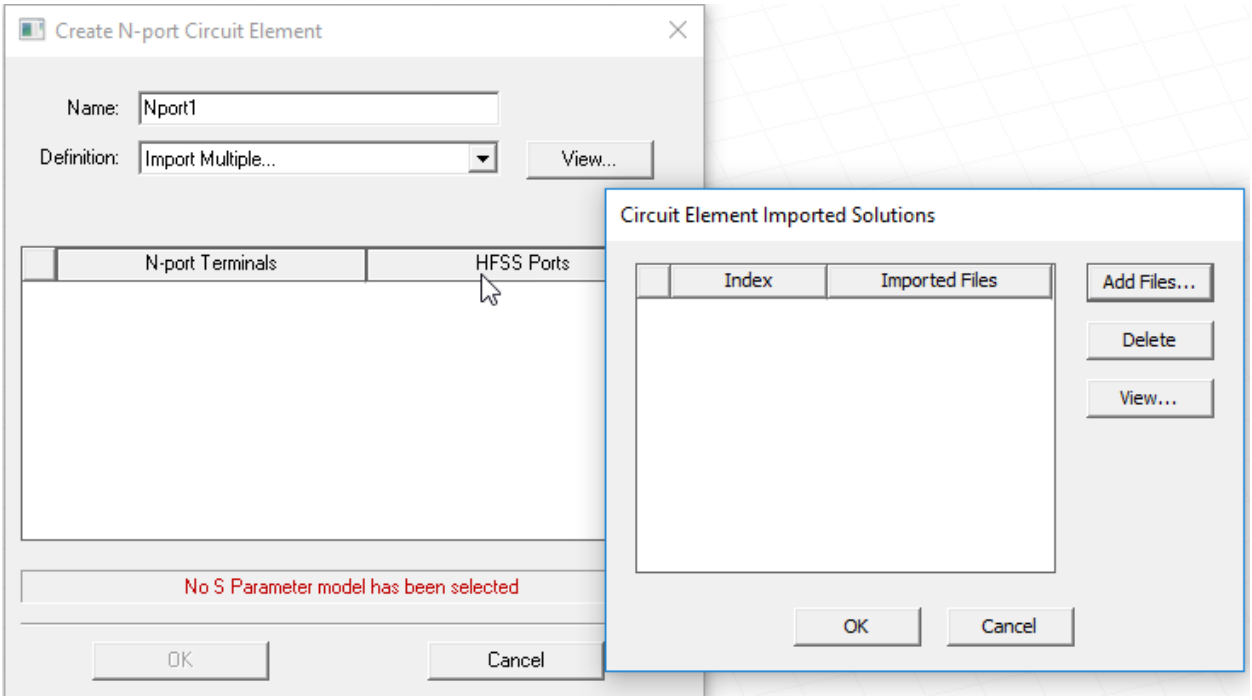


For a description of ndExplorer, see Network Data Explorer.

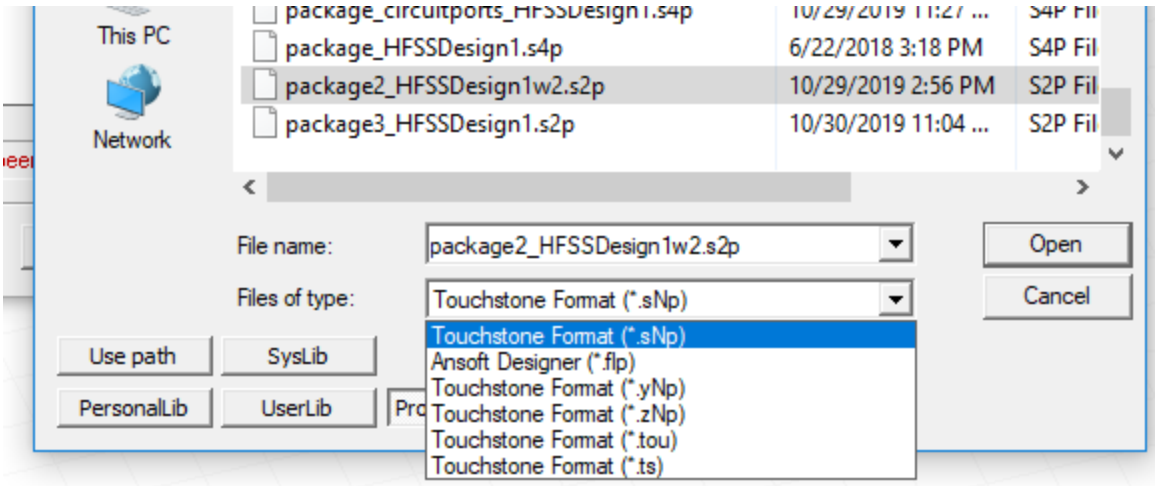
Using **Import Multiple ...** for Touchstone files.

If you select **Import Multiple...** in the **Create N-port Circuit Element** dialog box, the **Circuit Element Imported Solutions** dialog opens.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

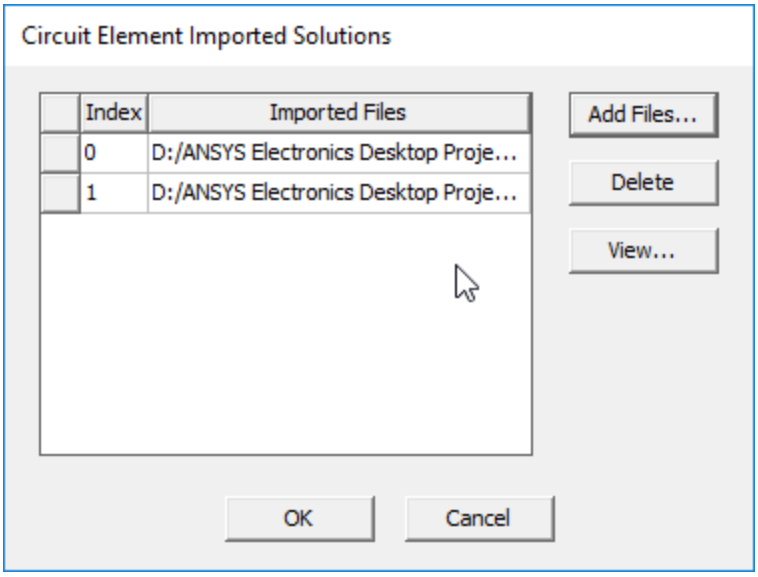


If no imported files are listed, you can click Add Files to open a file browser to add suitable touchtone solution files, such as those created when you Export Matrix data from the Solutions dialog.

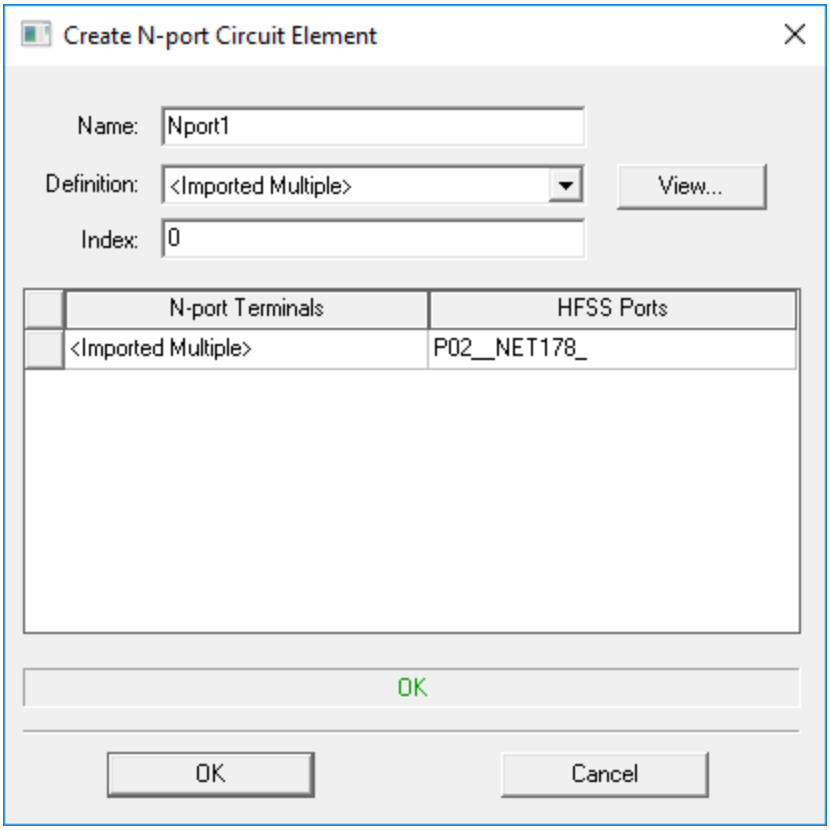


Added files are then listed as **Circuit Element Imported Solutions**. The **View...** button opens the selected solution file in NDExplorer.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

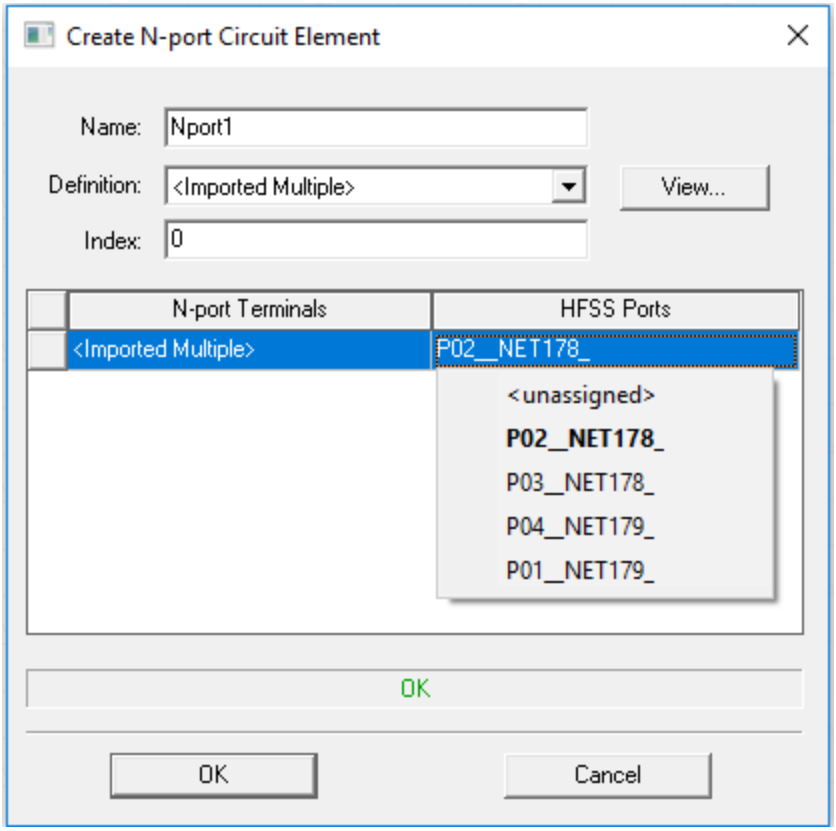


Click OK to close the dialog box. The Create N-port Circuit Element dialog shows an HFSS Port assignment, based on the solution file.

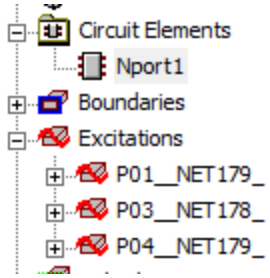


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the Solution files contains information for multiple ports, you can change the assignment by selecting from the drop-down menu.

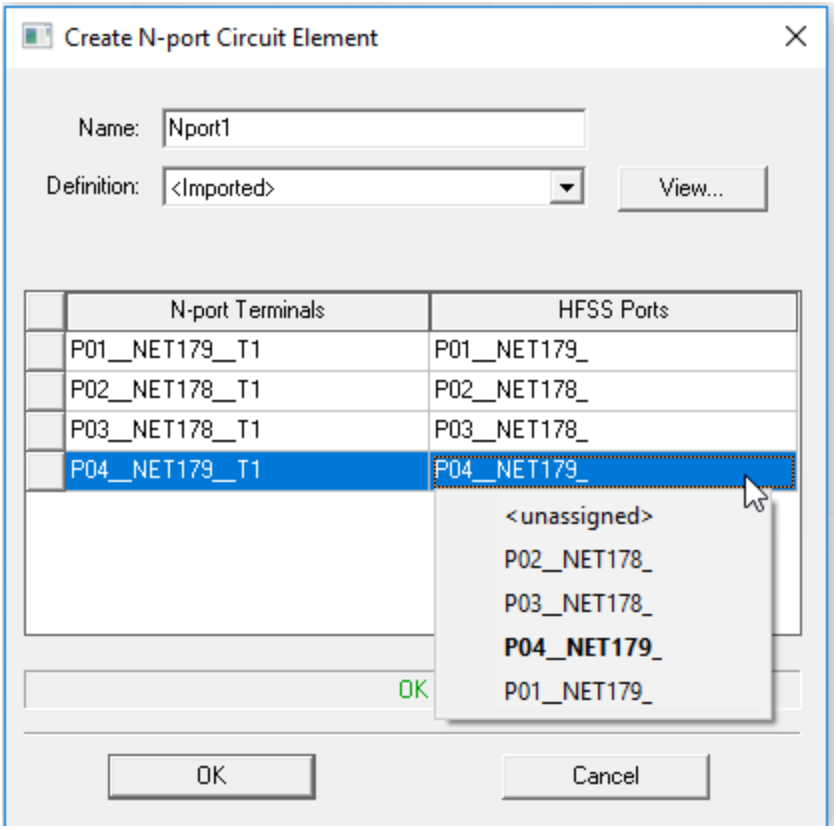


Click OK to close the dialog box. The Project tree will show the nPort listed as a Circuit Element, and the Excitation that corresponded to the assignment is not listed.

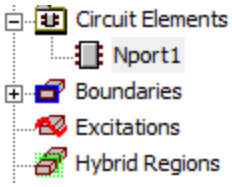


If you select **Multiple Terminal Model**, and the selected Touchstone file contains solution information for all N-port Terminals, the Create N-Port Circuit Element dialog allows you to make assignments for all available HFSS Ports.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



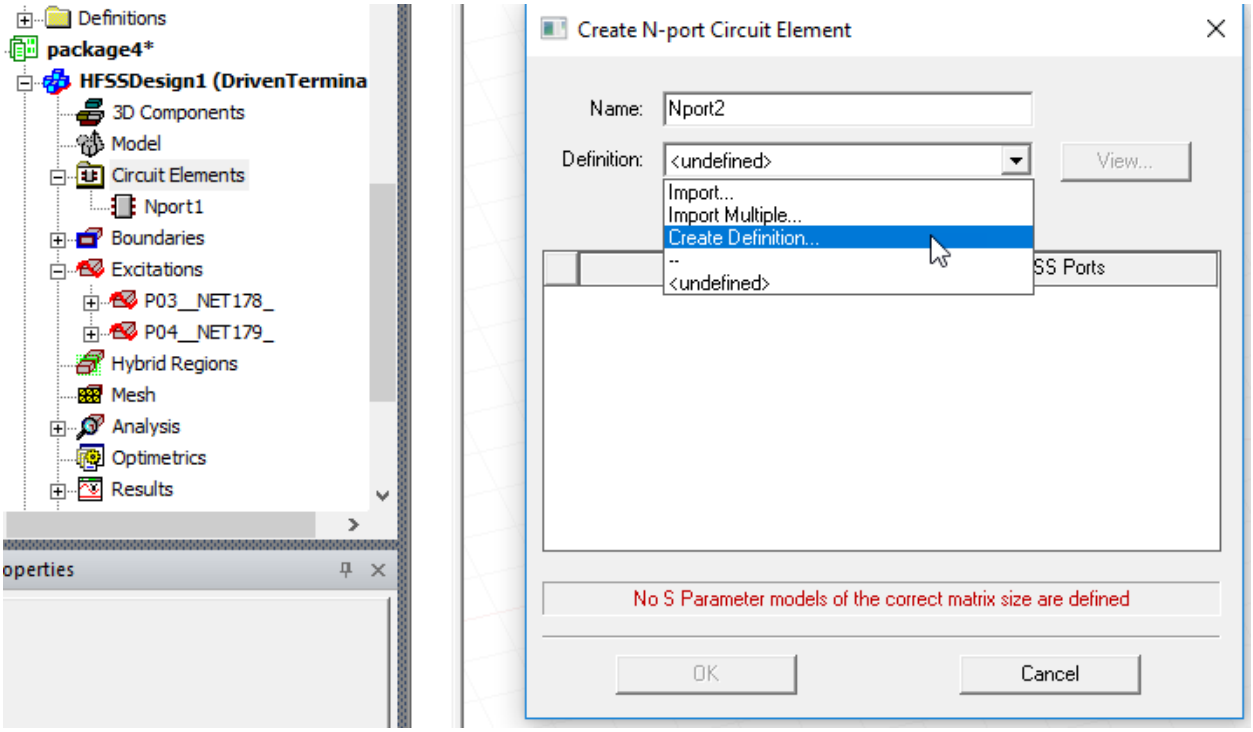
In this case, the click OK to close the dialog box. In this case the N Port replaces all assigned HFSS Ports.



Creating Circuit Models and Managing Libraries in HFSS

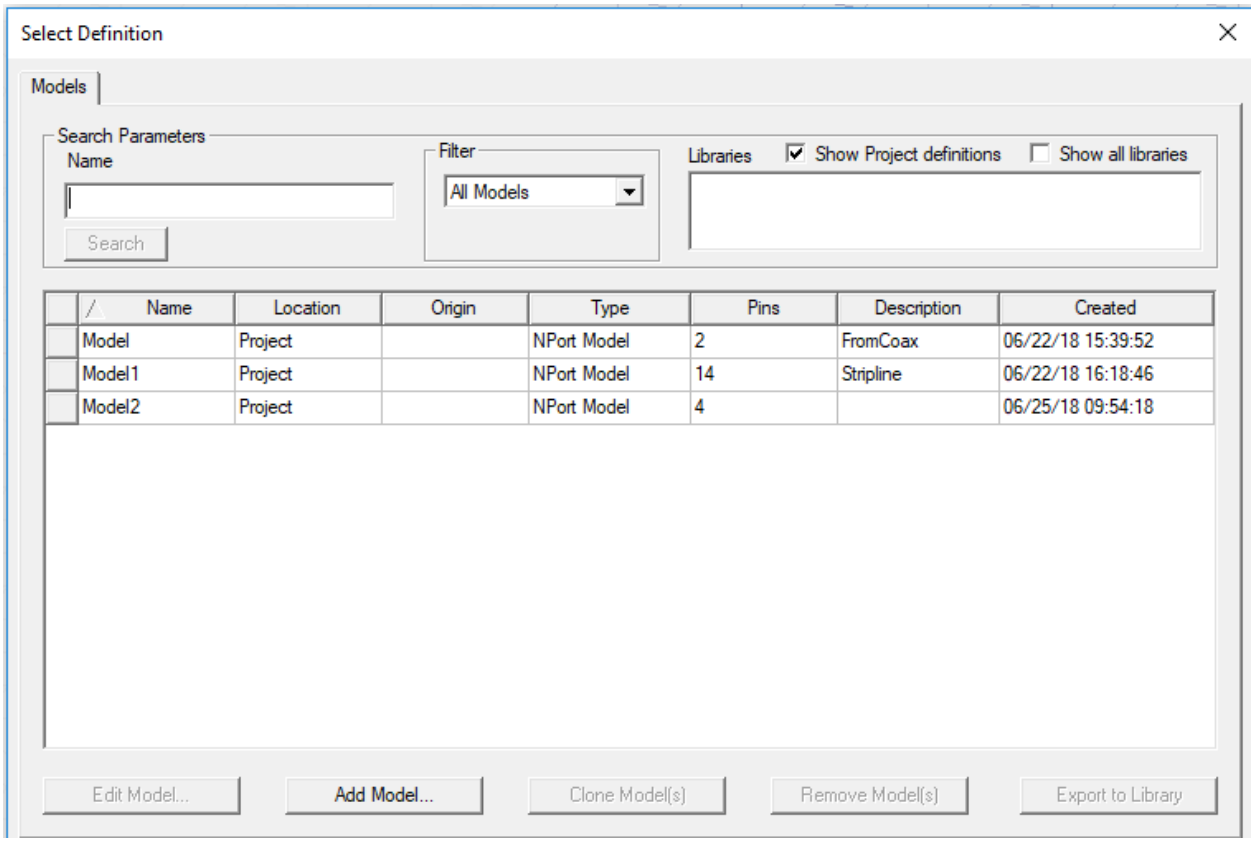
The **Create Definition...** command appears as a definition option on the **Create N-port Circuit Element** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

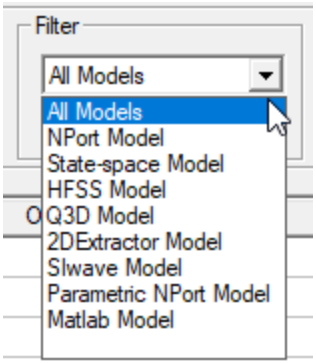


Selecting **Create Definition...** opens the the **Select Definition** dialog allows you to view, create, edit, filter and remove circuit models for the Circuit Elements feature in HFSS. It shows existing models in your library, and allows you add and organize your models.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

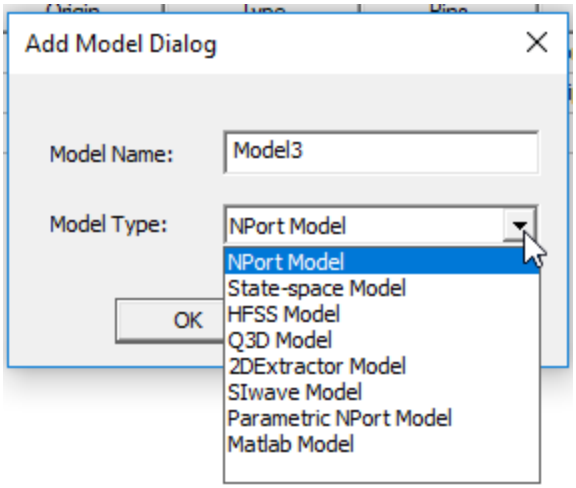


The dropdown menu in the Filter menu lets you display the models in your present library according to their original source.

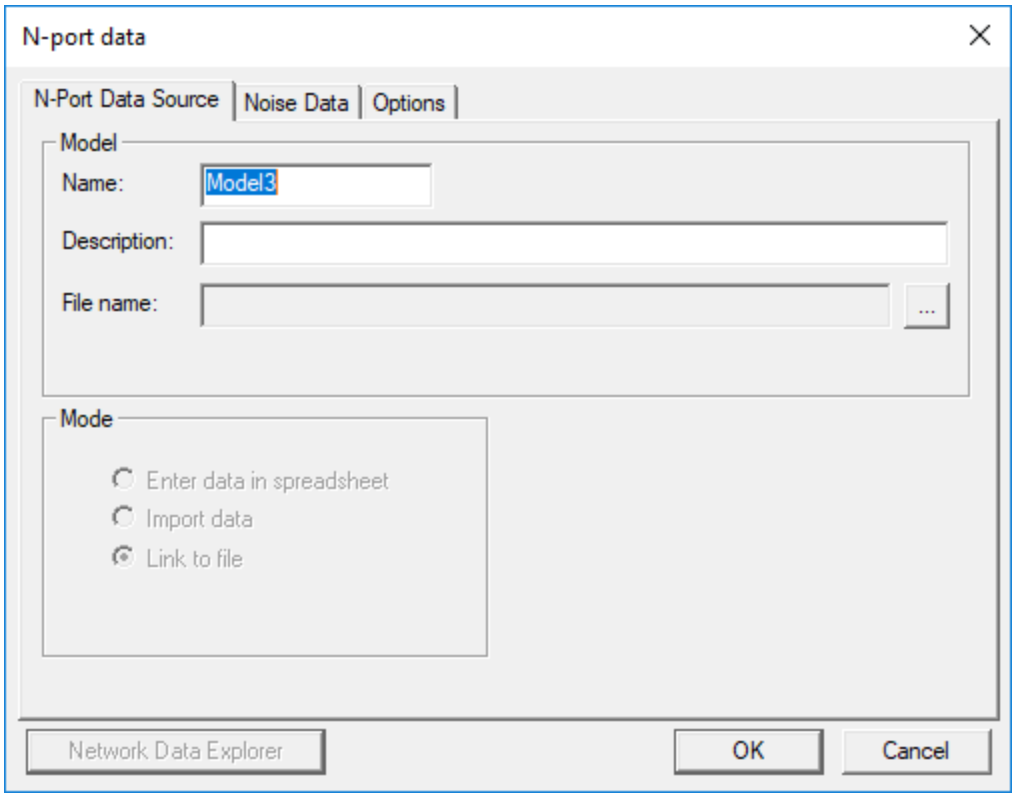


Click **Add Model...** to include additional models in your library. This opens the **Add Model Dialog.**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

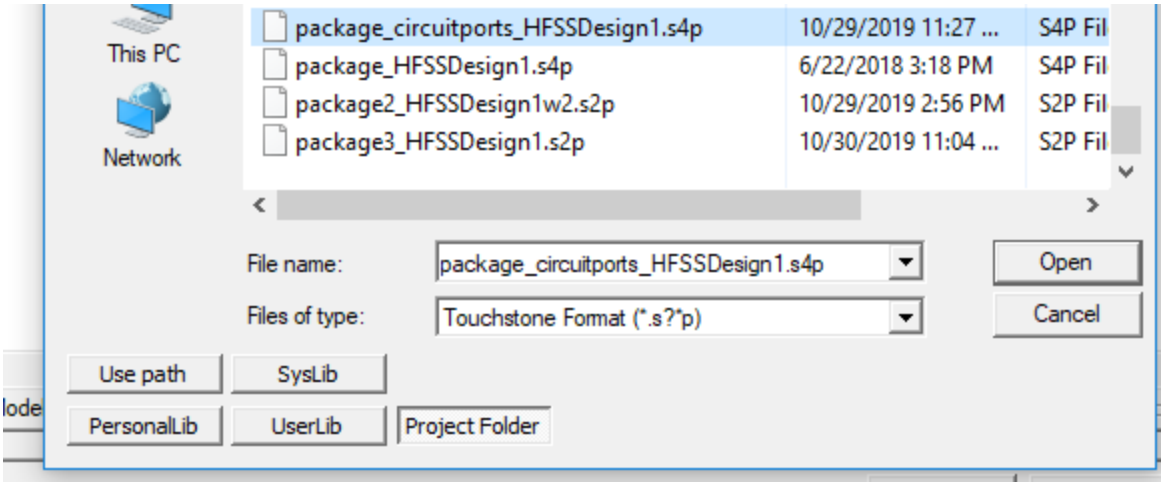


The **Add Model Dialog** contains a **Model Type** menu. Each selection leads to a dialog with options appropriate to that selection. For example, NPort Model or State space menu choices opens N-Port Data.

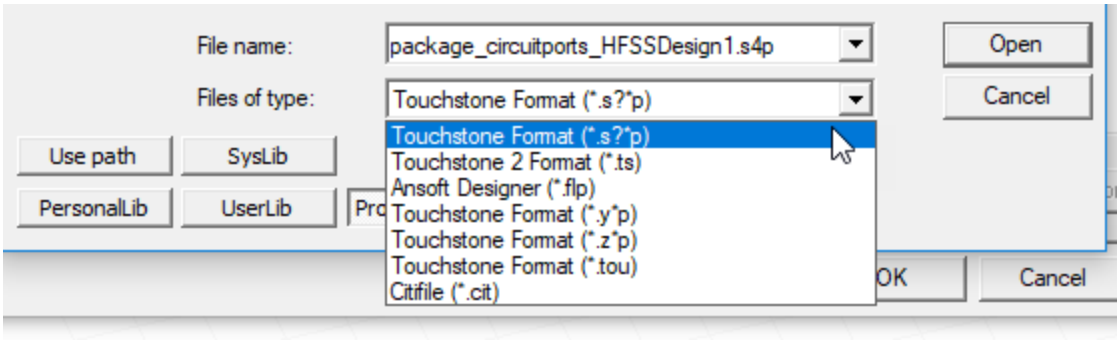


Here you can use the ellipsis button [**...**] to open a file browser to select a Touchstone file.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

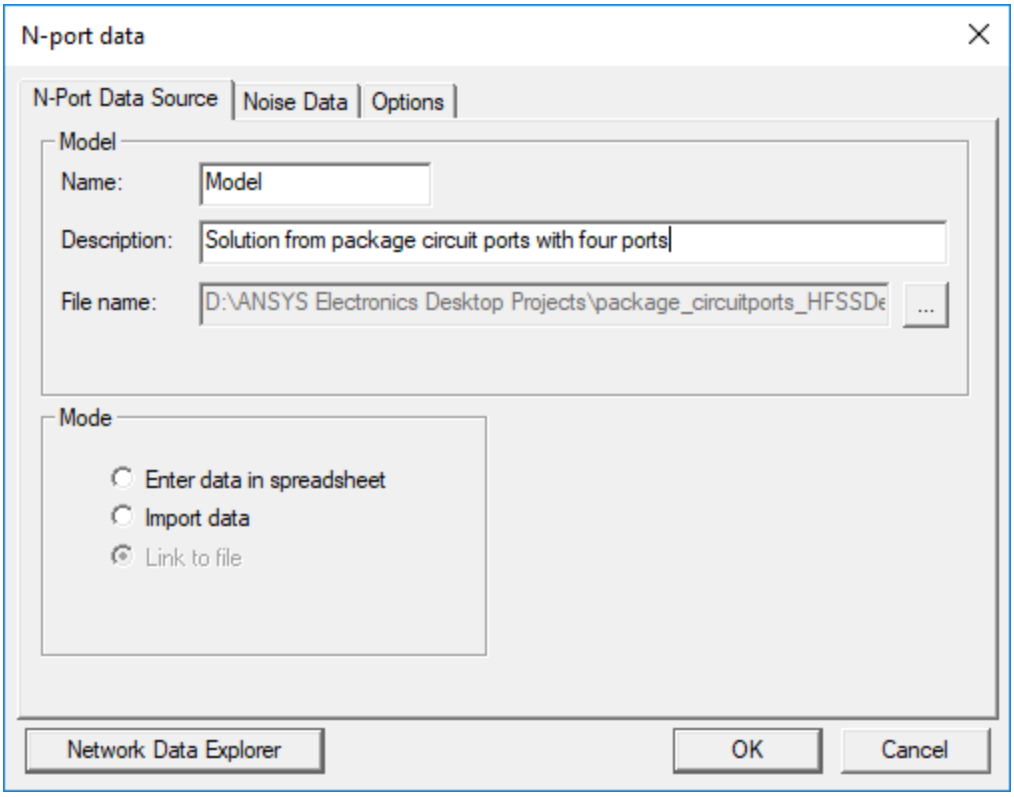


The **Files of Type** field includes a drop down listing the supported import formats.



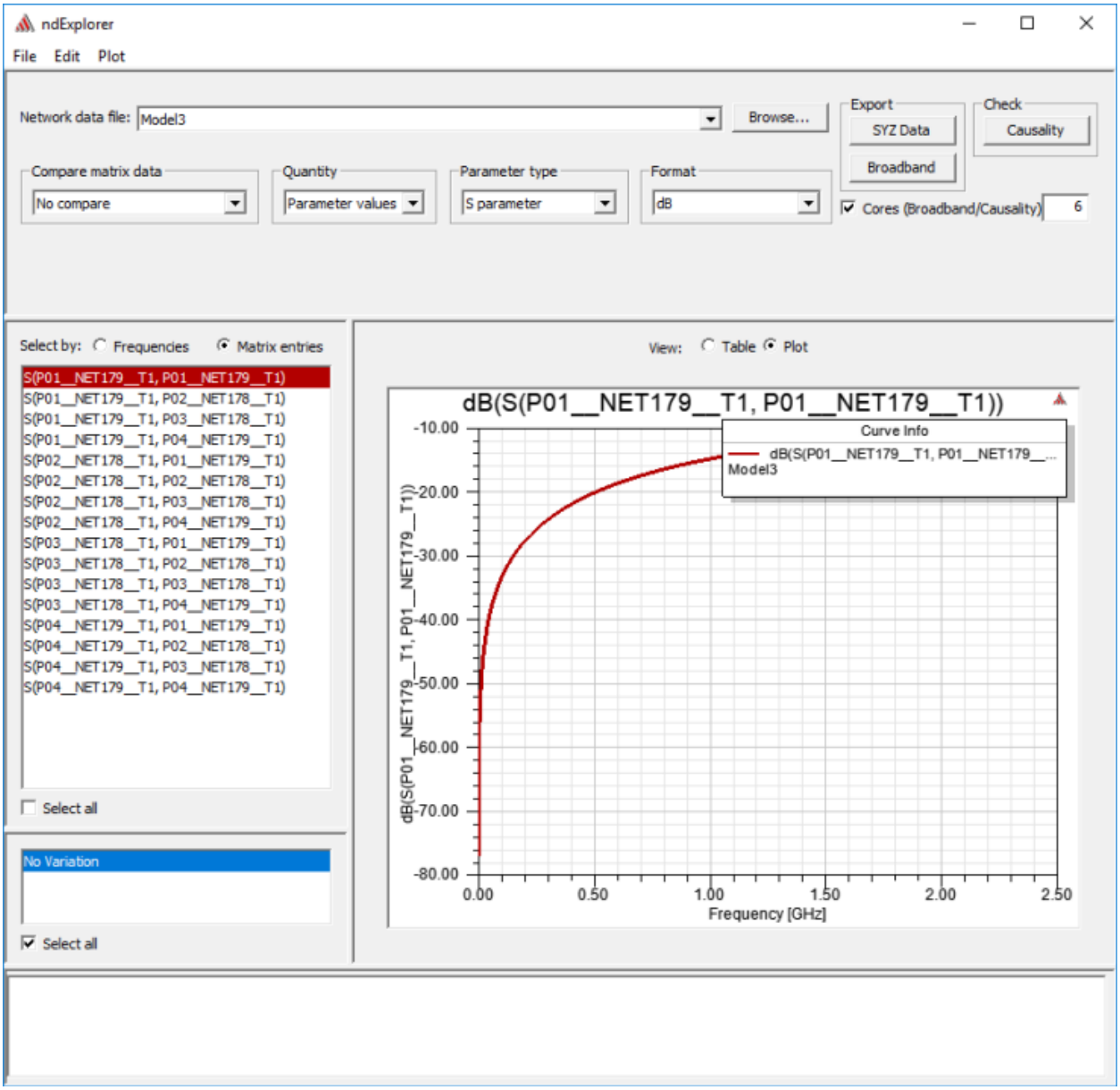
When you have selected the file, and clicked OK to close the browser, you can enter a description in the N-port data dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



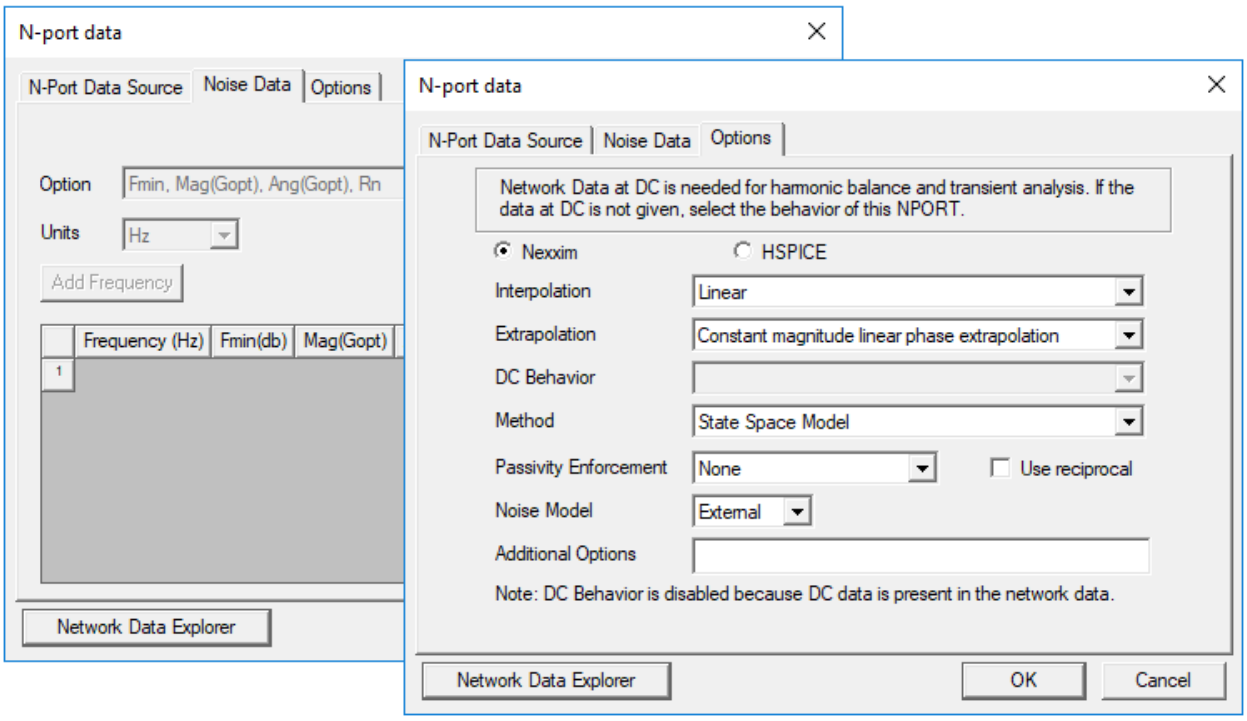
The **Network Data Explorer** button is enabled. You can use ndExplorer to view tables or plots of the matrix data.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



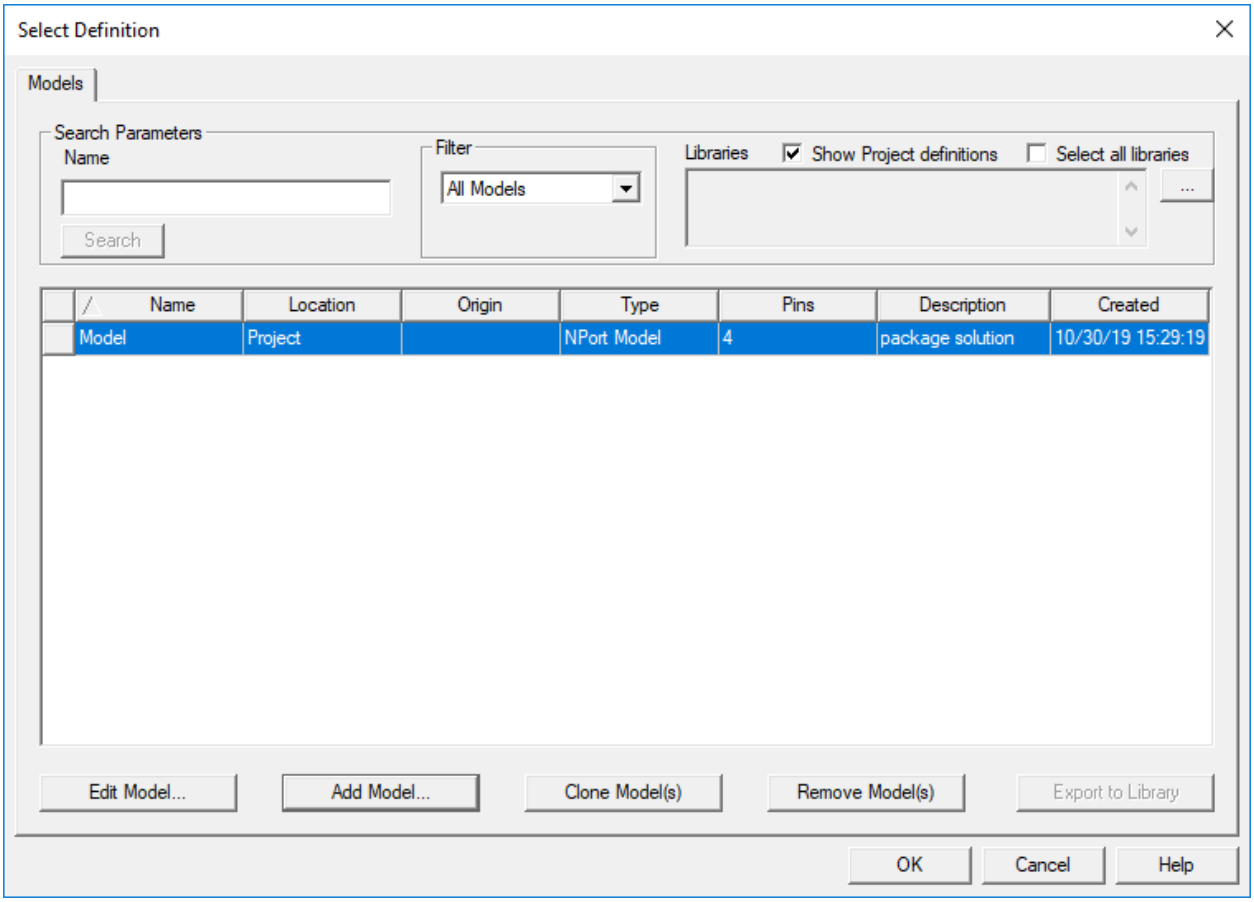
When you have selected a Touchstone file for the **N-port Data** dialog box, the **Noise Data** and **Options** tabs are enabled as well.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



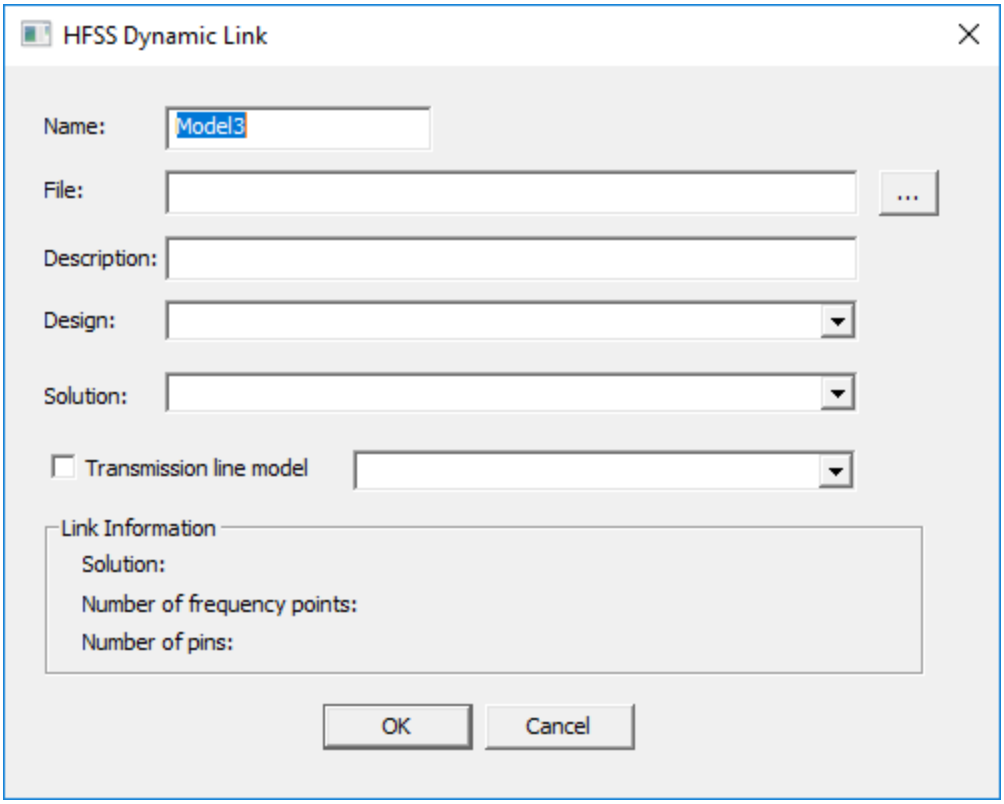
When you close the **N-port data** dialog box, the **Select Definition** window lists the newly added model.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



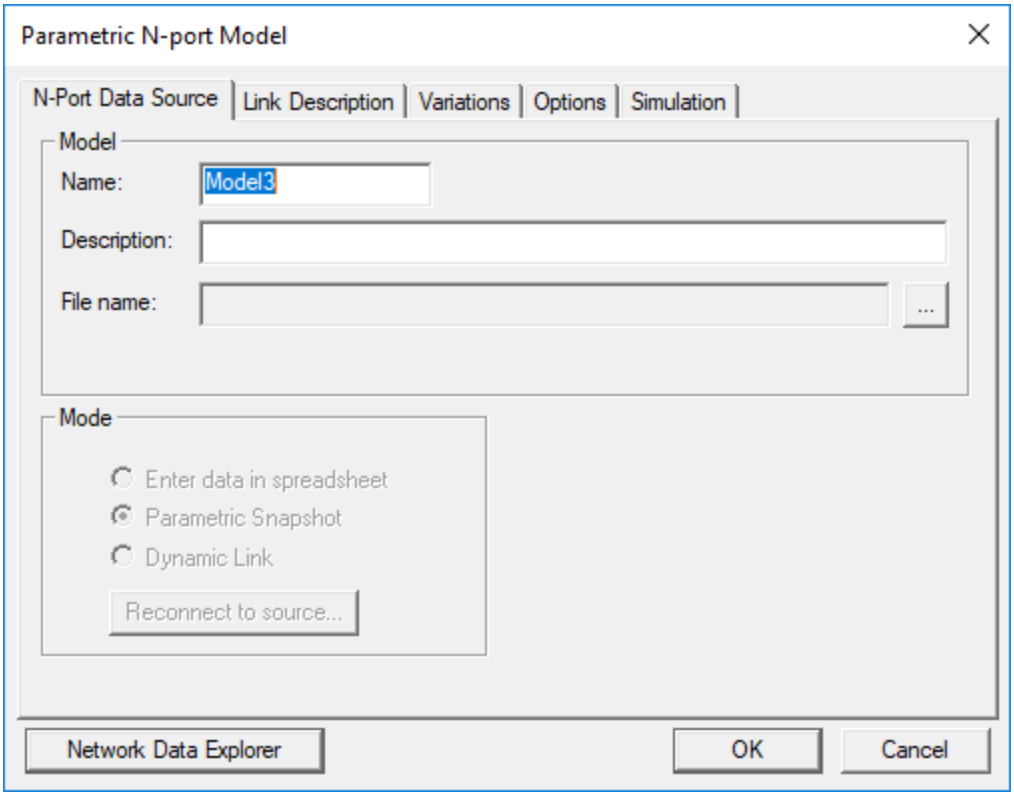
Add Model dialog choices for HFSS, Q3D, 2D Extractor, or SIwave open a **Dynamic Link** dialog box, that lets you select the project file, as well as the Design and Solution to use. If one or more transmission lines is available, you can optionally enable selection:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

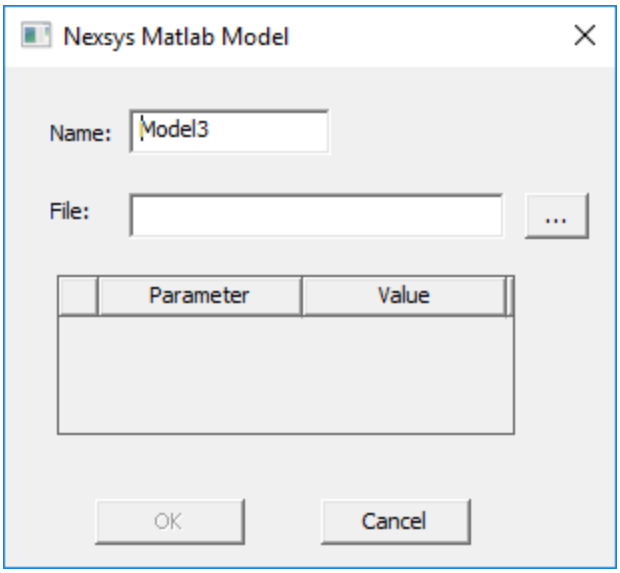


Add Model dialog choice for **Parametric N-port Model** opens a dialog with tabs for **N-Port Data Source**, **Link Description**, **Variations**, **Options**, and **Simulation**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Add Model dialog choice for **Matlab Model** opens a **Nexys Matlab Model** dialog that lets you select a file by opening a browser with the ellipsis button **[...]**.



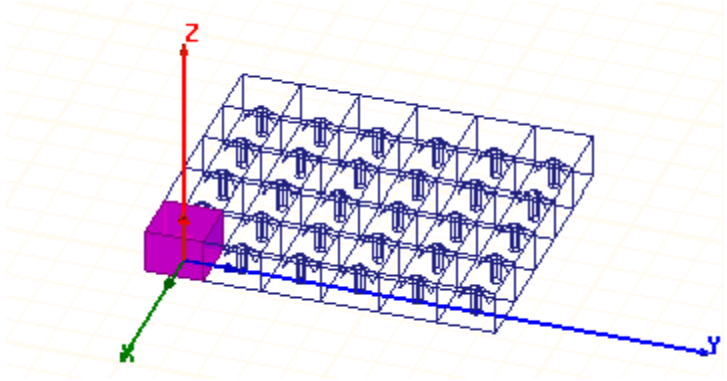
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

21 - Assigning Arrays in HFSS

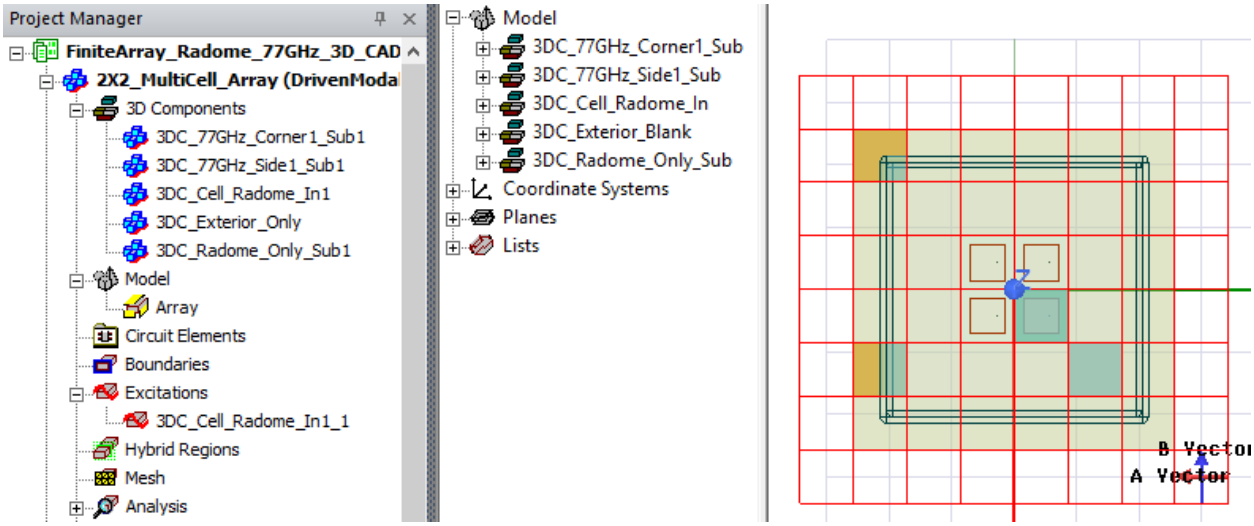
HFSS permits you to assign a virtual array based on a unit cell object, which can be a native design, a 3D component, or a SBR+ Parametric Antenna. For driven Modal and driven Terminal designs for HFSS with Hybrid and Arrays Solution Types, you can then simulate the array model using distributed processing, treating the instances as parent and child objects. This permits faster definition, display, and simulation of array based designs, such as antenna arrays. You can plot and animate array fields on cutplanes, lines or points. Post processing lets you view fields on any virtual instance.

The unit cells for an array can be rectangular, parallelogram, or hexagonal. HFSS SBR+ allows you to create Parametric Arrays from any of the parametric antennas. You can define the required primary and secondary boundaries so as to create offset arrays. You can only edit the settings in the physical cell and these settings will be applied to the corresponding instances in the virtual cells.

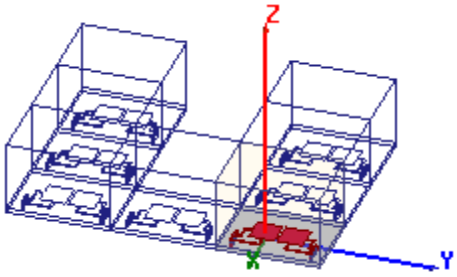


You can also create 3D components from unit cells, and use them to create a 3D Component Array.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once you have defined an array, you can designate any cell in the array as active or passive, or as padding. You can use the padding cell designation to define arbitrarily irregular arrays. Cells designated as padding are treated as background material for fields calculations.

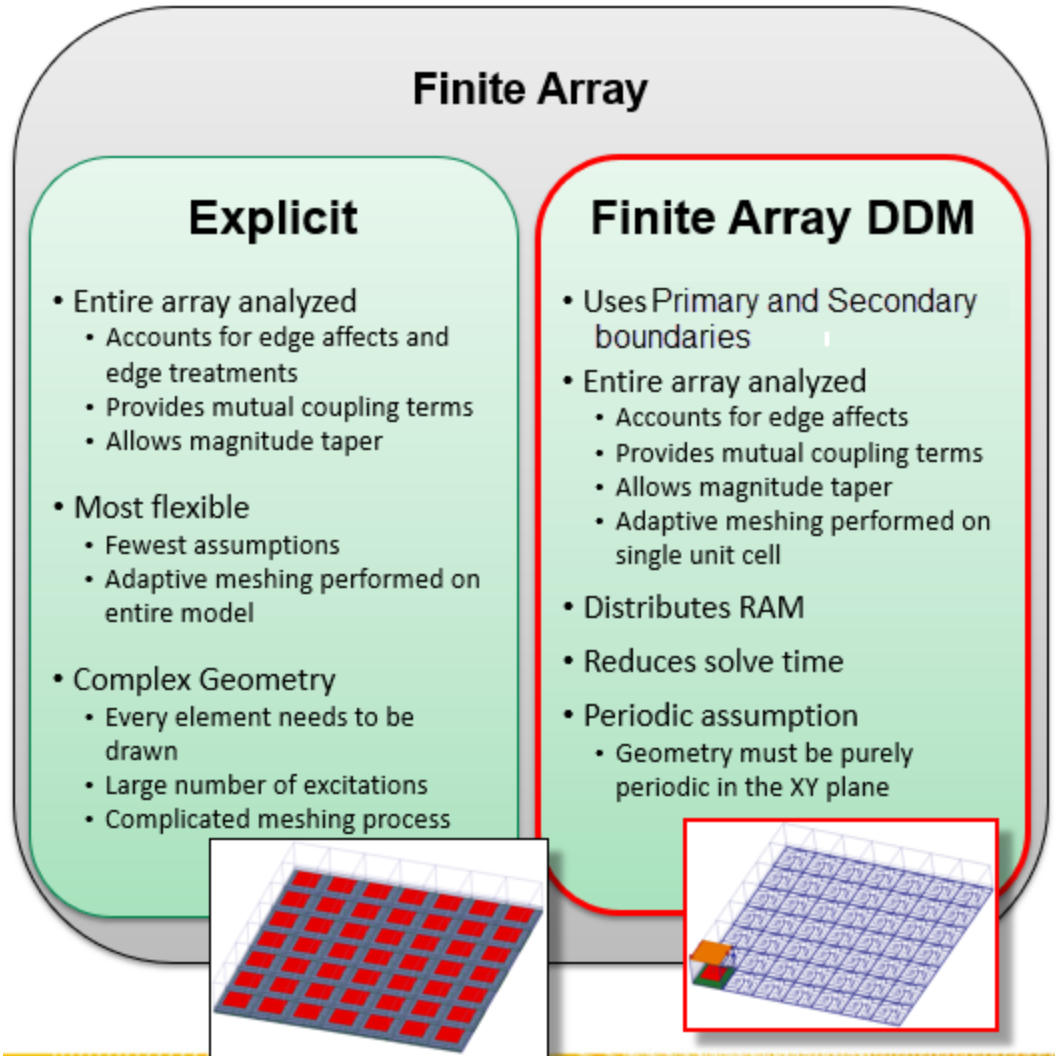


Most boundaries and excitations defined in the physical unit cell will have their corresponding instances in each virtual cell. The exception is incident wave, which is applied across the whole model and should include the 'expanded' model based on the array setup.

A Beta Feature allows you to have Multiple Component arrays in an HFSS design.

When the target design is a finite array, having both enforced E and enforced H options is not allowed. For all analytic incident waves except plane wave such as Gaussian beam, scatter field formulation is not supported for finite array design. HFSS supports all Incident Wave and Linked field sources to a target finite array DDM design. You can perform an explicit solve by modeling all cells in the modeler, or a Finite Array DDM solve by using Array Definition of a Unit cell object.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



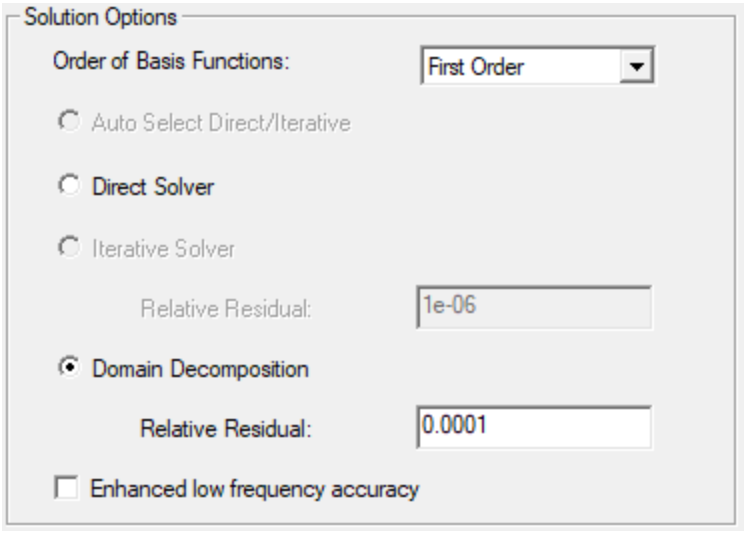
The basic process flow for using **HFSS>Model>Create Array...** or in the **Project** tree, right-click on the **Model** icon, and click **Create Array...** on the short-cut menu, is:

1. Draw the unit cell, containing all appropriate boundaries and source definitions. The design can contain at least one native geometry with no less than two pairs of lattice boundaries, or at least one 3D component with no less than two pairs of lattice boundaries. For native geometry unit cells, the regular array dialog is enabled. For 3D components, the 3D Component array dialog is enabled. Native geometries and 3D components cannot be mixed in an array.
2. Create the antenna array, including name, dimensions, Primary and Secondary boundaries where needed for conformal meshing, and selection of row and column primary and secondary pairs for implicit definition of lattice propagation vectors. Designate which cells are active, passive, and padding.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Set up the distributed processor pool. Designs with arrays require HPC licenses.
4. Provide a memory statistic for the amount of RAM guaranteed on each DSO processor.

The Solution Option for **Domain Decomposition** is automatically checked and Direct Solver and Iterative Solver options are disabled.



The domains are UI defined through the array definition, not solver defined domains. Given a valid configuration, an Array solve can use a distributed memory solution.

The UI will provide the antenna array definition to the domain manager. This will cause the following to occur:

1. Instantiation of domains to represent the cells of the antenna array plus surrounding air padding cells.
2. Creation of internal domain manager data structures that are needed to support the solve and post processing. This includes appropriate domain parent/child relationships, transformations from the physical domain, interface information per pair of domains, and support for locating a domain by row/column coordinates within the antenna array.

For linking to the Desktop, the network data from HFSS will include both physical and virtual cells. This applies to both port locations and push excitations.

For Optimetrics solution quantities of both virtual and physical cells can be used for calculation.

For 2D Reports for models with Arrays, matrix solution quantities of virtual cells will be expanded into a vector in the same fashion as without the array. The entries are listed according to their [row, column] order in the corresponding "expanded" matrix.

For Port Field Display there is no GUI change. Only physical ports/terminals will be listed. There is no need to support visualization of user-selected cell (like field overlay plot) because the field patterns of the virtual modes are the same as those in the physical cells.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For designs with an Array, the Edit Sources dialog listing order will be as follows: Sources will be listed according to their cell [row, column] order in the array. For each cell, port/terminals are listed in creation/assignment order with mode in each port listed sequentially. Other type of sources, such as incident waves and linked field sources, will be listed after ports/terminals. You can also create Source Groups for dealing with finite array sources.

There will be no change in the far/near field pattern setup and far/near fields will be computed from radiation surfaces on all cells (both physical and virtual).

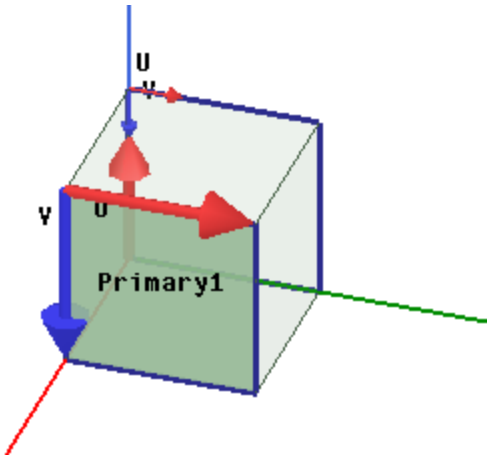
Copy/Paste design will copy an array. Copy/Paste geometry will NOT copy an array.

Note: A legacy regular array, with one component with at least two pairs of lattice boundaries, can be opened, but you cannot create a new regular array based on this condition. Regular array can only be created based on native geometries. If you delete a legacy with these conditions, you could only recreate it as a 3D component array.

Creating a Unit Cell for an Array

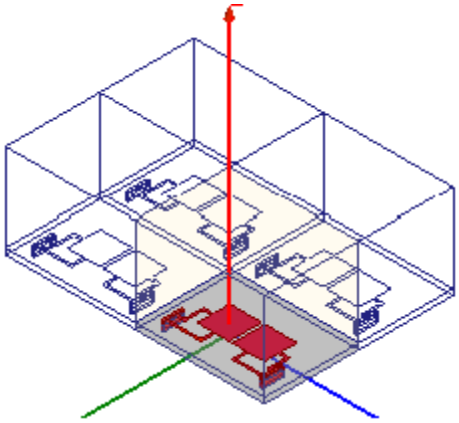
You use the Modeler to create a unit cell for an array.

- The unit cell can be rectangular, parallelogram or hexagonal.
- You must define appropriate Primary and Secondary boundaries, as well as other boundaries required for the model. Deleting a primary or secondary boundary that is referenced by an existing array also deletes the array. Typically, you define Primary and Secondary boundaries as opposite sides of a cell.

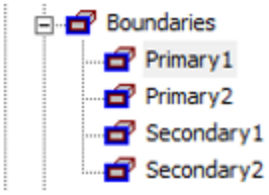


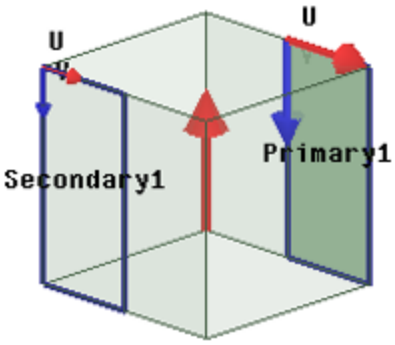
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

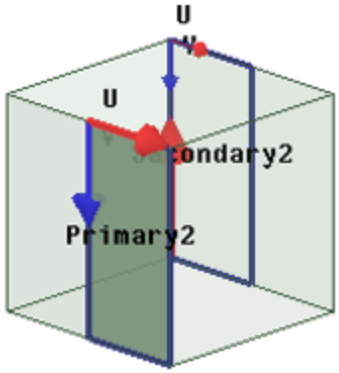
In such cases, with Primary and Secondary boundaries on opposite sides, the array sides align.



However, you can arrange Primary and Secondary boundaries to create an offset array. In such cases, you create both a Primary and a Secondary boundary for corresponding sections of a cell side, and assign Primary and Secondary to create offset alignment.

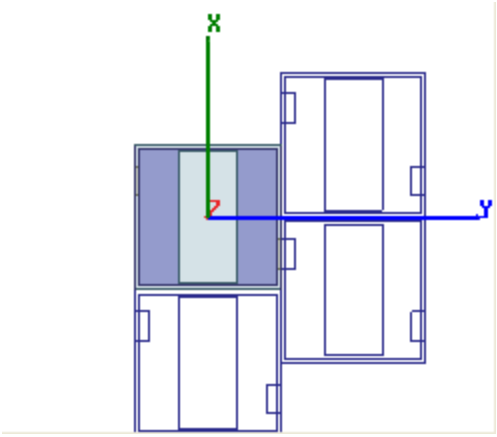






With all boundaries defined in this manner, the array can be offset.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- In making source definitions for unit cell for the unit cell, you cannot assign a Floquet port.
- Each driven Modal or driven Terminal design can contain a single array.

Editing the model object for an existing array invalidates any existing solutions. Once you have created a model, you may choose to make into a 3D Component. The Create Array... command provides different options, depending on whether you use a native geometry or a 3D component.

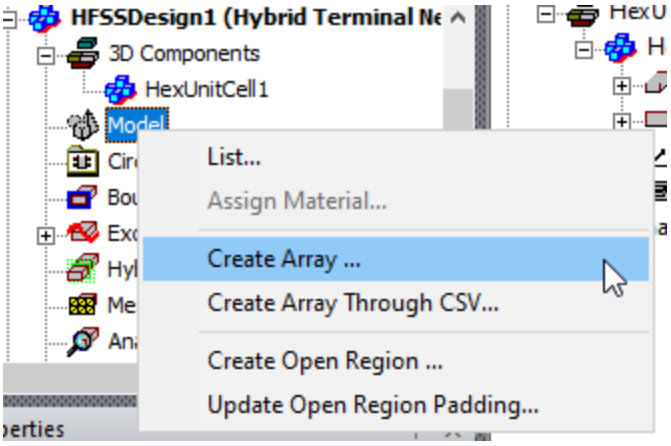
Create Array Command for Native Geometries

The **Create Array** command is enabled for Modal and Driven Terminal problems for the HFSS with Hybrid and Arrays Solution Types after you have assigned Primary and Secondary lattice pair boundaries to your unit cell model. You can access the command in three ways: click **HFSS>Model>Create Array**, right-click the Model icon in the Project tree and select **Create Array...** from the shortcut menu, or select the unit cell in the modeler window, and select **Create Array ...**from the short-cut menu. You also have the option to **Create Array Through CSV...**. This option is useful especially for defining large-scale arrays when defining through the UI dialogs requires a lot of manual work. This section describes the **Create Array...**. command. The **Create Array Through CSV...** command is described in a separate section.

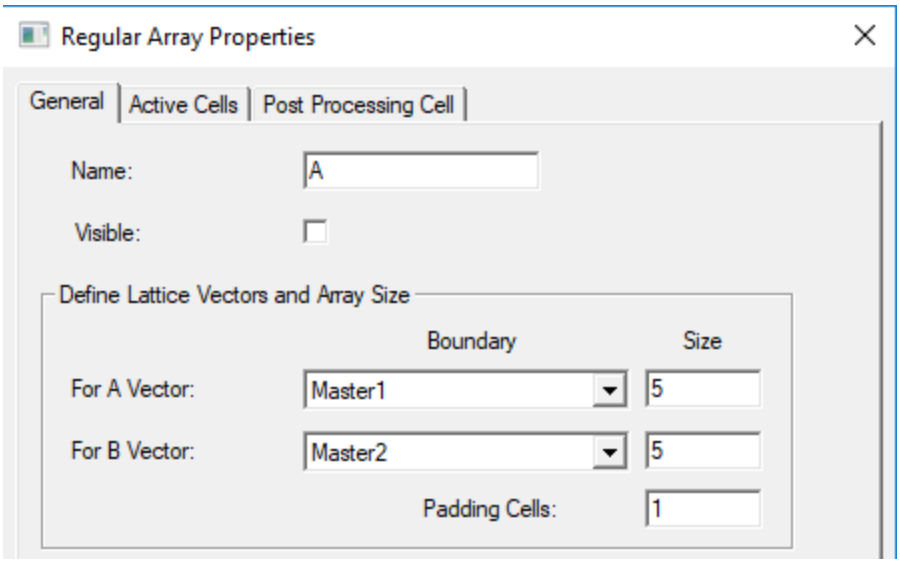
Create Array... Command Procedure

Right-click the Model icon in the Project tree and select **Create Array...** from the shortcut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



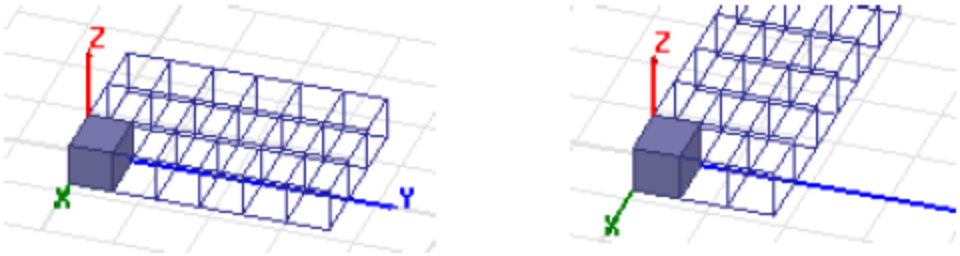
This displays the **Regular Array** dialog where you specify parameters for Number of Cells, Unit Cell Position and Lattice Vectors.



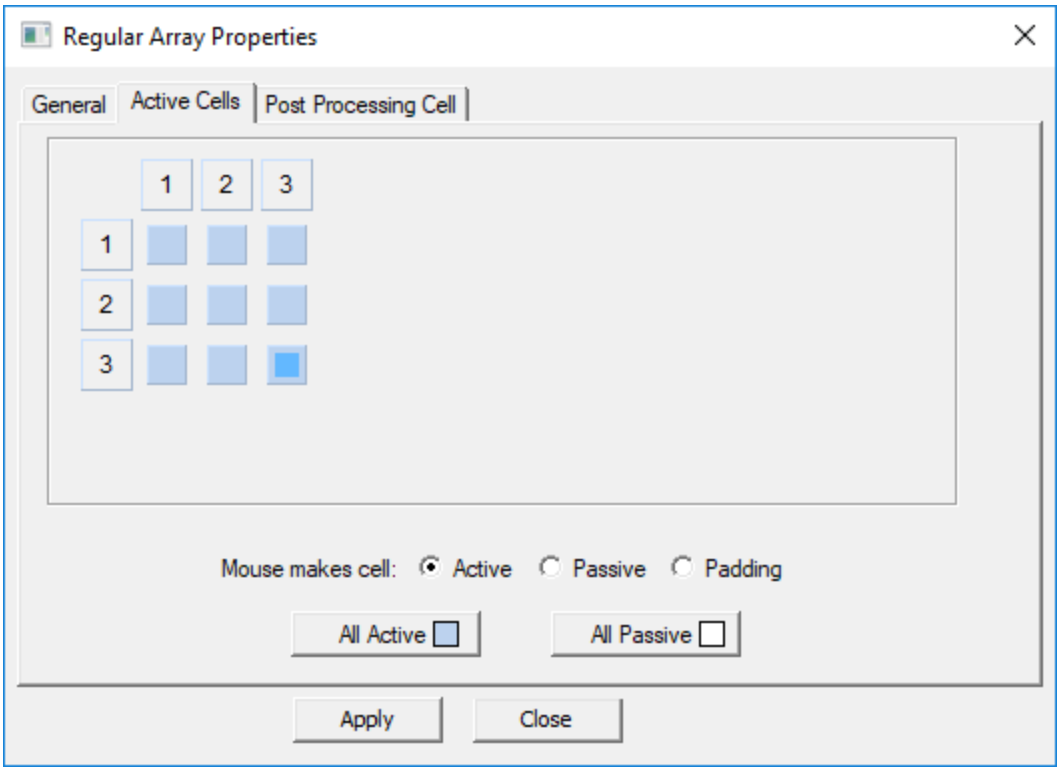
- If you have **Visible** enabled, you can see any changes.
- To Define Lattice Vector directions for A and B Vectors, select the different primary boundary from the drop down list for each..
- To define the array size, for the A and B vectors, specify the number of cells for each row and column, respectively.
- For example, the following figure shows the results when the initial Lattice Vectors are

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

swapped.

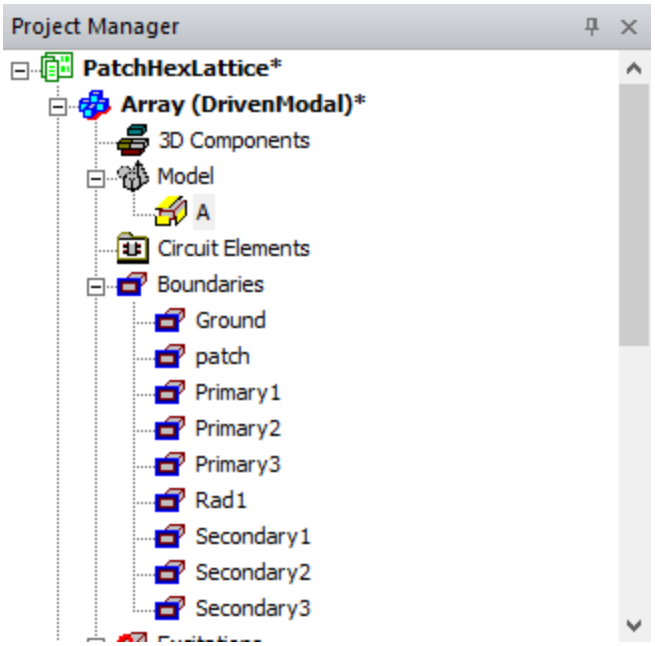


To create an irregular array, select the **Active Cells** tab, select the radio button for **Mouse makes cell** as **Padding**.

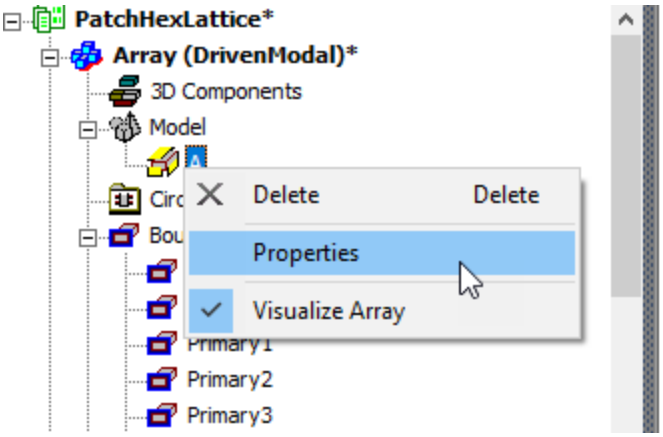


Once you have specified the parameters, the Array object appears in the Project tree under the Model. Only one Array is permitted for a model. The **Create Array** command is disabled if an array is defined.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

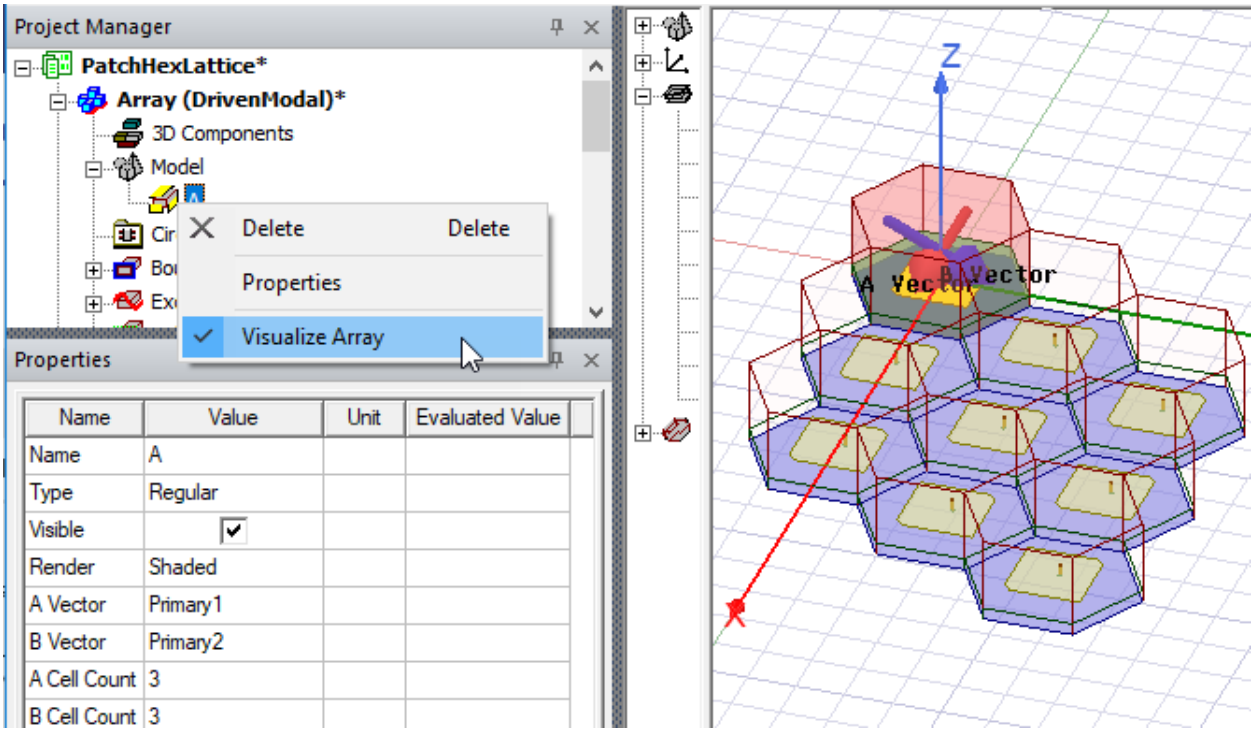


By selecting the Array icon in the **Project** tree, you right-click for the short cut menu.

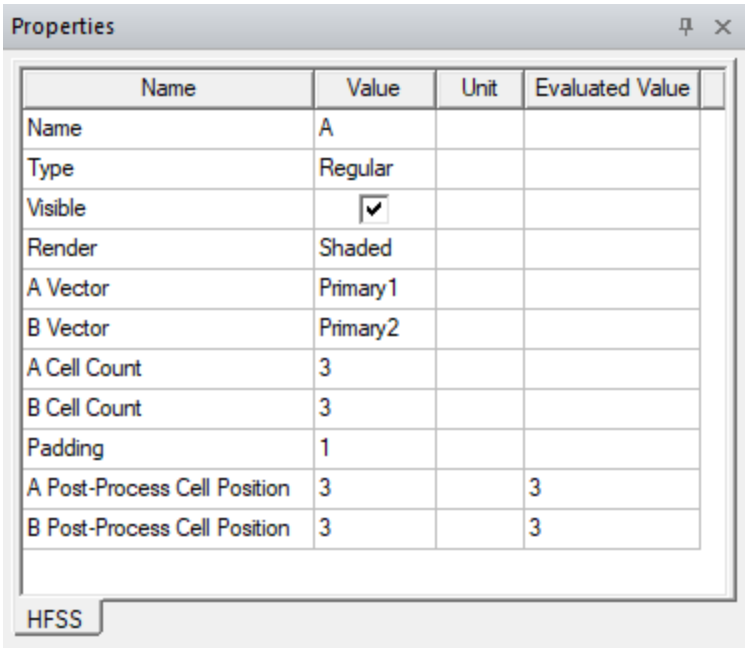


- Selecting **Delete** removes the array from the Model.
- Selecting **Properties** displays the **Regular Array** dialog.
- Selecting Visualize Array lets you toggle the array display. A check mark indicates that the array is being displayed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With the Array icon in the **Project** tree selected, if you have a docked Properties window displayed, you can see and edit Array Properties. All edits are undoable and informational messages will be posted to the Message window when design data is deleted.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

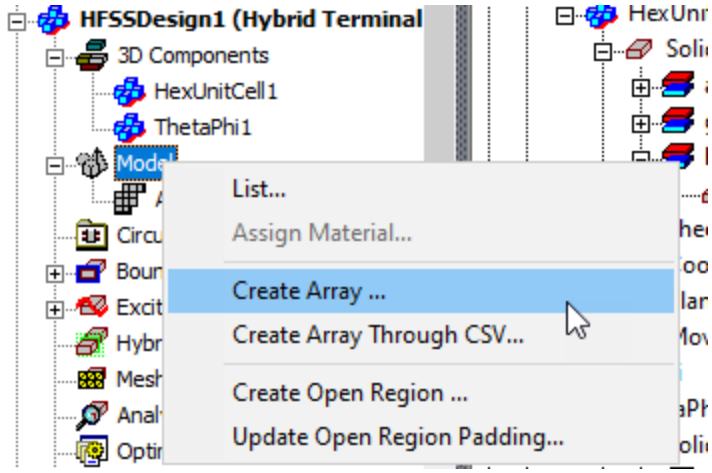
Multiple Finite Arrays (Beta)

For some HFSS 3D problems like Z-axis stacked up arrays, or multiple arrays on the same platform, the HFSS Multiple Component Arrays Beta feature allows multiple array setups with the same UI access as you use to create a single array. You must define each array center position by different relative coordinate systems, and each array has its own lattice vectors, component list and array mask.

Each array must be enclosed by an airbox. If multiple arrays have different airboxes, each airbox must be assigned as FEBI. Multiple arrays cannot have contact with lattice boundary faces.

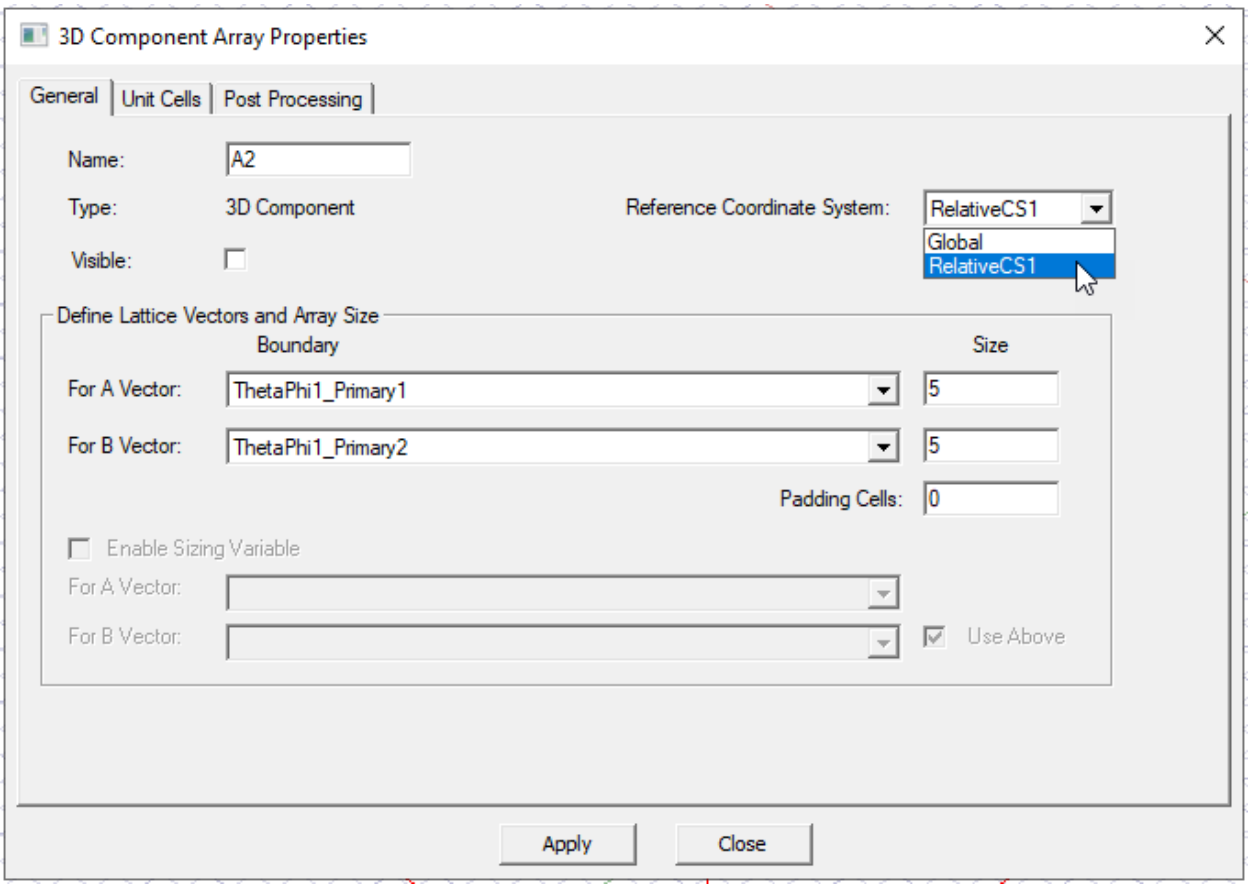
Creating Multiple Arrays

The process of creating each additional array uses the same commands as for creating a single array. Once you have a single array setup available, repeat the same array creation process by placing the Unit cell you want to use for the added array, and then select **Model > Create Array ...**



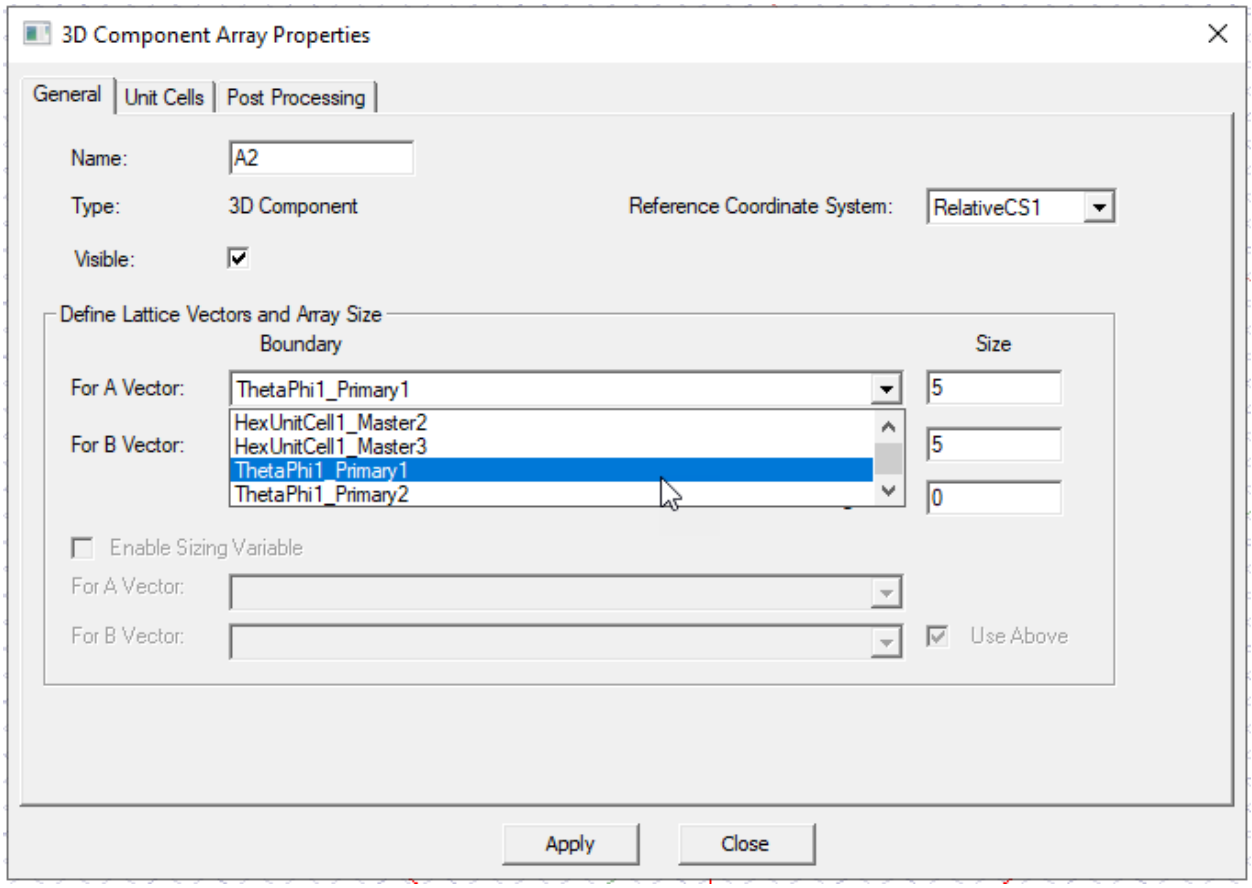
In the **3D Component Array Properties** dialog, first select the coordinate system that you associate with the Unit cell placement for the additional array.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



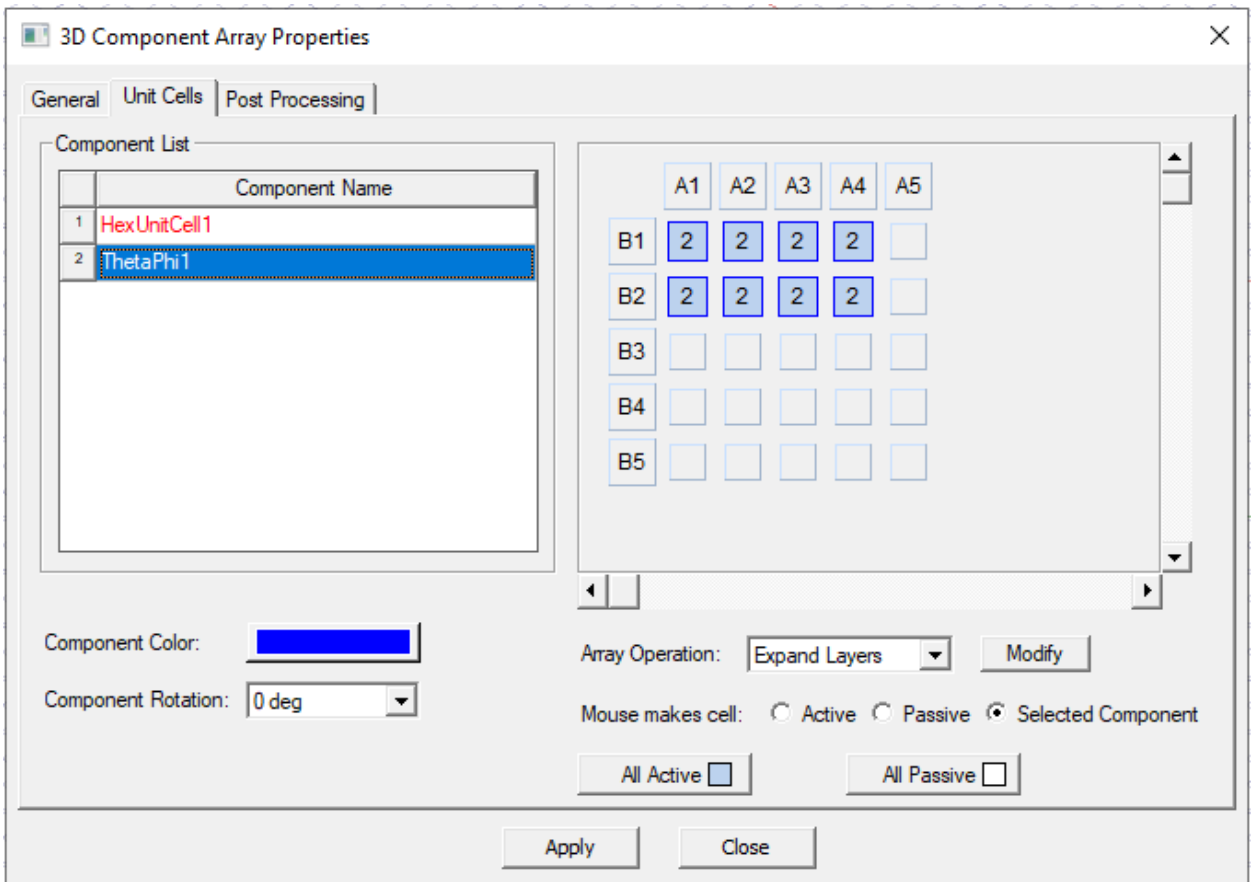
You can then select lattice vectors from available unit cell boundaries, and then define the array size and padding. A checkbox lets you make the array visible in the Modeler window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

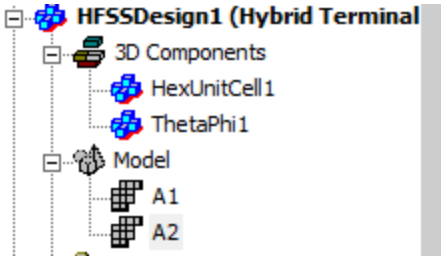


Then use the **Unit Cells** tab to select the array cells and define the component mask

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The added array appear in the Project tree and, if you have enabled visualization, in the Modeler window.



Create Array Through CSV

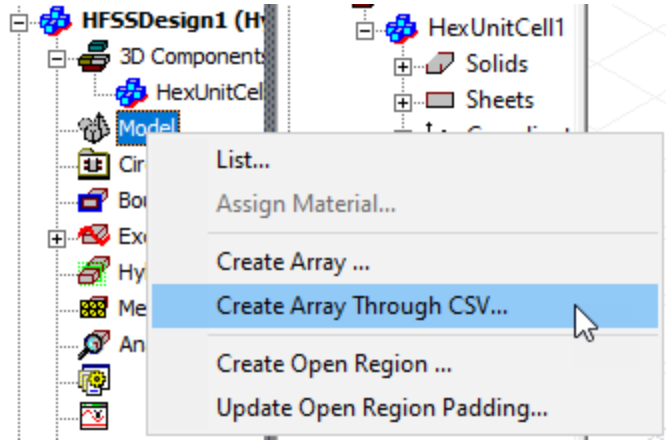
The **Create Array Through CSV...** command is enabled for Modal and Driven Terminal problems for the HFSS with Hybrid and Arrays Solution Types after you have assigned Primary and Secondary lattice pair boundaries to your unit cell model. This option is useful especially for

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

defining large-scale arrays when defining through the UI dialogs requires a lot of manual work. You can access the command in three ways: click **HFSS>Model>Create Array Through CSV...**, right-click the Model icon in the Project tree and select **Create Array Through CSV...** from the shortcut menu, or select the unit cell in the modeler window, and select **Create Array Through CSV...**from the short-cut menu. The **Export Array Though CSV** command section includes a description of the csv format.

Create Array Through CSV... Command Procedure

Right-click the Model icon in the Project tree for and select **Create Array Through CSV...** from the shortcut menu.

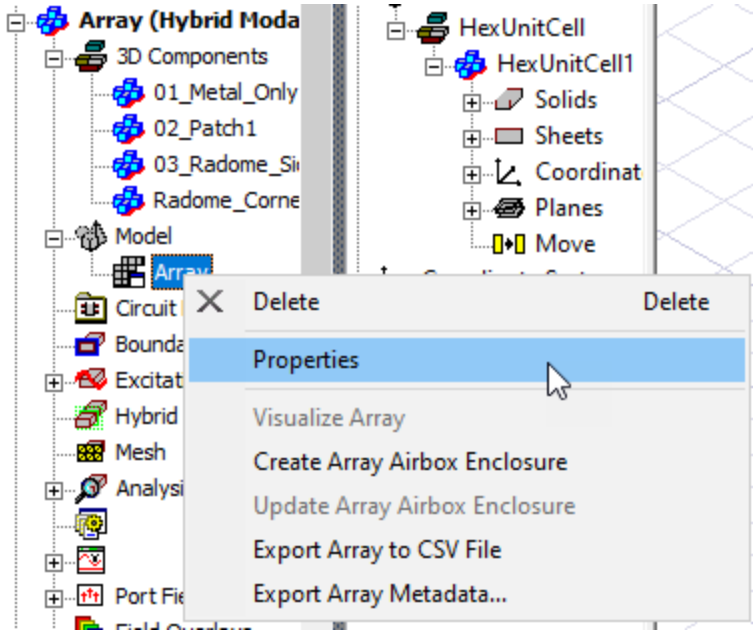


This opens a browser that you can use to navigate to and select the CSV file.

When you click Open, the array is created based on the .csv definition and the Array object appears in the Project tree under the Model. Only one Array is permitted for a model. The **Create Array Through CSV...** command is disabled if an array is defined.

By selecting the Array icon in the **Project** tree, you right-click for the short cut menu.

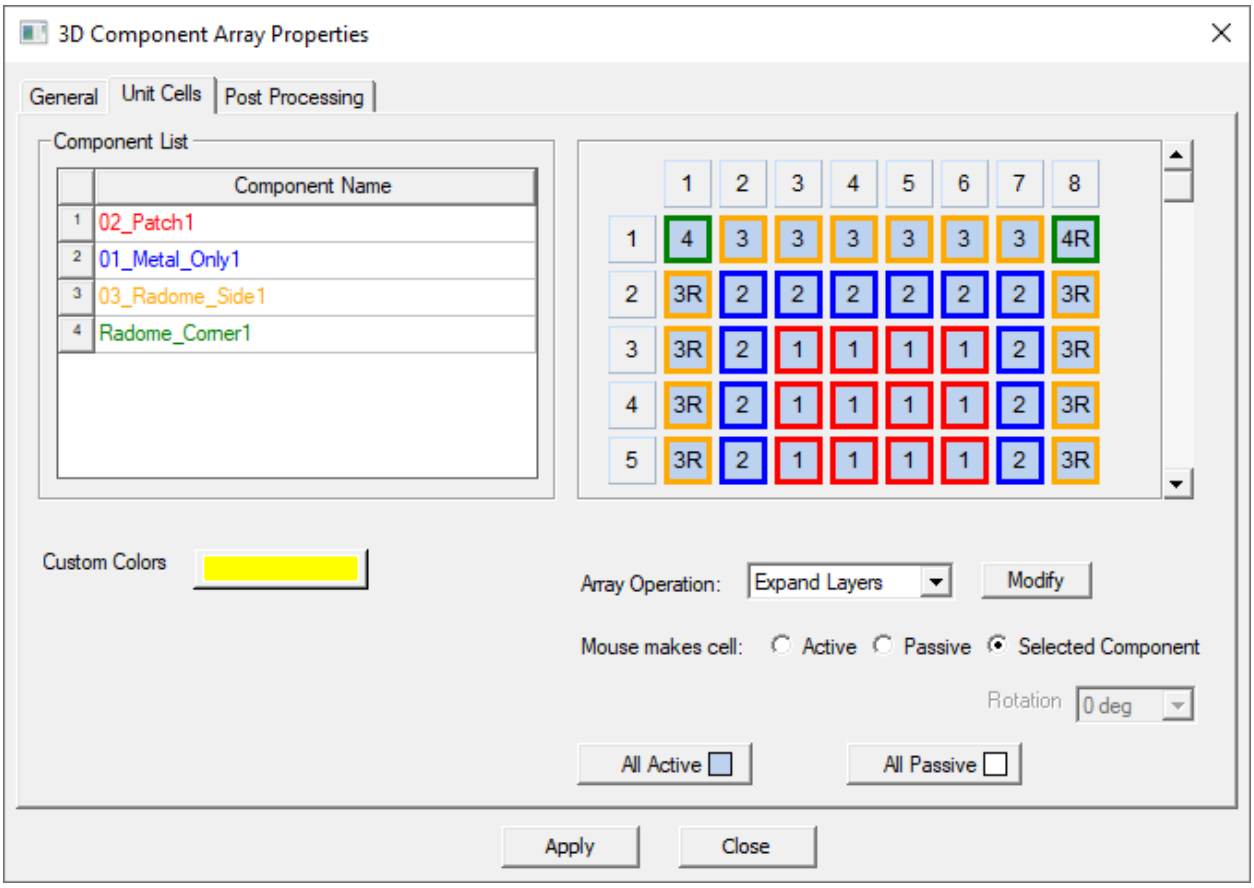
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Selecting **Delete** removes the array from the Model.

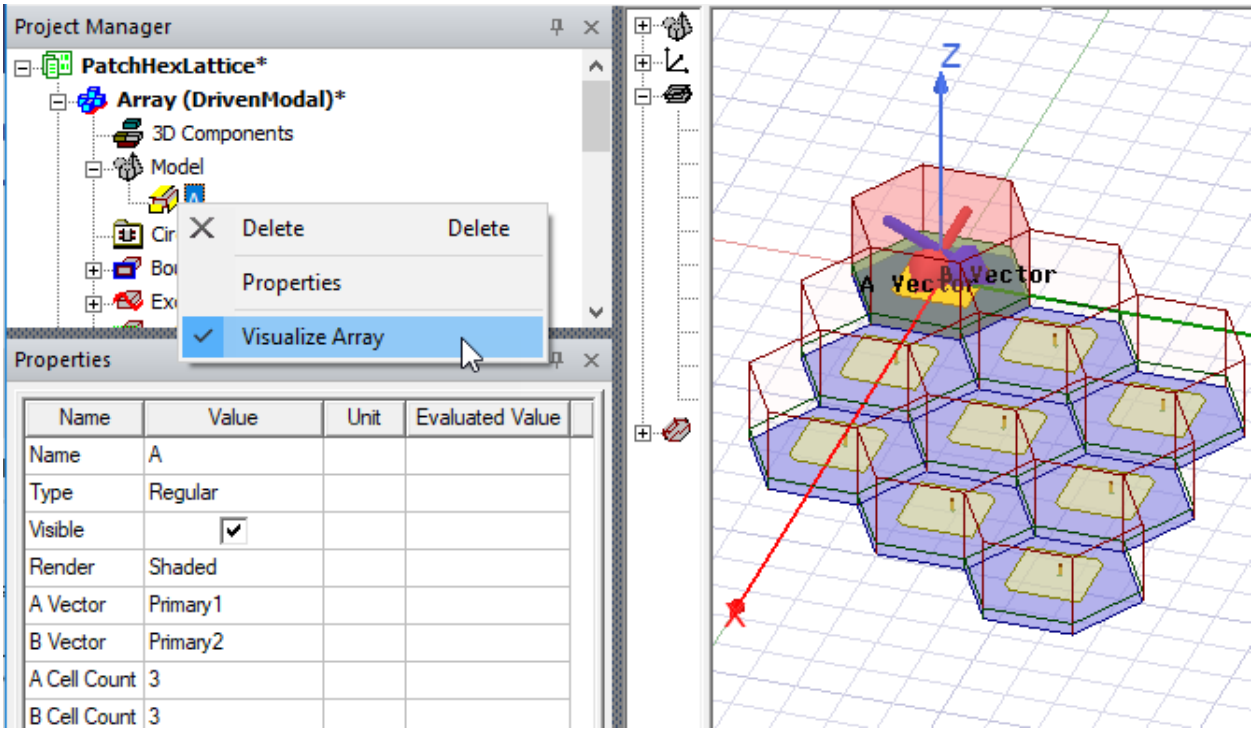
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Selecting **Properties** displays the **Regular Array** dialog. The Unit Cells tab shows the Component list for the array and the array definition.

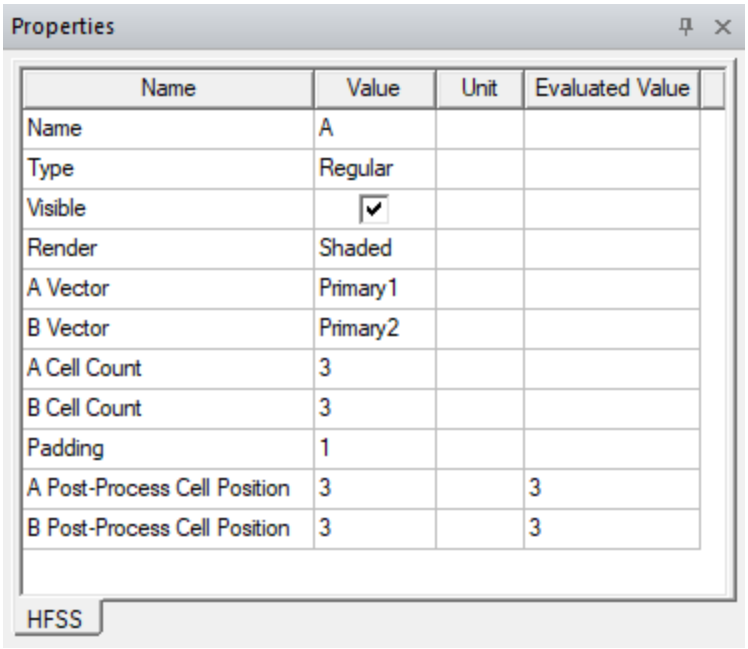


- Selecting Visualize Array lets you toggle the array display. A check mark indicates that the array is being displayed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With the Array icon in the **Project** tree selected, if you have a docked Properties window displayed, you can see and edit Array Properties. All edits are undoable and informational messages will be posted to the Message window when design data is deleted.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

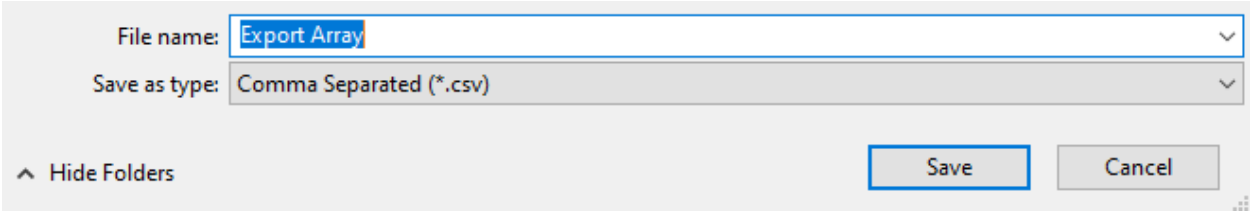
Exporting Array to CSV File

After you have defined an array, the **Export Array to CSV File** command is enabled for Modal and Driven Terminal problems for the HFSS with Hybrid and Arrays solution types. This option is especially useful for defining large-scale arrays when doing so through the UI would require a lot of manual work.

There are three ways to access the command:

- Select **HFSS** > **Model** > **Export Array to CSV File**.
- From the **Project Manager**, right-click **Model** and select **Export Array to CSV**.
- In the modeling window, right-click the unit cell and select **Export Array to CSV**.

Performing any of these three actions opens a browser you can use to navigate and select a location for CSV file:



Example CSV File

The exported file looks something like the one below. The imported file will be identical.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ArrayName	Array

Size Along A Vector	8
Size Along B Vector	8
A Vector	02_Patch1_LatticePair1
B Vector	02_Patch1_LatticePair2

Component Index	Component Name
1	02_Patch1
2	01_Metal_Only1
3	03_Radome_Side1
4	Radome_Corner1

Source Row	Source Column	Source Name	Magnitude	Phase
3	3	02_Patch1_1:1	ScanMag1*1	ScanPhase1 + (2*ScanPhaseShiftA + 2*ScanPhaseShiftB)
3	4	02_Patch1_1:1	ScanMag1*1	ScanPhase1 + (2*ScanPhaseShiftA + 3*ScanPhaseShiftB)
3	5	02_Patch1_1:1	ScanMag1*1	ScanPhase1 + (2*ScanPhaseShiftA + 4*ScanPhaseShiftB)
3	6	02_Patch1_1:1	ScanMag1*1	ScanPhase1 + (2*ScanPhaseShiftA + 5*ScanPhaseShiftB)
4	3	02_Patch1_1:1	ScanMag1*1	ScanPhase1 + (3*ScanPhaseShiftA + 2*ScanPhaseShiftB)
4	4	02_Patch1_1:1	ScanMag1*1	ScanPhase1 + (3*ScanPhaseShiftA + 3*ScanPhaseShiftB)
4	5	02_Patch1_1:1	ScanMag1*1	ScanPhase1 + (3*ScanPhaseShiftA + 4*ScanPhaseShiftB)
4	6	02_Patch1_1:1	ScanMag1*1	ScanPhase1 + (3*ScanPhaseShiftA + 5*ScanPhaseShiftB)

Array	Format: Component_index:Rotation_angle:Active_or_Passive						
4	4	3	3	3	3	3	4:90.0deg
3:270.0deg	2	2	2	2	2	2	3:90.0deg
3:270.0deg	2	1	1:0	1	1	2	3:90.0deg
3:270.0deg	2	1	1	1	1	2	3:90.0deg
3:270.0deg	2	1	1	1	1	2	3:90.0deg
3:270.0deg	2	1	1	1	1	2	3:90.0deg
3:270.0deg	2	2	2	2	2	2	3:90.0deg
4:270.0deg	3:180.0deg	3:180.0deg	3:180.0deg	3:180.0de	3:180.0de	3:180.0de	4:180.0deg

The CSV file contains four blocks:

- The first block provides the array name.
- The second block provides the A and B vector dimensions of the array, as well as vector IDs.
- The third block is for updating/editing source information:
  - Each line represents an update
  - The **Source Row** and **Source Column** fields identify cells that need to be updated.
  - The **Source Name** column shows the name of the source, formatted <SourceName>:<index>
  - The **Magnitude** and **Phase** columns show the weight of the sources. The default values are 0. If both values are 0, there is no need to list them here. If the magnitude

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and phase are variable, as in the example below, the units should be input as well:

```
ArrayName,A
X,Y
5,5
Component_index,Component_name
1,terminalComponent1
Source_row,Source_column,Souce,Magnitude,Phase
1,1,terminalComponent1_T1,1V,1deg
1,1,terminalComponent1_T2,2V,2deg
1,1,terminalComponent1_T3,3V,3deg
1,1,terminalComponent1_T4,4V,4deg
1,2,terminalComponent1_T1,6V,6deg
1,2,terminalComponent1_T2,5V,5deg
1,2,terminalComponent1_T3,7V,7deg
1,2,terminalComponent1_T4,8V,8deg
Array
1,1,1,,,
,,,,,
,,,,,
,,,,,
,,,,,
```

- The fourth block is the array matrix, with dimensions x and y. Each element in the matrix represents the information for one cell. The format for each cell is A:B:C where each value is optional:
  - A is the component index. B and C cannot be provided if there is no A.
  - B is the rotation angle (default: 0deg). B must always have the unit deg (for example, 1:90deg). Without the unit, A:B will be recognized as A:C.
  - C indicates whether a cell is passive. If a cell is passive, C is 0. Otherwise, it is active by default.

You can check scaling and magnitide assignments by using the Edit Source command.

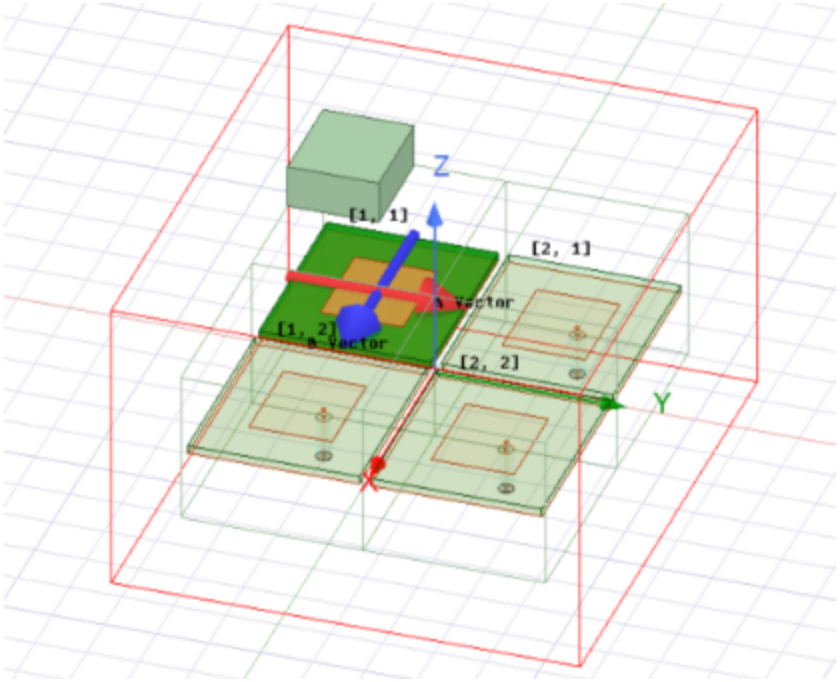
Create Array Command for 3D Component

The **Create Array...** command is enabled for Modal and Driven Terminal problems with the HFSS with Hybrid and Array Solution Type, after you have inserted a 3D Component that has assigned Primary and Secondary boundaries to your original unit cell model. 3D Component Array has non-conformal mesh interface between unit cells, which greatly reduces the memory

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

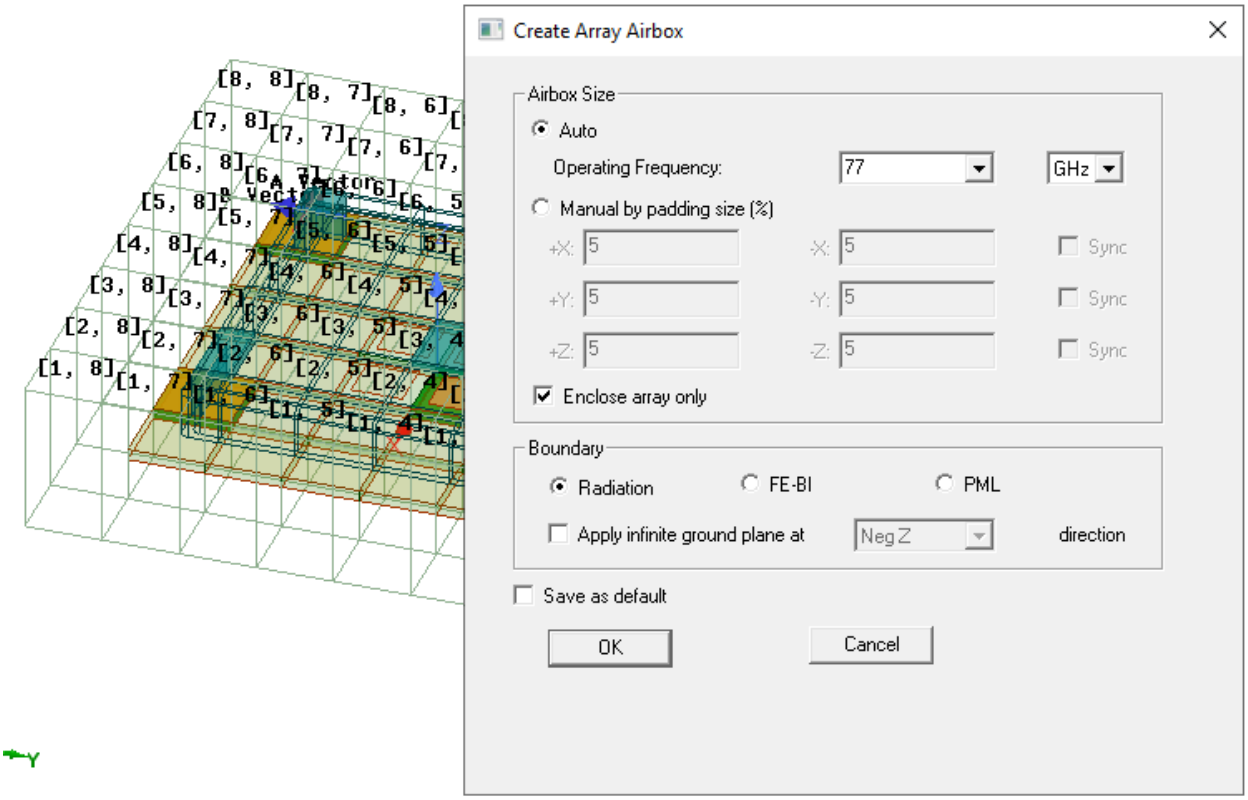
footprint and improves the performance. Once you have created a 3D Component Array, you can place it inside a region, and then assign a PML Boundary, a FEBI Hybrid Region, or a Radiation Boundary on the outside of the region. When a 3D component is placed inside of a region object, no native model object other than the region and its dependent objects are allowed.

The **Create Array Array Airbox Enclosure** command is enabled after you have used the **Create Array...** command to describe the array and designate active cells. Given an existing component array design, you can add arbitrary native FEM geometries around the component array, as long as all geometries are enclosed by an airbox. This feature greatly expands the use cases of existing component array definition.

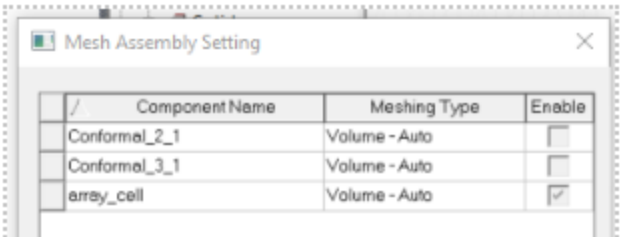


The **Create Array Airbox** dialog supports Auto definition of an airbox by frequency. It also has a checkbox to calculate airbox size based on either array itself, or array plus all FEM geometries.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

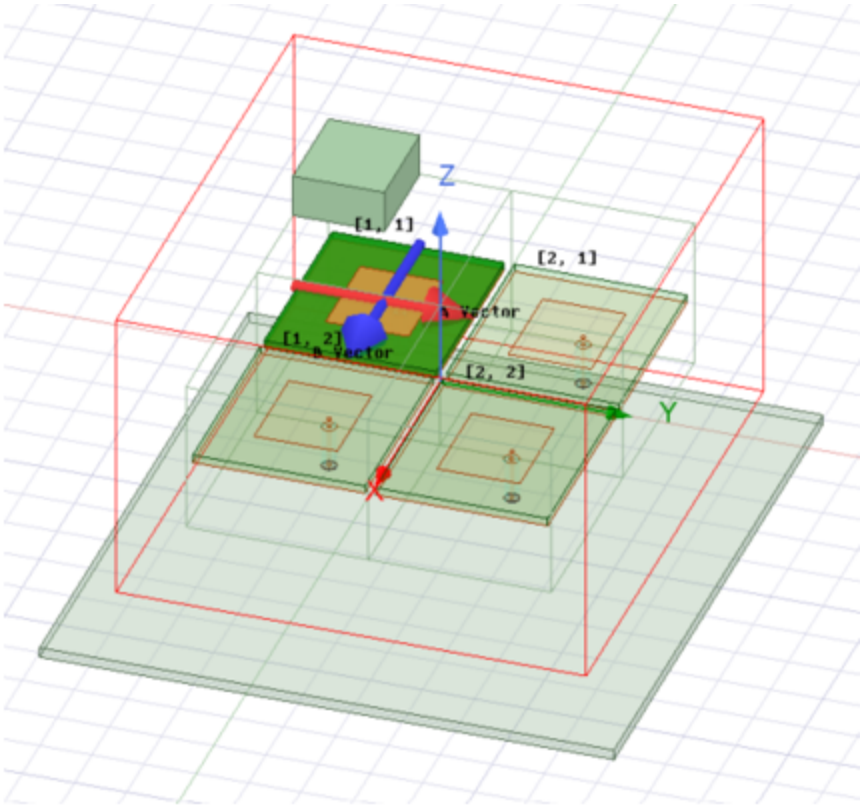


Component array design support non-array components (that is, components without at least two lattice boundaries). You can these components to array design just like native geometries. Mesh assembly is not supported in this case.



Component array design with surrounding geometries supports SBR+ regions. In this case, all geometries except SBR+ regions must be enclosed by an airbox. After you create a 3D Component array, you can define SBR+ geometry in 3D Component Array design to take advantage of leveraging the current component array and SBR+ techniques and workflows. This can provide significant enhancement to design and simulation efficiency. For use with SBR+ you must enclose the component array with an airbox and assign it as either FEBI, Radiation or PML.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



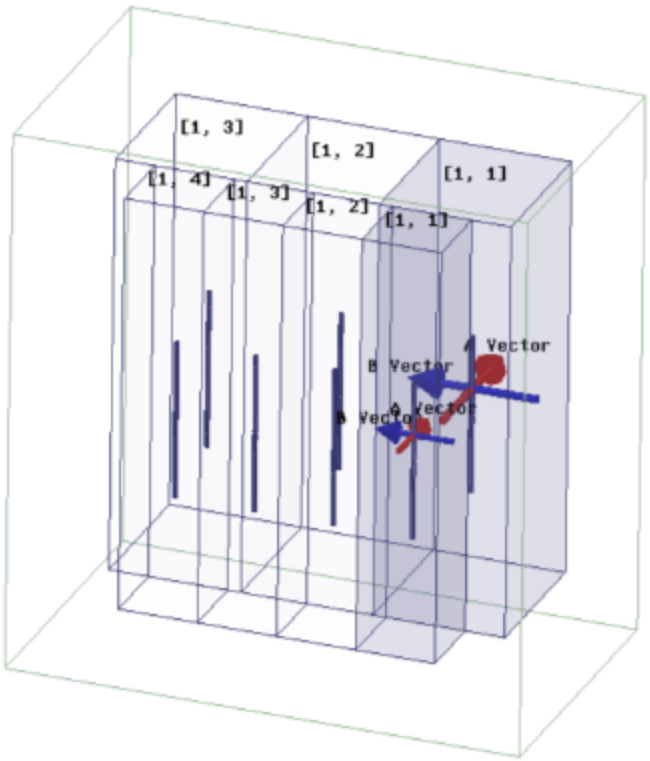
See the SBR+ and 3D Component Array section for the process, which including automatic airbox create/update, array reference CS, and hybrid coupling dialog.

Multiple Finite Arrays (Beta)

For some HFSS 3D problems like Z-axis stacked up arrays, or multiple arrays on the same platform, the HFSS Multiple Component Arrays Beta feature allows multiple array setups with the same UI access as you use to create a single array. You must define each array center position by different relative coordinate systems, and each array has its own lattice vectors, component list and array mask. You can define arrays within SBR+ Regions.

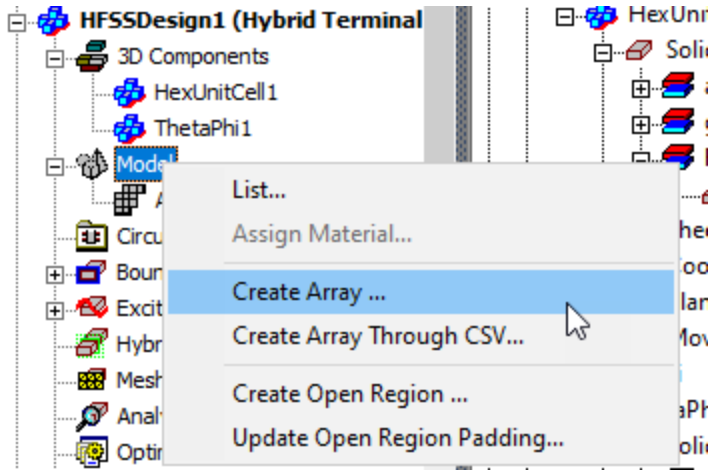
Each array must be enclosed by an airbox. If multiple arrays have different airboxes, each airbox must be assigned as FEBI. Multiple arrays can now have contact with lattice boundary faces.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



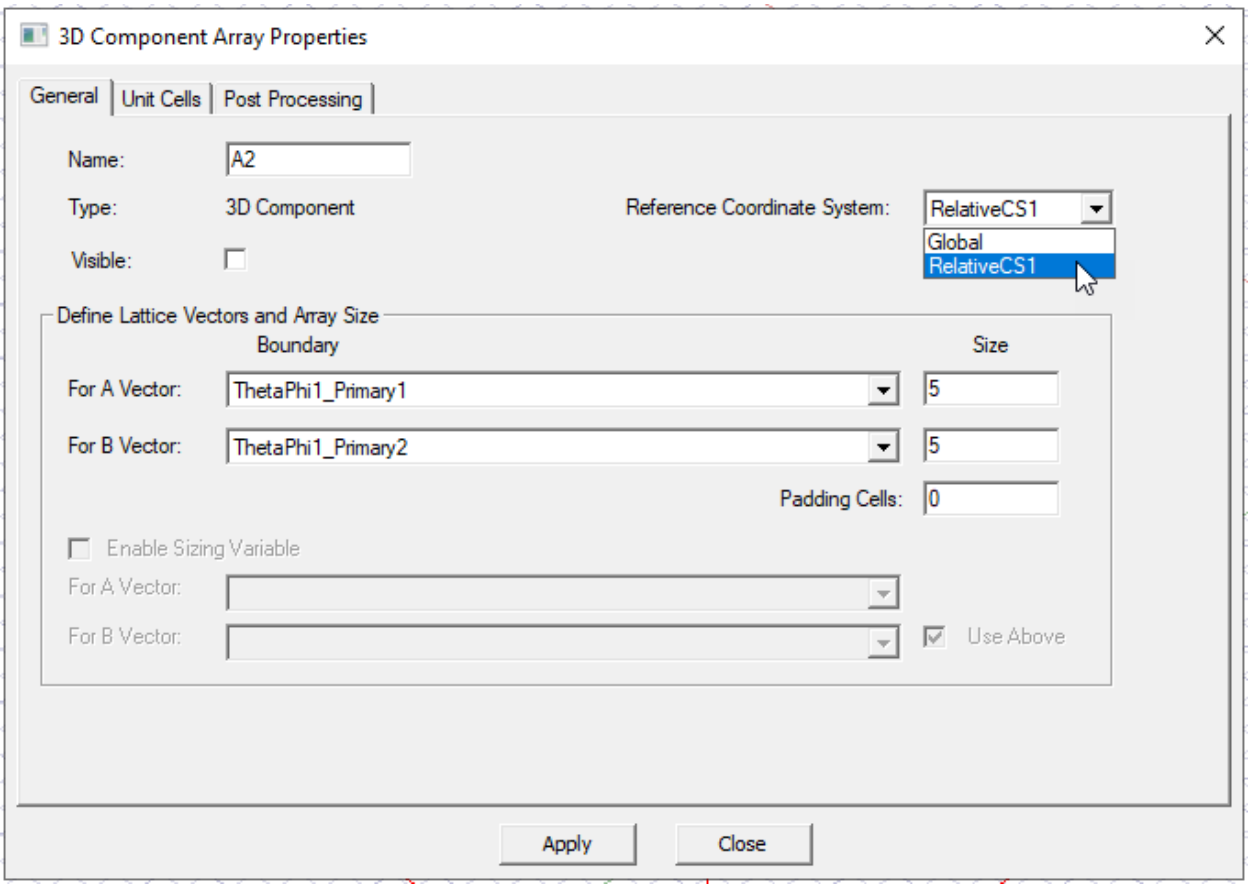
Creating Multiple Arrays

The process of creating each additional array uses the same commands as for creating a single array. Once you have a single array setup available, repeat the same array creation process by placing the Unit cell you want to use for the added array, and then select **Model > Create Array ...**



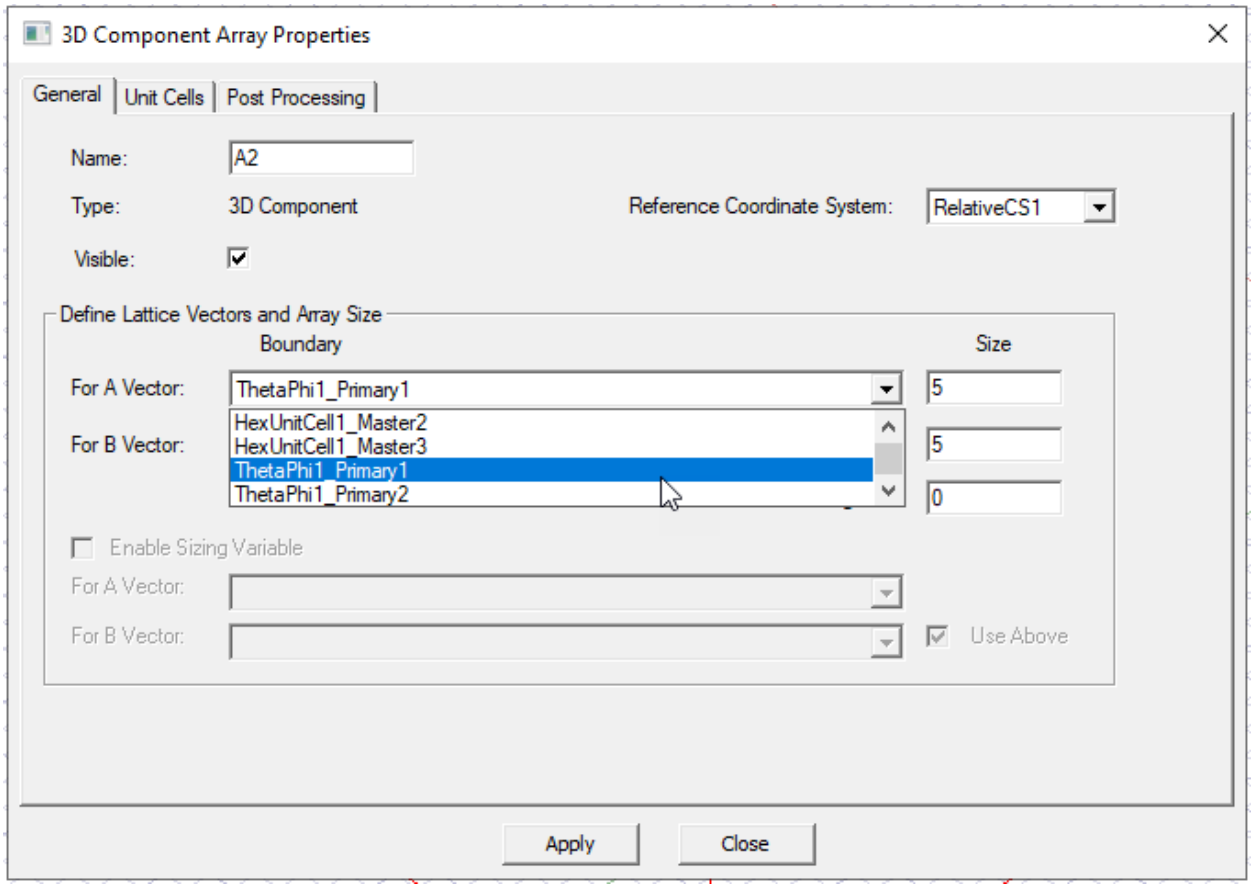
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **3D Component Array Properties** dialog, first select the coordinate system that you associate with the Unit cell placement for the additional array.



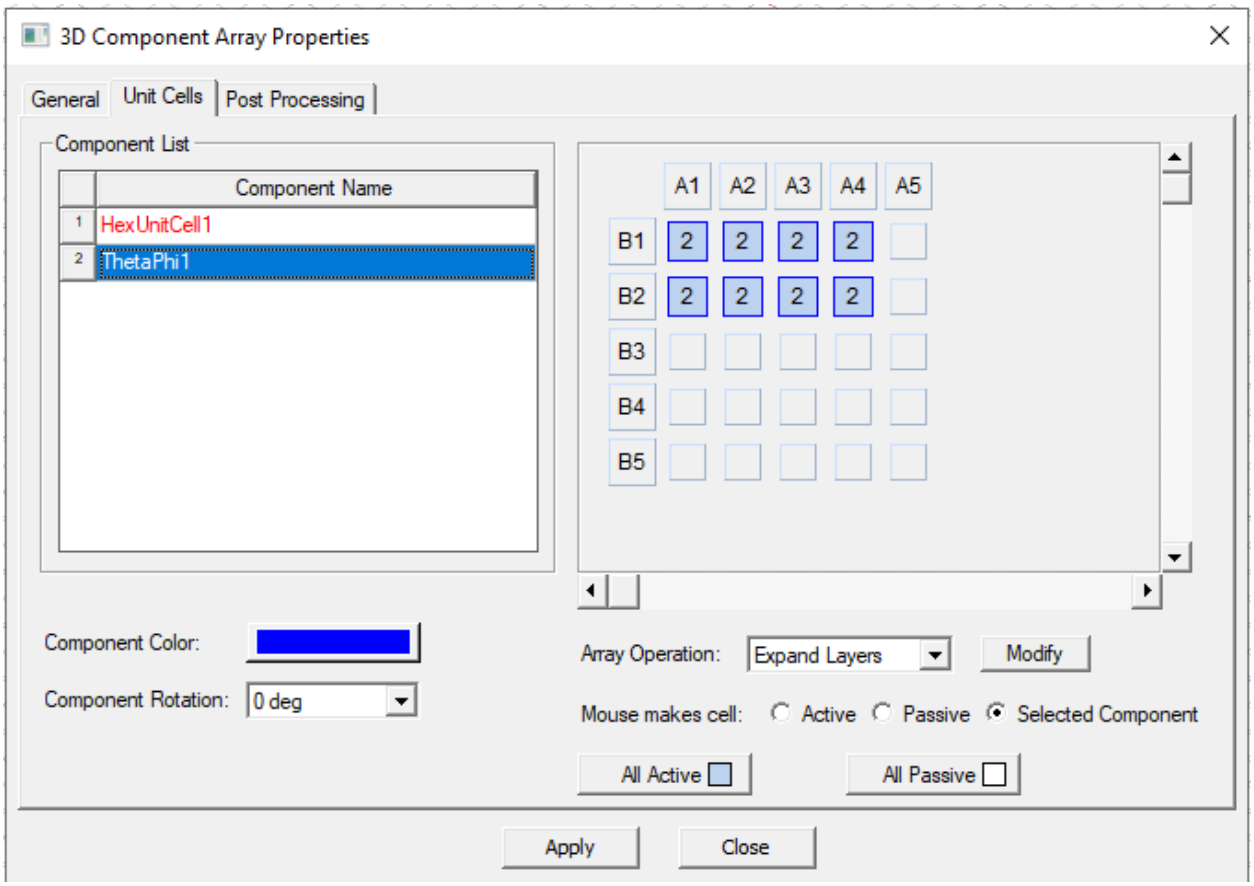
You can then select lattice vectors from available unit cell boundaries, and then define the array size and padding. A checkbox lets you make the array visible in the Modeler window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

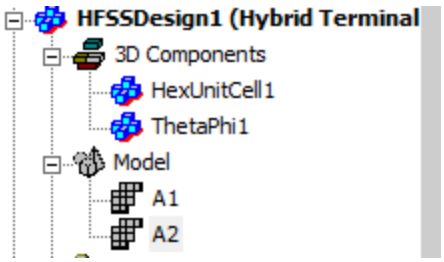


Then use the **Unit Cells** tab to select the array cells and define the component mask

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The added array appear in the Project tree and, if you have enabled visualization, in the Modeler window.



Copy/Paste for Arrays

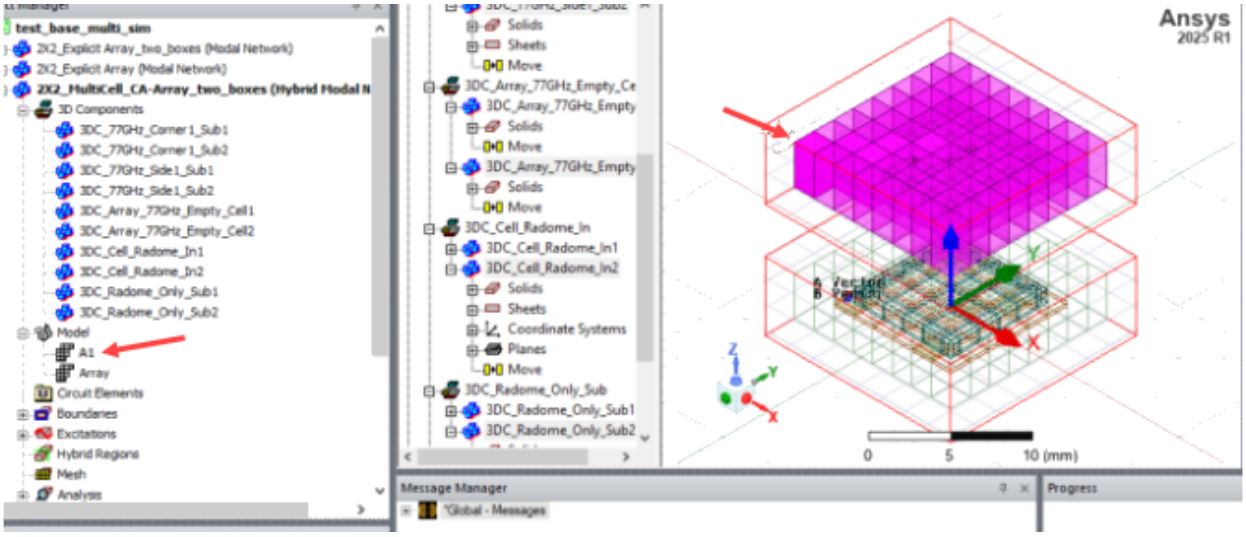
Since multiple arrays are supported in HFSS, it's natural to define new arrays based on existing arrays. HFSS supports array copy/paste like 3D component copy/paste. This operation first creates new array component instances and then duplicates array definitions, such as component mask and component rotation angles). By default, the pasted array uses the active coordinate system as the array reference coordinate system.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the Beta feature for Multiple Arrays is not enabled, the copy/paste functionality still works if the pasted design does not have an existing array.

Array Component Highlighting

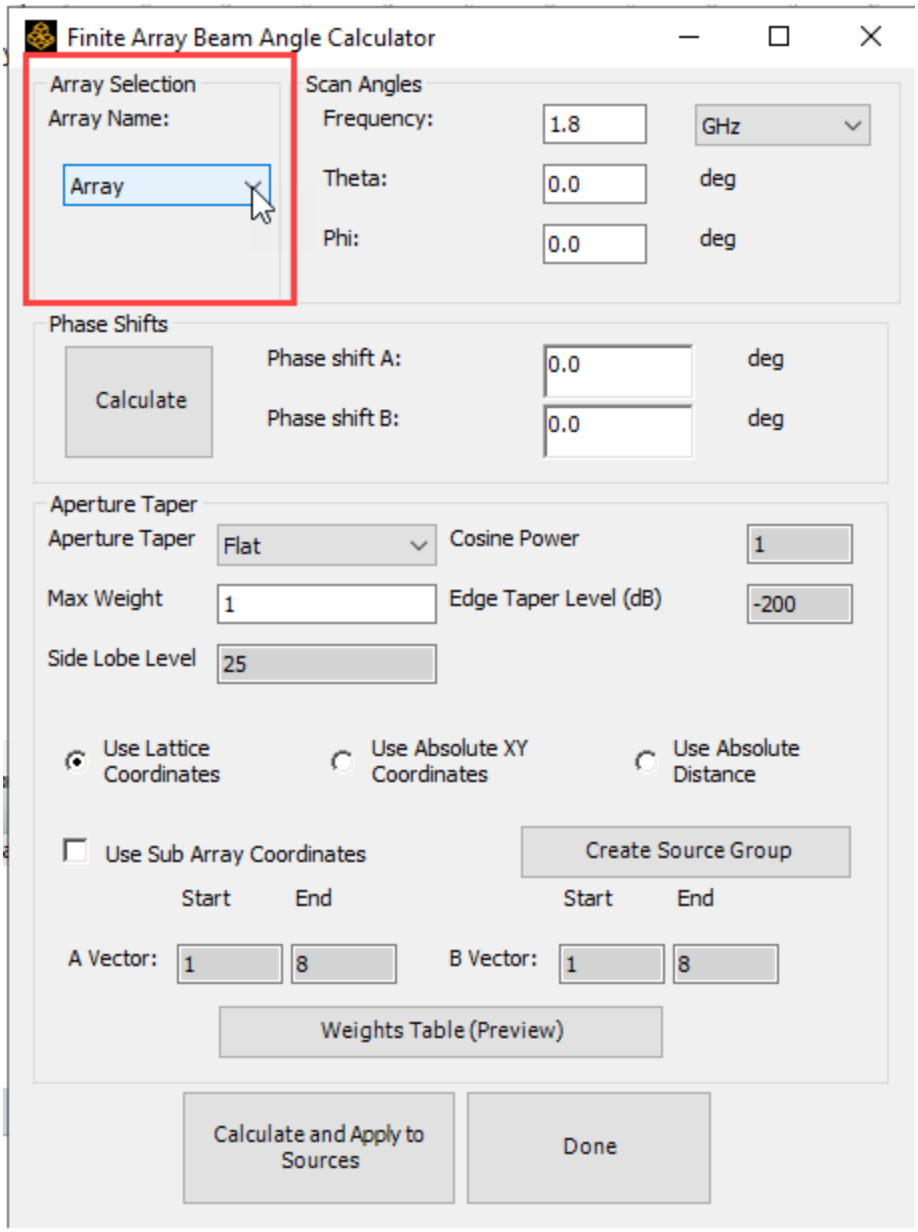
When you select an array item, all related components are highlighted. This new functionality works when the Beta feature for Multiple Arrays is off.



Multiple Arrays Beam Angle Calculator Toolkit

The Finite array beam angle calculator toolkit automatically calculates the edit source setting given aperture taper parameters and is very useful in array beam design. In 25.1 the feature extends to multiple array designs and each array can be calculated individually. When the Beta feature for Multiple Arrays is off, the new “Array Selection” control box still appears in the Toolkit, but only one array can be selected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

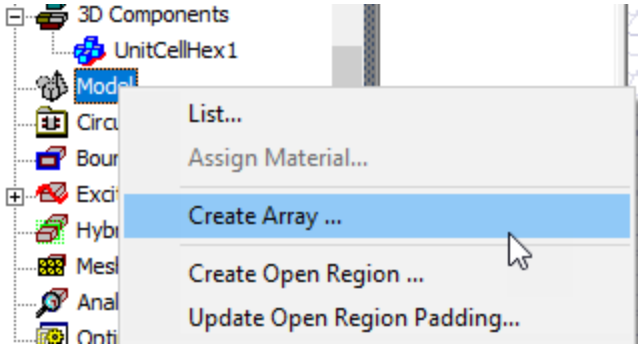


Create Array Command for 3D Component

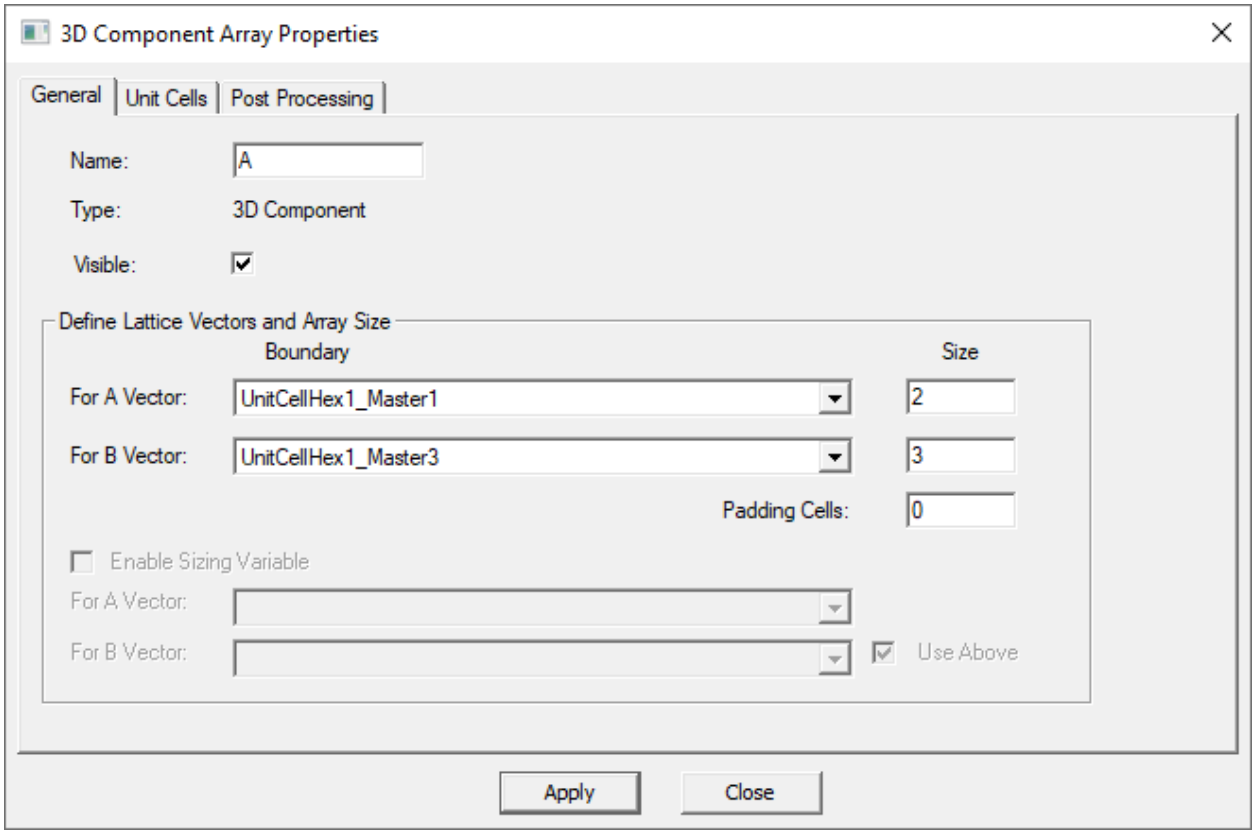
Create Array for 3D Component Process Flow

1. You can access the command in three ways: click **HFSS>Model>Create Array...**, right-click the Model icon in the Project tree and select ...**Create Array** from the shortcut menu, or select the 3D component in the modeler window, and select **Create Array...** from the short-cut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

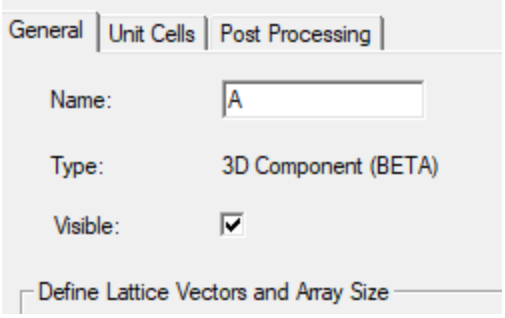


This displays the **3D Component Array Properties** dialog where you use the **General** tab to select the cell boundaries that define for Lattice Vectors, the number of cells in each vector, and any padding.

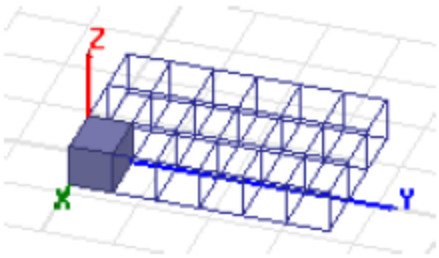


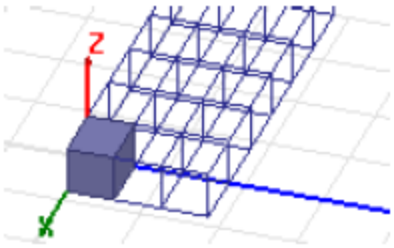
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you have **Visible** enabled, you can see any changes.



- To Define Lattice Vector directions for A and B Vectors, specify a different primary boundary for each. Drop down menus let you select from the available primary boundaries, two for square or trapezoidal components and three for hex components.
- To define the array size, for the A and B vectors, specify the number of cells for each row and column, respectively.
- The following figure shows the results when the initial Lattice Vectors are swapped.

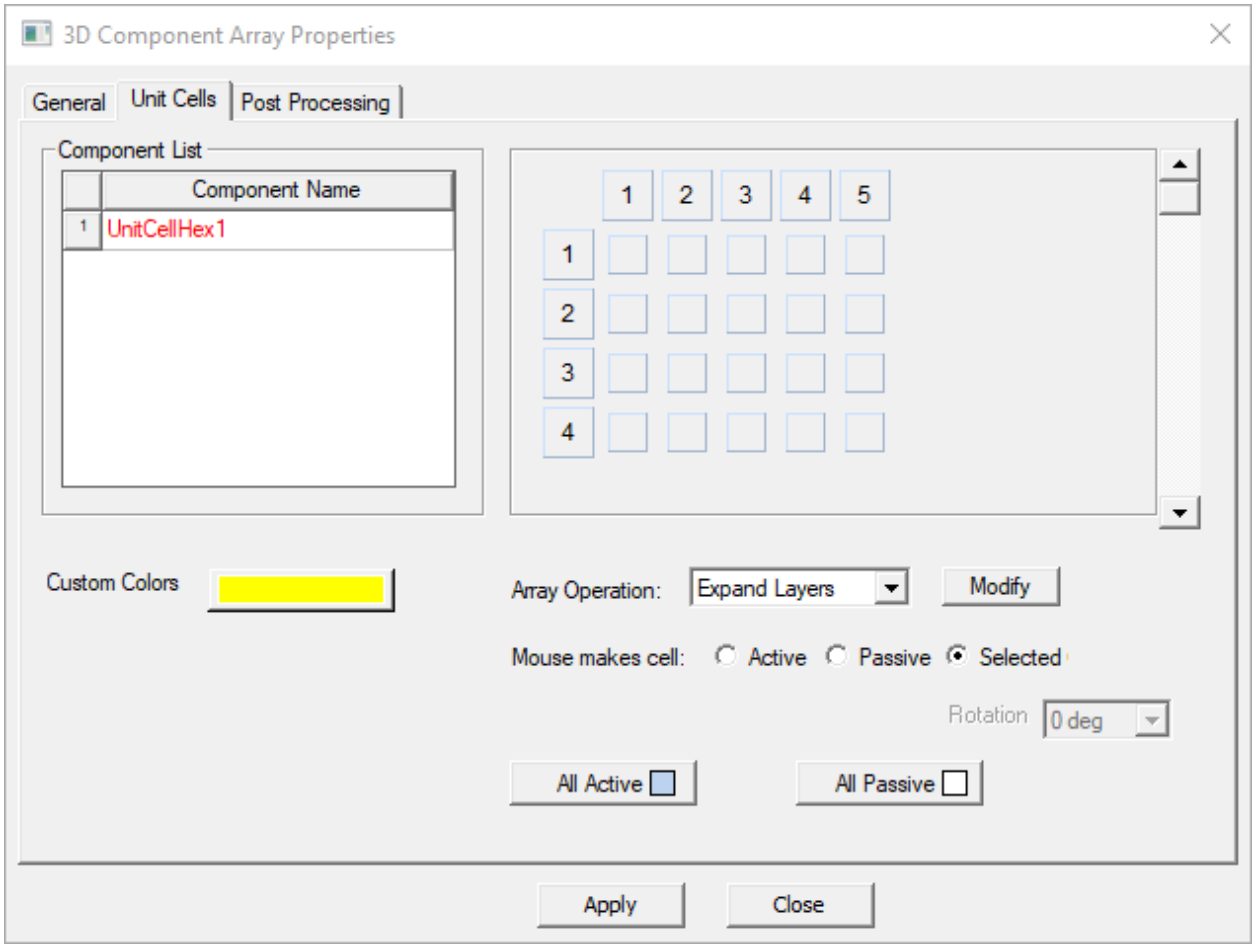




- If you have defined Design Variables for Length, the checkbox to Enable Sizing Variables is available. This feature should only be used when you create a new 3D Component. See Sizing Variables for New 3D Component Array.

2. Select the **Unit Cells** tab.

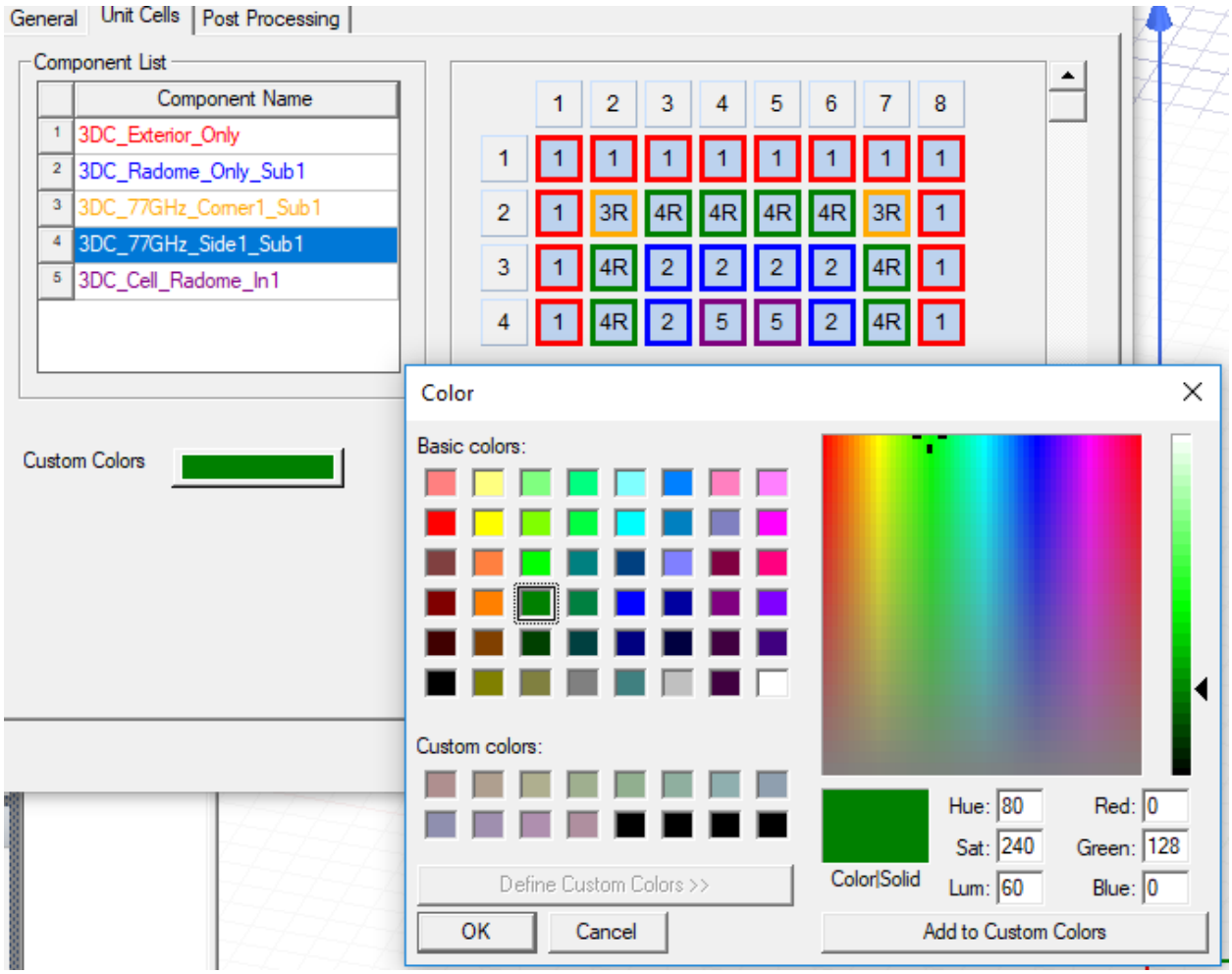
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The component list shows the available components. You must select at least one component as active.

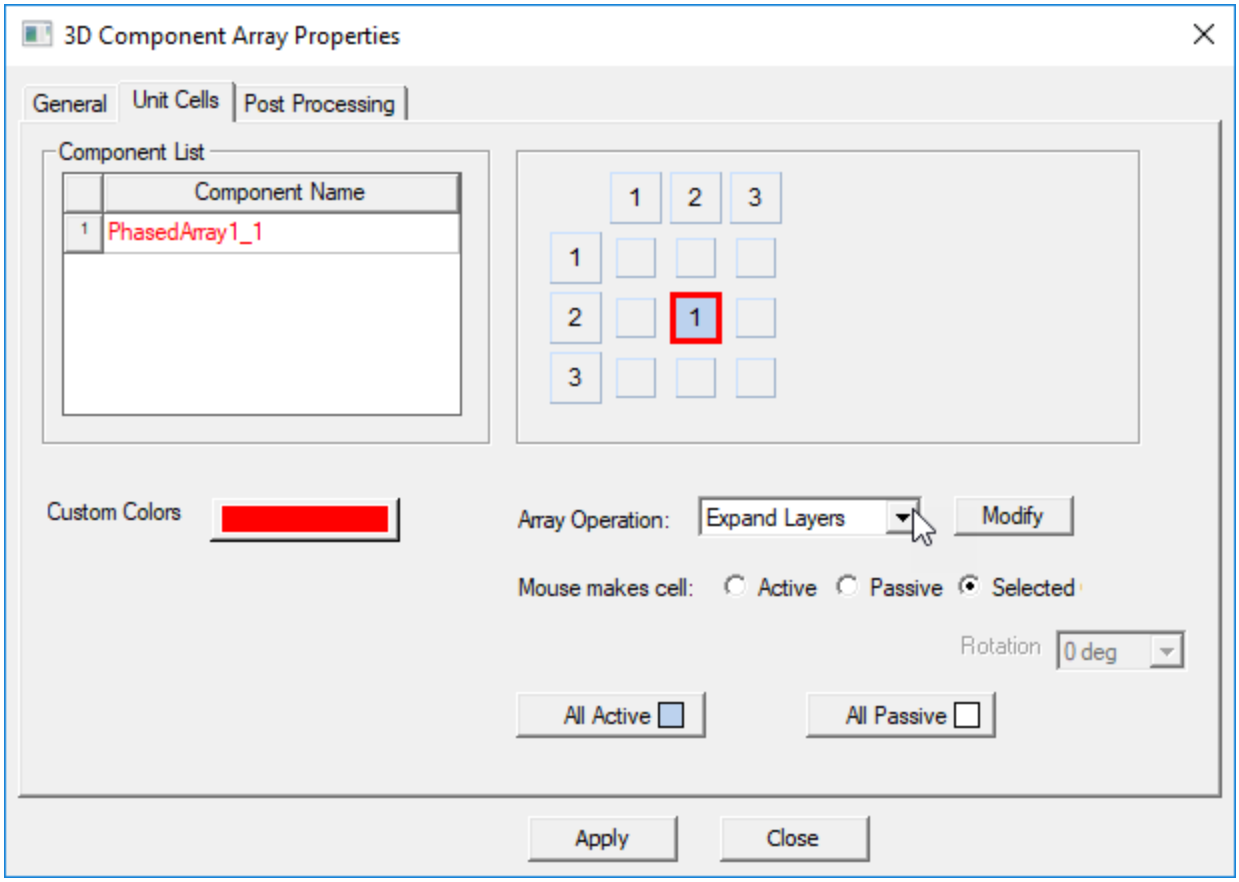
The **Custom Colors** button opens a dialog that lets you specify a color for how the selected component appears in the array.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

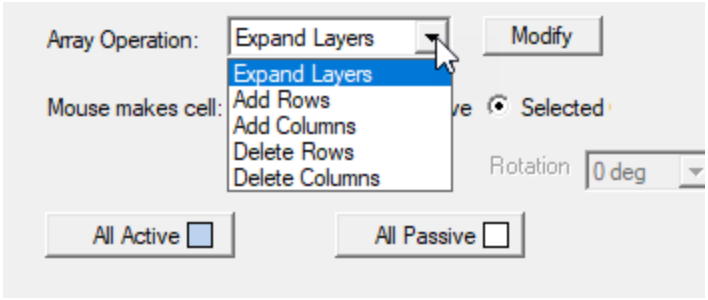


The selection pane will show the array as currently defined.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



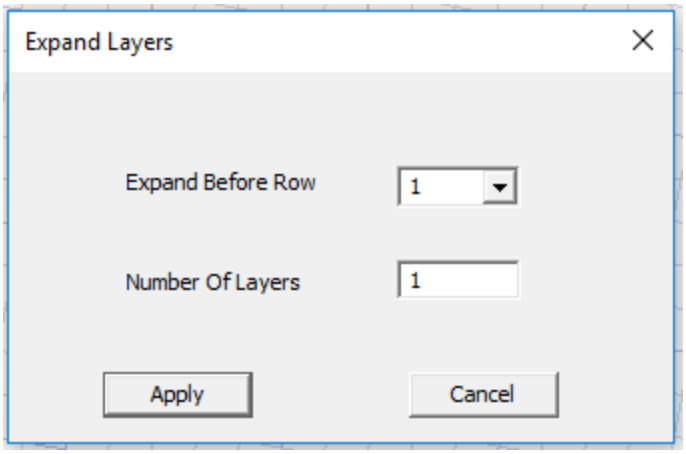
The Array Operation dropdown menu lets you select one of **Expand Layers**, **Add Rows**, **Add Columns**, **Delete Rows**, or **Delete Columns**.



Click **Modify** to open a dialog that lets you specify the location and parameters for the selected **Array Operation**. The component list shows the available components.

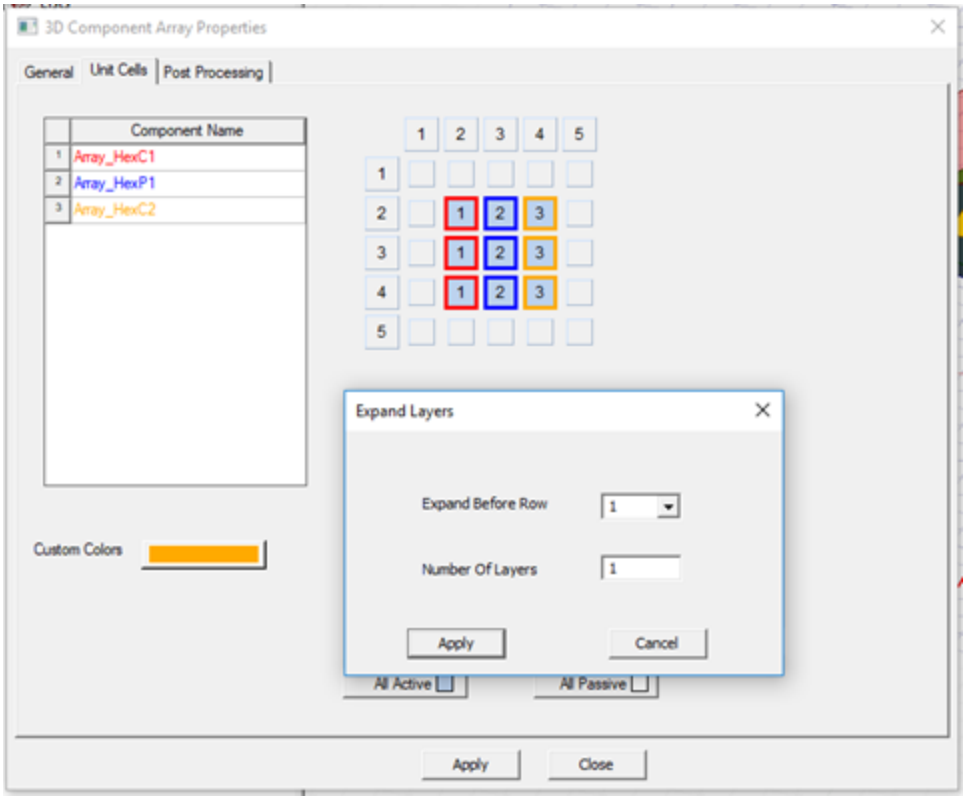
Expand Layers

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



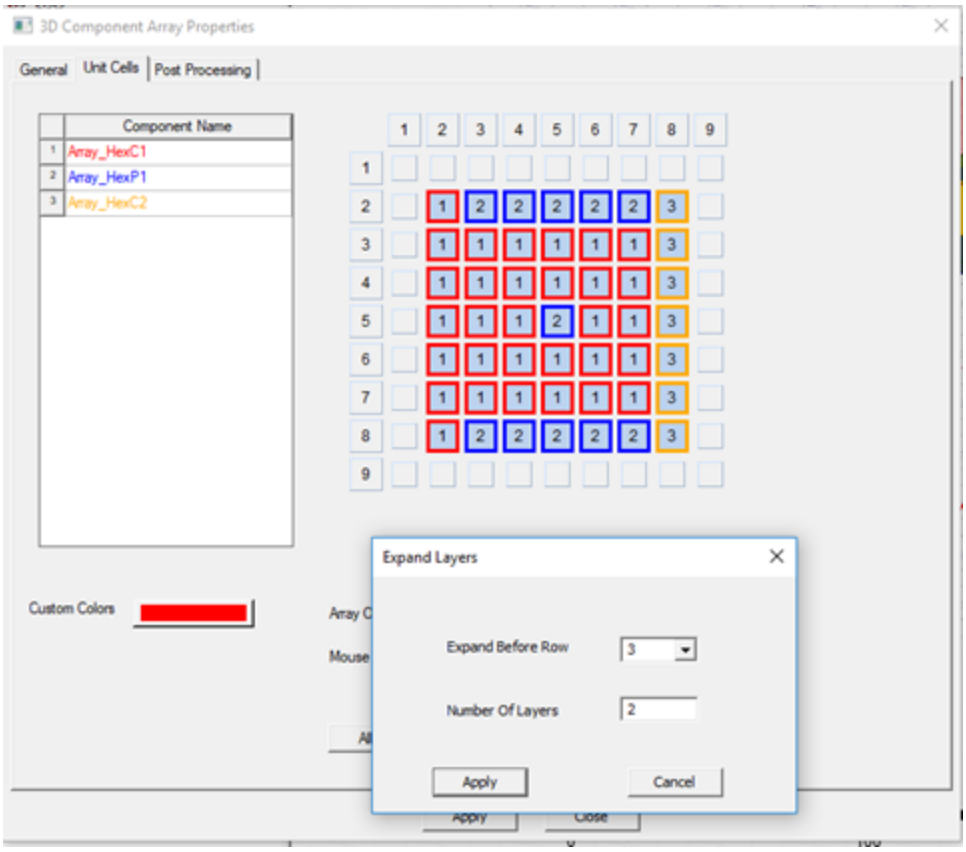
After selecting (or inputting) the row to start, inputting the number of layers to add, and clicking **Apply** button, the original array is expanded the corresponding number of layers before the inputted row from the center.

If no component name is selected in Component List, the newly added layers are filled in with air cells.

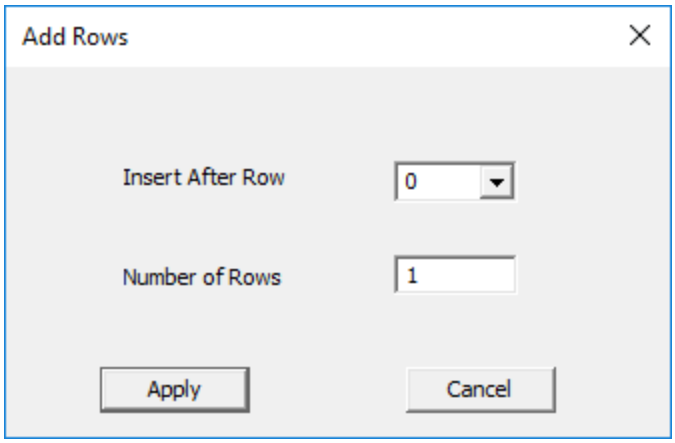


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If some component name is selected in Component List, the newly added layers are filled in with the selected component cells. The expanded layers have their middle cell repeated.



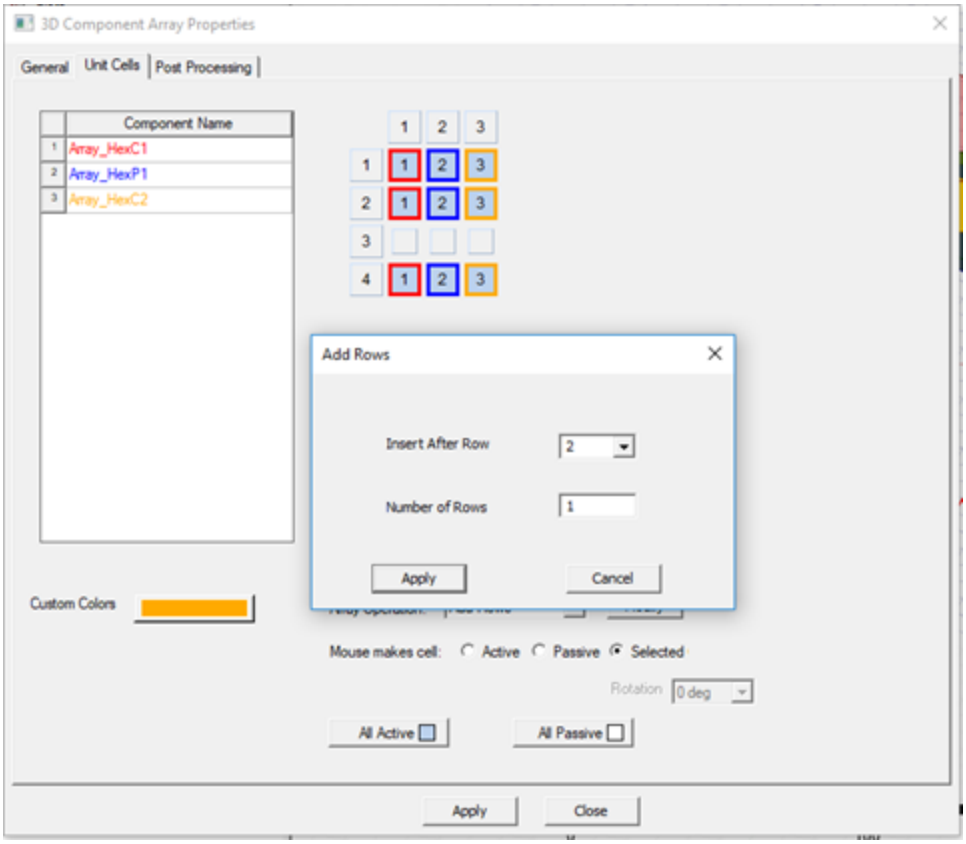
Add Rows



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

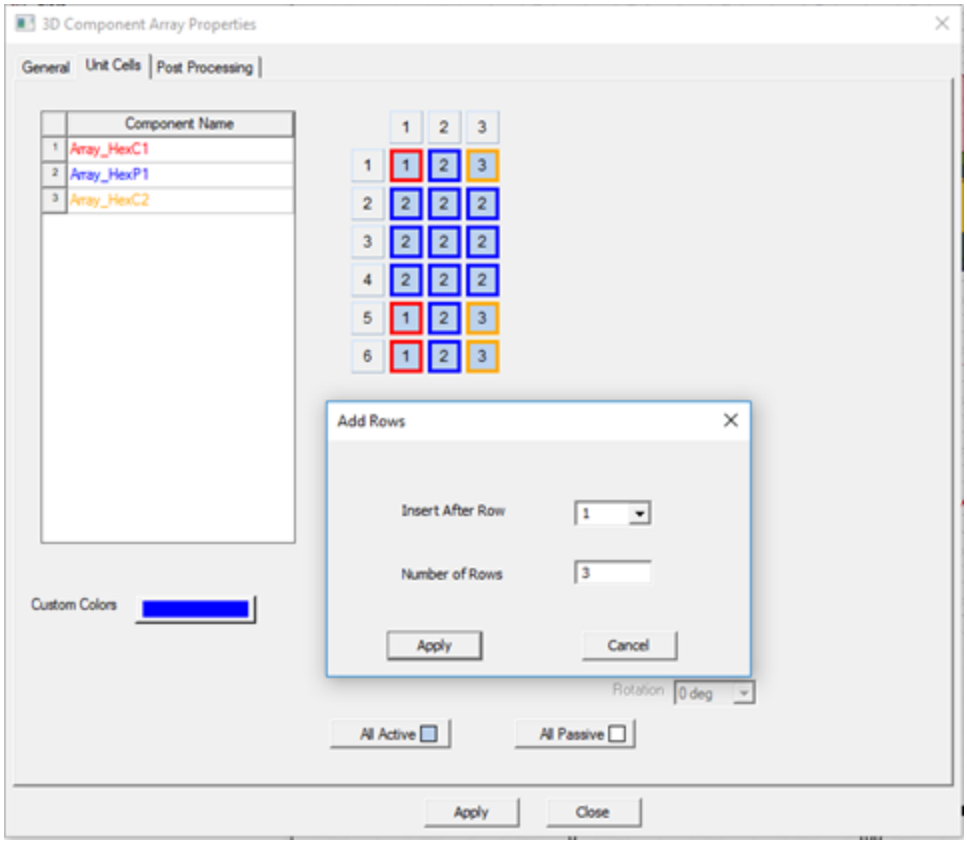
After selecting (or inputting) the row to start, inputting the number of rows to add, and clicking **Apply** button, the original array is added the inputted number of rows after the corresponding row.

If no component name is selected in Component List, the newly added rows are filled in with air cells.

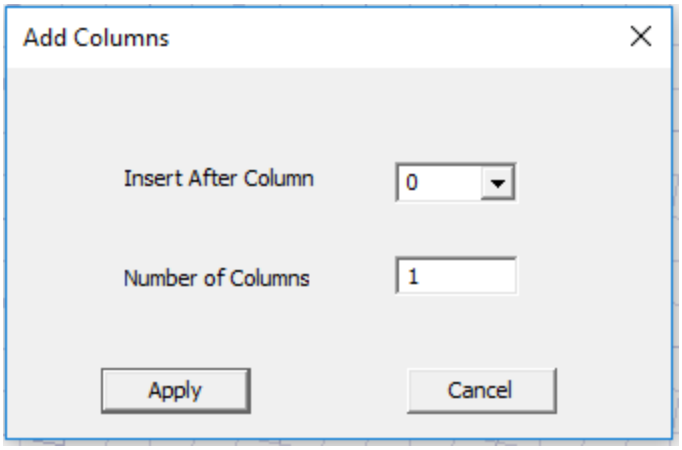


If some component name is selected in Component List, the newly added rows are filled in with the selected component cells.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



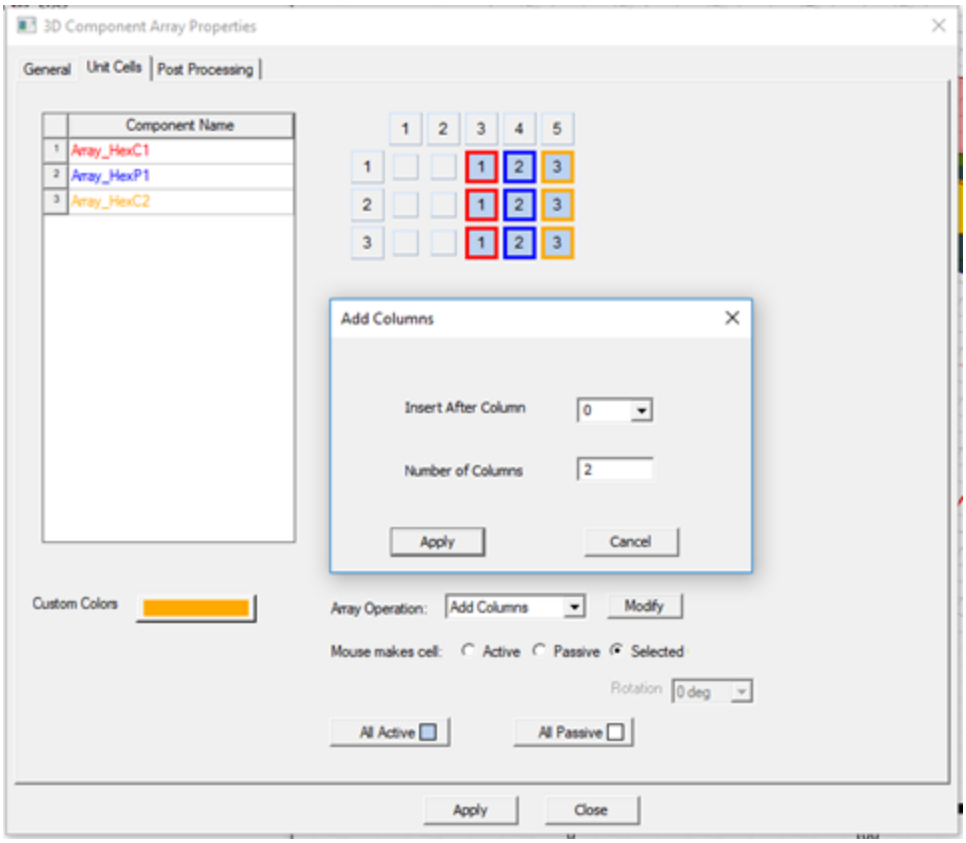
Add Columns



After selecting (or inputting) the column to start, inputting the number of columns to add, and clicking Apply button, the original array is added the inputted number of columns after the corresponding column.

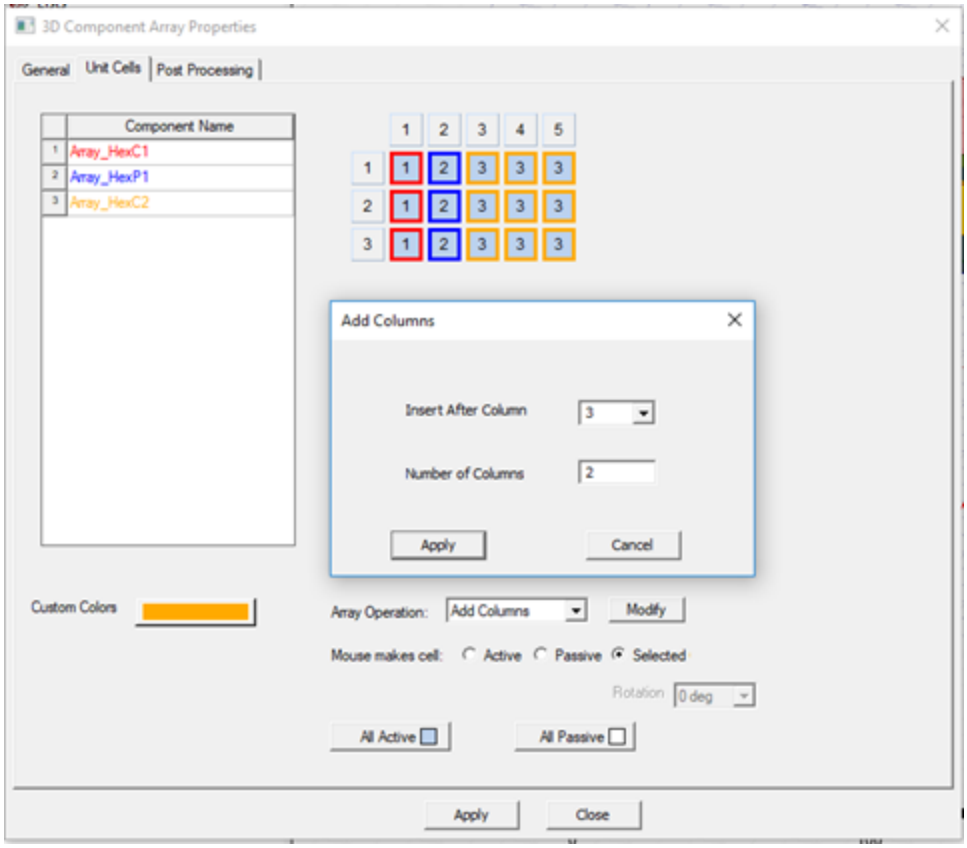
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If no component name is selected in Component List, the newly added columns are filled in with air cells.

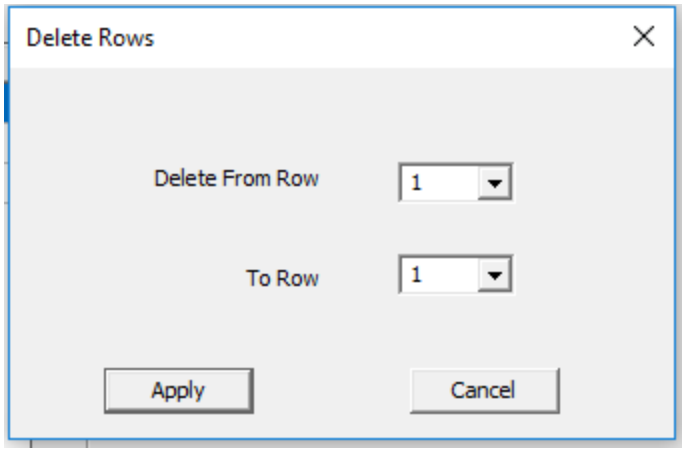


If some component name is selected in Component List, the newly added columns are filled in with the selected component cells.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

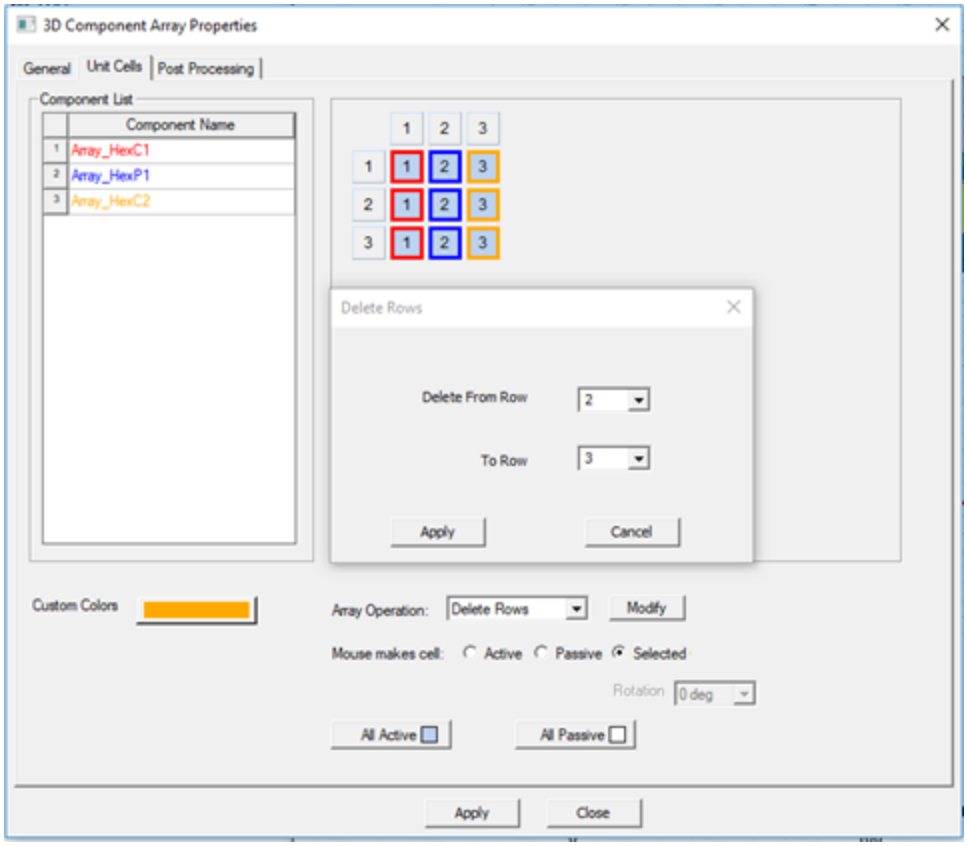


Delete Rows



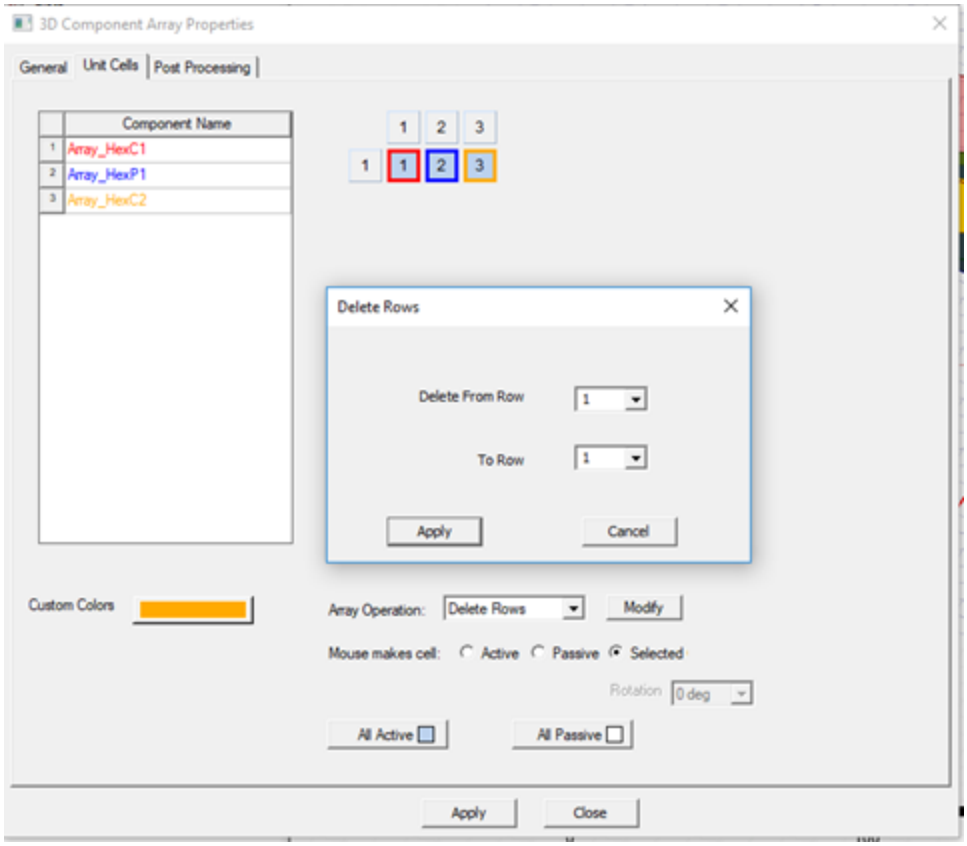
After selecting (or inputting) the row to start, selecting (or inputting) the row to end, and clicking **Apply** button, the original array is deleted rows within the range (include start and end rows). After deletion, if the inputted number of “Delete To Row” is larger than the row size of the new array, the two inputted numbers are reset to 1.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

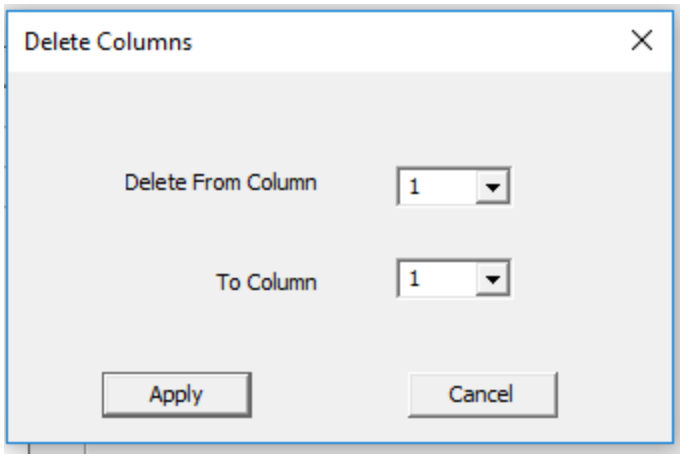


After selecting (or inputting) the row to start, selecting (or inputting) the row to end, and clicking Apply button, the original array is deleted rows within the range (include start and end rows). After deletion, if the inputted number of “Delete To Row” is larger than the row size of the new array, the two inputted numbers are reset to 1.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



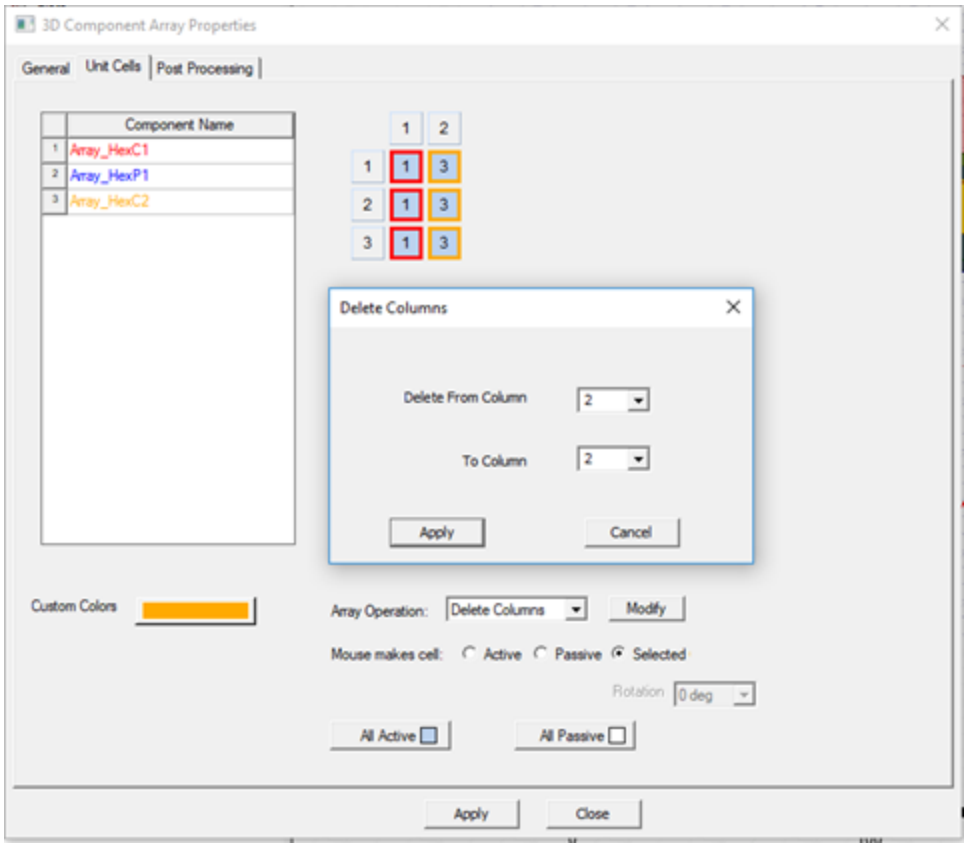
Delete Columns



After selecting (or inputting) the column to start, selecting (or inputting) the column to end, and clicking **Apply**, the original array is deleted columns within the range (include start and

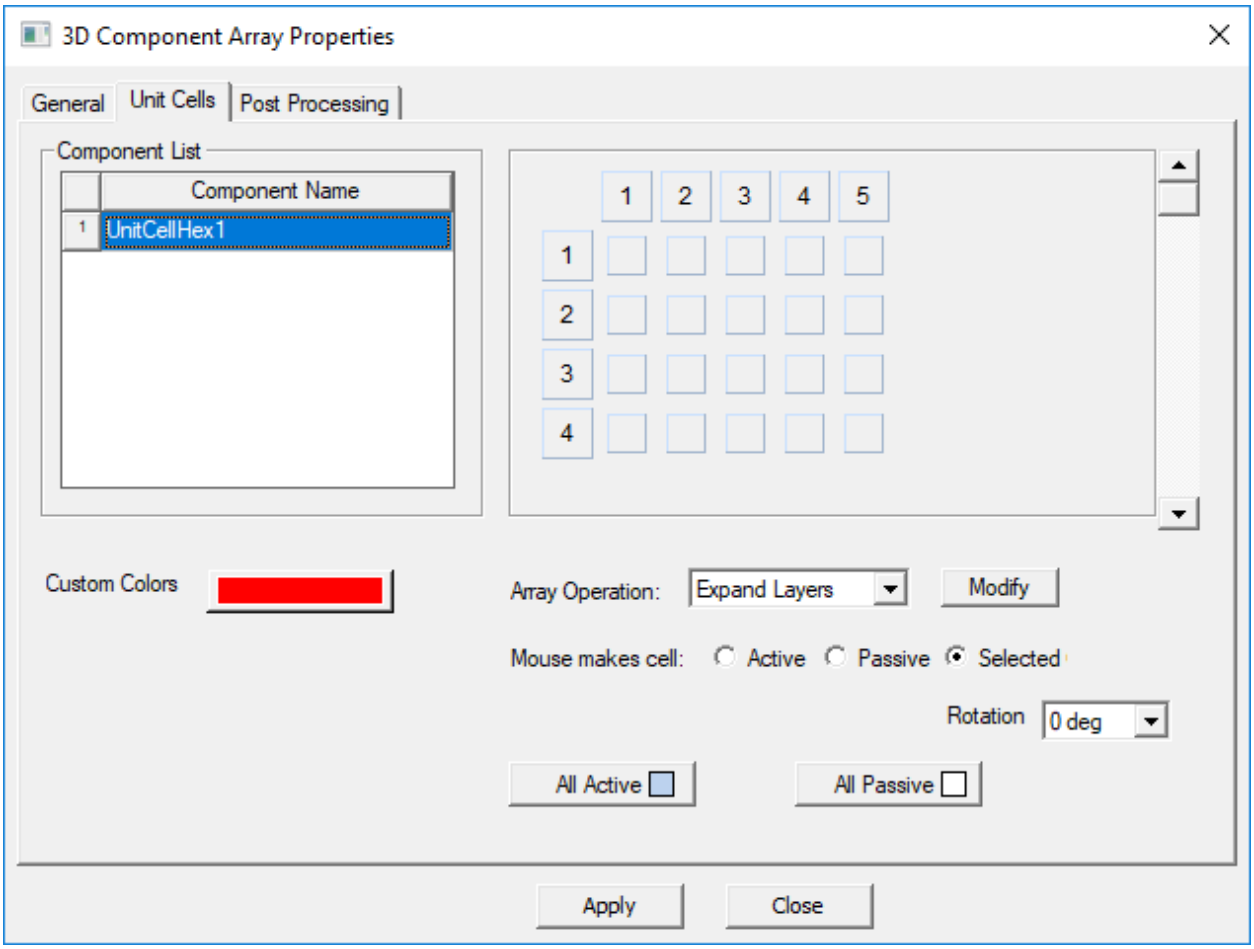
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

end columns). After deletion, if the inputted number of “Delete To Column” is larger than the row size of the new array, the two inputted numbers are reset to 1



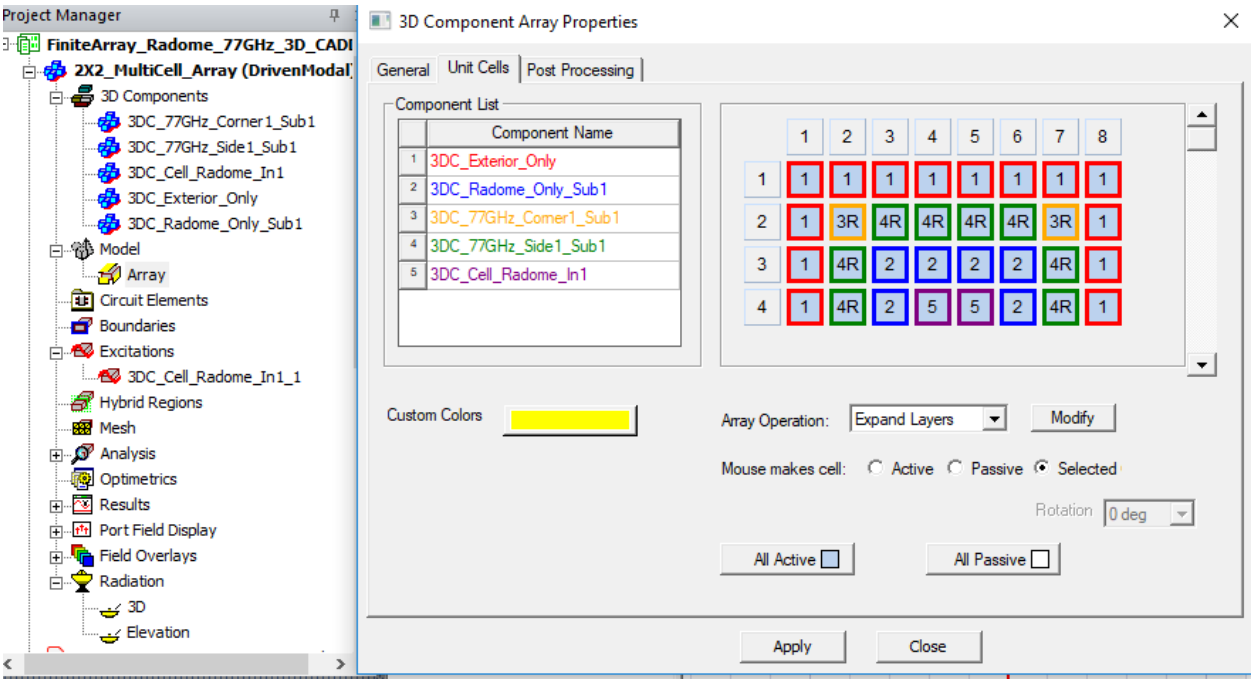
3. The component list shows the available components. Select the Component Name under the component list. You must designate at least one component as active.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

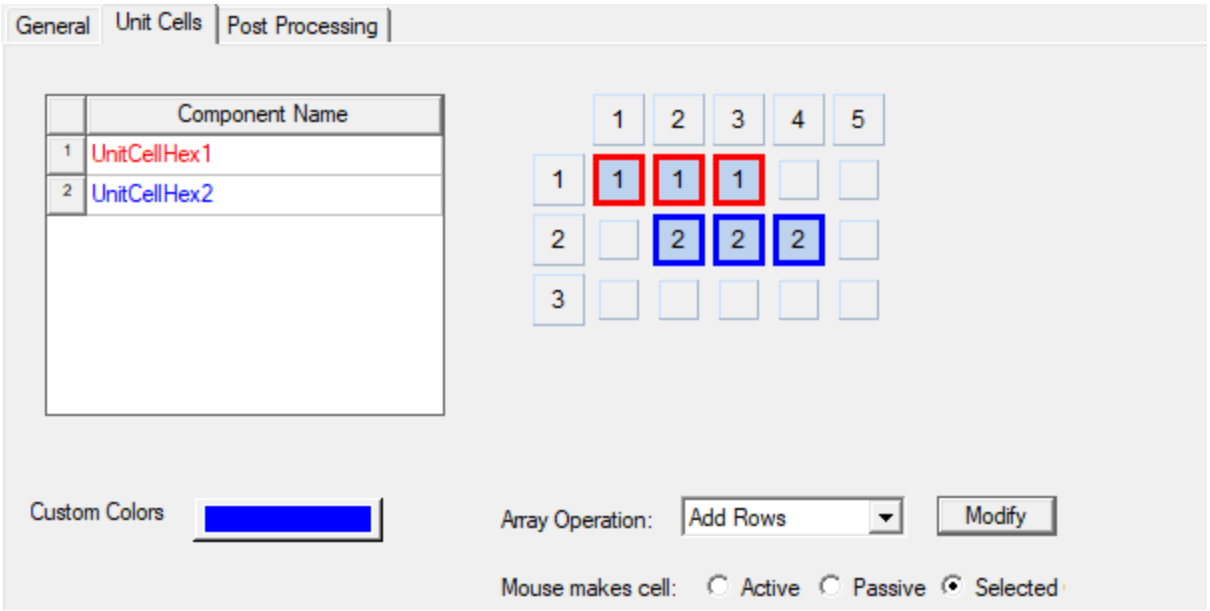


If the model contains more than one component you can select from the list.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

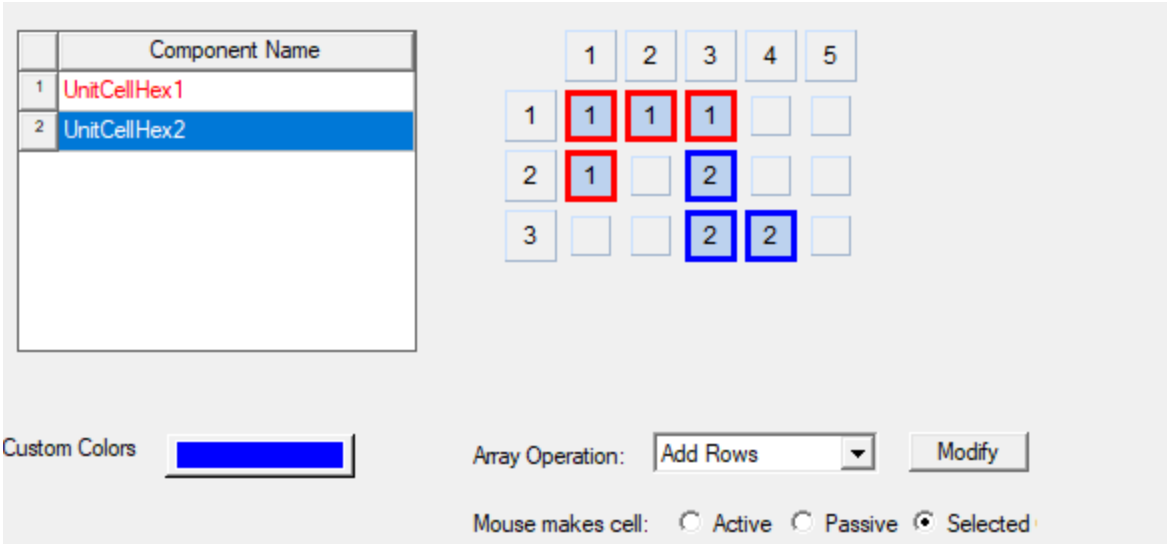


4. With **Mouse makes cell**: set to **Selected**, click on the cells to populate with that component.

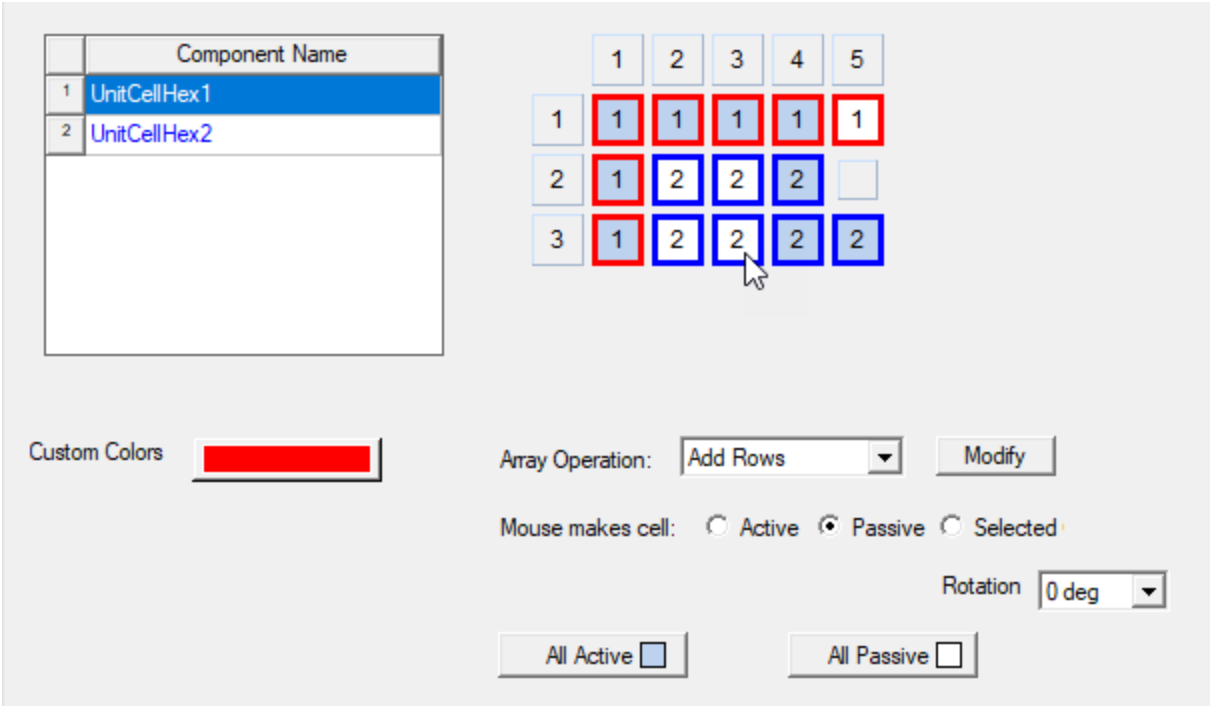


You can create an irregular 3D component array, depending on how you specify Selected components. Not selecting a 3D Component effectively sets the cell as an air cell.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also designate cells as Active or Passive. When designated as passive, the cells are displayed as light inside the color boundary.



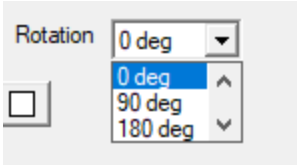
Rotation for Selected Cells

All component cells are initialized with 0 degree rotation (i.e. no rotation as default). You could first select the component name from Component List, then select the rotated degree from the rotation combobox at the bottom right, and then click the cell desired to rotate. The

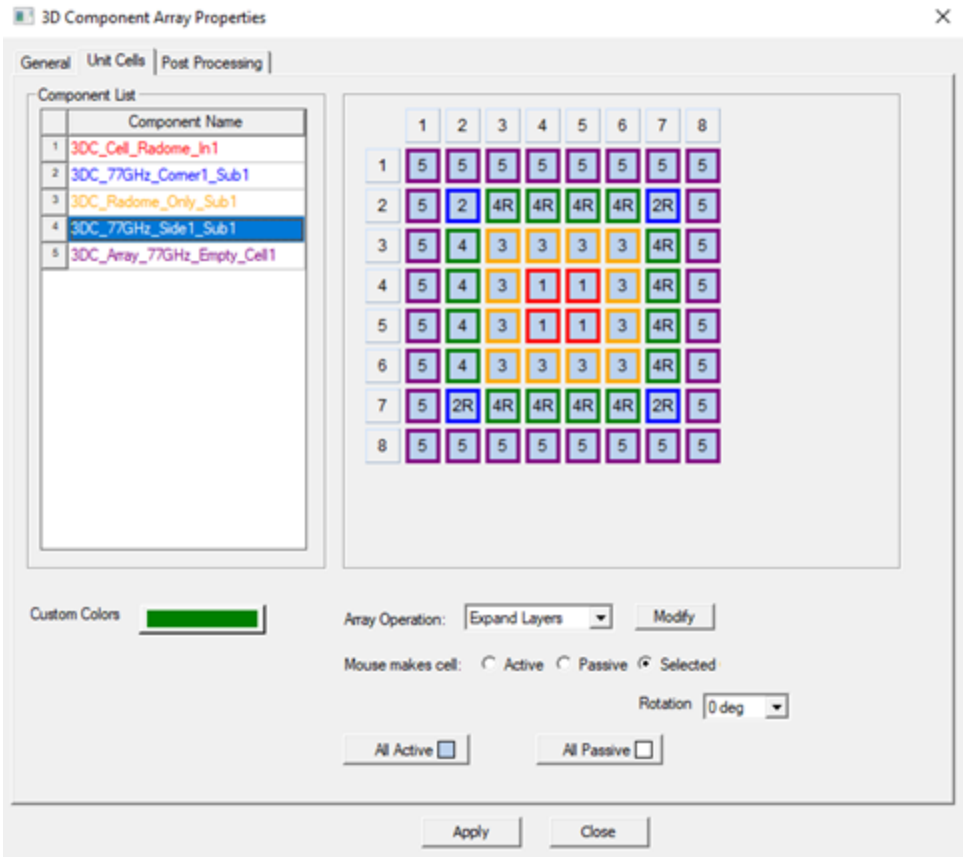
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

rotated cell is labeled “R” after the index to remind user. If no component is selected (i.e. air cell), the rotation combobox is disabled.

Note: rotation cannot be applied to any array cell containing a port.



After setting the rotated cells and clicking **Apply** button in the **3D Component Array Properties** dialog box, the array is visualized.

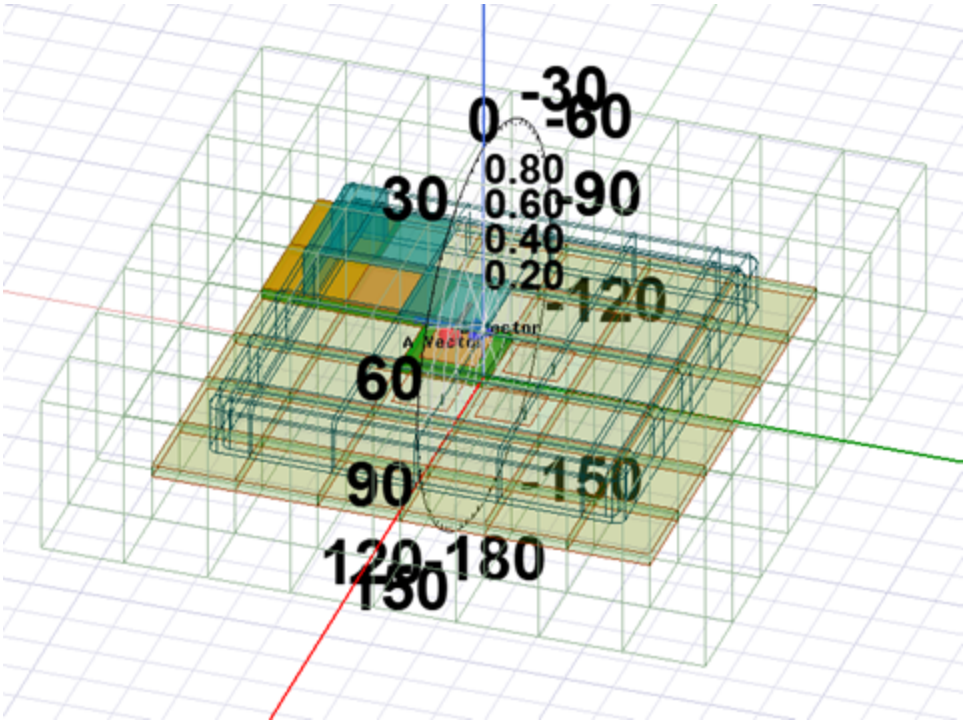


For the example above, the cells [2, 3], [2, 4], [2, 5], [2, 6], [2, 7] are rotated 270 deg, the cells [3, 7], [4, 7], [5, 7], [6, 7], [7, 7] are rotated 180 deg, the cells [7, 2], [7, 3], [7, 4], [7, 5], [7, 6] is rotated 90 deg. The rotated direction is anti-clockwise.

When you move the mouse on a component cell, a tooltip provides information including the domain name, state, and rotation angle.

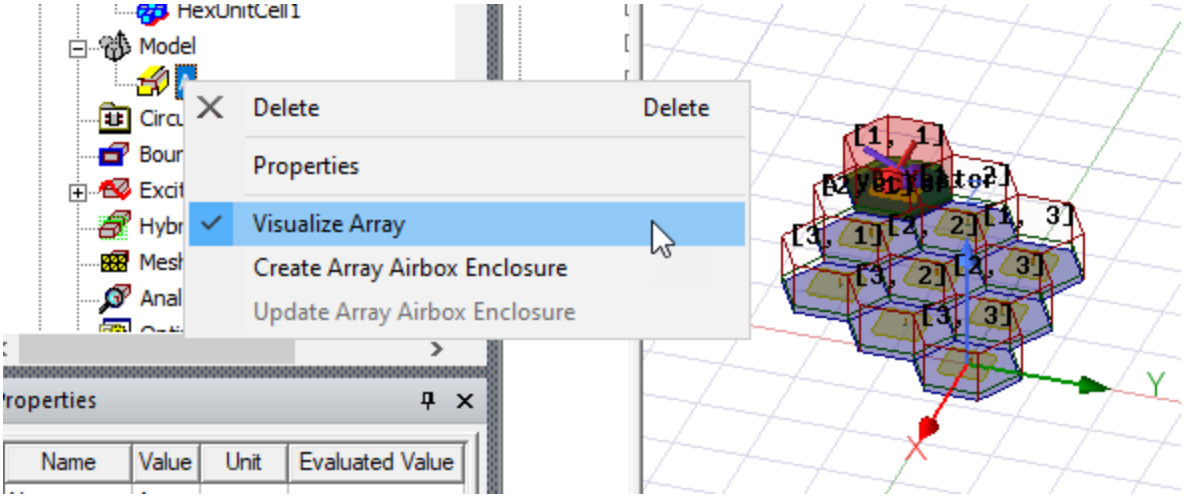
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Below is the visualized array for the above design:



5. Click **Apply** to create the array you have defined and displayed. Right click on the **Array** icon to see the short cut menu that lets you manage the model properties, and visualization.

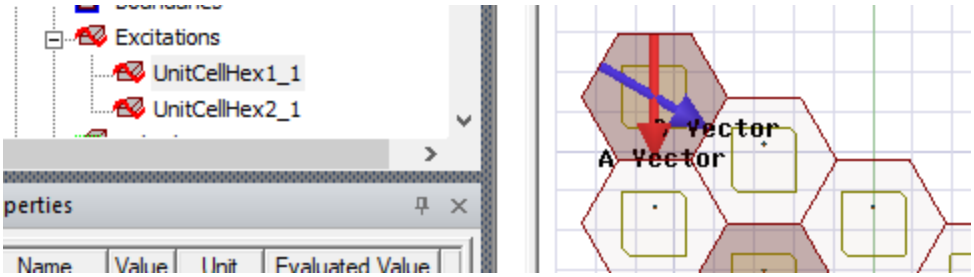
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When you have defined an 3D Component array, an array icon appears under the Model icon in the **Project** tree. Only one Array is permitted for a model. The **Create Array** command is disabled if an array is defined.

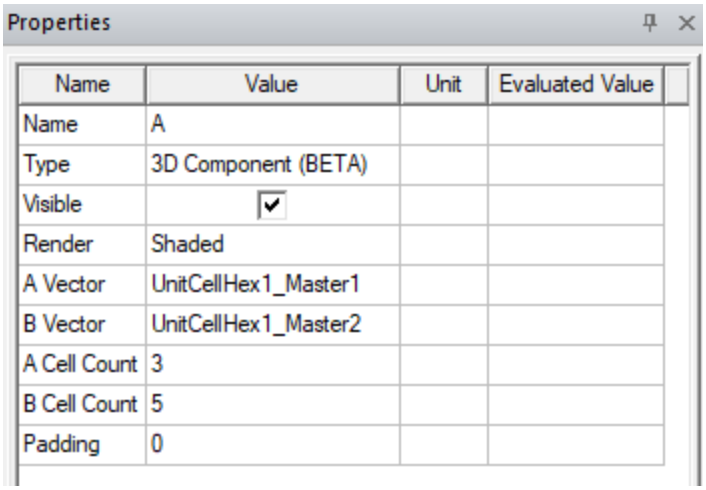
- Selecting **Delete** removes the array from the Model.
- Selecting **Properties** displays the **3D Component Array** dialog.
- Selecting Visualize Array lets you toggle the array display. A check mark indicates that the array is being displayed.

To view the lattice vectors, select the Excitation for the 3D Component.

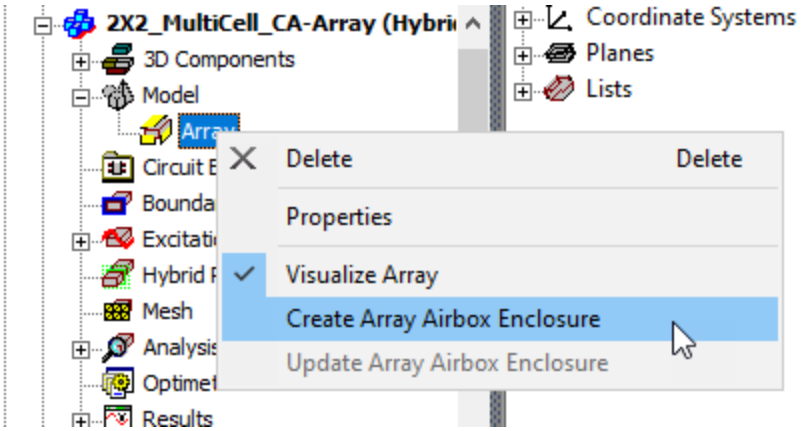


With the **Array** icon in the **Project** tree selected, if you have a docked Properties window displayed, you can see and edit array **Properties**. All edits are undoable and informational messages will be posted to the **Message** window when design data is deleted.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

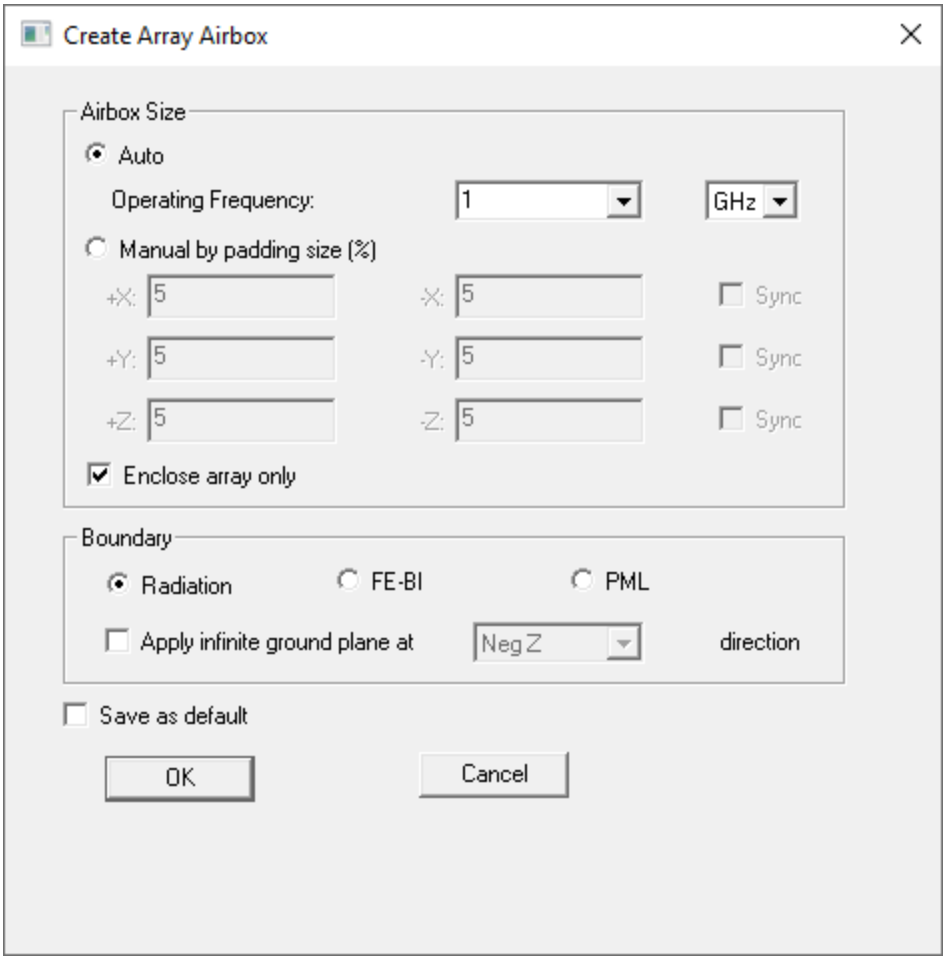


When you have defined the Component Array with Active cells designated, the **Create Airbox Enclosure** command is enabled.



This opens the **Create Array Airbox** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The array airbox dialog supports Auto definition by frequency and has a checkbox to calculate airbox size based on either the array itself (**Enclose array only**), or the array plus all FEM geometries. By option you can uncheck Auto and manually specify dimensions as padding. You can assign the airbox boundary as Radiation, FE-BI, or PML. And you can assign an infinite ground plane.

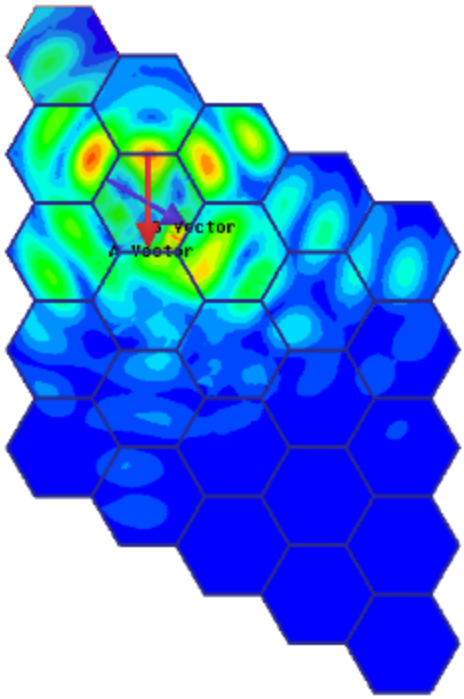
In setting **Analysis** options for an 3D Component Array simulation, if you select an **Advanced** setup, the **Options** tab lets you **Skip 3D Component Array Solve during Adaptive Passes**. This option is on by default. The mesh for each component is independently adapted, which significantly improves the performance compared to full array adapt with a minimum lose of accuracy. Even it is a composite design, the solver still adapts each component as non-composite during component adapt. As a result, even though it showa Max Delta Normalized Power due to UI constraints, the underlining calculations are all based on the Max Mag Delta S from each component.

"

Post Processing

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

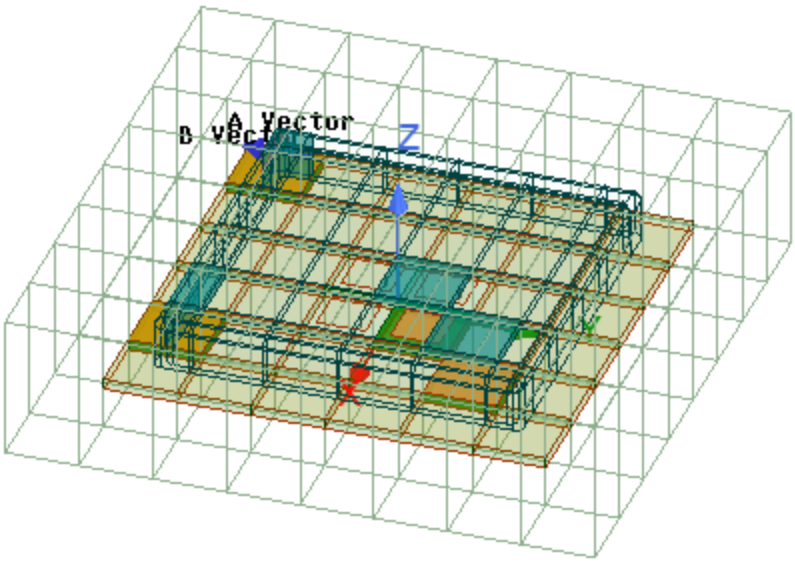
Once you have run analysis, you can view Solutions Profiles including matrix data, create reports and, if you have checked Save Fields on the analysis setup, you can create fields plots.



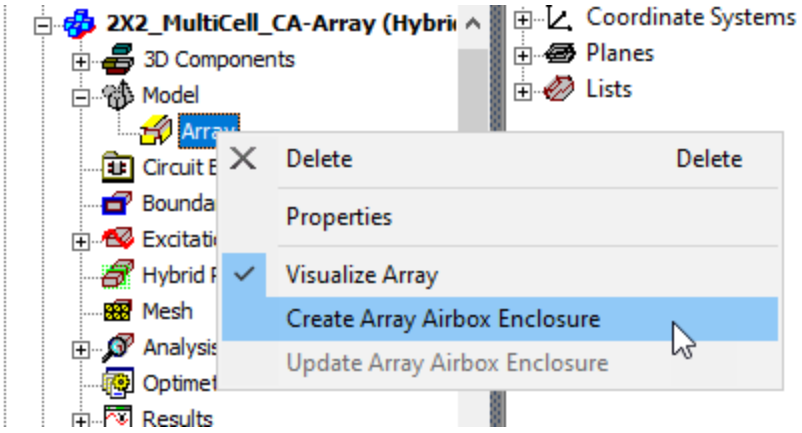
SBR+ and 3D Component Array

Assume that you have already created a 3D Component array, such as the “FiniteArray_Radome_77GHz_3D_CADDM” in Examples, HFSS>Antennas.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

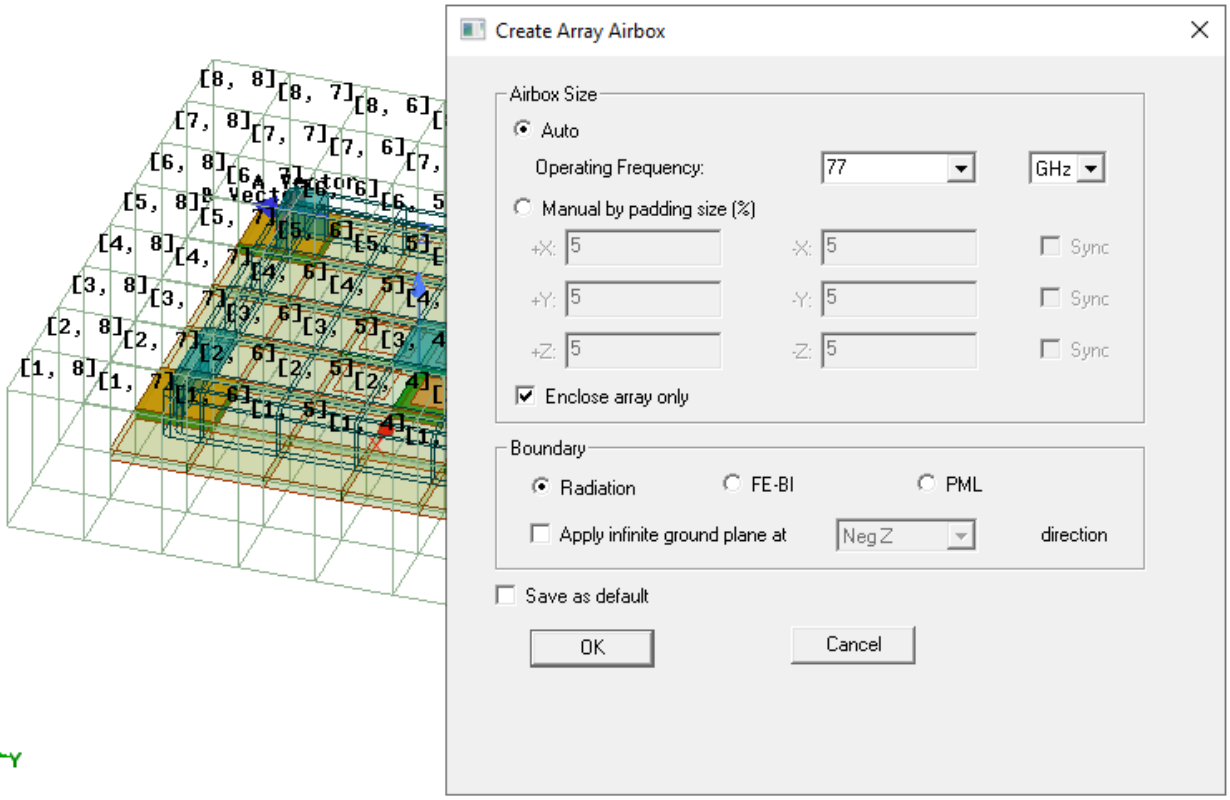


To create an SBR+ component array, you must have one FEM array airbox that encloses the entire array and add an SBR+ region. For the array airbox, you can manually define it as you wish, or right click the **Model > Array** icon in the Project and select **Create Array Airbox Enclosure** from the shortcut menu.



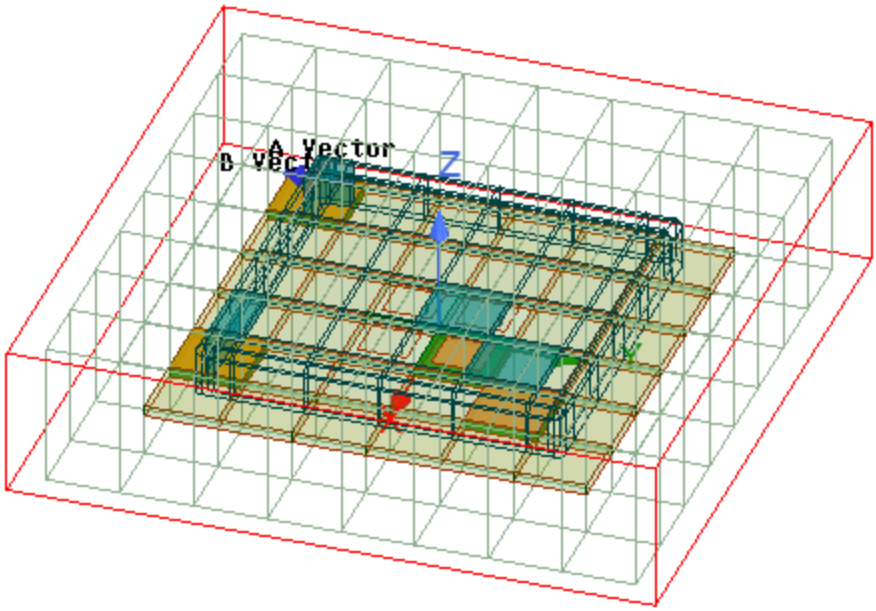
This opens the **Create Array Airbox** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



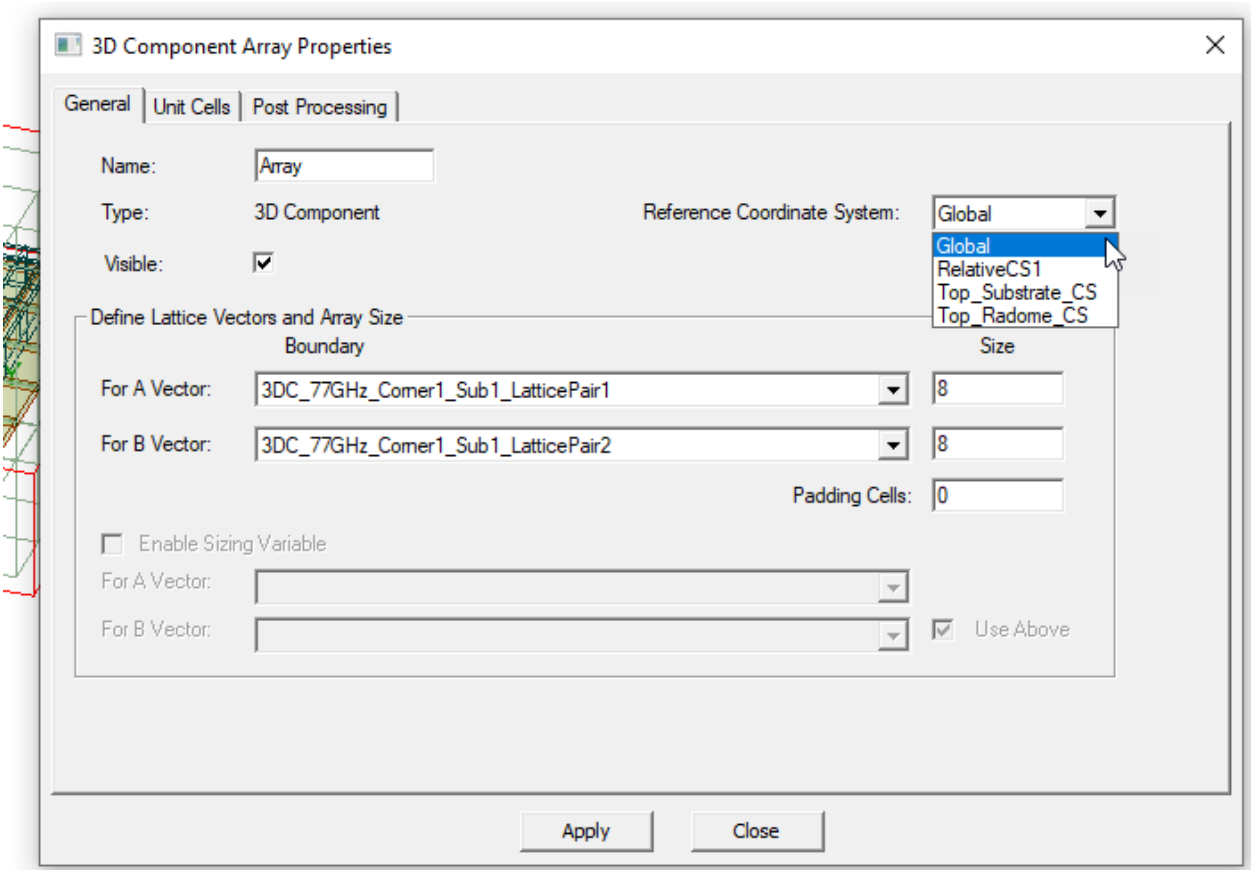
The **Create Array Airbox** dialog supports Auto, which is defined by frequency and enclose array only or calculated by array plus all other surrounding geometries.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



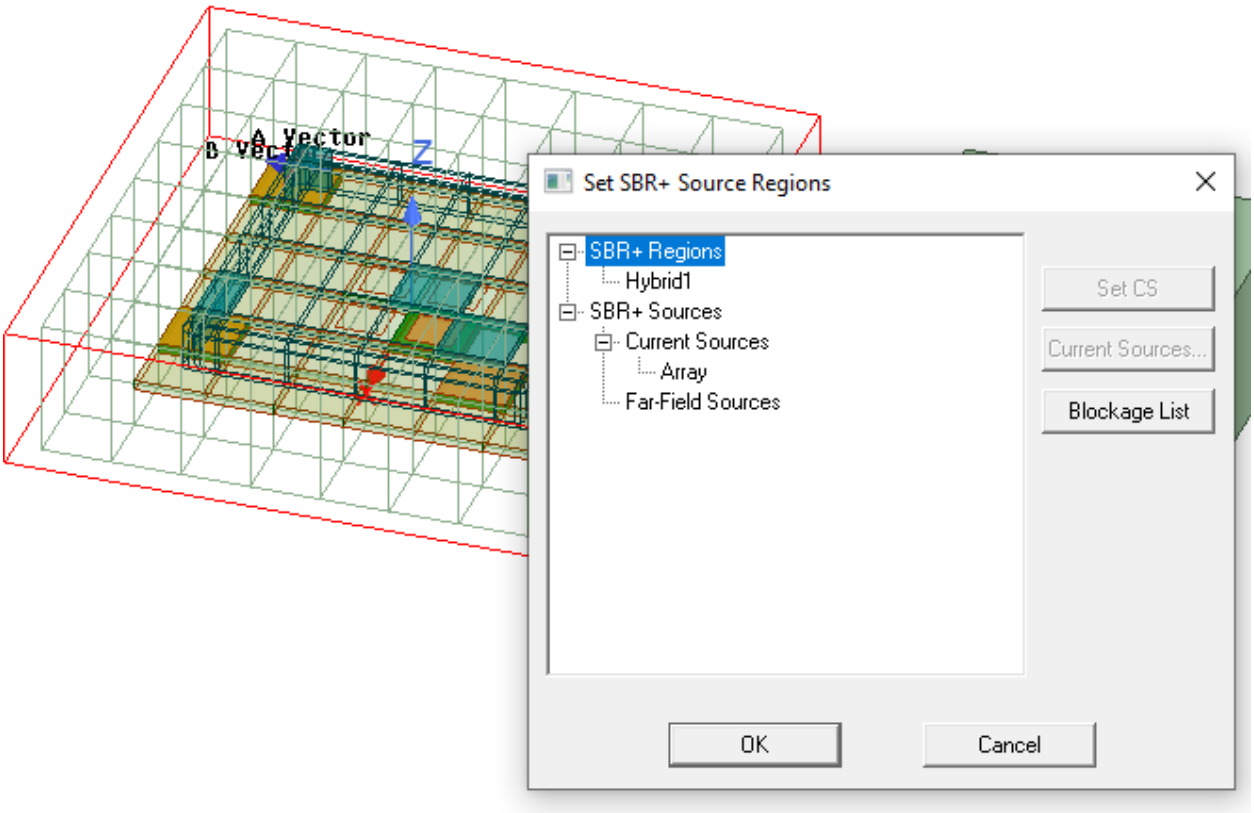
An option in the component array Property page lets you reposition the whole array (and array airbox, if it has been defined at this point) to a new reference coordinate system. This can be helpful if you define the SBR+ region in a reference CS.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The SBR+ component array design supports any SBR+ workflows like the normal SBR+ design, except that excitation is limited to port only and no incident wave is allowed. Some UI controls are also improved for component array use. For example, the “SBR Source Region” is simplified since the component array is the only SBR+ source in SBR+ component array design.

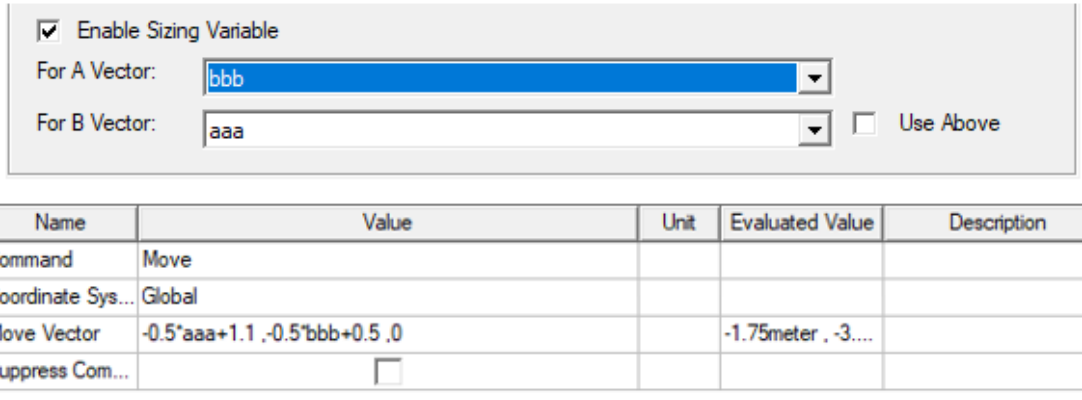
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



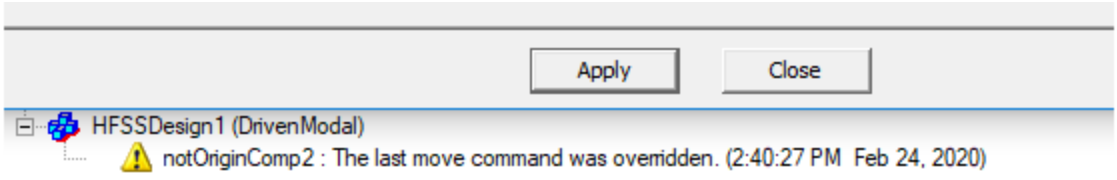
Sizing Variable for a New 3D Component Array

When you create a new 3D Component array, you can employ a sizing variable. This variable should only be used when you create a new component for a 3D component array. The feature does not appear for a regular array. The sizing variables control array's row vector and column vector (i.e. size of component's bounding box) and enable you to do related sizing research. The functionality involves passing sizing variables to the Move command during a component's automated placement. When you create a new variable, you ensure that the 3D Component expands or shrinks from the center point, rather than the starting point. Otherwise, the placement controlled by the variable will be incorrect.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

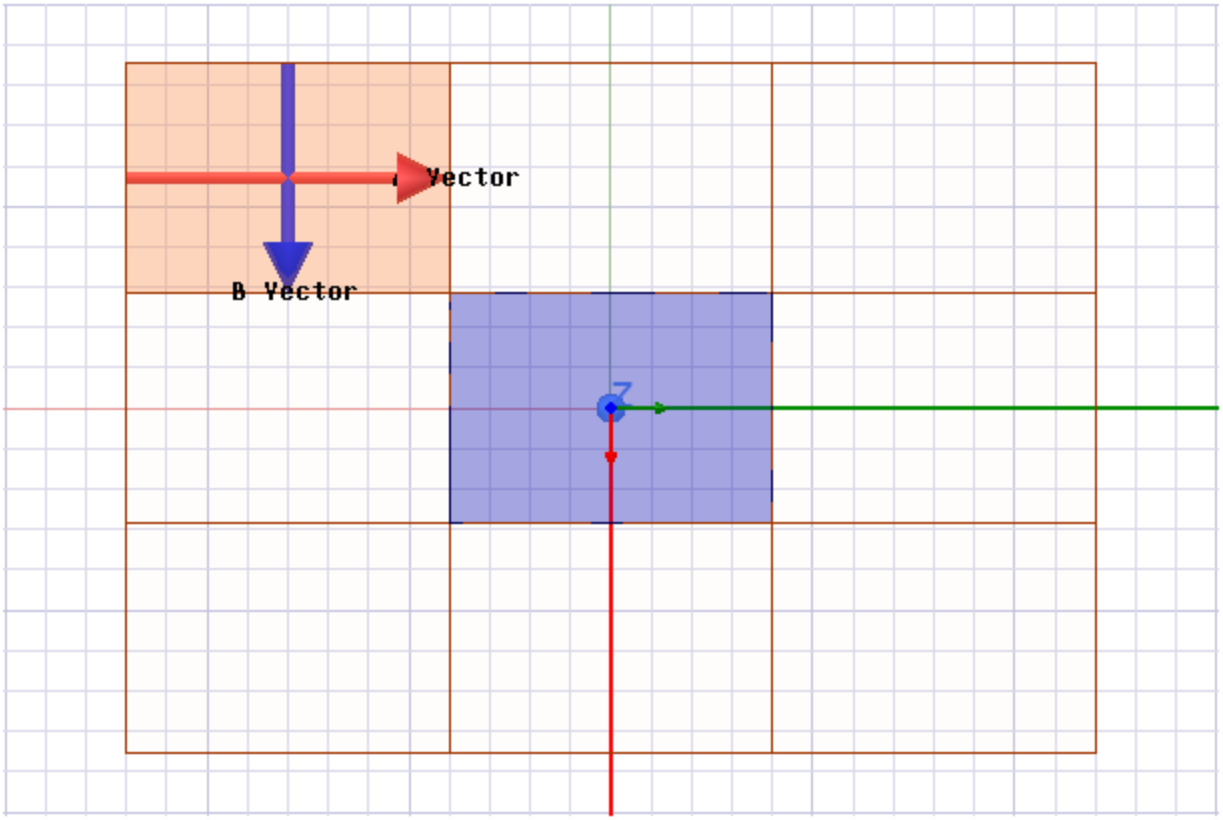


After you create or edit a 3D component array and hit the Apply button, if the component has had move commands with its last move command 1) based on global coordinate system and 2) different from the expression of the current move command, the Modeler directly overrides the component's last move command and shows a warning message in the Message Manager.

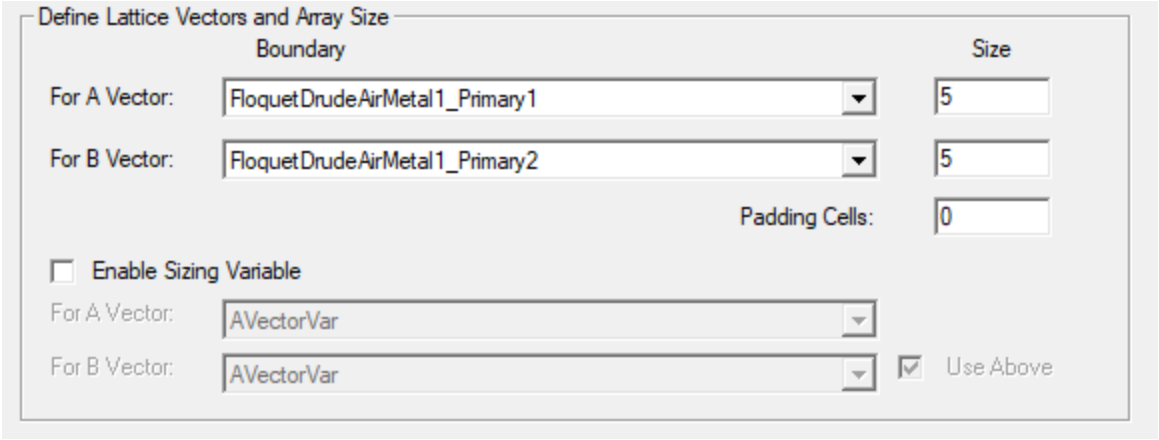


To be valid, the sizing variables must be created as Design Variables of Length, and must match their corresponding Row/Column Lattice Vector Length.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

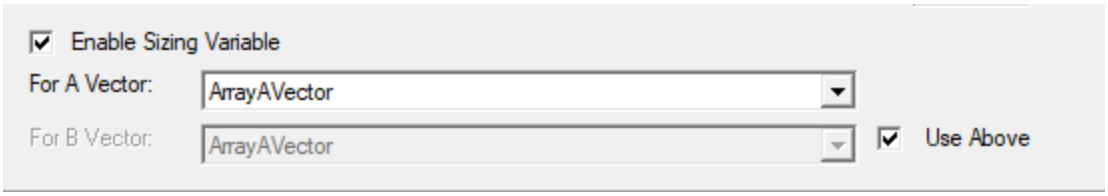


If you have defined Design Variables for Length, the checkbox to Enable Sizing Variables is available.



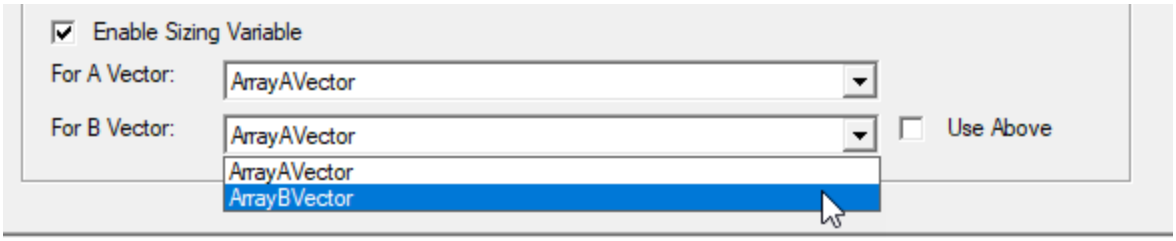
Check Enable Sizing Variable enable the selection menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can select available Length variables from the drop down. If you select a variable that does not match the corresponding vector, a warning message displays. Most of the time, array's row vector and column vector have the same length, which means one 1 sizing variable can control both vectors. By default, Use Above is checked, so For B Vector is grayed out. We only need to select the correct sizing variable For A Vector, and the currently selected variable in combo-box For B Vector is automatically changed to the same one as in A.

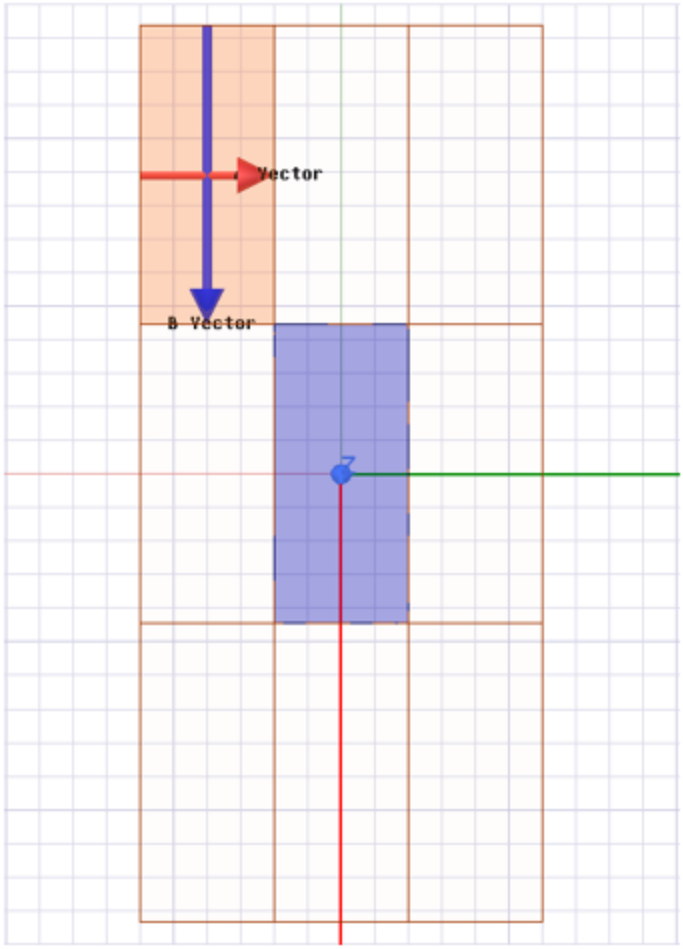
If you uncheck Use Above, you can select available Length variables from For B Vector drop down.



After you modify values of the sizing variables from Design Properties dialog or Design Properties property window, each component directly changes its size but is still placed in the correct position (that is, the array is still normal).

Name	Value	Unit	Evaluated Value	Type
aaa	8.9	meter	8.9meter	Design
bbb	4	meter	4meter	Design

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Important Notice:

Make sure that the component's bounding box expands or shrinks from the center point to the surroundings evenly.

Assume that the start position is on the top left with coordinate (x, y, 0), and the original center position is (x_0, y_0, 0) with x_0, y_0 constant. It's obvious we have relationship $x + 0.5a = x_0$ and $y + 0.5b = y_0$. Therefore, the start position should have the coordinate expression as ($x_0 - 0.5a$, $y_0 - 0.5b$, 0).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Value	Unit	Evaluated Value
Command	CreateBox		
Coordinate...	Global		
Position	-0.5*a ,-0.5*b ,0meter		-1meter , -0.5m...
XSize	a		2meter
YSize	b		1meter
ZSize	1	meter	1meter

In the above example, since the center point is originally located at the origin, the expression of the start point should be (-0.5a, -0.5b, 0).

a) Assume that the component has had several Move commands and some move commands except the last one also contain variables before its current placement. Then after the current placement, if we modify the value of some variable contained in the middle Move commands, it's very possible that the current placement is messed up (i.e. the array is abnormal).

The reason is that for each component, the Move commands are executed in order, and some intermediate positions become incorrect after we modify the variable value.

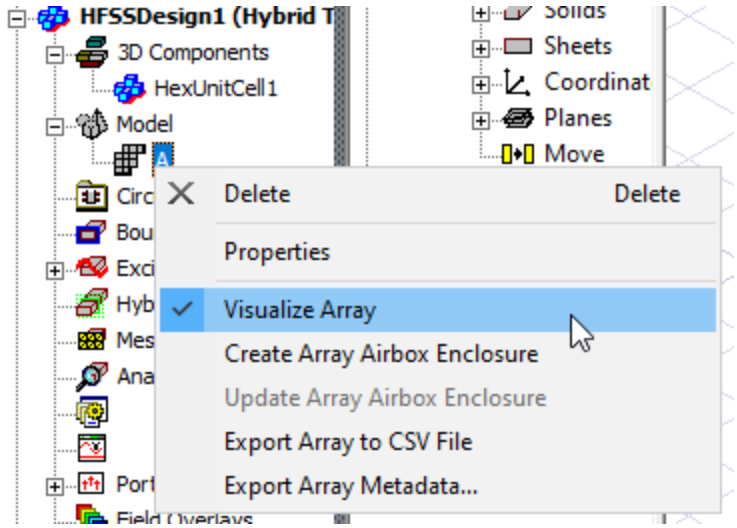
Therefore, before we execute the current placement, it's better that we either don't include variables in the previous move commands except the last one, or don't modify their values casually.

b) When you create the bounding box of the component, the start position is at either top left or bottom right. However, if you control an array via a Sizing Variable, the Modeler expands or shrinks the component bounding box from the center point. When you create the component, **you must set the center point** to make sure that the component expands from the center to the surroundings evenly. Otherwise the component expands from the top-left or bottom right, which creates problems. After you create a component, you cannot set the center point again.

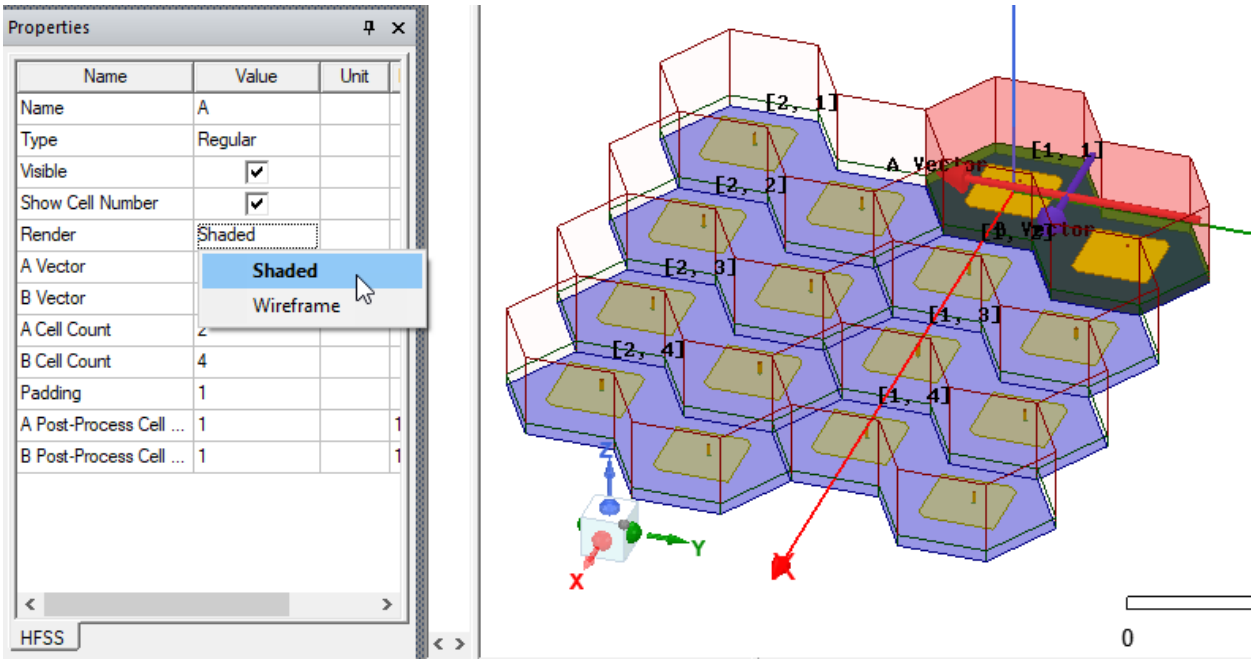
Array Visualization

You control the visualization of an array by selecting an existing Array in the Project tree and toggling the **Visualize Array** command on the short-cut menu or the Visble check box for the Array properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

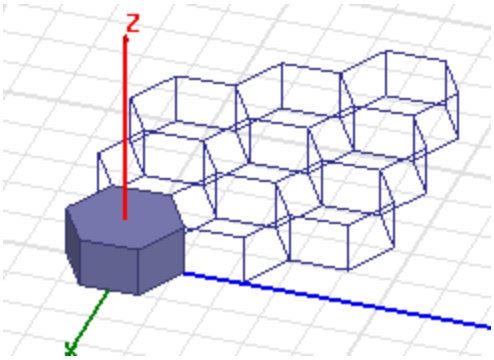


Virtual objects in an array can display as Wireframe or as Shaded, depending on your selection in for Render in the Array **Properties** window.



If the all virtual objects for an array do not appear in the view modeler window, perform a **View>Fit All** operation. If you change the view, you can also use **Ctrl+D** to fit the array into the current view.

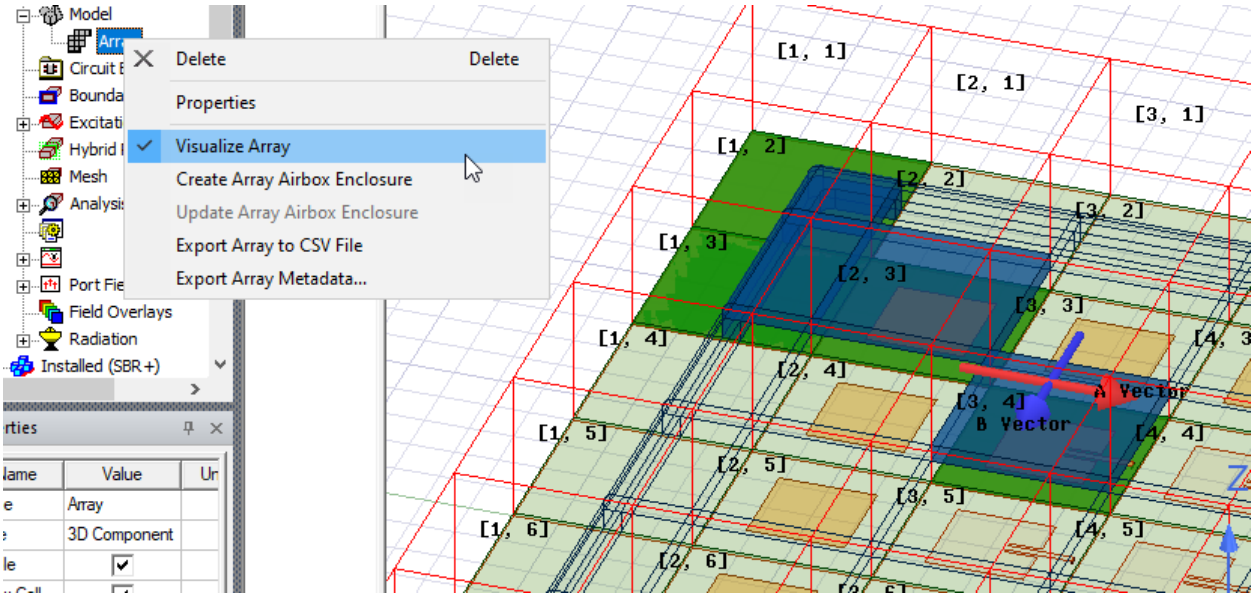
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can disable the array visualization by clicking on the **Visualize Array** option again. A check mark appears in front of **Visualize Array** if the option is already ON. This option will also be present in general active view visibility options, **View>Active View Visibility**.

The visualize array option will be applied to the current active view. You can have multiple windows in the modeler, each window will have its own **Visualize Array** setting. This allows you to use different windows to look at unit cell display and whole array display at the same time.

If the **Visualize Array** option is set to true, the array will be visible when you open the project.



Set up and Run an Array Simulation

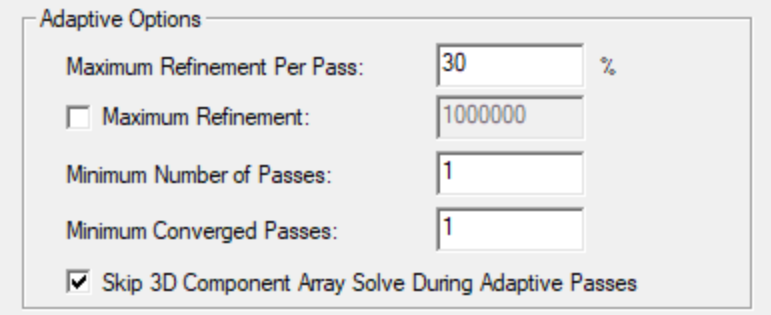
There are some changes in the GUI for setting up the adaptive simulation for designs with a virtual array and if the design uses 3D components However, the solution quantities of the virtual cells will be available for convergence setup (In both adaptive and interpolating sweep). There

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

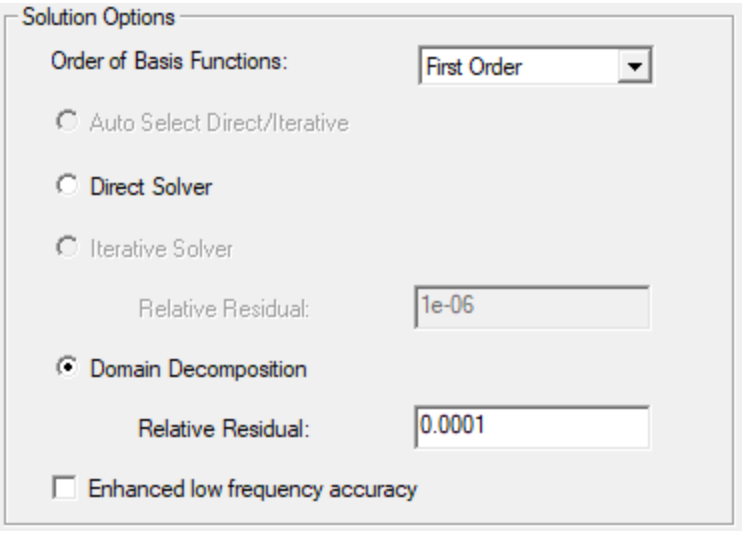
are no changes in the way convergence information is presented on the Convergence tab of the **Solution** Display panel.

If your design contains a virtual array and/or 3D Components, the **Solution** setup can have some differences.

- If an array contains 3D Components, the Solution Setup Options tab contains a **Skip 3D Component Array Solve During adaptive Passes** option.



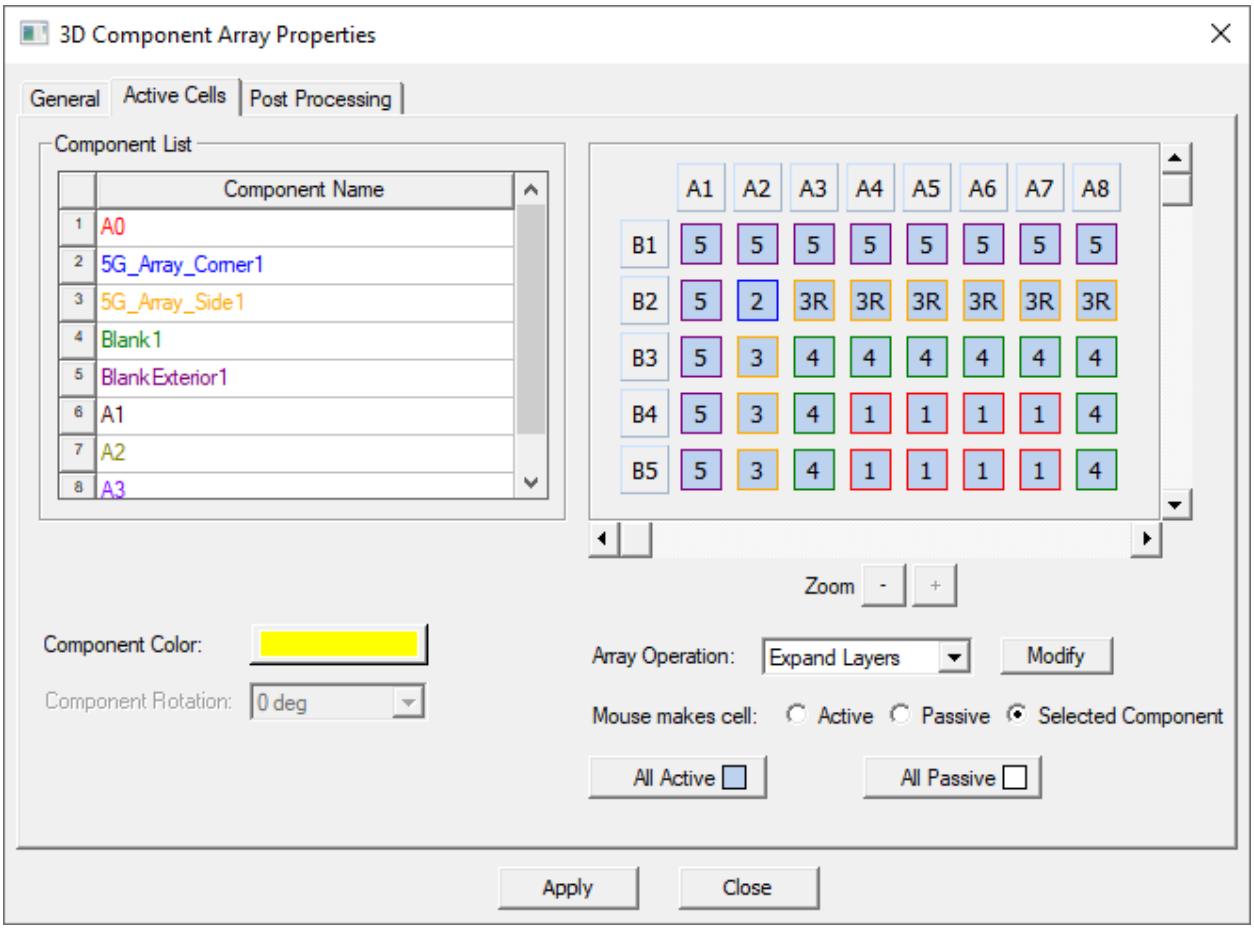
  This option is on by default. The mesh for each component is independently adapted, which significantly improves the performance compared to full array adapt with minimum lost of accuracy. Even it is a composite design, the solver still adapts each component as non-composite during component adapt. As a result, even though it's showing Max Delta Normalized Power due to UI constraints, the underlining calculations are all based on Max Mag Delta S from each component.
- The Solution Option for **Domain Decomposition** is automatically checked. You can choose Direct Solver. Iterative Solver options are disabled. The domains are UI defined through the array definition, not solver defined domains.



- Set up the distributed processor workgroup. Designs with arrays require HPC licenses.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- General Setup for Virtual Array Simulation for Matrix Convergence, if you choose Selected Entries.
- Interpolating Sweep Advanced Options for Array Simulation
- Fast sweep is not supported.
- You can also set up the expression cache at solve setup. The expression cache interface for accessing array elements is the same as those used in report setup.
- Use the **Active Cells** tab on the **Regular Array** dialog or **3D Component Array** dialog to designate which cells are active or passive for a simulation. You can make All Active, All Passive or select which cells are active or passive. The more active cells there are for a simulation, the more processing is required. By default, clicking the corresponding array elements toggles the current selection, You can also choose the **Mouse makes cell** setting to click for Active or Passive, whichever is most convenient. Clicking on a row or column number applies the mouse click command to all cells in that row or column. Dragging the cursor over cells performs the current operation on them.



It is important to understand the impact of passive ports on antenna parameters. For accepted power calculations, passive ports are not included when computing the total power

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

passing through the union of all port surfaces. This means that the passive ports can be viewed as a loss mechanism for the device and it is not equivalent to viewing the passive ports as active ports with zero excitations.

- Report setup for Arrays.

The solution/matrix quantities are grouped by category. The entries in each category are listed according to their [row, column] order in the corresponding matrix.

The entry in [row1, column1] will be listed first, followed by

[row1, column2], … [row1, columnN], [row2, column1], …

[row2, columnN], … [rowN, columnN]. Note that the [row, column] order of each entry in the matrix is controlled by the 'Matrix' order as specified by user.

The existing "Filter" capability can help locate the desired quantity from the potentially very long list.

Beta Feature: Parallel Component Mesh Adapt

If **Skip 3D Component Array Solve During Adaptive Passes** is selected and you enable the Beta feature **Parallel Component Mesh Adapt**, if a design contains multiple 3D Components in an array, and is run through HPC, the Solution Profile can show multiple hf3d solvers for frequency solves.

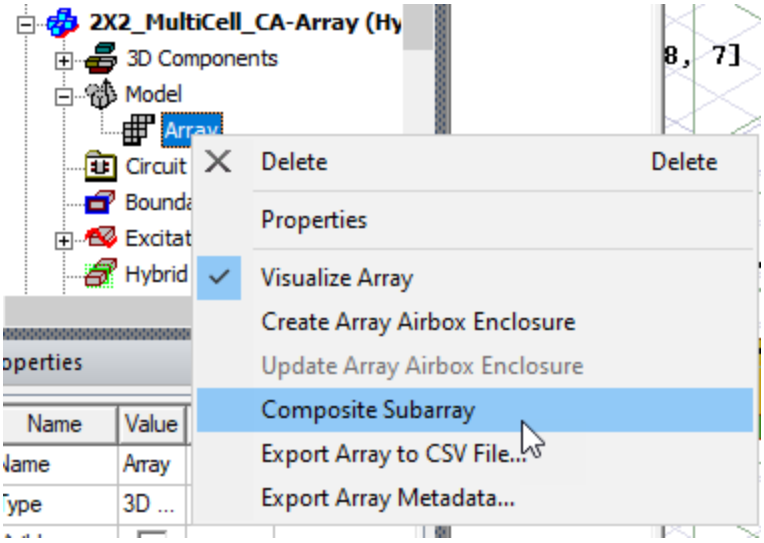
Beta Feature: Composite Subarray in HFSS Finite Array Design

A composite excitation is simply a specific superposition of defined sources. Using composite excitations can speed up analysis and reduce storage requirement for fields when there are many sources, and you are only interested in a few combinations of those sources (for example, for 5G antennae).

This feature resembles the composite solution type in which all source excitations are combined into a single source. Here, however, you can combine a subset of sources into a single source. This feature supports multiple, named, subsets of sources. When solved, each subset presents as a single source to the reporter. Each combined source (namely composite group) is accessible from **Edit Sources** to control its magnitude and impedance.

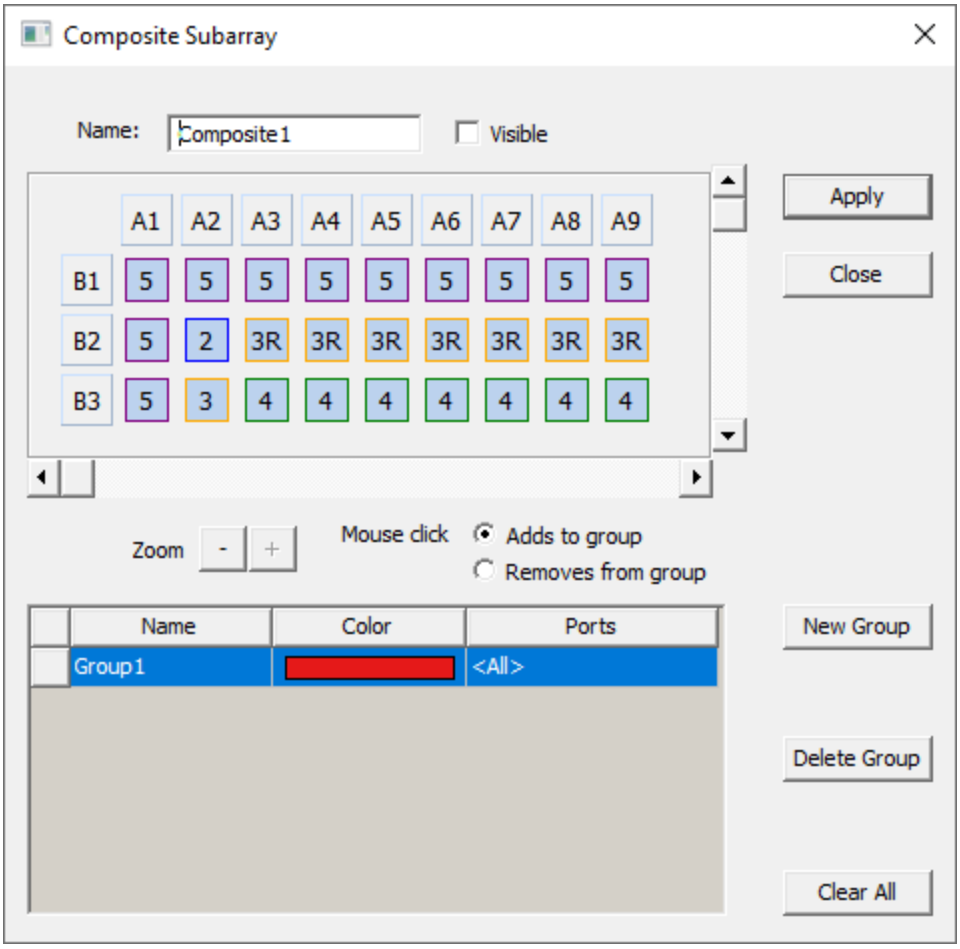
For an HFSS design with a finite array, and the Beta Feature **Composite Subarray in Network Analysis** enabled you can access the composite subarray functionality by right-clicking on the Array Item in the Project tree and selecting **Composite Subarray** in the menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



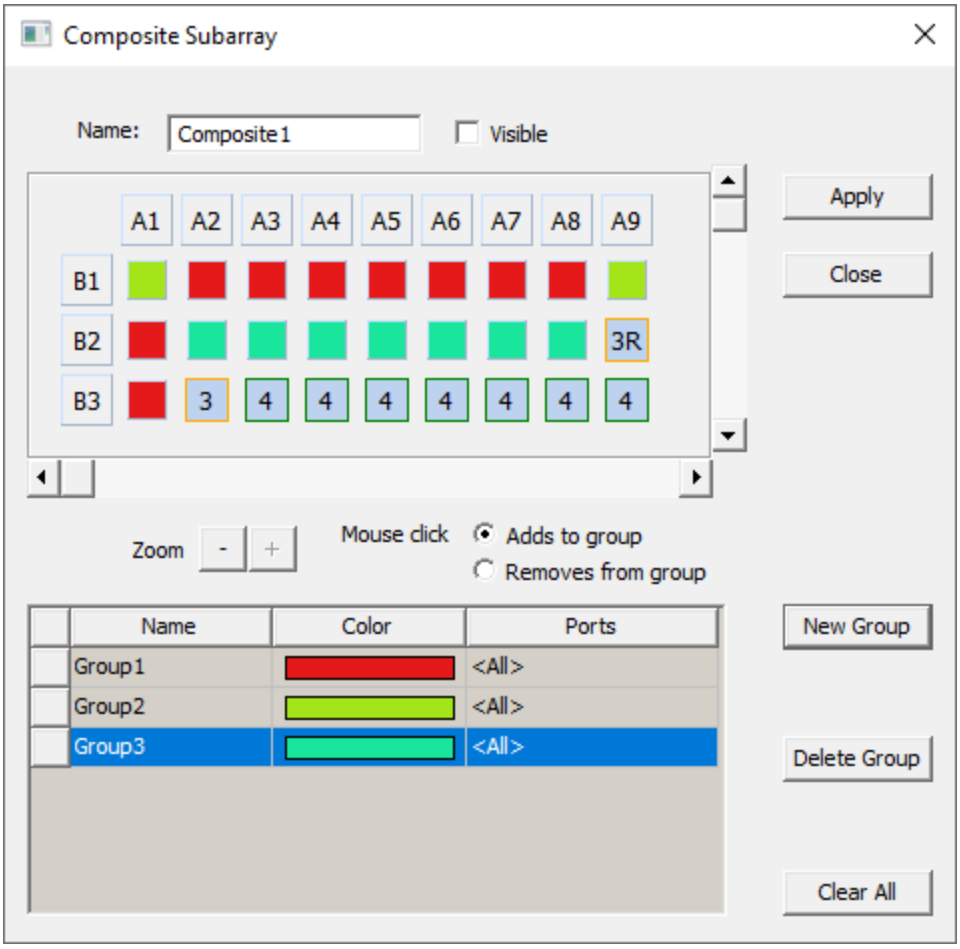
In the **Composite SubarrayArray** dialog, you can create/edit/delete composite groups and assign array cells to composite groups in a similar way that you defined the array itself. If the design includes an existing subarray, it is depicted in the dialog. Notice that the boundary colors for the defined cell are independent of those applied to define each new group. You can click on individual cells to assign them to the currently active group.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

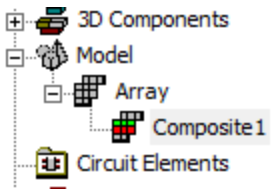


You can click on individual cells to assign them to the currently active group, or click on the Row or Column number to assign that row or column. You can resize the dialog and use the vertical and horizontal scroll bars to view array cells for selection. Click the New Group for a new one which will be given new colors, and by default, an incremented name.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

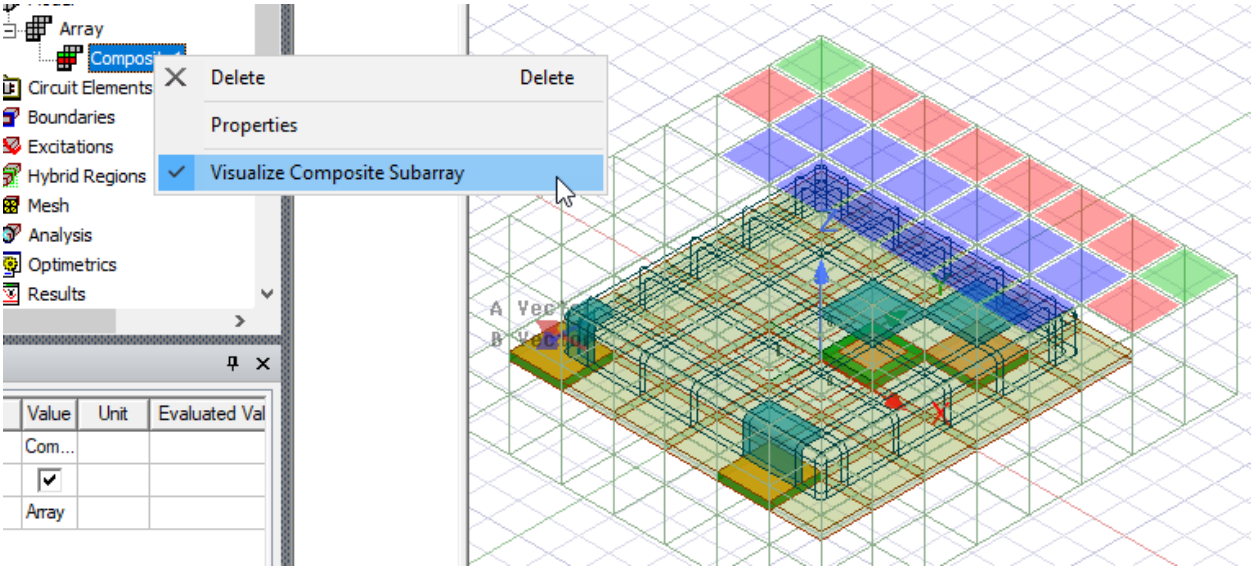


Click **Apply** to activate the Group Assignment in the Modeler window display. Once you have defined a subarray, it appears in the Project tree.



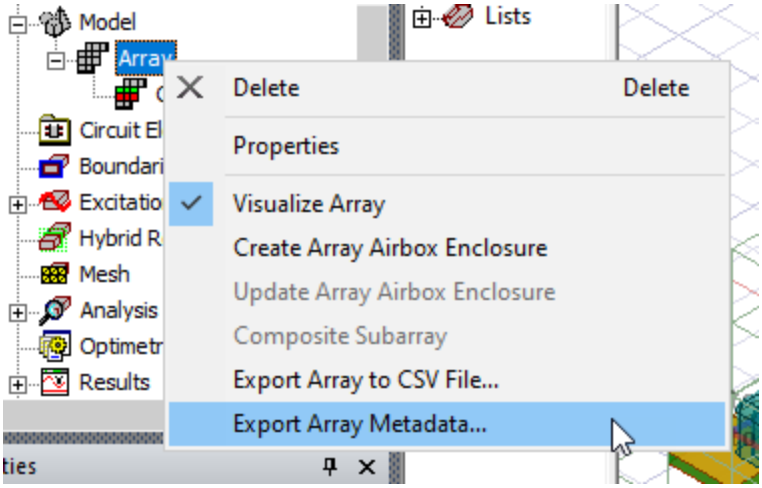
You can right-click on the Composite subarray to view the **Properties** window, and to Visualize Current Subarray assignments in the **Modeler** window. Once the composite array is defined and is set to visible, each composite group would be visualized using a unique color overlay in the modeler.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



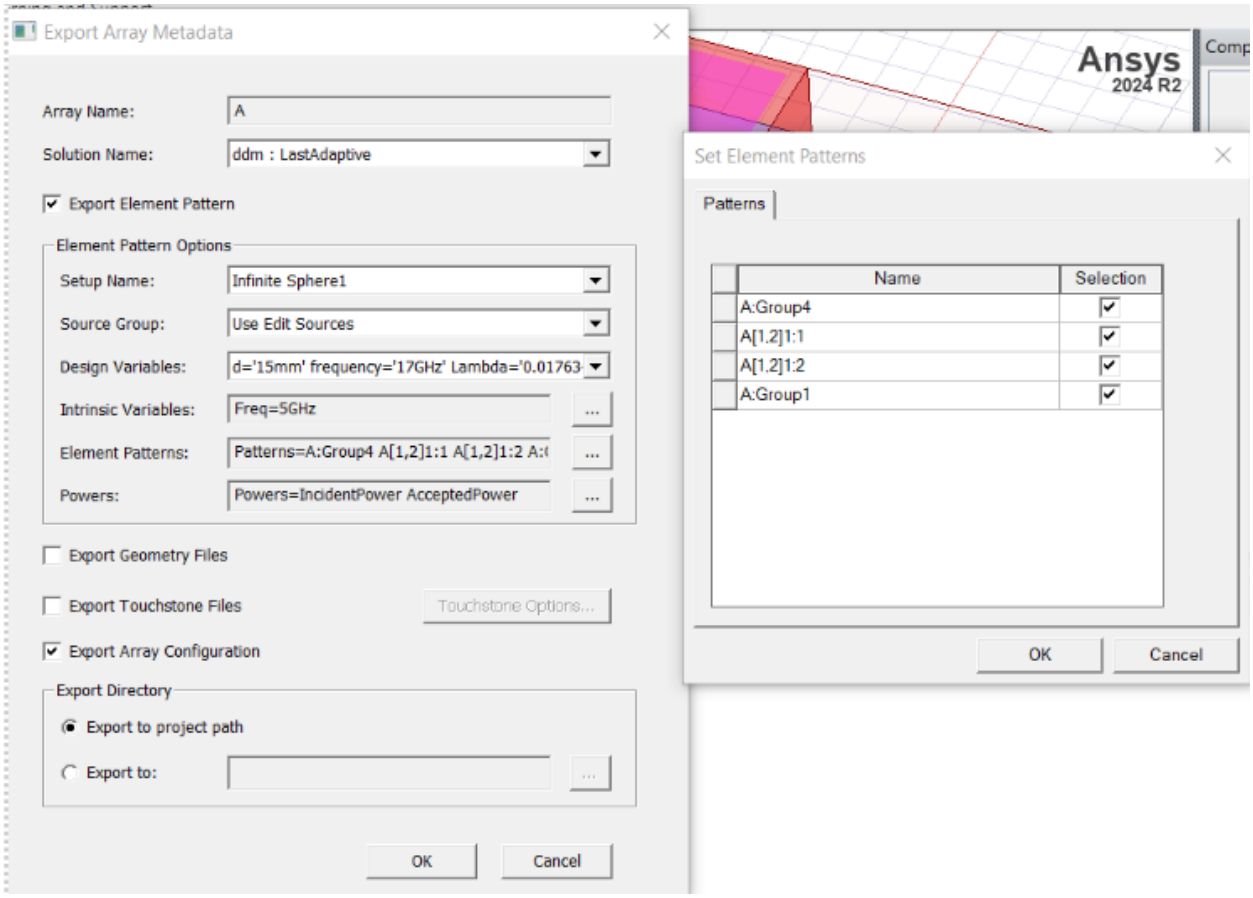
Note that in the example figure, the blue color for the cells in Row three in the dialog has been applied to the cells in the Row 3 modeler window.

After a composite subarray design is solved, you can export array metadata by right clicking the array item in project tree:, you can right-click on Array to Export Array Metadata.



In the **Export Array Metadata**dialog the element pattern selection shows composite groups instead of individual ports.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For designs that Link from Driven Modal Design to SBR+, the **Phase Reference Setting** dialog also offers choices for composite groups if you want to specify unique phase reference for each group.

Similarly, the powers in the export metadata header file would clearly show the group names and group element patterns (i.e., only this composite group is excited). For example:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
<ElementPatterns>
  <Source name='A:Group1'>
    <Filename>exportfield_0.ffd      </Filename>
    <ReferenceLocation>-0.075000,0.075000,0.300000      </ReferenceLocation>
      <PowerInfo>
          <Power Freq='1GHz'>
            <IncidentPower>1.000000              </IncidentPower>
            <AcceptedPower>0.264860              </AcceptedPower>
            <SystemPower>1.000000              </SystemPower>
            <RadiatedPower>0.271018              </RadiatedPower>
          </Power>
      </PowerInfo>
  </Source>

  <Array name='A'>
    <LatticeVectorA>0.000000,150.000000,0.000000      </LatticeVectorA>
    <LatticeVectorB>-150.000000,0.000000,0.000000      </LatticeVectorB>
      <CompositeSubArray name='Composite1'>
        <Group name='Group1'>
          <Cell>[1, 1] [1, 2] [1, 3]             </Cell>
        </Group>
        <Group name='Group2'>
          <Cell>[2, 1] [2, 2] [2, 3]             </Cell>
        </Group>
        <Group name='Group3'>
          <Cell>[3, 1] [3, 2] [3, 3]             </Cell>
        </Group>
      </CompositeSubArray>
  </Array>
</ArrayInfo>
```

Composite Array Post-Processing

Once a finite array design with composite array is solved, post-processing treats each composite group as if it is a single excitation. For example, in an S-parameter table, you can examine the S

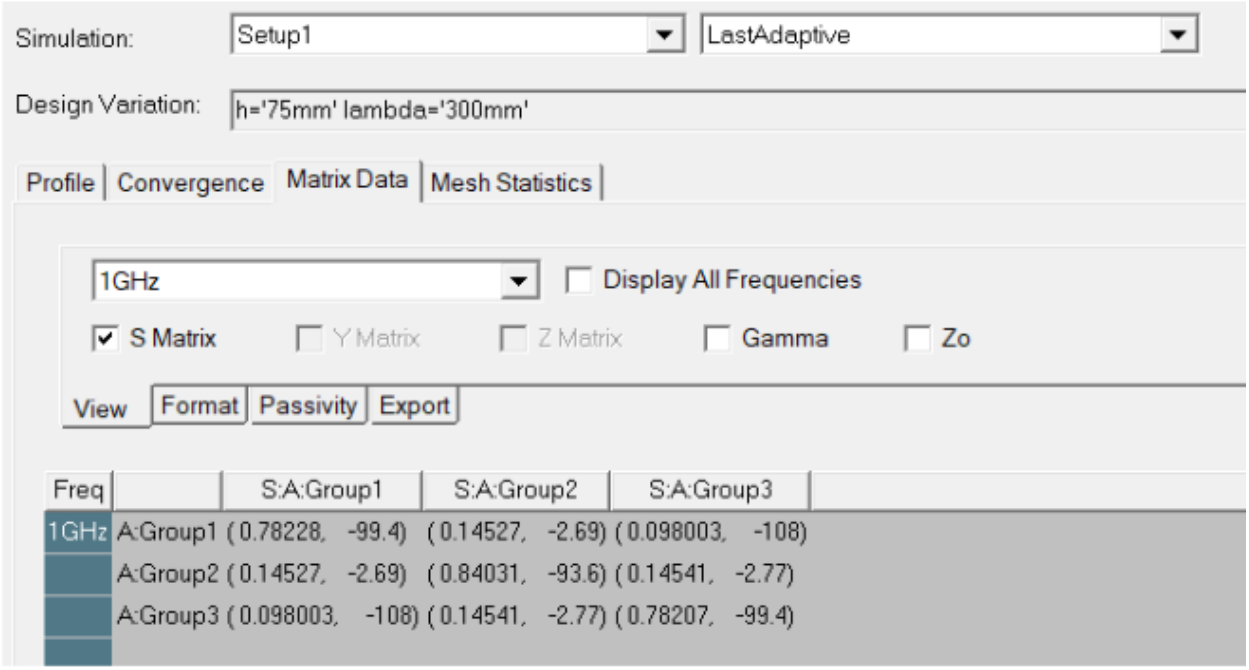
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

parameter between each group in modal solution reporter and matrix display, where the composite group name is shown instead of an individual port.

S Parameter Table 1

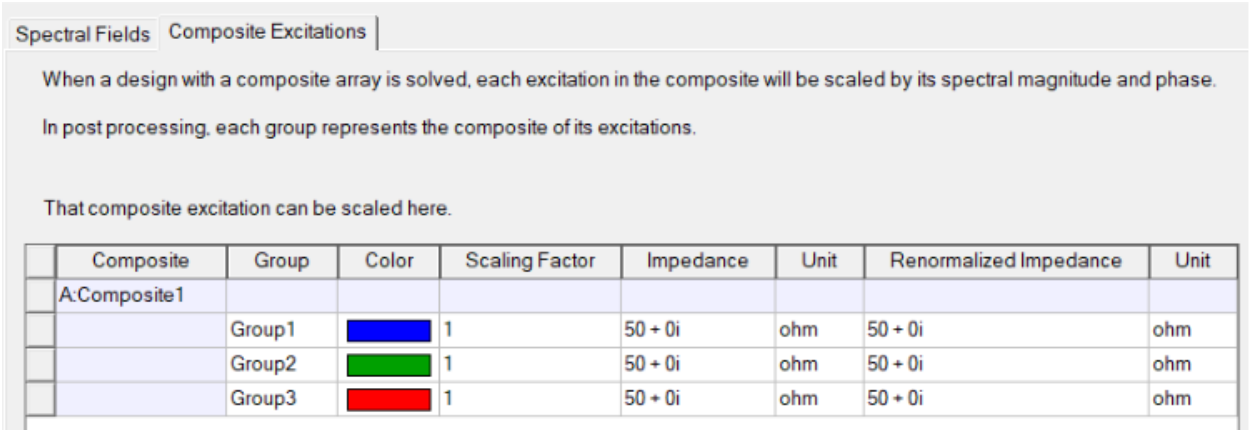
	Freq [GHz]	dB(S(A:Group1,A:Group1)) Setup1 : LastAdaptive	dB(S(A:Group2,A:Group1)) Setup1 : LastAdaptive	dB(S(A:Group3,A:Group1)) Setup1 : LastAdaptive	dB(S(A:Group1,A:Group2)) Setup1 : LastAdaptive	dB(S(A:Group2,A:Group2)) Setup1 : LastAdaptive
1	1.000000	-2.132799	-16.756288	-20.175226	-16.756288	-1.511248

The **Matrix Data** tab of the **Solution Data** does the same thing, showing composite group names, rather than port names.



For field post-processing, the **Edit Source** dialog includes the settings of each port, and each group has its own scaling factor, which can be defined in the **Composite Excitation** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



As with an individual port, you can also define impedance and renormalized impedance for composite groups, but the underlying logic is slightly different.

Due to limitations you must explicitly define the Z_0 of composite group, and the renormalized impedance will be used as Z_0 (i.e., composite group always applies post-processing effects). On the other hand, the concept of "Impedance" is introduced solely to account for the matrix renormalization effect (e.g. for S matrix normalization) which is based on the ratio of impedance and renormalized impedance.

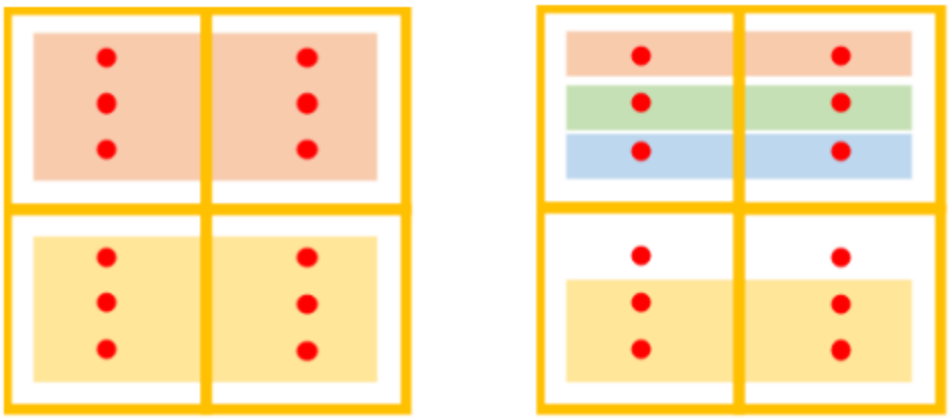
For example, a composite group with A impedance and B renormalized impedance can be seen as an individual port with A impedance and B renormalized impedance, with renormalization effect turned on. Therefore, if you do not want any renormalization for composite group, you must simply ensure that A and B are the same and match the expected impedance value.

The impedance and renormalized impedance is only used for matrix quantity (e.g., S parameter, Z_0) but not for fields results. The "Include Post Processing Effects" option is disabled to reflect this concept.

Multi-band Composite Subarray

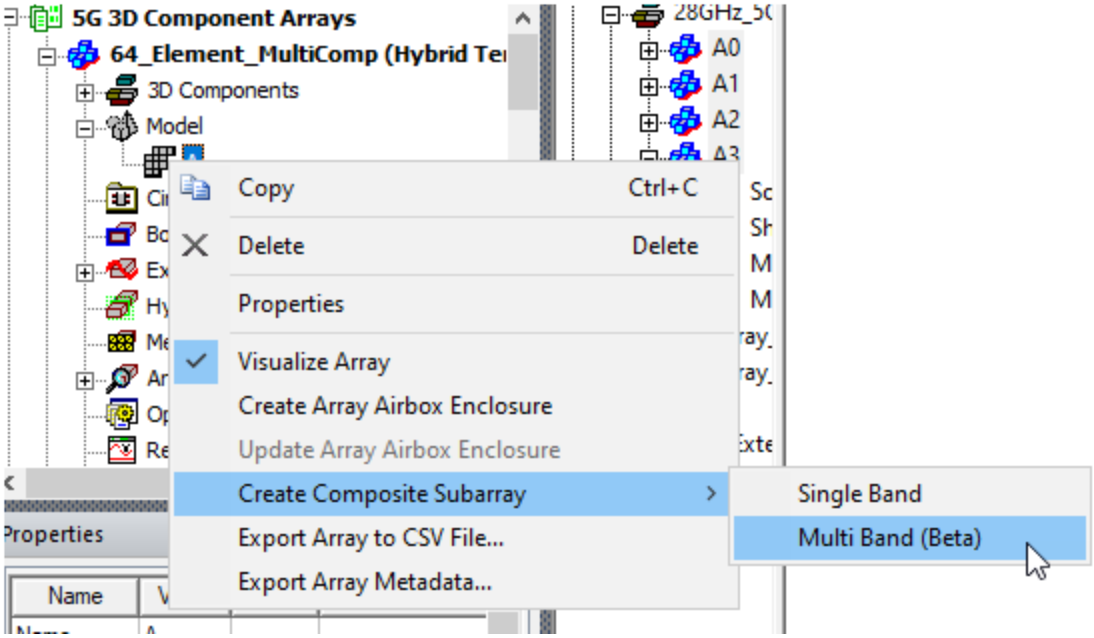
HFSS support multiple composite excitations within a given design, along with S-parameters between these excitations. A composite excitation is a specific superposition of defined sources. Leveraging composite excitations can significantly enhance analysis efficiency and reduce storage requirements for field data. A Beta feature allows multi-band composite subarrays, addressing realistic and complex scenario requirements. If array cells have multiple ports, you can further subdivide existing composite groups to include specific ports, as shown in the figure below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



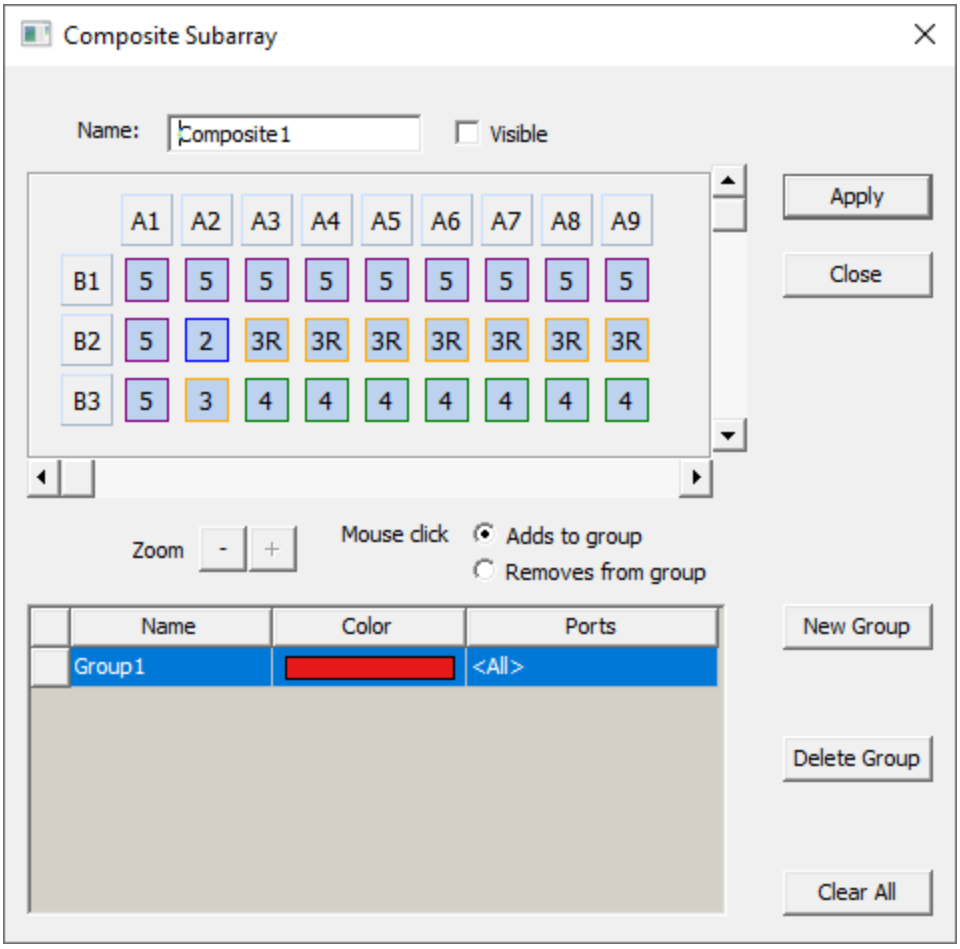
Single Band Multi Band

When feature is enabled, and the design has an array and multiple ports defined click the array right-click menu to select **Create Composite Subarray>Multi Band (Beta)**:

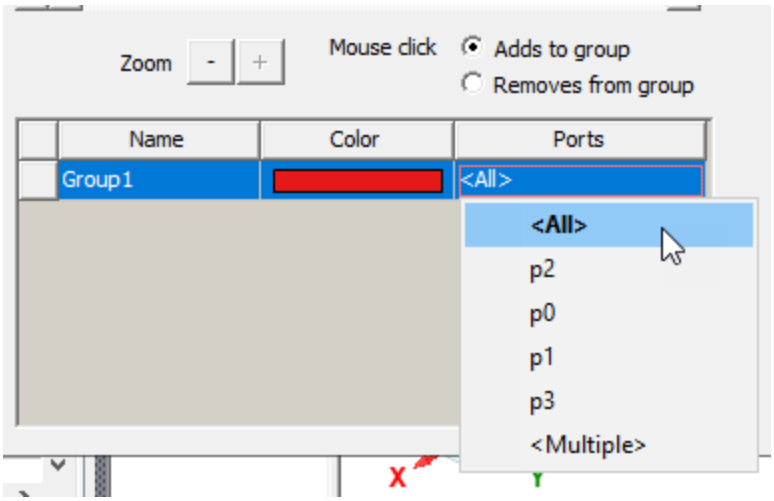


This opens the **Composite Subarray** dialog, with Rows and Columns appropriate to the design to select which port(s) to be included in a composite group. The interface differs from the Single Band version in that it contains a Ports column with a field that allows you to select single or multiple ports to define additional bands.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Right-click on the Ports for the group to see a drop down with the available ports listed.



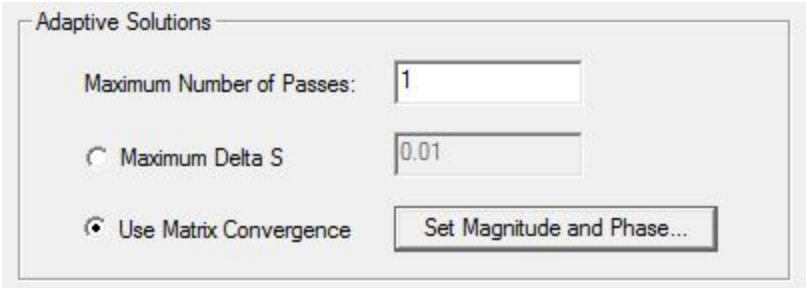
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can select either all ports (making a single-band composite subarray), one port or <Multiple> for a subset of ports.

You can define additional groups and edit the contents of groups.

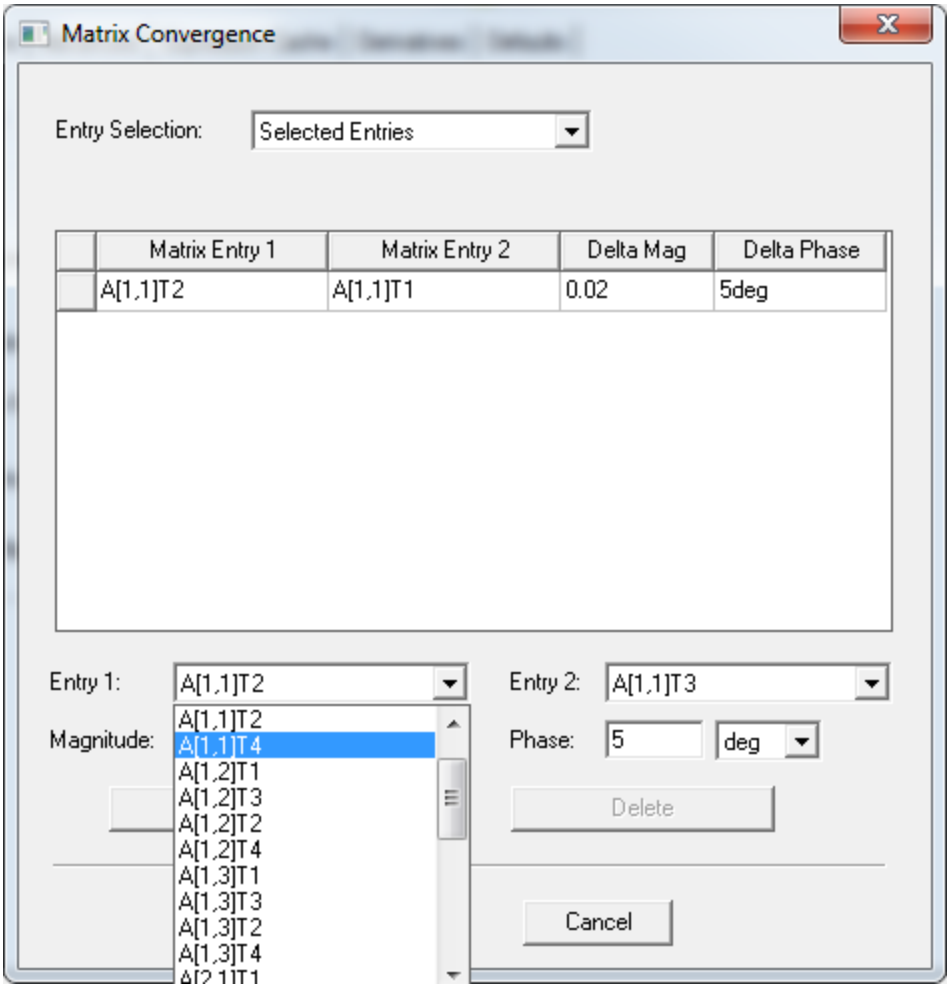
General Setup for Virtual Array Simulation

For a project with an array the General Setup some differences appear in how you can specify Matrix convergence.



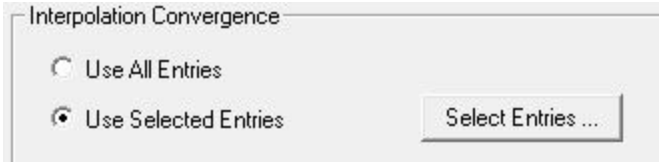
If you select Matrix Convergence, and click **Set Magnitude and Phase**, you will see the Matrix convergence dialog box. In the Matrix Convergence dialog box, if for **Entry Selection** you choose Selected Entries, (rather than All or Diagonal/Off Diagonal), you will see scrollable drop-down menus that let you select from all Array elements to define pairs of Matrix entries.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



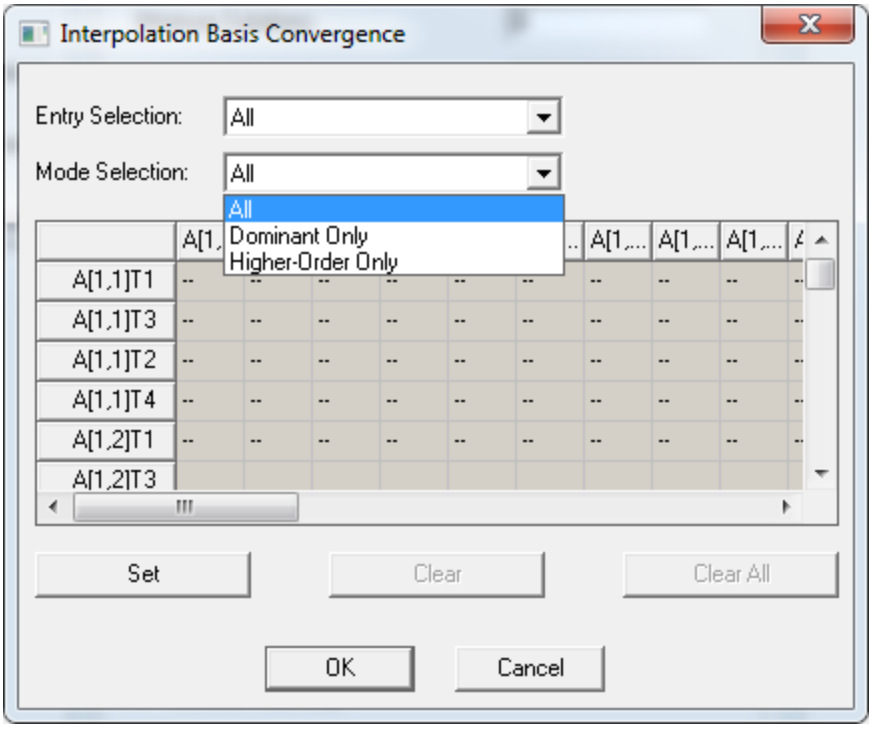
Interpolating Sweep Advanced Options for Arrays

For an Interpolating Sweep Advanced Options, if you select **Use Selected Entries**,



Clicking the **Select Entries** button displays an Interpolation Basis Convergence dialog that lists the Array elements.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note that the matrix entries are listed according to the Matrix Sort Order as specified by user.

Entry Selection can be All, Diagonal, or Off-Diagonal.

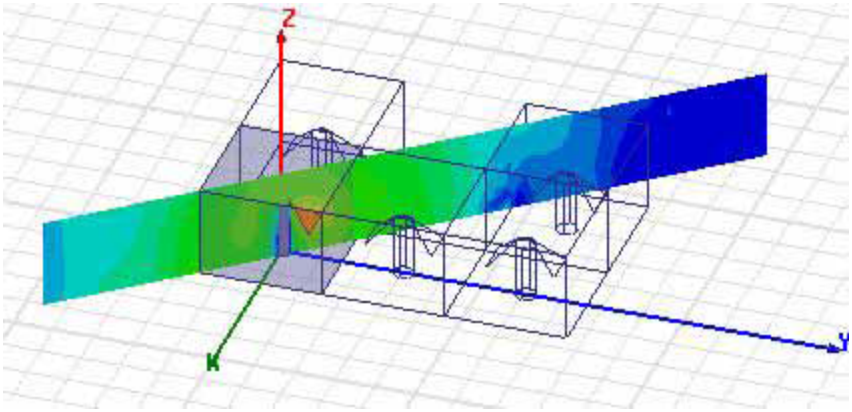
Mode Selection can be All, Dominant Only, or Higher-Order Only.

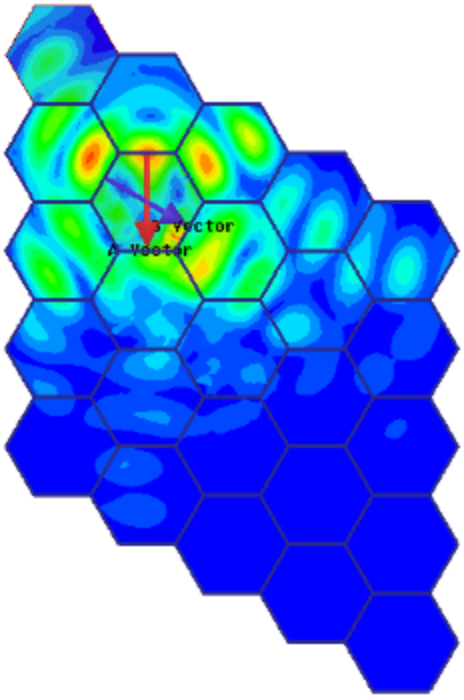
Post Processing for Array Models

Solution quantities of both physical and virtual cells are available for post-processing. You must select **Save Fields** in the Solution setup before running an analysis to enable the **Plot Fields** selections.

Radiation fields (near and far) post processing is over the whole array. You can plot and animate fields on non-model cut planes, vectors, and points, as well as on selected model object faces. Any padding cells in or around the array are treated as background material. You can plot and animate fields on cut planes, non-model objects set on the same coordinates as object or array surfaces, vectors, and points.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





You can use the Fields calculator to define calculated expressions.

HFSS can also post process individual cells in an array one at a time. You can select an arbitrary cell and do fields post processing on that cell. HFSS post processes on a single user selected cell in the array.

For Port Field Display there is no GUI change. Only physical ports/terminals will be listed. There is no need to support visualization of user-selected cell (like field overlay plot) because the field patterns of the virtual modes are the same as those in the physical cells.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Reports for Arrays
- Plotting Fields for Array Models
- Fields Calculator Applications for Geometries in Arrays
- Fields Post Processing for Selected Cells in Arrays

Reports for Arrays

There are some differences in the Report setup for Arrays.

1. The solution/matrix quantities are grouped by category. The entries in each category are listed according to their [row, column] order in the corresponding matrix.

   The entry in [row1, column1] will be listed first, followed by

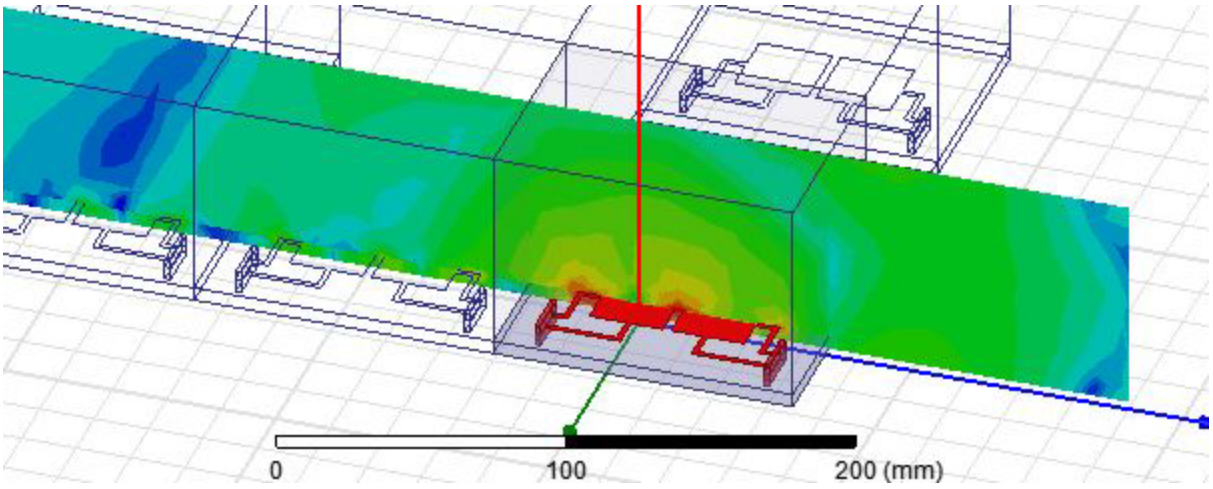
   [row1, column2], … [row1, columnN], [row2, column1], …

   [row2, columnN], … [rowN, columnN]. Note that the [row, column] order of each entry in the matrix is controlled by the 'Matrix' order as specified by user.

2. The existing "Filter" capability can help locate the desired quantity from the potentially very long list.

Field Plots for Arrays

You can generate field plots on object faces, as well as on non-model planes, lines, and points. The plots display calculated fields where ever the selected geometry intersects the array cells based on whether the cells are active, passive, or padding in the array properties. Locations designated as padding are treated as background material in field calculations. The virtual cells do not need to be visible to affect the generated field plots.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also create animations of field plots. If the plotted geometry is controlled by a swept variable (such as plane angle or a point location) the animation displays the correct values where ever they intersect the array based on the designations as active cells, passive cells, or padding cells.

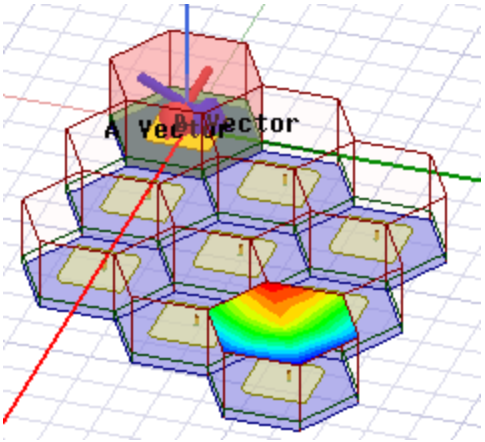
Fields Calculator Applications for Arrays

Geometries selected for Fields Calculator expressions can intersect any active, passive, or padding cells defined for the array. You can generate animated field output in which each frame is a snapshot of the fields on a different plane of the modeled volume. The Fields Calculator cookbook describes an example of the technique. Any derived field quantity can be plotted in this manner.

Fields Post Processing on a Designated Array Cell

For post processing fields information for arrays, you can select an arbitrary cell and do fields post processing on that cell.

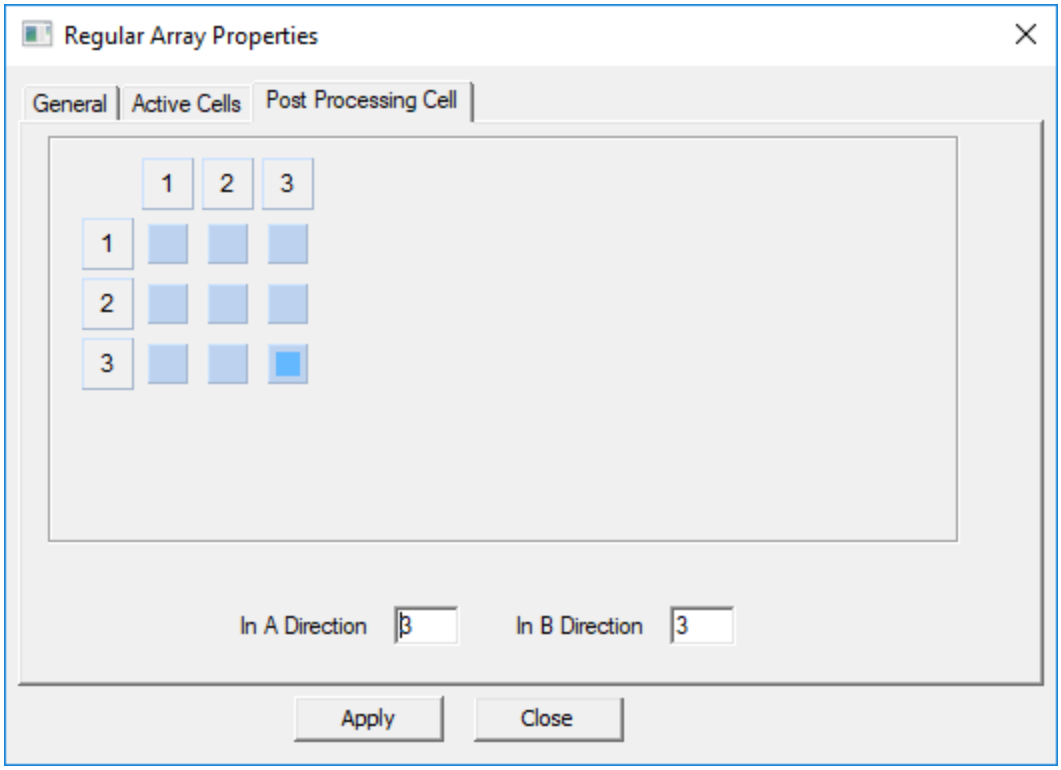
The unit cell mesh from position (1,1) will be translated to the user selected cell and fields will be plotted in the user selected cell.



The procedure is:

1. Select objects in the unit cell
2. Use the Array dialog to specify the cell to post process for fields, either by clicking on a the corresponding array position indicator in the Post Processing Cell tab in or by giving the A and B direction indices (row and column respectively).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Field plot will be visible in the user selected cell.

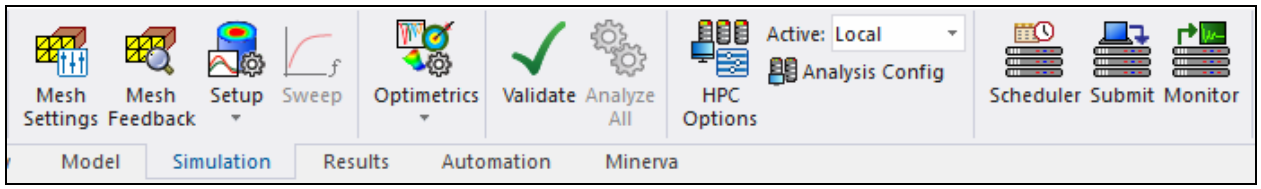
Radiation fields post processing will be over the whole array.

Jsurf is calculated inside the geometry of the cell but it will be equal to 0 along cell borders.

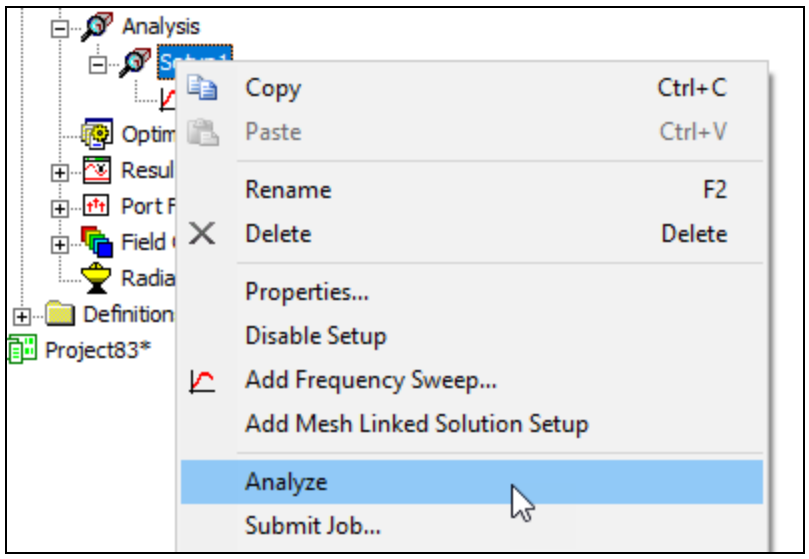
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22 - Running Simulations

After you specify how Ansys Electronics Desktop is to compute the solution, you need to begin the solution process. You can access the **Analyze** commands for a specific setup by right-clicking on a solution or sweep in the **Project** tree and selecting from the shortcut menu, you by selecting the **Analyze All** from the solver menu, or the **Analyze All** icon on the **Simulation** tab of the ribbon.

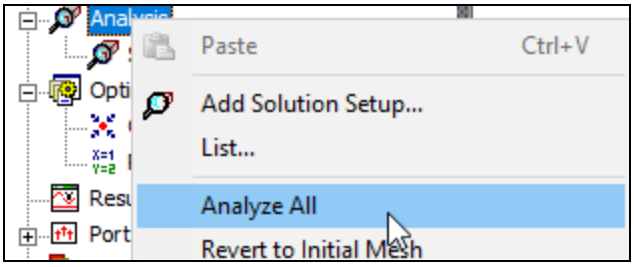


In general, the **Analyze** command on the shortcut menu applies to the selected setup and associated sweeps, if any, or to a selected sweep. To use this command, right-click on a setup or sweep in the Project Manager and click the command on the context menu.



The **Analyze All** command applies to allenabled setups,dependent setups, and sweeps at or below the level invoked in the Project Manager. To use this command, either click **HFSS**> **Analyze All** or right-click the **Analysis** icon in the *Project Manager* and select **Analyze All**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



What do you want to do?

- Local Analysis
- Solve a single setup with or without sweeps (when applicable)
- Solve a specific sweep
- Enable a queue so that multiple simulations can run sequentially as resources become available.
- Run more than one simulation, whether multiple setups, or multiple sweeps under a single setup, or setups with dependencies.
- Enable or Disable one or more solution setups or sweeps.
- Submit Job to RSM or HPC Scheduler
- Monitor queued simulations
- Monitor the solution process
- Change a solution priority for system resources
- Abort an analysis
- Re-solve after modifying a design
- Re-solve after Ansys Workbench Feedback

Local Analysis

Ansoft RSM Service does not need to be installed or running for local analysis and should not be running if the user only will be running local analysis. If running a distributed analysis, consult Distributed Analysis.

> **Note:**
>
> MPI distribution may be affected by starting or stopping a VPN, even if the MPI distribution is limited to the local host.

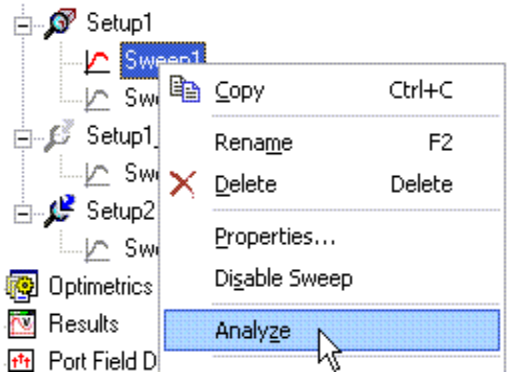
To run more than one analysis at a time, follow the same procedure while a simulation is running. If you have enabled queuing, the next solution setup will be solved when the previous solution is complete.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solving a Single Setup or Sweep

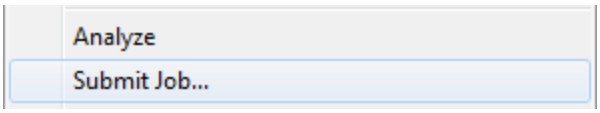
To run a single setup or sweep:

1. Select a solution setup or sweep in the Project Manager.
2. Right-click and select **Analyze** from the shortcut menu. The graphic shows a single sweep selected for analysis.



The 3D field solution is computed inside the structure for a solution. For a select sweep, it is computed for the sweep variables.

When you right-click a Setup, rather than a Sweep, the shortcut menu also includes the **Submit Job...** command.



For more information on the **Submit Job...** command see, Distributed Analysis and High Performance Computing (HPC) Integration.

To run more than one analysis at a time, follow the same procedure while a simulation is running. If you have enabled queuing, the next solution setup will be solved when the previous solution is complete.

> **Note:**
>
> If a linked dependency in the setup is already simulating (for example, due to setup links to the same external source for a near or far field wave, or a magnetic bias), Ansys Electronics Desktop will not allow another dependent simulation to start until the first use of the source has completed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

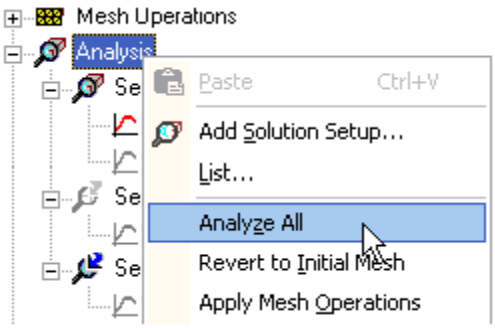
Running More than One Simulation

To solve every enabled solution setup in a design:

1. In the project tree, under the design you want to solve, select **Analysis**.
2. Click **HFSS**> **Analyze All**.

Each enabled solution setup is solved in the order it appears in the Project Manager.

The example here shows an analysis invoked from the Project Manager shortcut menu with three setups: one disabled, two enabled. The first setup has one sweep enabled, and one disabled (shaded icon). The second setup is disabled, and the third is enabled, with a disabled sweep.



Note:

The **General** tab for the Setup includes an **Enabled** check box. By default, this is checked. Clearing the **Enabled** check box excludes a setup from running

Note:

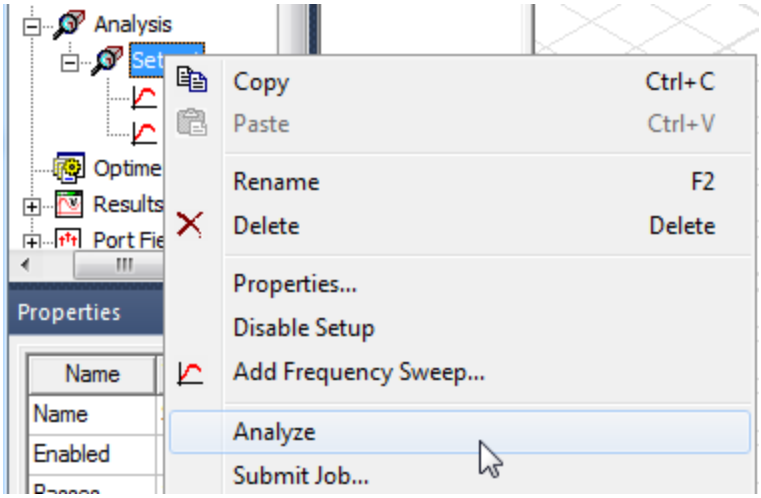
You can simulate all designs in a project by clicking **Project** > **Analyze All**.

To solve *two or more sweeps or two or more parametric analyses under a setup*:

1. In the project tree, under the design you want to solve, right-click **the setup** icon that includes the sweeps of interest.
2. Click **Analyze** on the shortcut menu.

   Each solution sweep under that setup is solved in the order it appears in the Project Manager, using the available machines. The following example shows a setup with two enabled sweeps.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Submitting a Job

Ansys Electromagnetics Desktop supports its own Remote Simulation Management (RSM) and Ansys Cloud Direct along with other High Performance Computing (HPC) software management programs (See: HPC Integration). The **Simulation** tab of the ribbon includes icons for setting HPC Options, creating and selecting analysis configuration, selecting the scheduler, submitting jobs, and monitoring jobs.

There are two ways that the GUI may be used to submit jobs. The first (and most common) mode requires that the Desktop (UI) process run on a host which is also a submission host for the job scheduler. This mode is called local mode or working mode.

The second mode is only supported on Linux in the Ansys Electromagnetics Suite. In the second mode, an administrator configures the RSM Service to act as an interface to the job scheduler, and starts the RSM Service on a submission host for the cluster. The user runs the Desktop (UI) process on another host (which may be called the post-processing host). To submit a job, the user specifies the host where the RSM Service is running, and the Desktop process connects to the RSM Service over the network to submit the job. In this mode, some configuration is required, and the RSM Service typically must run as a privileged user (for example, root), so that it can launch processes as any user. This mode is useful for cases in which the submission hosts are not able to run graphical processes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

For certain methods of resource selection for Windows HPC job submission, Ansys Electronics Desktop checks for nodes being online. In an auto-scaling cluster, nodes can fail the online check because they're not online until a job is running. An environment variable allows customers to bypass this check:

ANSYSEM_SKIP_NODE_ONLINE_CHECK

Set the variable value to 1 to disable the online check. Debug logging fully verifies what aspects of the cluster are causing this check to fail.

Job submission in Windows works without changing settings if environment variable ANSYSEM_SUBMIT_JOB_REQ_NODE_ONLINE is set with value 0. The installation doesn't set this environment variable. ANSYSEM_SUBMIT_JOB_REQ_NODE_ONLINE accepted values: 0 or 1 (0 = disable; 1 = enable, i.e. require nodes to be online for submission). Default: 1 (i.e. if not set, Ansys Electronics Desktop requires nodes to be online for submission).

To submit a job:

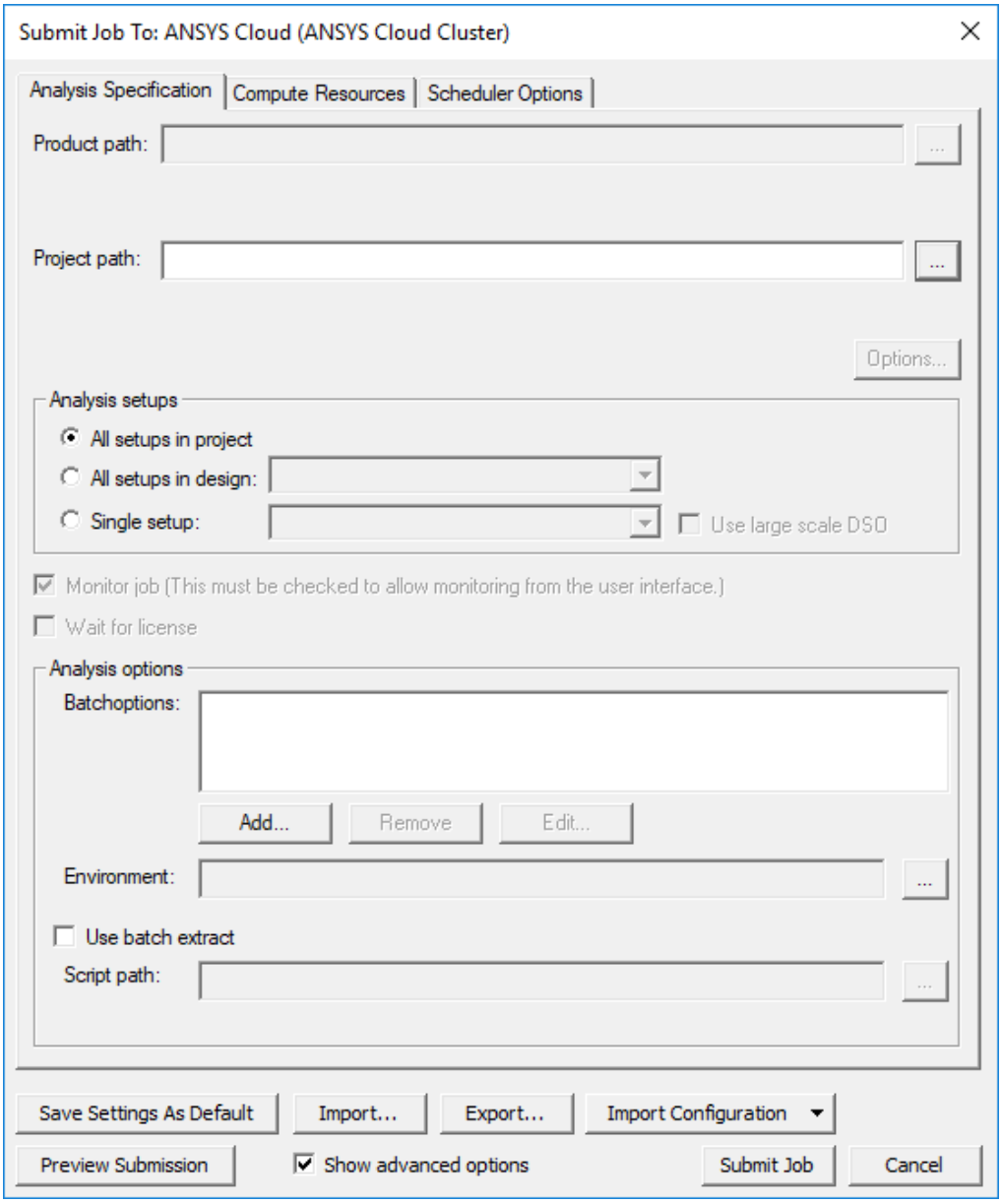
1. Prepare your design.

   For HFSS, Ansys recommends you use **HFSS** > **Design Settings** to turn on S parameter export for frequency so that they can be downloaded quickly after the simulation is complete. They must be exported to default project path for download to work correctly.

2. Open the **Submit Job To** window one of the following ways:
   - On the **Simulation** tab, click the **Submit** icon.
   - Select **Tools** > **Job Management** > **Submit Job**.
   - Select **HFSS** > **Submit Job**.
   - Right-click a solution setup and select **Submit Job**. *In this case, the information in the **Submit Job To** window will be pre-populated from the setup*.
   - From a Command window, use the -showsubmitjob command.

   The **Submit Job To** window appears. The window header indicates your selected scheduler.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This window contains up to three tabs:

- **Analysis Specification** – allows you to specify the product path, project name, setup and analysis options, batchoptions, and environment variables (for advanced users).
- **Compute Resources** – allows you to specify whether to use automatic settings, and to set resource selection and job distribution parameters.
- **Scheduler Options** – allows you to specify the job name and priority. *This tab does not appear if you have selected local RSM as the scheduler*.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. On the **Analysis Specification** tab, specify your desired options.

   The **Product path** and **Project** fields support mapped drives. Click the ellipses (**...**) to select files.

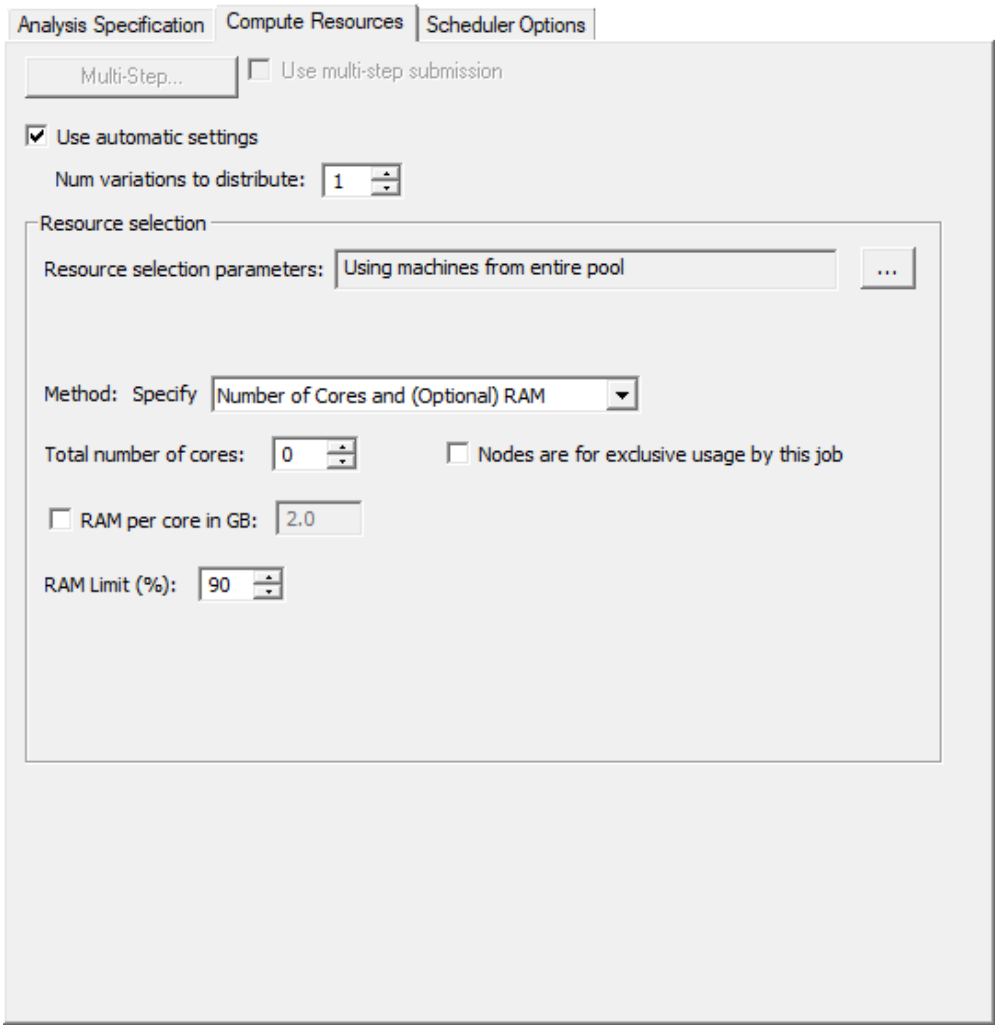
   The project can be an archive. The project file pathname must be a UNC path that is accessible from each compute host used for Ansys Electromagnetics jobs. After clicking the **Project path** field's ellipsis button (**...**), a check box allows you to **Use converted UNC path if mapped drive specified**. If you select a project or product on a mapped drive, and check the option, the converted UNC path equivalent to the mapped drive pathname is used.

   In the **Analysis Setups** area, select the radio button for **All setups in project**, **All setups in design**, or a **single setup**.

   For Parametric setups, you have the option to select **Use Large Scale DSO**. See: Large Scale DSO for Parametric Analysis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Select the **Compute Resources** tab.



Options may vary slightly depending on your selected scheduler.

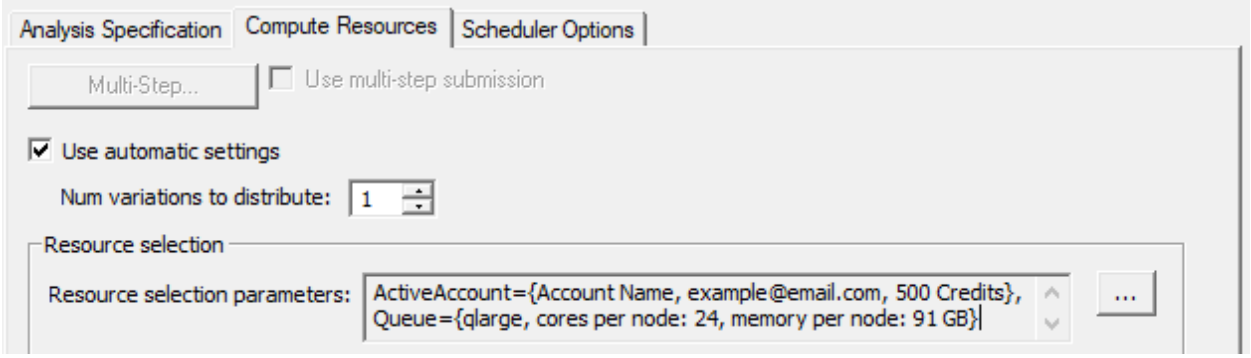
5. If applicable, determine whether or not to **Use multi-step submission**.
6. Determine whether or not to **Use automatic settings**.

> **Note:**
>
> Ansys strongly recommends using Automatic settings.

7. Use the buttons in the **Resource selection** area to allocate resources. See: Job Management for Large Scale DSO. If you are using Ansys Cloud Burst Compute select an active account using the ellipsis […] button to the right of the **Resource selection parameters:** text box and ensure that it has sufficient Core Hour Balance. This balance

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will be displayed in the **Resource selection parameters:** text box.



8. If you opted not to use automatic settings, you can also set **Job distribution** parameters. See: Distributed Analysis.
9. If applicable, select the **Scheduler Options** tab and set the **Job name** and **Priority**.

10. If desired, import or export job configurations.
11. If desired, click **Save Settings as Default** to save the current settings and overwrite defaults. These settings are saved on a per-scheduler basis.
12. If desired, click **Show advanced options** to see options for advanced users. See: Running Electronics Desktop from a Command Line.
13. If desired, click **Preview Submission** to view the commands to be sent to the scheduler. Ansys Cloud Direct submissions contain queue (pool) configuration details, including the job's hourly cost in Ansys Elastic Units (AEUs).

    The preview text can be copied to the clipboard.
14. Click **Submit Job** to submit the job to your selected scheduler.

Monitoring Queued Simulations

If you have multiple setups for a design and have selected **Analyze All**, the simulations can be queued until there is a machine available. You enable queuing in the HPC and Analysis Options: Options tab. If queuing is enabled, and you run multiple setups, they are solved in the order that they appear in the Project Manager. You can prioritize setups by changing their order in the queue.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. To view the solution queue, click **Tools> Show Queued Simulations**.

   This displays a dialog box showing all the simulations and their current status. You can select and remove any simulation from the queue.

   You can also select any setup and use the **Move up** and **Move down** buttons to prioritize them.

2. To remove a simulation from the queue, select the simulation and click **Remove from Queue**.

   This removes the selected simulation from the queue.

Monitoring the Solution Process

While a simulation is running, you can monitor the solution's progress in the **Progress** window. The progress bar shows the relative progress of the simulation.

You can also view the following solution data at any time during or after the solution:

- The convergence data
- The matrices computed for the S-parameters, impedances, and propagation constants
- A profile showing the status of the analysis process, including the progress and number of valid passes completed for adaptive solutions, elapsed time, memory usage, and more
- Mesh statistics (numbers of elements, lengths, volumes, and volume standard deviation; all on a per-object basis).

To view the *Solutions* window:

1. Right-click the solution **Setup** in the Project Manager.

   A shortcut menu appears.

2. Select **Convergence**, **Solutions**, **MatrixData**, **Mesh Statistics**, or **Profile** from the shortcut menu.

   The *Solutions* window appears with the corresponding tab selected and the current data displayed.

For "out of core" problems, quite different amounts of memory may be used for factorization and for solution. So if the amount for factorization is displayed under the progress bar and the amount used is calculated for the profile at the end of the solution, they may be quite different numbers.

To view the solution status:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click **HFSS** > **Results** > **Browse Solutions**.

  The *Solutions* dialog box appears with the **Browse** tab selected. It displays data about the number of valid passes completed (when adaptive solutions are applicable). It contains a tree structure showing the solutions listed according to Setup, Solution, and Variation. A table lists the setup, the solution, the sweep variable (when applicable), and the state of the solution.

- You can use the **Properties** button to display a dialog box that lets you change the way the Setup, Solution, and Variation are listed in the tree structure of the *Solutions* dialog box.
- You can delete one or more solutions by selecting from the table and clicking **Delete**. Click on a solution to select it, and use Ctrl+click to select multiple solutions, or Shift+click to select a range of solutions. You can also select all solutions using the **Select All** button.

  > **Note:**
  >
  > If a license is lost, the software waits for the license to be regained, checking every 2 minutes or until you abort.

- The **Statistics** tab of the *Solutions* dialog box displays path information as well as format, number of files, and size.

Changing a Solution Priority for System Resources

You can modify the priority of Ansys Electronics Desktop simulations so that system resources are allocated to other computer processes before the solver. If you reduce the priority of Ansys Electronics Desktop simulations, your other software tools will respond as they normally would, but Ansys Electronics Desktop simulations may take longer.

> **Note:**
>
> The Windows Task Manager does not indicate a reduced priority for the Ansys Electronics Desktop solvers. It only lists the priority of the engine manager, which appears normal, not the actual engine. The actual engine is in a separate thread, whose priority is not visible in the Windows Task Manager.

To change the priority of simulations for the system's resources:

1. While a solution is running, right-click the **Progress** window, and click **Change Priority** on the shortcut menu.
   - To affect priority for future simulation runs, click **Tools> Options> HPC and Analysis** and click the **Options** tab of the resulting dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. From the **Change Priority** menu (or the **Default Process Priority** drop-down menu), select one of the following priorities:

Highest	
Above Normal	
Normal	The default.
Below Normal	
Lowest Priority	

3. Click **OK**.

Related Topics

Monitoring Queued Simulations

Aborting an Analysis

To end the solution before the currently running process is complete:

- Right-click in the **Progress** window and click **Abort**.

  The solver ends the analysis immediately. The data for the currently running adaptive pass, frequency point, time step, or variation is lost. Previously completed solutions may or may not be retained (depending on what process was interrupted by the Abort command). If you want to ensure that the results of all completed solutions are saved, use *Clean Stop* instead of *Abort*.

To abort the solution after the currently running process is complete:

- Right-click the **Progress** window and click **Clean Stop** on the shortcut menu.

  The analysis ends after the currently running adaptive pass, frequency point, time step, or variation has been solved. Solutions completed before the stop request are retained.

If you request a clean stop during the third adaptive pass, the solution for the third pass will be available once the third pass has finished solving, but the fourth pass will not run.

Ansys EM Application as an LSF Job

If you have an Ansys EM application running as an LSF job, you can use the command "bkill -s SIGTERM *jobid*" to terminate that application. Here *jobid* is the LSF job id. The response will be "Job <jobid> is being signaled." The response is the same whether the job is actually being signaled or not.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In cases where the SIGTERM parameter is ignored, the command kills the LSF job, but does not clean the lock files, and other files may not be in a consistent state. See http://www.vital-it.ch/support/LSF/programmer/advanced.html for a detailed description under *Signal Handling in Windows*.

Linux

For Linux, you can use TERM commands. Sigterm handling is done in Desktop library. You can abort a running batchsolve by sending a TERM signal to hfss.exe.

Re-solving after Modifying a Design

In some cases, if you modify a design after generating a solution, the solution in memory will no longer match the design. In such cases you receive a warning message that "Solutions have been invalidated. Undo to recover."

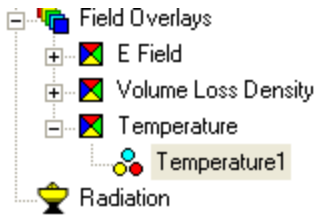
To generate a new solution after modifying a design, follow the procedure for running a simulation.

Also see Re-Solving with Ansys Workbench Thermal Feedback.

Re-solving after Ansys Workbench Thermal Feedback

With the Enable Feedback box in Setting the Temperature of Objects dialog box is checked, you can manage analysis with feedback in Ansys Workbench. After solving an HFSS or Q3D Extractor or Maxwell design, after performing the corresponding linked thermal analysis in Ansys Workbench, you can receive a temperature distribution back from the thermal solution. Ansys Workbench will write the feedback files directly to the Project Solution directory.

After an analysis that includes thermal feedback from Ansys Workbench, you can see temperature changes expressed in Temperature field overlays (both visually in the overlay and in the color key) as well as in the Solution data.



In the **Solution** data Profile tab you will see a new entry for Maximum Delta T, for the change in temperature from the previous simulation. The solver calculates delta in the first iteration by comparing the temperature distribution output from thermal with the initial temperature setting in

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HFSS/Maxwell/Q3D. Subsequent simulation iterations provide a number for the temperature delta.

Solver MCS4	00:00:00	00:00:00	31.4 M	Disk = 0 KBytes, matrix size 3468 , matrix bandwidth
Field Recovery	00:00:00	00:00:00	31.4 M	Disk = 310 KBytes, 2 excitations
				Maximum Delta T = 7.0685

This simulation feedback loop from Ansys Electronics Desktop to Ansys Workbench and back can continue until you decide that Temperature delta reported in the Solution Report low and stable for the designs.

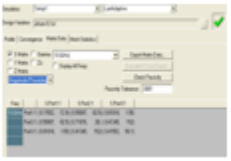
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

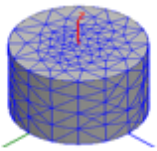
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

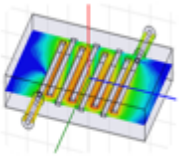
23 - Postprocessing and Generating Reports

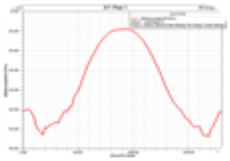
As Ansys Electronics Desktop completes a solution, you can display and analyze the results:

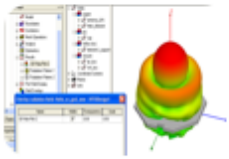
View Solution Data

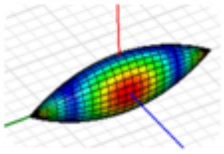

View Optimetrics Results

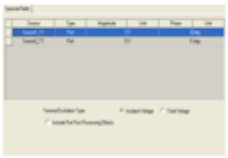

Create Mesh Plots

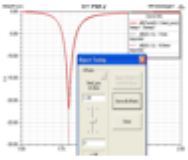

Plot Field Overlays

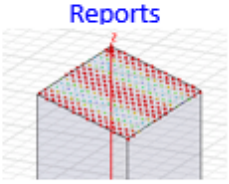

Create or Modify 2D or 3D Reports


Overlay 3D Field Plot on Model


Create Animations

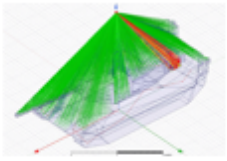

Scale Excitation Mag/Phase

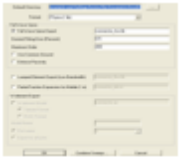

Tune Reports


Port Field Display

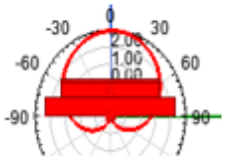
Specify Output Variables

Radiated Fields Observation Regions

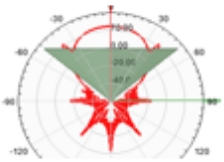

Visual Ray Trace

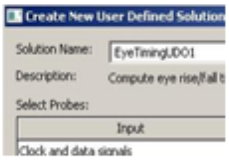

Export Equivalent Circuit Data

Fields Calculator / Derived Field Quantities



Overlay 2D Radiation Fields


Overlay Monostatic/ Bistatic RCS



User Defined Outputs

User Defined Documents

Network Data Explorer

- View solution data including the following: convergence information, computing resources that were used during the solution process, mesh statistics, and matrices computed for the S-parameters, impedances, and propagation constants during each adaptive, non-adaptive, or sweep solution. For eigenmode solutions, you can view the real and imaginary parts of the frequency and quality factor Q computed for each eigenmode. Solution data can also be viewed while Ansys Electronics Desktop is generating a solution.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- View analysis results for Optimetrics solutions.
- Plot field overlays - representations of basic or derived field quantities - on surfaces or objects.
- Overlay Field Plots on Models
- Create 2D or 3D reports of S-parameters, basic and derived field quantities, and radiated field data.
- Plot the finite element mesh on surfaces or within 3D objects.
- Create animations of field quantities, the finite element mesh, and defined project variables.
- Scale an excitation's magnitude and modify its phase.
- Apply Derivative Tuning to Reports

> **Note:**
>
> Except in the case of non-model boxes drawn in the global coordinate system (CS), non-model objects cannot be used for any fields post processing operation You can use non-model boxes drawn in the global CS for post processing operations, including integration and solution domaining.

Viewing Solution Data

While Ansys Electronics Desktop is generating a solution, or when it is complete, you can view the following information about the solution:

- Convergence information.
- Computing resources, or profile information, that were used during the solution process.
- Matrices computed for the S-parameters, impedances, and propagation constants during each adaptive, non-adaptive, or sweep solution.
- Mesh statistics
- For eigenmode solutions, view the real and imaginary parts of the frequency and quality factor Q computed for each eigenmode.
- For Characteristic Modes solutions, a CMA Data tab reports the the Number of Modes, Characteristic Angle and current, Modal Significance and Quality Factor, and Voltage per port based in edit sources weighting.
- The state of solved solutions.
- For transient solutions, Transient Data.

To access the **Solution Data** window, in which the information above can be accessed, do one of the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click **HFSS**> **Results**> **Solution Data**.
- Right-click **Results** in the Project Manager, and then click **Solution Data** on the shortcut menu.
- Select the **Results** tab on the ribbon, and click the **Solution Data** icon.

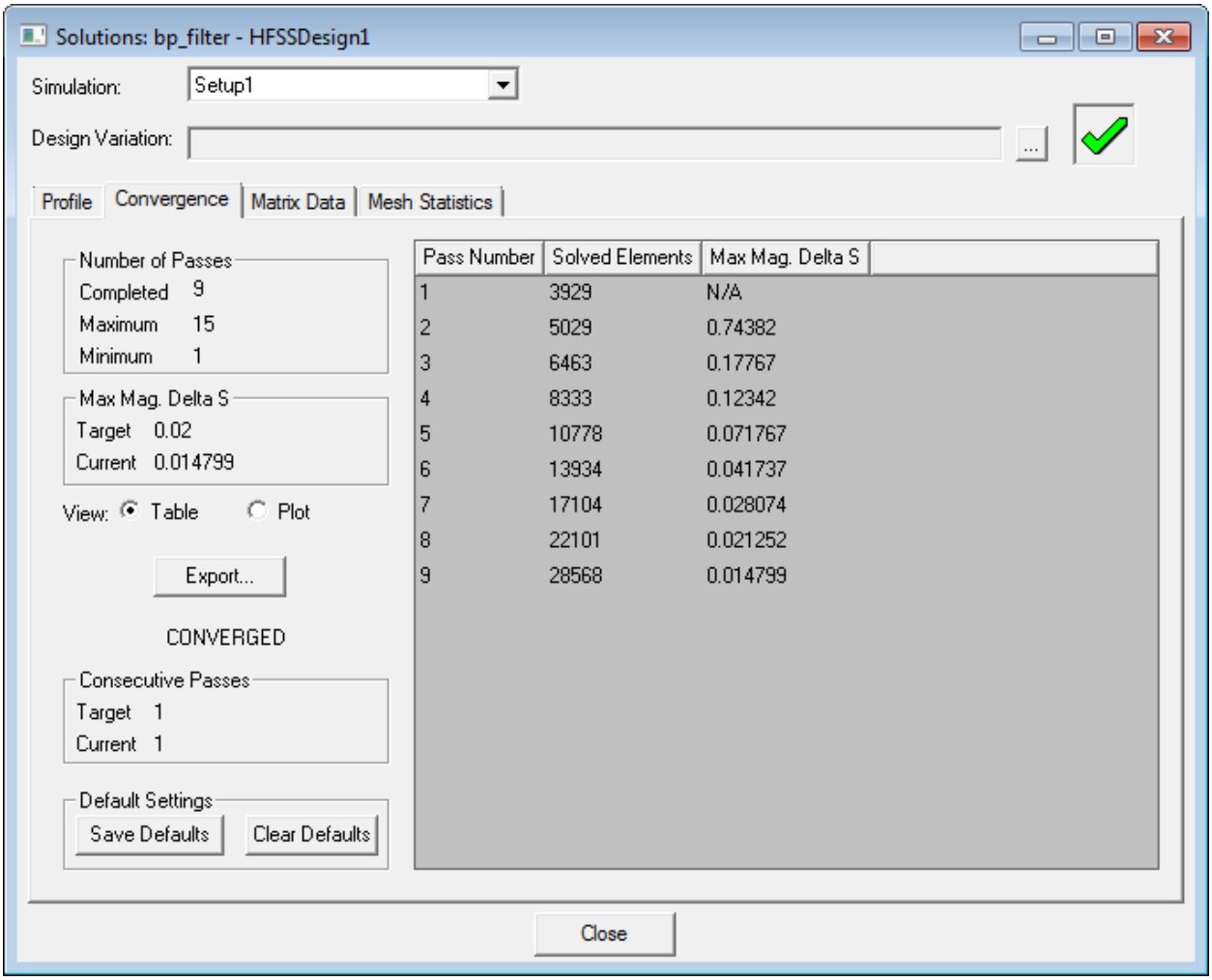
Viewing Convergence Data

To view an adaptive solution's convergence information, either during or after the solution process:

1. In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

   The **Solution Data** window appears. The **Convergence** tab is selected. You can also select the **Results** tab of the ribbon, click the **Solution Data** icon, and click the **Convergence** tab when the **Solution Data** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. If you have run multiple simulations, from the **Simulation** list, select the solution setup for which you want to view convergence data.

   By default, the most recently solved solution is selected.

3. Under the **Convergence** tab, depending on your design setup, you can review the following convergence data:
   - Whether the solution is converged or not converged.
   - Number of adaptive passes completed and remaining.
   - The Solved Elements at each adaptive pass, which includes solve inside tetrahedra, and, for projects using IE Regions for metal objects, also includes the number of solved IE surface triangles.
   - Maximum magnitude of delta S between two passes.
   - Maximum delta Energy between two passes.
   - Magnitude margin between passes.

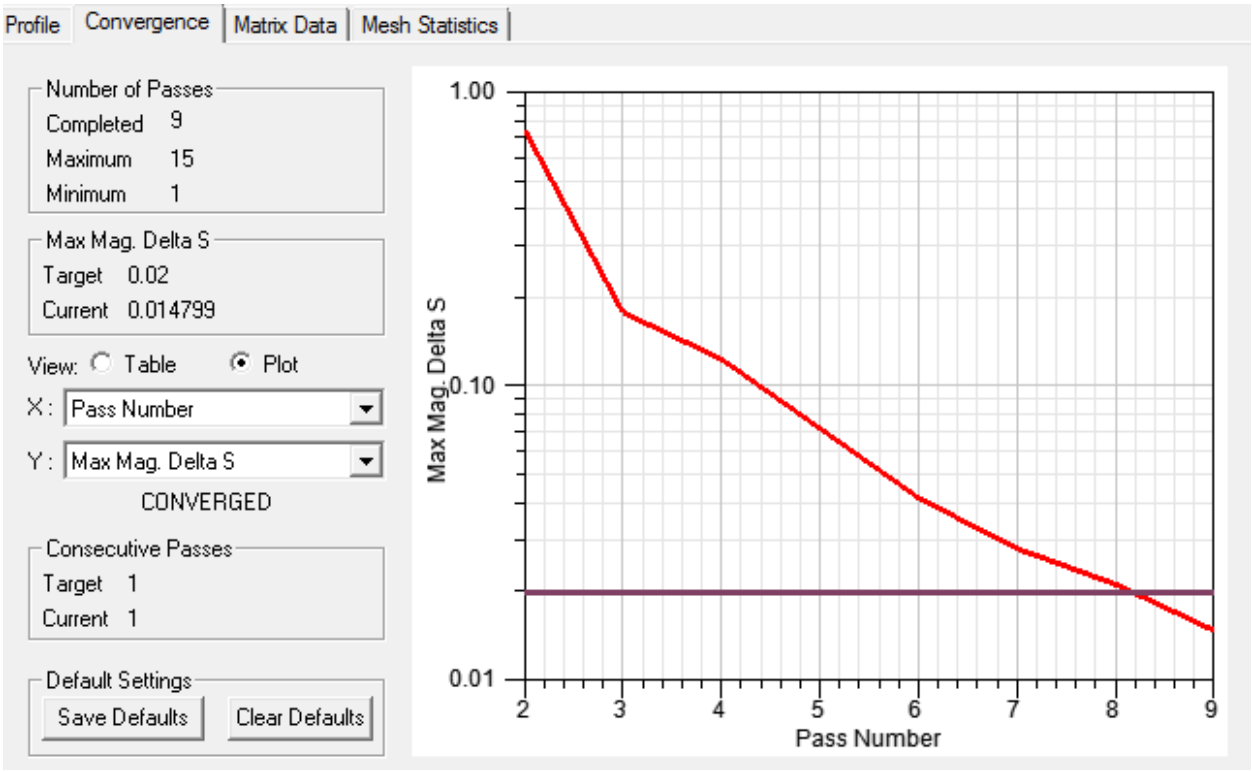
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Phase margin (deg) between passes.
- Maximum delta frequency between passes.

If for the Solution Setup, you elected to **Use Matrix Convergence**, and selected specific table entries for the Magnitude and Phase, the **Convergence** tab also shows the following values with the Magnitude Margin and Phase Margin:

- Max Delta (Mag S)
- Max Delta (Phase S)

4. Select **Table** to display the convergence data in table format or **Plot** to plot the convergence data on a rectangular (X - Y) plot.



5. If you select **Save Defaults**, the settings for Table vs. Plot, and for Plot, the X and Y axis selections, those settings apply, based on the design-type, each time you open the desktop. You can also select **Clear Defaults** to reset to the original settings.

Note:

If you receive a message that the eigenmodes have not converged, it may indicate that the existing mesh is too coarse. You may need to refine the mesh.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Viewing the Number of Completed Passes

At any time during the solution process, you can view the number of adaptive passes (solve — error analysis — refine cycles) that have been completed and that have yet to be completed. When the solution is complete, you can view the number of adaptive passes that were performed. If the solution converged within the specified stopping criteria, fewer passes than requested may have been performed.

To view the number of passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

The **Solution Data** window appears. The **Convergence** tab is selected.

The number of completed and remaining passes is listed in the **Number of Passes** area.

Viewing the Max Magnitude of Delta S Between Passes

For solutions with ports.

At any time during or after the solution process, you can view the maximum change in the magnitude of the S-parameters between two consecutive passes. This information is available after two or more passes are completed.

To view the maximum magnitude of delta S between passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

The **Solution Data** window appears. The **Convergence** tab is selected.

The **Max. Mag. Delta S** column lists the maximum magnitude of delta S from one pass to the next.

The **Max. Mag. Delta S** area lists the target change in magnitude of delta S and the change in magnitude of delta S between the last two solved passes.

> **Note:**
>
> Delta S is computed on the appropriate S-parameters - modal or terminal - after the S-parameters have been de-embedded and renormalized.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> You can renormalize mathematically, without having to re-solve, by accessing the postprocessing tab on the port definition panel and de-selecting the **Deembed** selection box.

Viewing the Delta Magnitude Energy

For designs with voltage sources, current sources, or incident waves. Not applicable to designs with ports.

At any time during or after the solution process, you can view the difference in the relative energy error from one adaptive pass to the next. The change in the magnitude of delta energy is available after two or more passes are completed.

To view the delta magnitude E between passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

The **Solution Data** window appears. The **Convergence** tab is selected.

The **Delta Mag. Energy** column lists the delta energy from one pass to the next.

The **Delta Mag. Energy** area lists the target change in delta energy and the change in delta Energy between the last two solved passes.

Viewing the Magnitude Margin

For solutions in which convergence criteria for specific S-matrix entries were specified.

At any time during or after the solution process, you can view the solution's proximity to the target delta magnitude, which was specified in the **Matrix Convergence** dialog box. The magnitude margin is available after two or more passes are completed.

To view the magnitude margin between passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

The **Solution Data** window appears. The **Convergence** tab is selected.

The **Magnitude Margin** column lists the magnitude margin from one pass to the next.

Viewing the Phase Margin

For solutions in which convergence criteria for specific S-matrix entries were specified.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

At any time during or after the solution process, you can view the solution's proximity to the target delta phase, which was specified in the **Matrix Convergence** dialog box. The phase margin is available after two or more passes are completed.

To view the phase margin between passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

The **Solution Data** window appears. The **Convergence** tab is selected.

The **Phase Margin** column lists the phase margin from one pass to the next.

> **Note:**
>
> When the Mag S becomes small (near to zero) its phase becomes indefinite and insignificant due to mathematical issues so that Phase Margin will be discarded.

Viewing the Max Delta (Mag S)

For solutions in which convergence criteria for specific S-matrix entries were specified.

At any time during or after the solution process, you can view the maximum difference of the S matrix magnitudes between two consecutive passes. The Max Delta (Mag S) is available after two or more passes are completed.

To view the Mag S between passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

The **Solution Data** window appears. The **Convergence** tab is selected.

The **Max Delta (Mag S)** column lists the Max Delta (Mag S) from one pass to the next.

Viewing the Max Delta (Phase S)

For solutions in which convergence criteria for specific S-matrix entries were specified.

At any time during or after the solution process, you can view the maximum difference of the S Matrix phase between two consecutive passes. The Max Delta (Phase S) is available after two or more passes are completed.

To view the Max Delta (Phase S) between passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

The **Solution Data** window appears. The **Convergence** tab is selected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Max Delta (Phase S)** column lists the Max Delta (Phase S) from one pass to the next.

Viewing the Maximum Delta Frequency

For Eigenmode solutions.

At any time during the solution process, you can view the maximum delta frequency, the largest percent difference in the resonant frequencies from one adaptive pass to the next. It is a measure of the stability of the computed frequencies from pass to pass and is available after two or more passes are completed.

To view the maximum delta frequency between passes:

- In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

The **Solution Data** window appears. The **Convergence** tab is selected.

The **Max Delta Freq. %** column lists the maximum delta frequency from one pass to the next.

The **Max Delta Freq. %** area lists the target maximum delta frequency and the maximum delta frequency between the last two solved passes.

Plotting Convergence Data

To display convergence data vs. pass on a rectangular (x - y) plot:

1. In the project tree, right-click the solution setup of interest, and then click **Convergence** on the shortcut menu.

The **Solution Data** window appears. The **Convergence** tab is selected.

2. In the lower-left corner of the window, select **Plot** as the view type.
3. Select the data you want to plot on the x-axis from the **X** drop-down menu.
4. Select the data type you want to plot on the y-axis from the **Y** pull-down list.

The x -y plot appears in the view window.

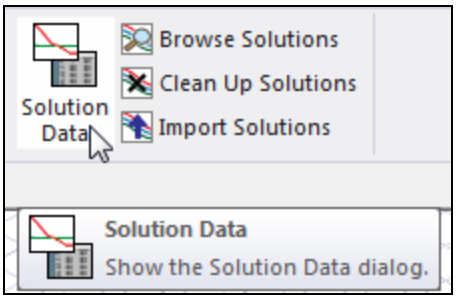
Viewing a Solution Profile

At any time during or after the solution process, you can examine the computing resources or profile data that were used by Ansys Electronics Desktop solvers during the analysis. The profile data is essentially a log of the tasks performed by Ansys Electronics Desktop during analysis. The log indicates the length of time each task took and how much physical and disk memory was required.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

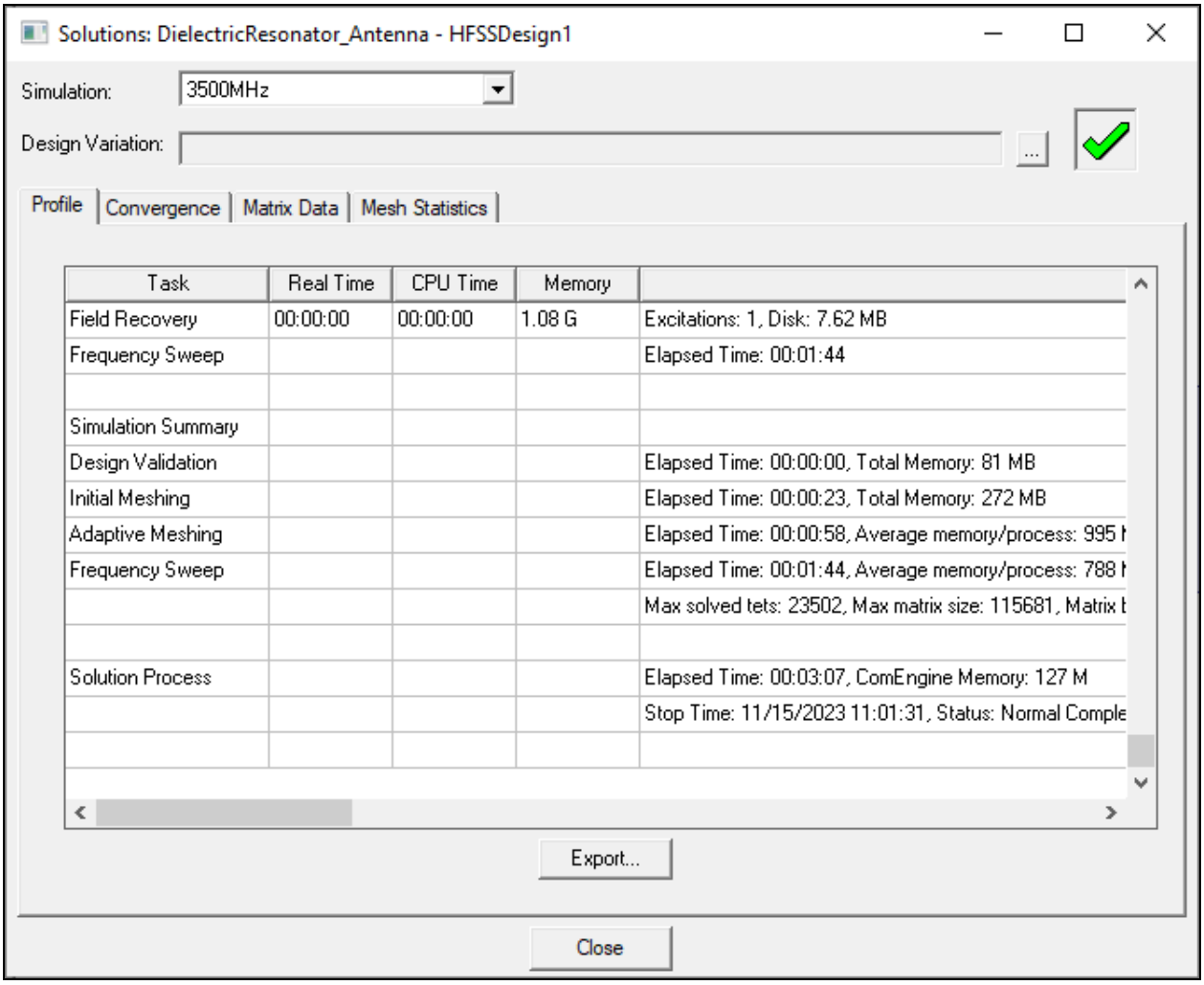
1. In the project tree, right-click a solution setup, and select **Profile**.

   The **Solutions** dialog box appears, with the **Profile** tab selected. You can also click the **Solution Data** icon on the ribbon to open the **Solutions** dialog box, and then select the **Profile** tab.



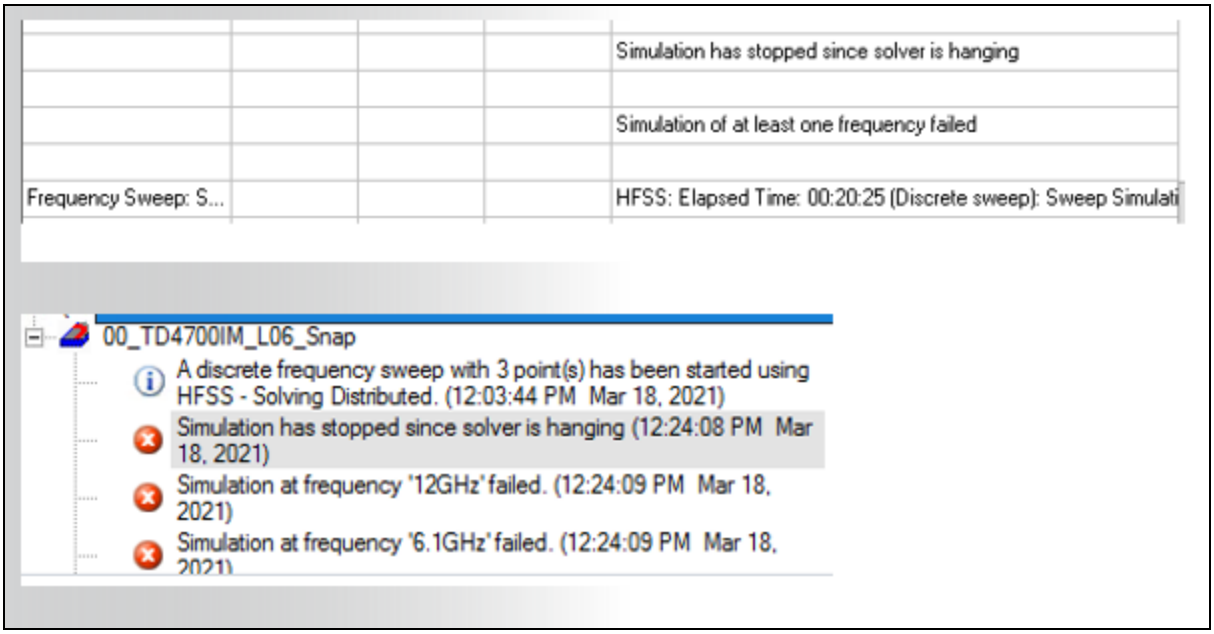
   The displayed data depends on the type of problem and solution setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the simulation hangs, equivalent messages appear in the Message window and the Solution **Profile** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. If one or more dependent setups exist, you can select the desired profile information from the **Simulation** drop-down menu. In general, the information displayed includes:

Task	Lists the type of task that was performed. Tasks include Start, various Mesh tasks, Simulation Setup, Port Adaptation, Adaptive Pass tasks, including simulation setup, Matrix Assembly, Solver tasks, and Field Recovery, Sweep tasks, and Solution Process summary and Totals for time.
Real Time	The difference in time between the start of the task and the end of the task (elapsed time).
CPU Time	The amount of CPU time required to perform the task.
Memory	The peak amount of physical memory (RAM) used by the individual executable running the task. The memory is freed for other uses after each task is complete.
Information	General information about the solution (for example, the number of tetrahedra used in the mesh, disk use, solver information, sweep information, and totals).

The matrix solver writes specific information in some of these fields, as outlined below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Task	The matrix solver task reports the type of solution performed by the solver, based on the physics of the problem. It has the form "Solver *pdsn*" (e.g., Solver MRS2, Solver DCS4-L2), where: • *p*, the precision type is: M (mixed for direct solver), or Mxx (for distributed direct solver), or D (double for direct and iterative solver) or E (extended precisions solve) • *d*, the matrix data type is: R (real) or C (complex) • *s*, the symmetry type is: S (symmetric), A (asymmetric), or H (hermitian) • *n*, the number of processors used. You specify the number of available processors on the local machine in the solver options. If a solve does not use all available processors (local or distributed), the number reported may be less than the number available. If a simulation uses the iterative solver, the Solver designation can include a level indicator appended. to an Iterative solver designation (L2 in the example above). The higher the Level number the lower the memory, you will never see L1 (this would be equivalent to direct solver. And a first order solve will only display L2 since it only has one level of order to go down for preconditioning. A second or mixed order solve may display L3 depending on the mesh quality. If the solver switches from the Iterative Solver to the Matrix solver, you see a Matrix solver warning: Switch from Iterative Solver to Direct Solver.
Information	The matrix solver information line includes, for example, Disk = 0 KBytes, matrix size 11137 , matrix bandwidth 20.3. • **Disk** – The amount of hard disk space used during the calculation of the matrix solution. If the disk usage for matrix solver is non-zero in profile, it usually indicates off-core matrix solver. If the matrix solver must solve off-core, smaller blocks of the data to be solved are created on disk, each block is then solved in physical memory, and then the matrix solution is reassembled. As a result of this additional processing, the time required to calculate a solution is higher. • **Matrix size** – The size of the matrix that was solved (the number of unknowns) • **Matrix bandwidth** – An FEM matrix is a sparse matrix. The solver only stores the non-zero entries. The matrix bandwidth is the average number of non-zeros per row. It gives an idea of the sparsity of a FEM matrix.Storage for the sparse matrix is proportional to the total number of nonzeros = #rows x bandwidth. The higher the bases order, the larger the bandwidth. • **Iterations** – For the Iterative Solver.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To export the profile data:

1. Open the **Solutions** dialog box and select the **Profile** tab.
2. Click **Export Profile**.

   This opens a file save dialog box that lets you provide a file name and location.

3. Click **Save**.

   The data is saved in a text file with a .prof extension.

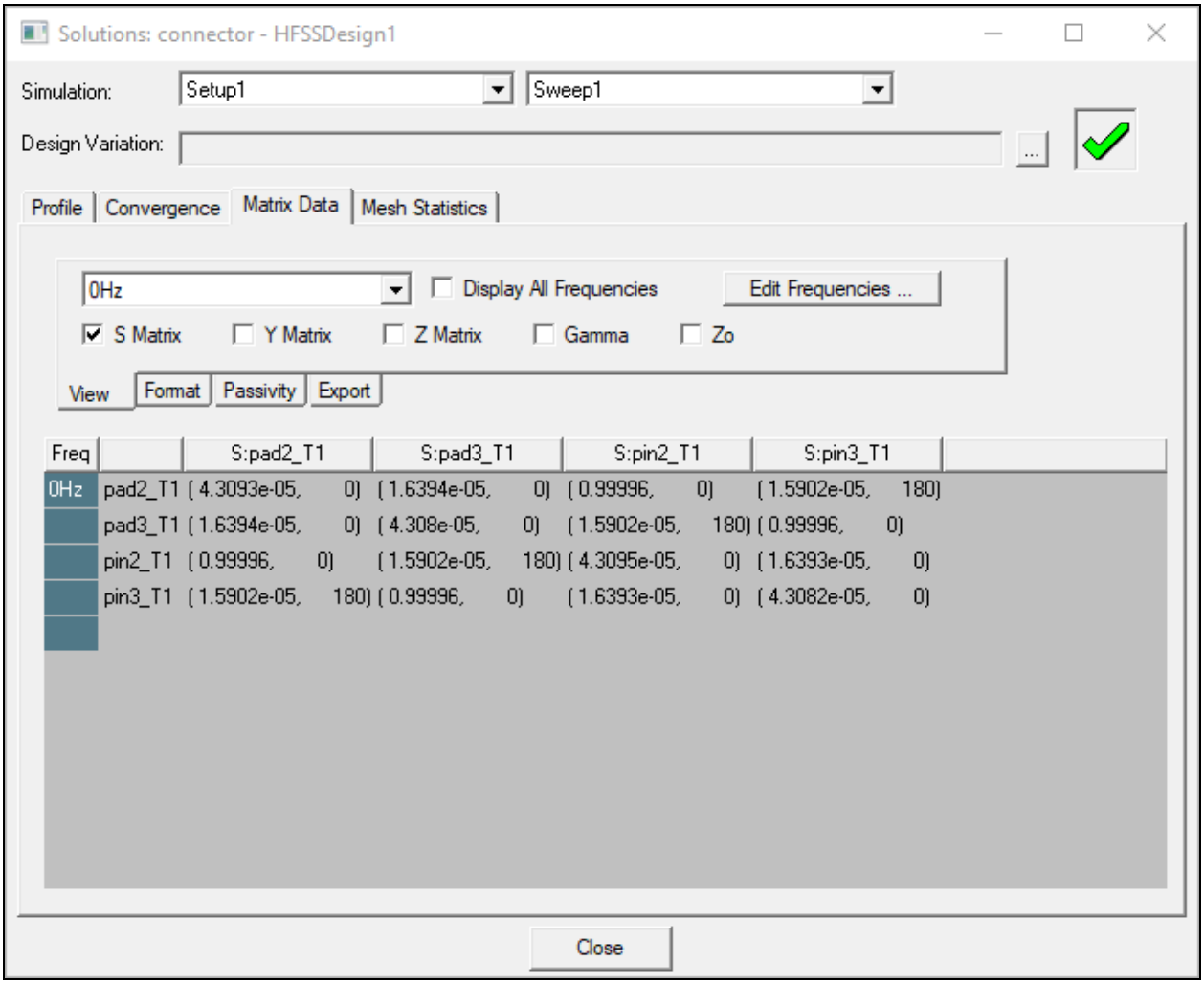
Viewing Matrix Data

To view matrices computed for the S-parameters, impedance, and propagation constants during each adaptive, non-adaptive, or sweep solution:

1. In the **Project** tree, right-click the solution setup of interest, and then click **Matrix Data** on the shortcut menu, or on the **Results** tab of the ribbon, click **Solution Data**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Solution Data** dialog box appears. The **Matrix Data** tab is selected.

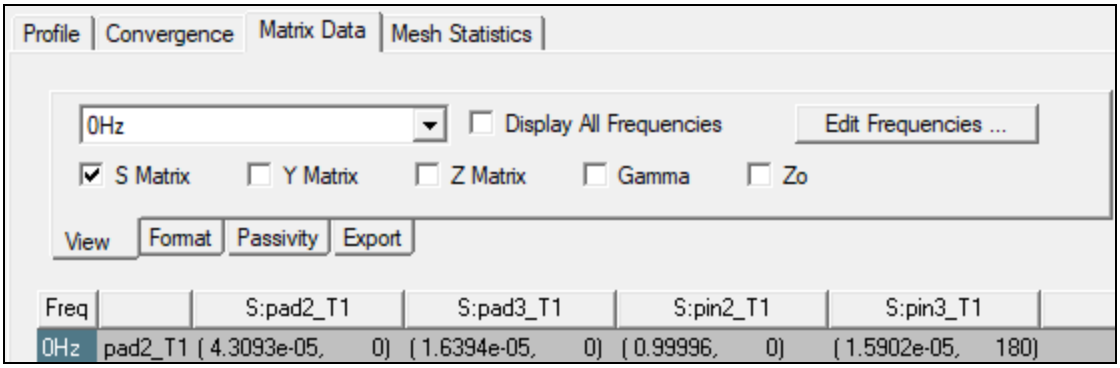


2. Select which pass to view in the drop down list to the right of the **Simulation** list. To select a specific adaptive pass:
    a. Choose **Adaptive Pass**.
    b. On the **View** tab, select the pass you want from the **Pass** list.
3. If design variations exist, in the **Design Variation** text box, specify the design with the matrices you want to view.

    Optionally, choose a design variation solved during an Optimetrics analysis from the **Set Design Variation** dialog box. This lists all the solved variations in the design. This dialog box is accessible from the **Solution Data** window by clicking the ellipsis button on the right of the Design Variation field, and via the **[solver]** > **Results** > **Apply Solved Variation** command.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. In the **Simulation** drop-down menu, click the solution setup and solved pass - adaptive, single frequency solution, or frequency sweep - for which you want to view matrices.
5. Select the type of matrix you want to view: **S-matrix**, **Y-matrix**, **Z-matrix**, **Gamma**, or **Zo** (characteristic impedance.) The available types depend on the solution type.



6. Select the solved frequencies to display:
   - To display the matrix entries for all solved frequencies, select **Display All Freqs**.

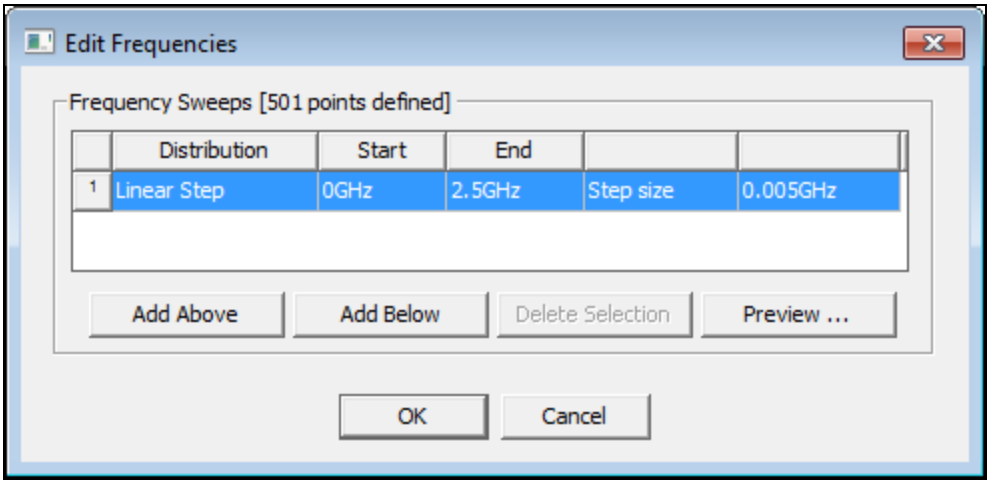
     If **Display All Freqs** is enabled, the drop-down frequency select list is disabled.
   - To show the matrix entries for a selected solved frequency, ensure that **Display All Freqs** unchecked and use the dropdown list to select the solved frequency for which you want to view matrix entries. You can use a scroll bar for selecting from long frequency lists.

     For adaptive passes, only the solution frequency specified in the **Solution Setup** dialog box is available. For frequency sweeps, the entire frequency range is available.
   - To insert or delete one or more displayed frequencies, click **Edit Frequencies**.

> **Note:**
>
> This command is only available if the sweep type is Fast or Interpolating.

Clicking **Edit Freqs** displays the **Edit Frequencies** dialog box. The current values are displayed in a table.

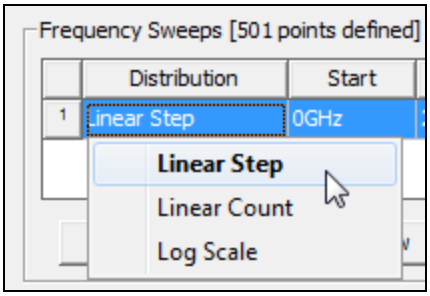
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

Changes to the Start value and End value cannot be outside of the initial range. No message is issued: rather the range is implicitly restricted.

Use the **Add Above** or **Add Below** button to add a new frequency to the table above or below the currently selected value. If no value, or the start value is selected, the new frequency repeats the current Start value and increments the count at the start a new row. You can select the type of distribution for a new Row by selecting from the drop down.

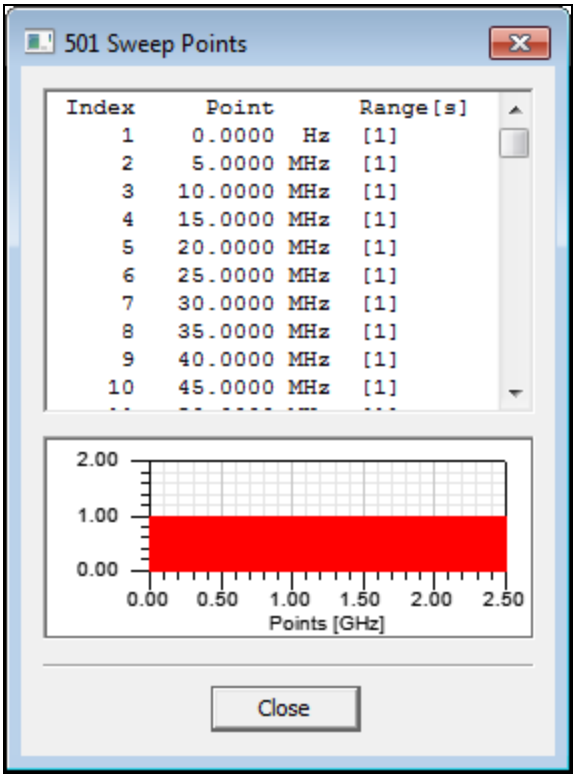


You can then edit the **Start** and **Stop** fields as appropriate for the new row.

The **Delete Selection** button enabled only if a value is selected. **Delete Selection** removes the selected value and decrements the Number of Values field.

The **Preview** button opens a dialog showing the defined sweep points.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



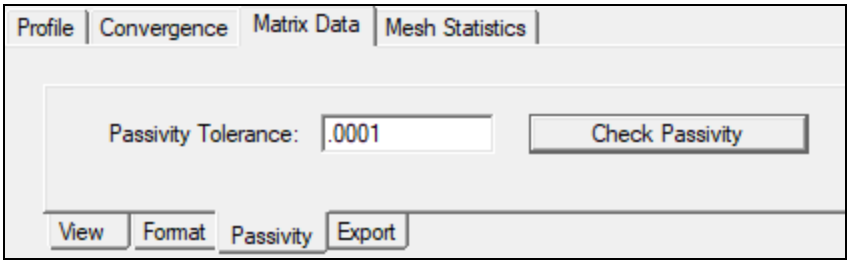
7. Select the display format — On the **Format** tab, select from the drop down menu which has the options **Magnitude/ Phase (deg)**, **Real/ Imaginary**, **dB/Phase (deg)**, **Magnitude**, **Phase (deg)**, **Real**, **Imaginary**, and **dB** — in which to display the matrix information and then in the next drop down menu, select **Terminal Data** or **Differential Pairs**.
8. Click **OK** to apply the changes to the **Solutions** dialog **Matrix Data** tab and close the **Edit Sweep** dialog box, or **Cancel** to close the dialog without applying the changes.

   If you choose to export the matrix data for the Fast or Interpolating sweep after modifying the frequencies in the **Edit Frequencies** dialog box, only those frequencies displayed under the **Matrix Data** tab will be exported.

   The data is displayed in the table. By default, wave ports are listed in alphabetical, then numerical order, just as they appear in the excitation tree. To change the port order, change setting for Default Matrix sort order in the HFSS General options. You may also want see how you can Reorder Matrix Data.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Optionally, select the **Passivity** tab and **Check Passivity**.



This passivity check tests whether the S-parameter data from HFSS is passive or not. If the S-Matrix is not passive at one or more frequencies, this check displays a dialog saying that the solution is passive for all frequencies or that identifies the worst frequency violation and identifies the passivity in that case. A uniform renormalization of 50 ohms is performed on the solution data for Passivity checking.

Selecting the Matrix Display Format

The **Solutions** dialog offers a range of formats for the Matrix display. The available formats depend on the matrix type being displayed. When selected, dB formatting only applies to S - matrix data, even if other matrix types are displayed. The column heads in the display identify the format for the matrix type. You can display matrix data in the following formats.

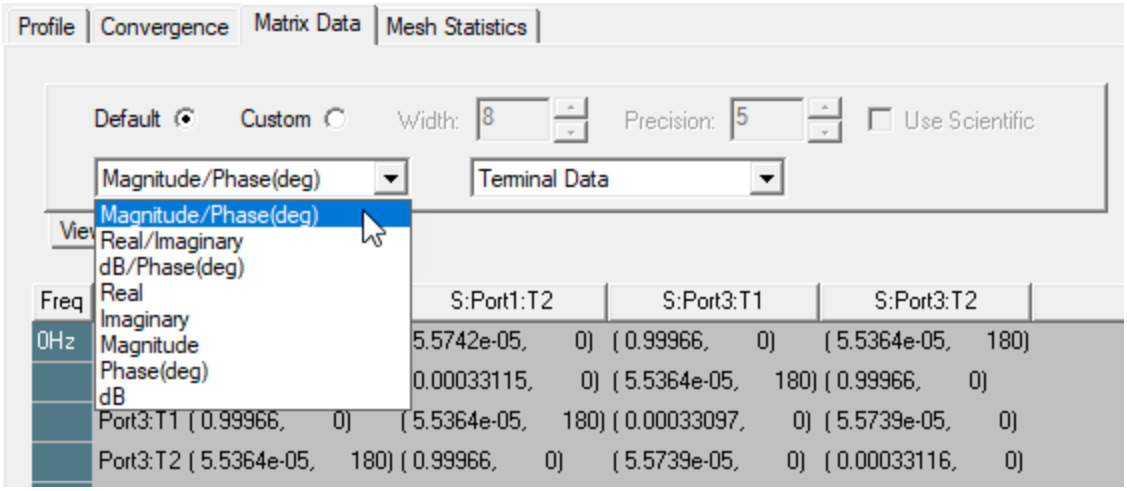
Magnitude, Phase (deg)	Displays the magnitude and phase (in degrees) of the matrix type.
Real, Imaginary	Displays the real and imaginary parts of the matrix type.
dB, Phase (deg)	Displays the magnitude in decibels and phase in degrees of the matrix type.
Magnitude	Displays the magnitude of the matrix type.
Phase (deg)	Displays the phase in degrees of the matrix type.
Real	Displays the real parts of the matrix type.
Imaginary	Displays the imaginary parts of the matrix type.
dB	Displays the magnitude in decibels of the matrix type.

After you select a format, choose whether to display terminal data or differential pairs.

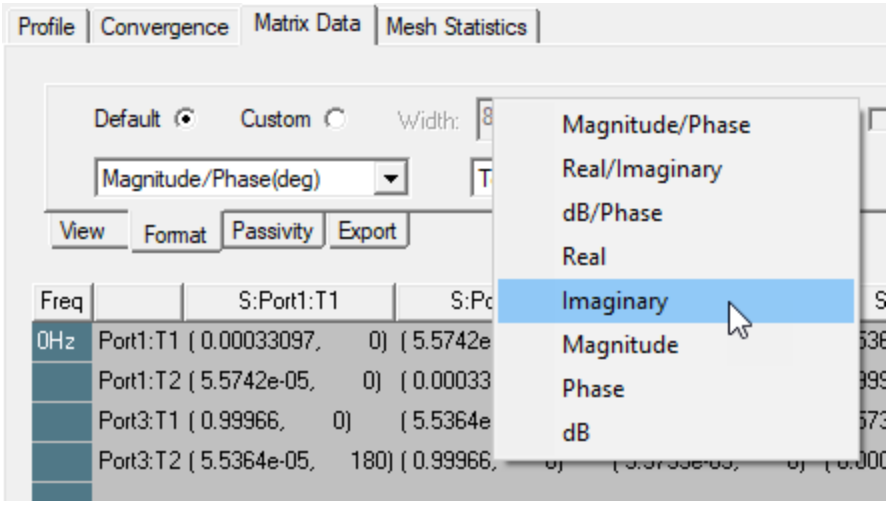
In the **Matrix Data** tab of the **Solutions** window, open the **Format** tab to select the display format in one of two ways:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select the drop-down menus under the formats:



- right-click the column headings to display a pop-up format menu.



- Selecting from this menu also adds the name of the selected format to the column head, if the format is anything other than dB.

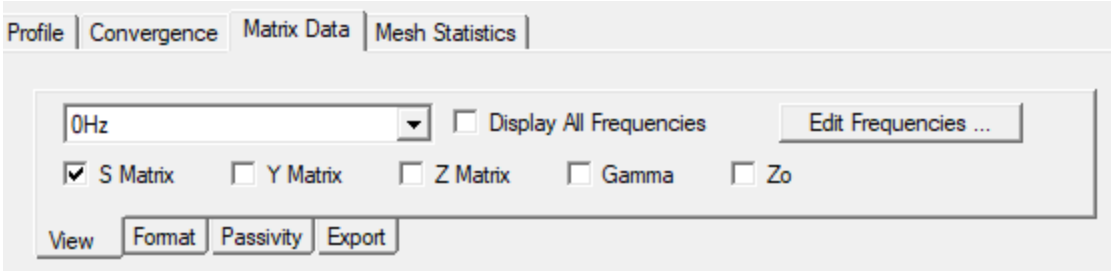
Freq		S:pad2_T1 (Imag)	S:pad3_T1 (Imag)	S:pin2_T1 (Imag)	S:pin3_T1 (Imag)
5 (GHz)	pad2_T1	-0.10414	0.056068	-0.75051	-0.1483
	pad3_T1	0.056068	-0.104	-0.14825	-0.75036
	pin2_T1	-0.75051	-0.14825	-0.16368	0.064232
	pin3_T1	-0.1483	-0.75036	0.064232	-0.1635

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

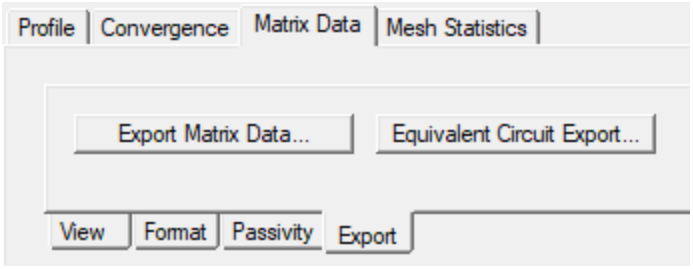
Exporting Matrix Data

1. In the project tree, right-click the solution setup of interest, and then click **Matrix Data** on the shortcut menu.

The **Solution Data** window appears. The **Matrix Data** tab is selected. You can also access the **Solution Data** window from the **Results** tab on the ribbon by clicking the **Solution Data** icon.

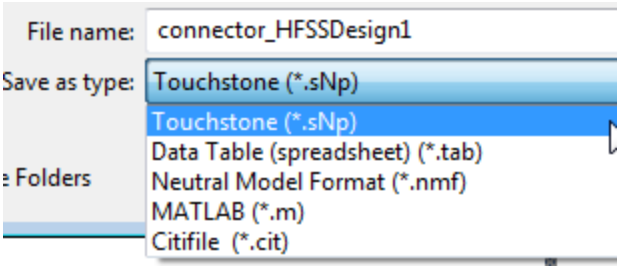


2. Select the type of matrix you want to view: **S-matrix**, **Y-matrix**, **Z-matrix**, **Gamma**, or **Zo** (characteristic impedance.)
3. Select the **Export** tab and click **Export Matrix Data**.



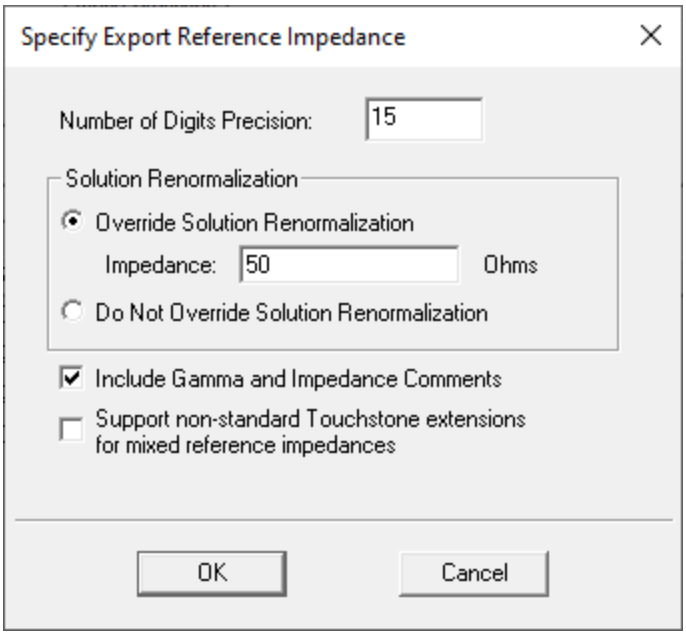
A file browser appears.

4. Type the name of the file you are exporting to in the **File name** text box.
5. Select one of the following file formats from the **Save as type** drop-down menu:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Format	Type	Description
(spreadsheet) *.tab	Data Table	A text file in which the elements of the S-matrix are arranged in a series of columns that are tab-separated and include a first row of headings. The file may be imported into a spreadsheet or similar utility.
***.sNp**	Touchstone	A Touchstone S-parameter file in which the number of ports is indicated by *n*. For example, a Touchstone file with one port would have the file extension .s1p. When you export this format, you are presented with a dialog.



You can specify:

- **Number of Digits Precision** (Default 15)
- **Override Solution Renormalization**. This refers to overriding the renormalization impedances that may have been asked for in the port setup. If so, the export renormalizes impedance
- **Do Not Override Solution Renormalization.** This leaves any port setup renormalization options in place, and grays out the Impedance setting on the dialog.
- **Include Gamma and Impedance Comments**
- **Support for non-standard Touchstone extensions for mixed reference impedances**. Writing gamma and Z_0 is not Touchstone standard, but rather an HFSS "feature" that only HFSS will

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		recognize.
		If you want to export raw S-Parameter data for later use, you may choose to not renormalize the solution.
		If all ports and associated modes/terminals are normalized to the same impedance and you choose **Do Not Override Solution Renormalization** during export, the Touchstone file header will indicate the normalized impedance.
		The comment header in the Touchstone file lists the port and mode numbers to show which column contains which port name (in case of confusion between alphabetical and force repriority ordering of ports and associated modes).
		`! Touchstone file exported from HFSS 2025.1`
		`! File: D:/Program Files/AnsysEM/v242/Examples/HFSS/Signal Integrity/package.aedt`
		`! Generated: 1:56:02 PM Apr 01, 2020`
		`! Design: HFSSDesign1`
		`! Project: package`
		`! Setup: Setup1`
		`! Solution: Sweep1`
		`!`
		`# GHZ S MA R 50.000000`
		`! Terminal data exported`
		`! Port[1] = P01__NET179__T1`
		`! Port[2] = P02__NET178__T1`
		`! Port[3] = P03__NET178__T1`
		`! Port[4] = P04__NET179__T1`

***.nmf**	Neutral Model format	Neutral Model file format defined by the MAFET Consortium. When you save to this format, if the design includes variables, you are presented with the NMF Parameters dialog that lets you select which variables to select as parameters in the NMF file. Non-selected variables will be give the constant value shown. You are then presented a dialog that lets you specify the number of digits precision for the renormalizing impedance.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***.m**	MATLAB	The Mathworks' MATLAB file in which the elements of the S-, Y-, or Z-matrix are arranged in a series of rows. Matlab only allows creation of complex numbers by specifying real and imaginary parts, not by specifying magnitude and phase.
***.cit**	Citifile	Common Instrumentation Transfer and Interchange file format. It is an ASCII format defined by instrument and CAE designers.

> **Note:**
>
> For Touchstone files, you no longer see a **Combine Sweeps** option on the **Export Network Data** solution dialog box. This is because the current Sweep setup allows you to define multiple sweeps for a single simulation, meaning any needed combining is defined with the sweep setup.

6. Click **Save**.

The data is exported to the file.

- By default, wave ports are listed in alphabetical, then numerical order, just as they appear in the excitation tree. You can change this order to creation order and back without invalidating the solution on the HFSS Options dialog box.
- If you select Touchstone format, you are first presented with a dialog in which you can specify the Number of Digits precision, and Override the Solution Renormalization. If so, you can specify the export renormalizing impedance (an integer value). Here you also can specify and whether to include **Gamma and Impedance Comments**, and **Number of Digits Precision** (Default 15).
- If you select Neutral File Format, you are presented with a **Specify Export Renormalizing Impedance** dialog that lets you specify the Number of digits precision for the save file.

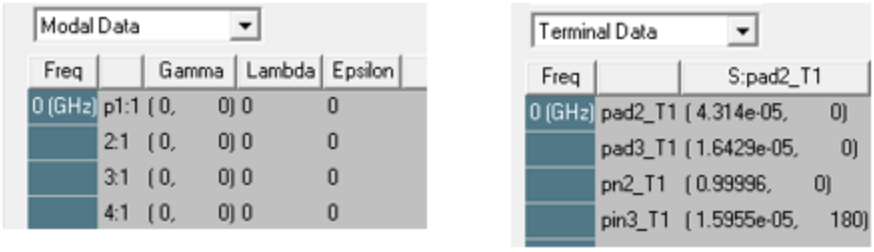
> **Note:**
>
> If you modify the display of solved frequencies in an Interpolating or Fast sweep under the **Matrix Data** tab (by clicking **Edit Freqs** and then modifying the values in the **Edit Sweep** dialog box,) only those frequencies listed will be exported to the file.

Renaming Matrix Data

In the project tree, you can right-click a port excitation to rename it.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you rename a port excitation, the associated Matrix data is reordered so that it can be presented in the same manner. The reordering is done to match the tree-sort order presented for the ports (renamed matrix data is reordered so that alphabetic values appear before numeric values).



Exports of the matrix data are ordered in the same manner. This reordering is conducted as part of post processing and does not force a re-solve.

Reordering Matrix Data

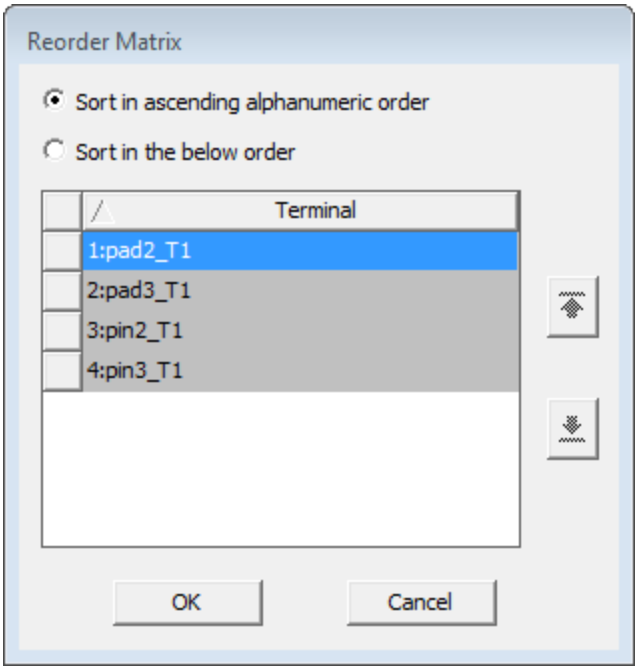
HFSS lets you reorder the matrix data as a post-processing step.

1. To re-order the matrix data, either as the default ascending alphanumeric order, or a user specified order, click **HFSS>Excitations>Reorder Matrix**, or right-click Excitations in the Project tree, and click **Reorder Matrix** on the shortcut menu.

This displays the **Reorder Matrix** dialog. It lists the ports for a modal solution, or the terminals for a terminal solution. For Transient solutions, the dialog displays separate lists for

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

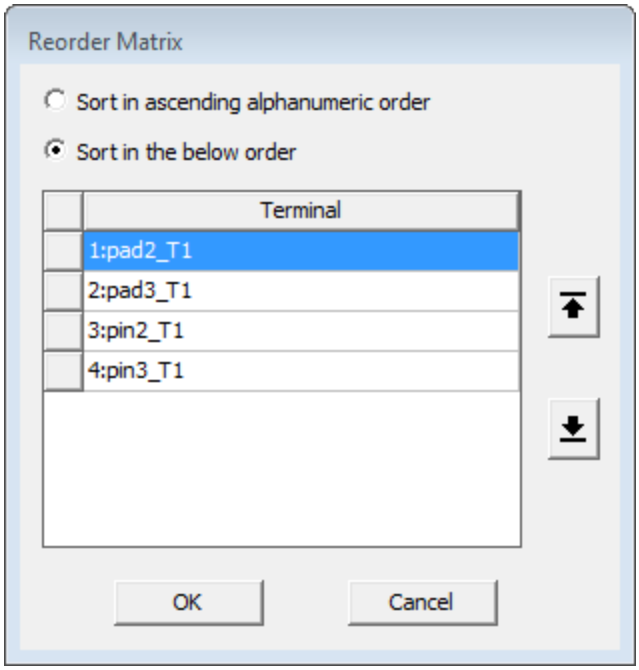
Active and Passive ports.



2. You can select the radio button for **Sort in ascending alphanumeric order**, or **Sort in the below order**.

If you select **Sort in the below order**, you can select any port or terminal to enable the up and down arrow keys. Use the keys to move the ports into any desired order. In the case of Transient, the arrow keys operate only within a partition, that is only within Active or Passive lists.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also use Ctrl+click to select multiple arbitrary ports, or hold Shift to select a range of ports. Clicking outside the list deselects all selections.

Note:

If there are differential pairs, the sort order is still specified) in terms of the underlying terminal names, but the entries that make up the pair should appear in the appropriate sort location for the terminals that are used to define them.

3. Click **OK** to accept the order specified.

For Transient solutions, removing an active source will not affect the solve but might affect the matrix order. Adding an active source will require a resolve if that source has not already been solved.

Exporting Equivalent Circuit Data

You can export S-parameter data from a Driven Terminal solution to PSpice, HSPICE, Spectre or Maxwell Spice format. Importing the new data file to PSpice, HSPICE, Spectre or Maxwell Spice will enable you to include wave effects in the circuit simulations. You can also export a W-Element model for a port.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

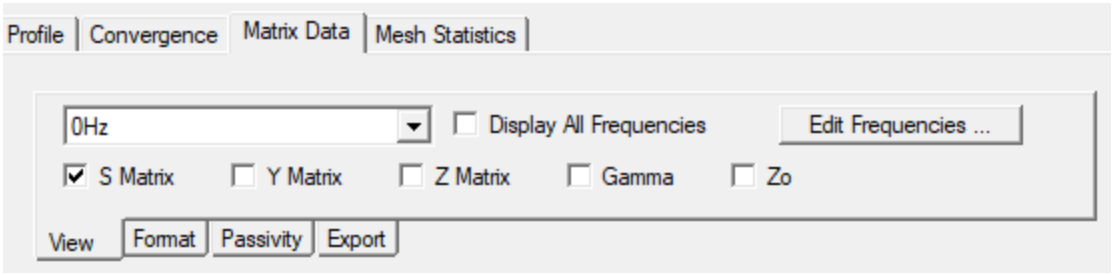
You must have a frequency sweep solution and five or more frequency points to successfully export an equivalent circuit data file. See the *Choosing Frequencies for Full-Wave SPICE* topic of the help for suggestions about the frequency range of the sweep.

The GUI lets you export full-wave Spice for a model that contains differential pairs, but it will silently export the data in its original single-ended form. The full-wave Spice model is a "broadband" equivalent circuit (that is, its S-parameters match those of the solution across the whole frequency sweep range.)

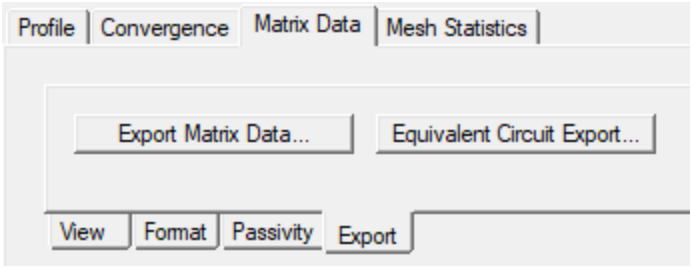
Certain discrete sweeps permit Full-Wave SPICE exports. It is allowed if the discrete data is evenly spaced, includes DC, and has at least 500 frequency points.

1. In the project tree, right-click the solution setup of interest, and then click **Matrix Data** on the shortcut menu.

The **Solution Data** window appears. The **Matrix Data** tab is selected. You can also access the **Solution Data** dialog from the **Results** tab on the ribbon by clicking the **Solution Data** icon.



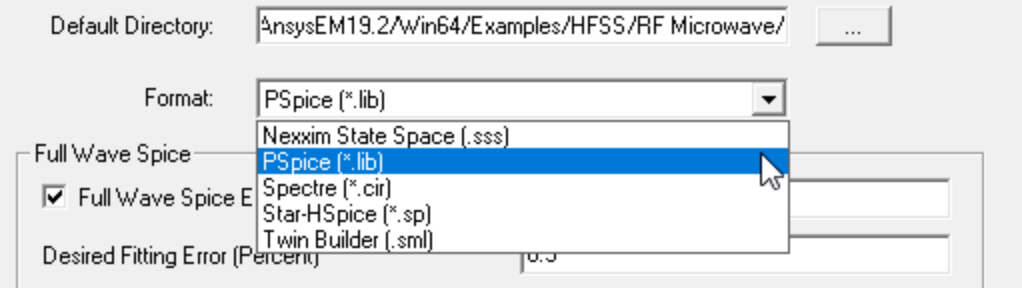
2. Select the **Export** tab and Click **Equivalent Circuit Export**.



The **Equivalent Circuit Export Options** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Type the name or browse to the directory in which you want to store the data.



4. Click one of the following formats in the **Format** list:

   PSpice (*.lib)
   Nexxim State Space (.sss)
   Spectre (*.cir)
   Star HSpice (*.sp)
   Twin Builder (.sml)

Your format selection affects the options available under **Full Wave Spice Export**. When Twin Builder format is requested, both *.sml and *.png are created. The latter contains the image in GIF format.

> **Note:**
>
> The Export to Twin Builder here does not use the same settings as Nexxim or Network Data Explorer. If you intend to export to Twin Builder, you should use the NdExplorer.

5. If the **Full-Wave Spice Export** check box is enabled, you can select it. Checking the box enables the text field for the file name, and depending on the format selection, other options may be enabled.
6. Desired Fitting Error (percent) has a default value of 0.5. You can edit this.
7. Maximum Order has a default value of 10000.
8. HFSS supports Full Wave Spice Export from a driven modal design as long as all ports have exactly one mode each. However, HFSS does not support definition of differential pairs in a driven modal design.
9. By default **Use Common Ground** is checked and produces circuit models with a "common" (suppressed) ground terminal. Uncheck Use Command Ground to apply one negative reference terminal per port.
10. Optionally, select **Enforce Passivity**. Selecting this enforces passivity in the output file. Passive devices can only dissipate or temporarily store energy, but never generate it.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(You can also check passivity from the **Matrix Data** tab using the Check Passivity button.)

This option is useful in cases where the transient simulation fails due to passivity violations in the circuit model. This circuit model is based on fitting a rational function to the S-parameter data computed by the field solver. Small errors in the data fitting can result in non-passive behavior. Selecting the **Enforce Passivity** option will take more CPU time, but ensures that the resulting model will be passive. The **Enforce Passivity** check box uses "Iterated fitting of passivity violations (IFPV)" method to do the passivity enforcement.

The passivity check tests whether the S-parameter data from HFSS is passive or not. For more information see Passivity.

11. Optionally, select **Lumped Element Export (Low Bandwidth)** if you want to save the data as a low-frequency circuit model using simple lumped elements (resistors, capacitors, inductors, and dependent current sources). The low-bandwidth model is only going to be accurate in a limited frequency range around the adaptive solution frequency

This option is not enabled for Spectre export.

12. Optionally, select **Partial Fraction Expansion for Matlab** if you want to specify a file that expands the partial fractions for use in Matlab. The partial fractions involved describe the frequency response of the low-bandwidth model from the previous step.
13. You can also select **Combine Sweeps** to select and combine available sweeps into a single output file.

By option, for Nexxim State Space, Spectre, and Star-HSpice, you can Export W-Element Data.

14. Click **OK**.

The S-matrices are written to the data file that you specified in the equivalent circuit data format.

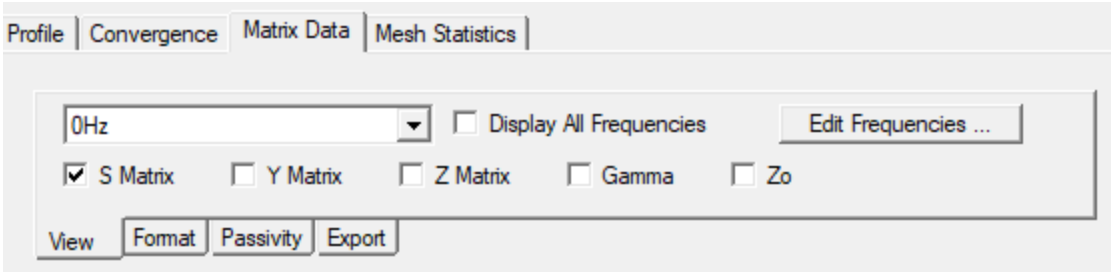
Exporting W-element Data

It is possible to extract a W-element model for a port. This W-element model can be used in a SPICE model to represent a length of transmission line of the same cross-section as the port. A W-element model can be extracted for a port only solution and for a full 3D solution.

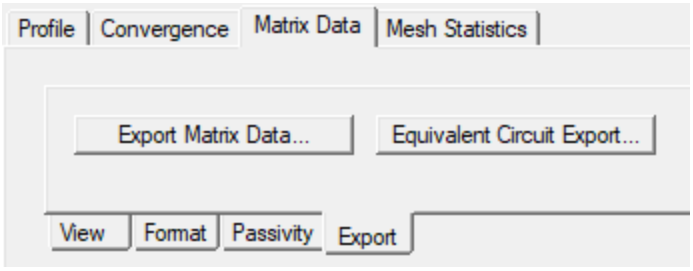
1. In the project tree, right-click the solution setup of interest, and then click **Matrix Data** on the shortcut menu.

The **Solution Data** window appears. The **Matrix Data** tab is selected. You can also access the **Solution Data** dialog from the **Results** tab on the ribbon by clicking the **Solution Data** icon.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

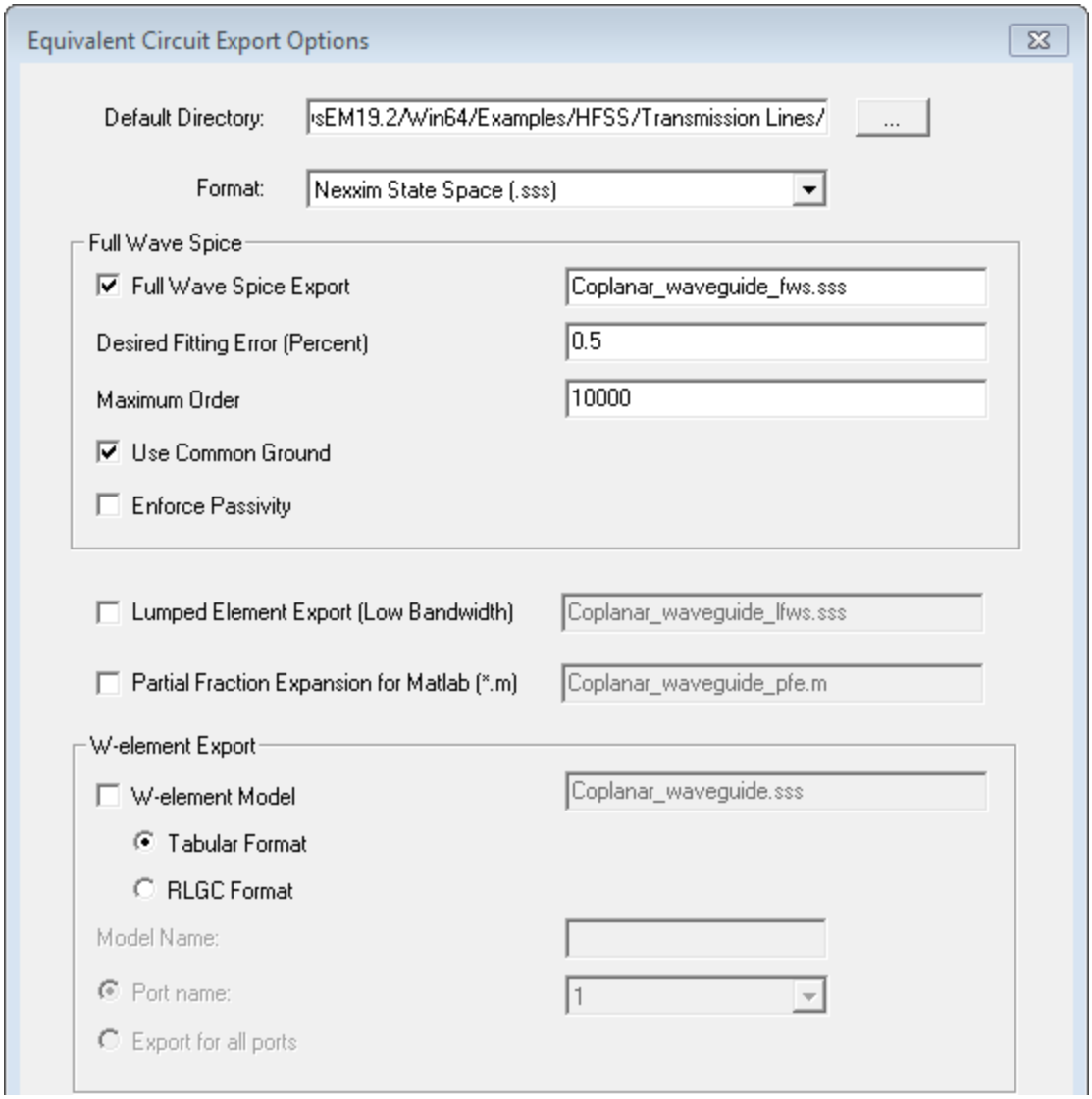


2. Select the **Export** tab and click **Equivalent Circuit Export**.

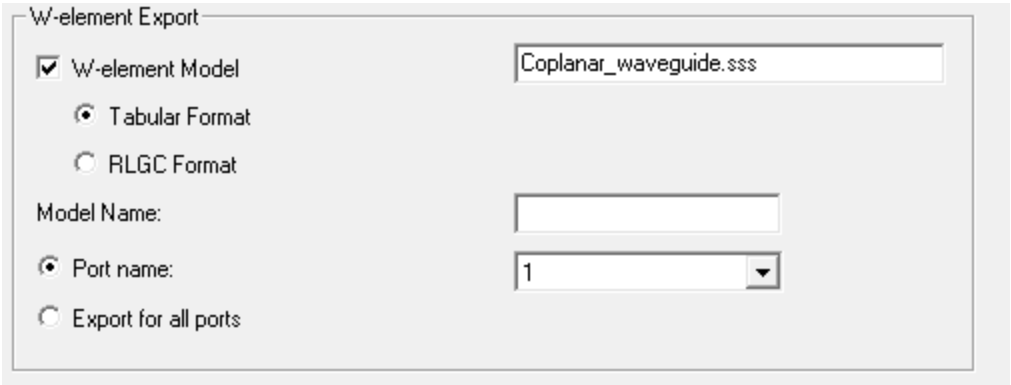


The **Equivalent Circuit Export Options** dialog box appears. If you select the format as Nexxim State Space, Spectre, or Star-HSpice, at the bottom of the dialog you see the W-element model check box enabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click the W-element model check box to enable the W-element fields.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. The W-element model name field has the project name by default. You can change this if desired.
5. Choose the format as Tabular Format (the default) or RGLC format for W-element export.

Tabular Format: provides a unique RLGC model for each frequency in the solution.

RLGC Format: provides a RLGC fit over a frequency range based on Ro, Lo, Go, Co, Rs and Gd parameters.

> **Note:**
>
> For the RLGC Format, if only a single frequency solution is selected (e.g., LastAdaptive) then Rs and Gd parameters are ignored.

6. In the **Model name** field, provide a model name.
7. Either select the port from the **Port Name** pull down, or, to export a W-element model for all ports, select the **Export for All Ports** radio button.
8. Click **OK**.

The W-element model is written to the data file that you specified.

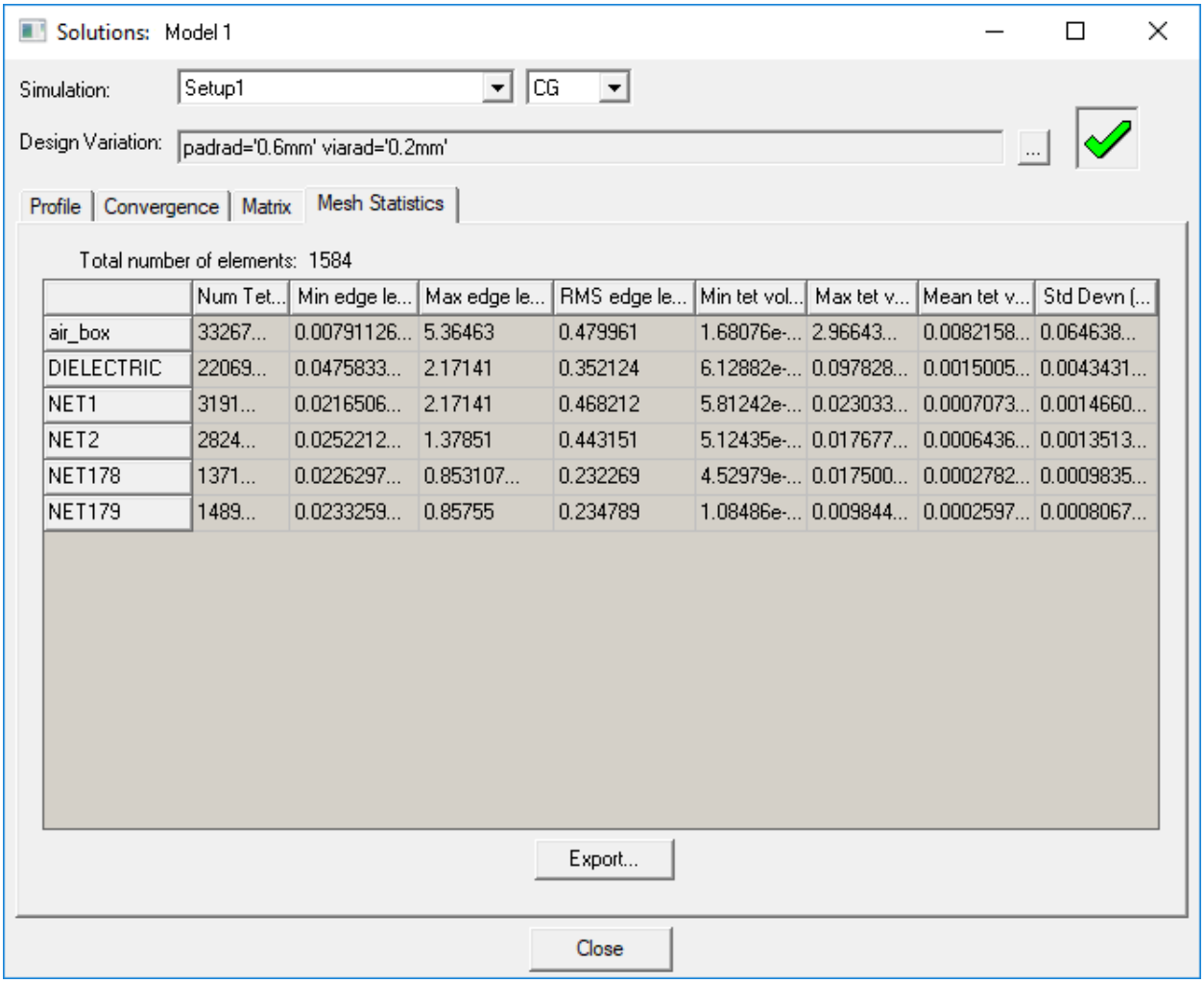
Viewing Mesh Statistics

To view an adaptive solution's mesh information, either during or after the solution process:

1. In the Project Tree, right-click the solution setup of interest, and then click **Mesh Statistics**. Alternately, click the **Solution Data** icon on **Results** tab of the ribbon.

   The **Solutions** window appears with the **Mesh Statistics** tab displayed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The table lists design elements and for each includes: Num Tets, Min edge length, Max edge length, RMS edge length, min tet vol., max tet vol., mean tet vol. and standard deviation (min elem area, max elem area, mean elem area and standard deviation for 2D designs).

For projects with HFSS-IE regions, the Mesh data table is similar but includes min, max, and mean element area information for triangles, instead of tet volume information for solve-inside portions of the project.

If mesh repairs have been performed, two additional columns appear in the table: **Recovered %** and **Repaired %**. These columns indicate the fraction of an object that was successfully recovered, and the fraction that needed some repair.

To sort mesh statistics by column from low to high values or vice-versa, click a column header. This displays a shadowed triangle pointing down to indicate a list ordered from highest to lowest; and a triangle pointing up to indicate a list ordered from lowest to highest. Clicking again inverts the order.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

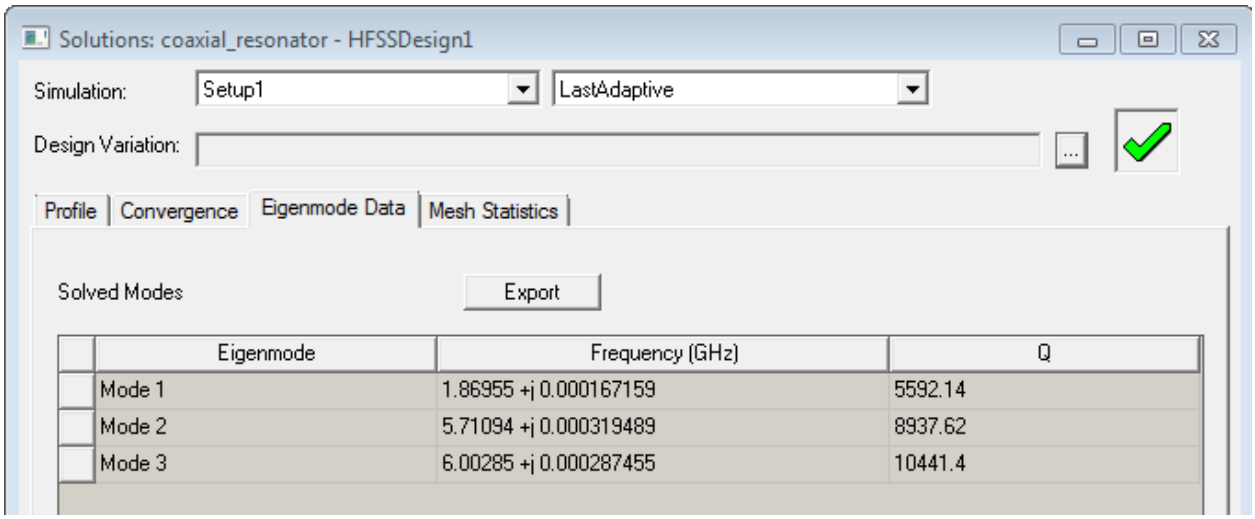
Click on a blank cell above the object list to invert the order of objects. In this case, the cell does not display a directional triangle.

Viewing Eigenmode Solution Data

To view the real and imaginary parts of the frequency and quality factor Q computed for each eigenmode:

1. In the project tree, right-click the solution setup of interest, and then click **Eigenmode Data** on the shortcut menu.

The **Solution Data** window appears. The **Eigenmode Data** tab is selected. You can also select the **Results** tab on the ribbon, and use the **Solution Data** icon.



The solved eigenmodes are listed in the table.

The **Frequency** column lists the real and (for lossy materials) imaginary parts of the frequency (or resonant frequency) for each solved eigenmode.The display uses the *re* + j *im* format.

For lossy Eigenmode solutions, a **Q** column appears, which lists the unloaded quality factor Q computed for each eigenmode.

2. In the **Simulation** drop-down menu, select the solution setup and solved pass - adaptive or single frequency solution - for which you want to view data.
3. To export the Eigenmode solutions to a text file with an eig extension, click the **Export** button.

This displays **Save As** dialog box. You can provide a file name, and if desired, change to a non-default location. click **Save** to save the text file and close the Save As dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

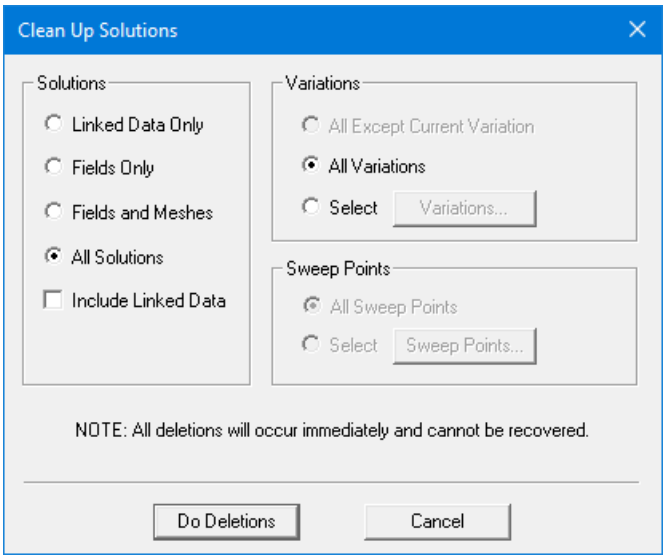
Deleting Solution Data

You can use **Clean Up Solutions** to selectively make deletions, or remove all solutions from the results.

To use **Clean Up Solutions**:

1. Click **HFSS** > **Results** > **Clean Up Solutions**.

   The **Clean Up Solutions** dialog box appears.



2. Under **Solutions**, select whether you want to delete only fields data, only fields and mesh data, only linked data, or all solution data. Deleting all solution data erases all mesh, matrix, and fields data for all adaptive passes and frequency sweeps for the selected **Variations**.

   By option, you can **Include Linked Data** in the deletions.

   Linked data can be mesh, field or some other post-processing data that the source design generated. The target design for the link caches these data internally to minimize the need to activate the source design. If you don't need to resolve the source design, for example, if it has a lengthy solution time, you should not check **Include Linked Data**. If you need to make some change in the source design that would affect the solution, and you want to ensure that the next solve result includes those changes, you should check Include **Linked Data**.

3. Under **Variations**, select which solution data you want to delete:
   - Select **All Except Current Variation** to delete all solution data that does not correspond to the current project and design variable values for the current design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select **All Variations** to delete all solution data for the current design.
- Select **Select** to specify the variations you wish to delete. Click **Variations** to select the variations for deletion.

4. Click **Do Deletions**.

   The solution data you selected are deleted. Any post processing reports or field overlays you created that included data you deleted will be marked with an X in the project tree. They will be invalid until new solution data are generated.

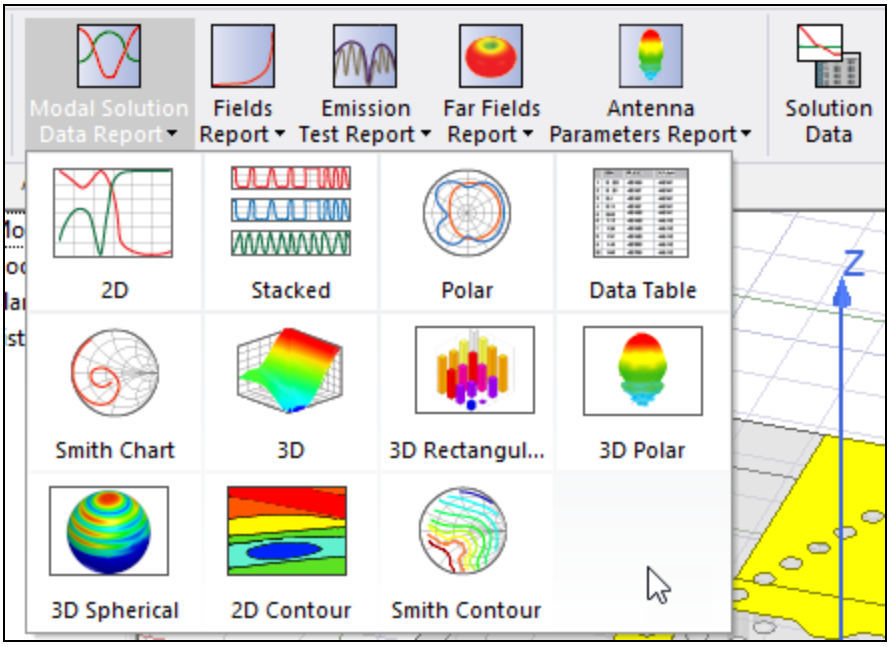
Creating Reports

After a solver generates a solution, you can analyze all the results for that solution. Ansys Electronics Desktop lets you create 2D or 3D plots. A 2D or 3D plot shows the relationship between a design's values and the corresponding results of the analysis. You can create reports using either the **Create Quick Report** option or the **Create *<type>* Report** commands. The **Quick Report** feature lets you select from a list of predefined categories (such as S-parameters) from which to create a rectangular plot.

For each solution **<type>** (for instance, Modal, Terminal, Transient, Eigenmode, Characteristic mode, Fields, Far Fields, Emission test, Partial Discharge, Multipaction, and so forth), the **Results** menus present a list of **Create *<type>* Report** commands based on the solution data of direct interest for the design. For example, for the Eigenmode solution type, the **Results** menu contains templates for Eigenmode Parameters and for Fields. These appear on the menus as **Create Eigenmode Parameters Report** and **Create Fields Report**. For the Modal and Terminal Solution types, several different types appear, appropriate to each solution type. Each of these **Create *<type>* Report** menu items includes a further cascading menu that lists the Display Types available for that report. For some reports you can modify the Display Type from the Properties for that Report. For reports for Transient designs, see this discussion.

The **Results** tab for the Ribbon will show icons and drop-down menus for available report types for the active project.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you have created custom report templates (for example, including your company name or other format changes), you can also create a report based on that template by selecting **HFSS** > **Results** > **Report Templates** > <***templateName***>. You can also access previously defined 2D templates using **Report2D** > **Report Templates** > **Apply Settings...**. You can save the properties for a modified report to provide the custom default settings for all new reports.

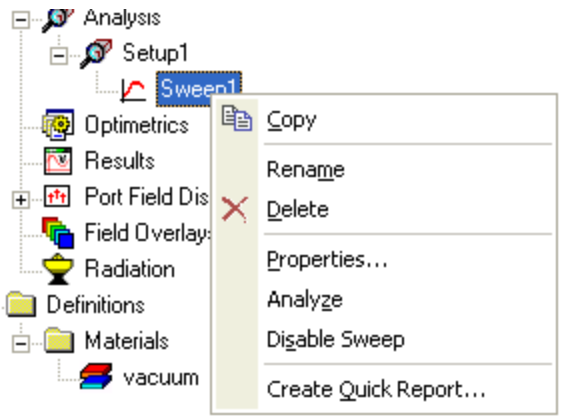
You can also use the **Report2D** > **Export** or **Report 3D** >**Export** feature, select the *Ansoft ReportData Files (.rdat)* format, and **Save** the file, which you can later select to Create Report from File.

Creating a Quick Report

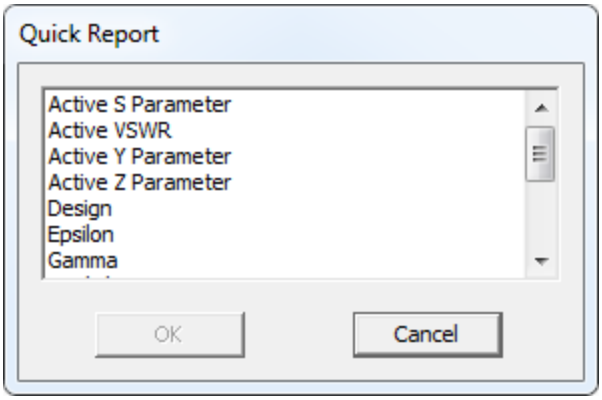
Following is the procedure for creating a quick report.

1. On the Project tree under Analysis, select a **setup** or **sweep icon**, or the **Results** icon.
2. Right-click to display the shortcut menu and select **Create Quick Report**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Quick Report** dialog appears.



The list of available reports differs depending on the Solution type. The figure shows reports for a Modal solution. Eigen mode solutions and Terminal solutions provide different selections.

3. Select the one or more categories for the report from the list and click **OK**.

A rectangular plot for each selected category displays. The new plot or plots appear in the Project tree under the Results icon. The default Report Name that appears derives from the report type specified in the **Quick Report** dialog box.

Creating a New Report

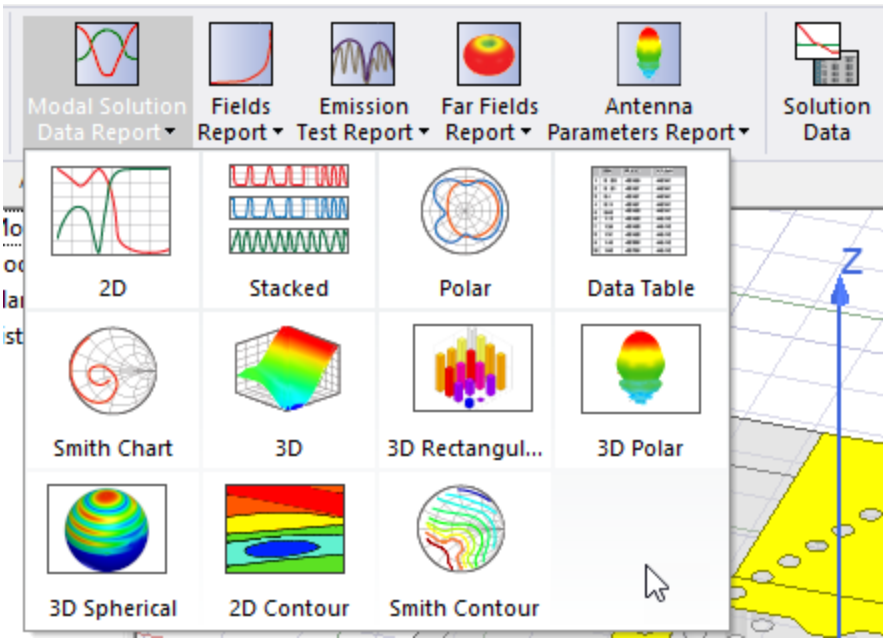
Following is the general procedure for creating a new report:

1. On the **HFSS** menu or the Project Manager, point to **Results**, select **Create *<type>* Report**, and select the **Display Type** template. There are more **Report Type** templates available for terminal solutions (terminal, modal, fields, near fields, and far fields) and for modal solutions (modal and fields). For Eigenmode solutions, the **Report Type *<templates>*** are for Eigenmode Parameters and for Fields. Characteristic Modes

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solution reports include quantities for Significance, Value, and Angle.

The **Results** tab for the Ribbon will show icons and drop-down menus for available report types for the active project.



If you have created custom report templates (for example, including your company name or other format changes), you can also create a report based on that template by selecting **HFSS** > **Results** > **Report Templates** > **PersonalLib** > *<templateName>*. You can also make such changes the default for new reports by right-clicking a modified report and selecting **Report Templates** > **Save Settings as default**.

When you have selected the report and display type from the **Results** menu, the **Report** dialog box appears, with the **Trace** tab selected by default.

2. In the Context section, you make selections depending on the design and solution type.
3. In the **Y** Component section of the dialog box, make selections for the following:
    a. Categories – those depend on the Solution type and the design. For example, Eigenmode quantities include Eigenmodes, variables, output variables, and the design. Driven solutions include such categories as S parameters. Report categories for Transient designs include Spectral and Transient. For a Transient Network design with differential pairs defined, the Reporter interface allows selection of single-ended or differential signals just as for driven terminal. Characteristic Mode Data Reports include Characteristic Mode quantities for Significance, Value, and Angle. Report categories for SBR+ designs that include an Incident Plane wave and an RCS selection as Monostatic do not require a geometry selection and include a range of Monostatic Quantities when select Monostatic RCS

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

as the Report Category. For SBR+ designs and RCS Monostatic you can choose between Freq, IWaveTheta and IWavePhi variables for specifying sweeps. The selected Category provides the default plot name. You can edit the plot names in the project tree and the plot header text in the report synchronizes.

b. Quantities for Y are relative to the selected category.

> **Note:**
>
> The Quantity text field can be used to filter the Quantity list by typing in text, or by using the four predefined selections. This is useful if the Category selected produces a lengthy Quantities list. See Filtering Quantity Selections for the Reporter.
>
> When the matrix is very large, the number of quantities can be correspondingly huge. Therefore, the Quantities field can optionally use a tree structure to divide matrix quantities into groups by their first element name. The initial display shows groups, without initially listing group members. See Report Setup Options.

c. **Function** list to apply to the Y quantities.

d. Value field displays the currently specified Quantity and Function. You can edit this field directly.

> **Note:**
>
> Color shows valid expression.

e. **Range Function** button – opens the **Set Range Function** dialog box. This applies to the currently specified Quantity and Function.

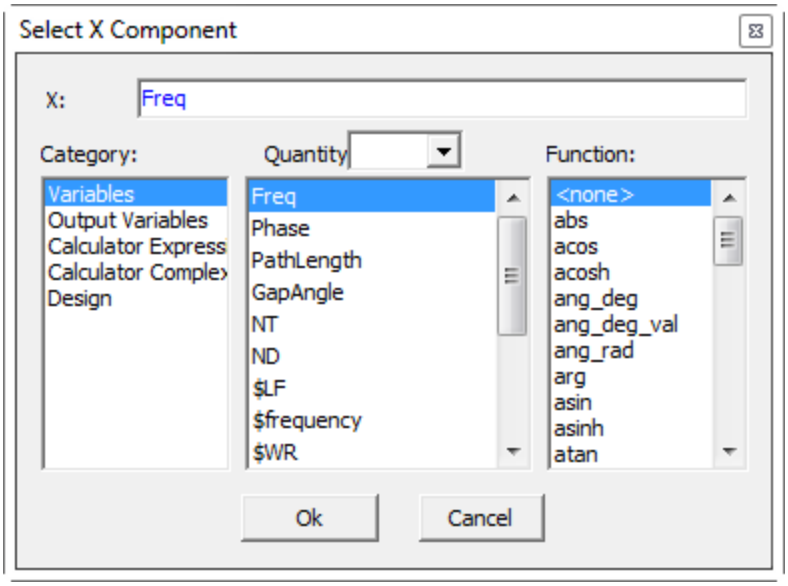
f. To use a dataset in this report, set the **Y** component to a Piecewise Linear Function `pwl(dataset_expression, primary_sweep)` where is the `primary_sweep` is what you set in the next step.

4. In the **X (Primary Sweep)** section, make selections for the following:

a. Select the Primary value(s) from the drop-down menu.

To select an X component that is different from the Primary Sweep, uncheck the Default field to enable the X field and **Browse** [...] button. Click **Browse** [...] to display the **Select X Component** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This lets you specify the X component as you do the Y; that is, in terms of Categories which define the selectable Quantities, and Functions to apply. After making selections, **OK** the dialog box to assign the X component.

b. If sweeps are available, you can select **Browse** [...] to display a panel that lets you select Use all values, or selected sweep or sweeps, or access an Edit Sweep dialog box with further editing options. Post-Processing variables are Post-Processing sweeps/editable sweeps, so you can use the **Edit Sweep** dialog box to create your own sweep.

c. The **Families** tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (for example, for a parametric sweep). If so, the variables other than the one chosen as the **X (Primary sweep)**, are listed under the **Families** tab with columns for the variable, the value, and an Edit column with an ellipsis [**...**] button. See Using Families tab for Reports.

5. **Update Report** setting
   - **Real Time** checked -- enable real time updates for all reports while the reports are being edited.
   - **Real Time** unchecked -- enables drop down menu to **Update All Reports** or **Update Report**. Reports will only be updated with one of these user selectable update options or upon exiting the report dialog box. This can be useful if you expect a trace to take time to display. You can then add additional traces without having to wait.
6. The **Report** dialog command buttons permit you create a new report with the settings you provide, or to modify an existing report.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Output Variables** – opens the **Output Variables** dialog box.
- **Add Trace** – this is enabled when you have created or selected a report. Add one or more traces to include in the report.
- **Update Trace** – updates the selected traces in a report based on further processing or changes.
- **New Report**. Adds a report to the Project tree under the Results icon. The new Report is displayed in the Project window.
- **Options** – opens the **Report Setup Options** dialog box. This contains a check box for using the advanced mode for editing and viewing trace components. This mode is automatic if the trace requires it. It also contains a field for setting the maximum number of significant digits to display for numerical quantities.
- **Close** – closes the **Report** dialog box.

7. Click **New Report** to create a new report in the Project Manager.

   The report appears in the view window. It will be listed in the project tree under Results, with the default name based on the Report Category you selected, for example, S Parameter Plot *n* or Output Variables Plot *n*. You can edit the plot names in the project tree and the plot header text in the report synchronizes. Traces within the report also appear in the project tree.

   Some plots may take time to complete. Performing a **File>Save** in such cases after the plot has been created will permit you to review the plot later without having to repeat the calculation time when you reopen the project later.

8. To speed redraw times for changed plots, perform a **Save**. This saves the data that comprises expressions. For example if re(S_{11})*re(S_{22}) is requested over multiple widths, each of the S_{11} and S_{22} are stored when you save. If you do not do a save of a changed plot, the changed version is not stored.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Remember that the evaluated value of an expression is always interpreted in SI units. However, when an angle quantity is plotted in a report, you have the option to plot values in units other than SI. If you want to plot the polar angle of a complex simulation result, S11 say, you can choose between ang_deg(S_{11}) and ang_rad(S_{11}). Both of these return the exact same angle quantity but in degree and radian units respectively.

Note that when used in expressions, some surprising outcomes might result. For example, the expression "1+ang_deg(S11)" represents an 'angle' and the number "1" is treated as "1 rad". The angle SI unit is attached to any unitless number that is added/subtracted from an angle value. If you want to treat "1" as degrees, make it explicit and use "1deg + ang_deg(S_{11})" instead.

If you are interested in unitless degree values, two additional functions exist: ang_deg_val(S_{11}) and cang_deg_val(S_{11}). These return simple numbers and are treated as such by any expression. If the complex S_{11} lies on the positive Y axis say, ang_deg_val (S_{11}) would be 90 and "1 + ang_deg_val(S_{11})" will be 91.

Context Section for Reports

In the **Context** section, make selections from the following field or fields, depending on the design and solution type.

1. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes. The selections for HFSS Transient include Transient, and for Transient Network includes Spectral.
2. Domain field with a drop down selection list. Whether this field appears, and domains are listed, depends on the Solution type and the report ***<type>*** selected. For modal and terminal solution data reports, the domain can be **Sweep** or **Time**.

   Before you can examine the time domain, you must perform an Interpolating sweep for a driven solution (Modal or Terminal). If you select **Time**, the **TDR Options** button is enabled. Select it and follow the directions for time-domain plotting.

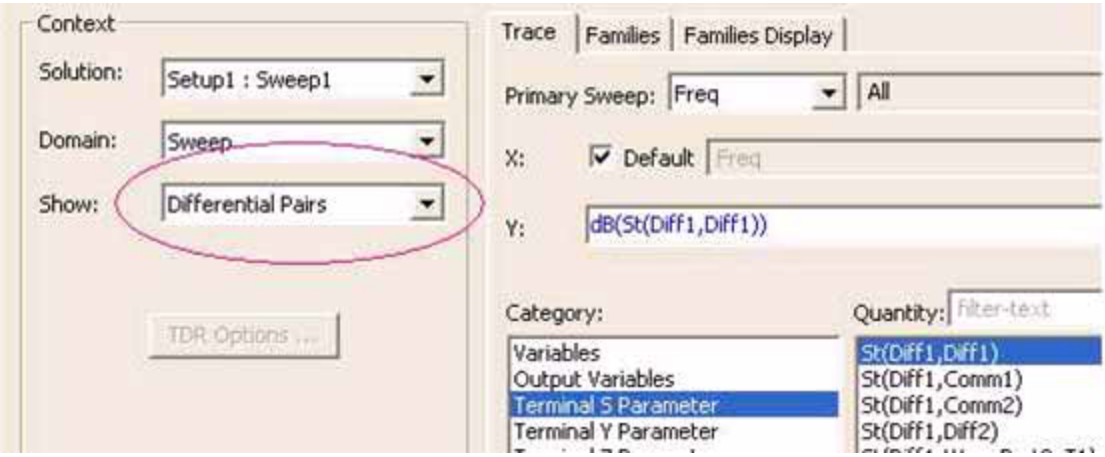
   For Near Field or Far Field report, for a Rectangular Contour Plot, the Domain can be Theta, Phi, or Sine Space. Before you can create a Sine Space plot, you must create the appropriate Radiation Setups. If you manually enter Phi or Theta component of a far field quantity when you using Az/El or El/Az far field infinite sphere definition, then the Report uses the Z axis of the Az/El or El/Az coordinate system definition as pole to calculate theta/phi component for rE.

3. Geometry field with a drop-down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup. For SBR+ designs with an

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Incident Plane Wave and Monostatic RCS selected, the Geometry field is unavailable. In this case, the solver computes the scattered field in the direction of the plane wave.

4. Show field with a drop-down selection list for Differential pairs or Terminals. This field appears for designs using terminal solutions that have differential pairs defined. It lets you plot either differential pair data, or single-ended terminal data, or both in the same plot without having to disable or enable differential pairs under the Excitations heading in the Project Manager. Note that single-ended quantities are computed as if no differential pairs existed. So in the unlikely case of several terminals where only a subset is combined into pairs, the results may not be as expected.



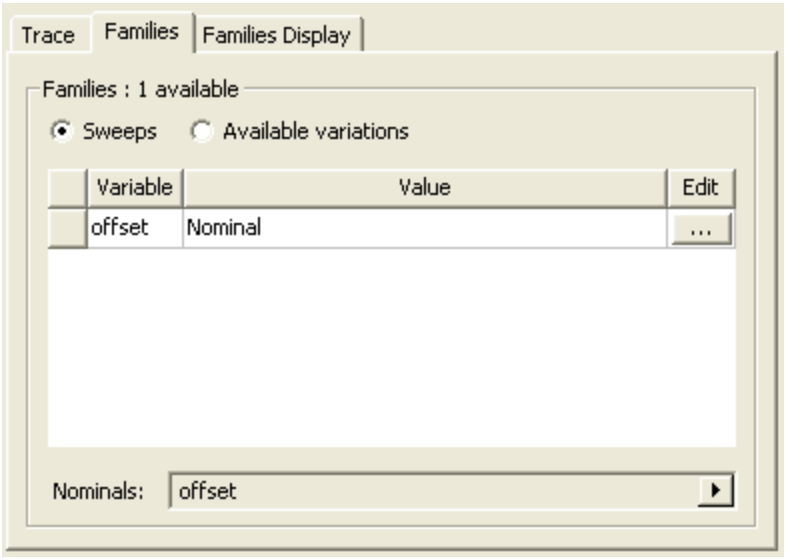
5. Derivative field with a drop-drown selection list of none, all, and specific variables for which you specified **Use** on the Derivatives tab of the solution setup. You can use derivatives in some Optimetrics situations, Far Field reports, non-Port excitations including incident wave, linked field, voltage source, current source, and magnetic bias with the Derivative Tuning feature in the Reporter.
6. A Sources combo box appears in the Reporter when you have specified at least one source in the **Edit Sources** dialog box. See Specifying Source Contexts for Creating Radiated Field Reports. For a usage example, see: *User Defined Solution for MIMO Calculations*.

Using Families Tab for Reports

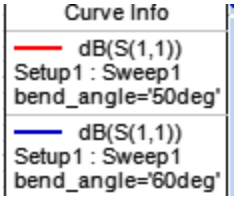
The **Families** tab of the **Report** dialog box provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined and solutions exist for multiple variable values (for example, for a parametric sweep or re-running an analysis with a different variable value). If no variables are defined, or none have solutions for different values, 0 families will be available. If so, the variables other than the **X (Primary sweep)**, are listed under the **Families** tab with columns for the variable, the solution value (which may be All, Nominal, or a Specific value), and an Edit column with an ellipsis [**...**] button. Families gives the number available. If an existing variable is specified as Nominal, only that value is currently available.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can set any solved variables as Nominal, All, or select from values provided for Available solutions.



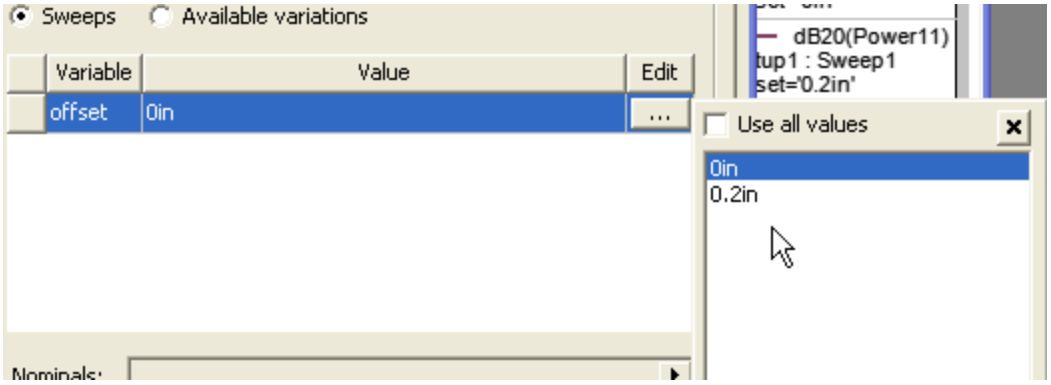
When you select a variable with multiple solved values, a trace for each solved value appears in the Report, with the variable value appended to the trace name in the Report legend.



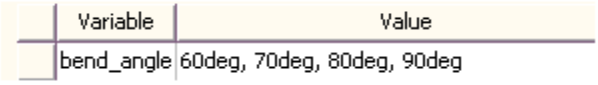
When families are available, you can make selections for the following:

1. Select the **Sweeps** radio button (the default) to list the swept variables you can select or the Available variations button to list and select variation values for which solutions exist.
2. With the **Sweeps** radio button selected, click the ellipsis [**...**] button to display a list of variable values for a particular variable. If many variables exist, you can use a scroll bar to navigate the list.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

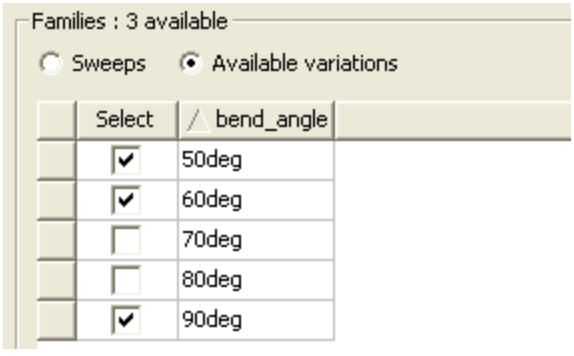


- To select all values, click the check box for **Use all** values. This writes "All" in the value field for that variable. You can also select individual values by clicking on them.
- To select a range of values, hold down the shift key, and click again.
- To select intermittent additional values, hold the Ctrl key and click additional. The values you select are highlighted in the list, and are also listed in the Values column for that variable.
- To select all, use the **Select All** button. This highlights the complete list, as well as listing all values for the variable in the Value field.



- To clear the selections, use the **Clear All** button.

Select the **Available variations** radio button to list the choices that derive from variable combination.

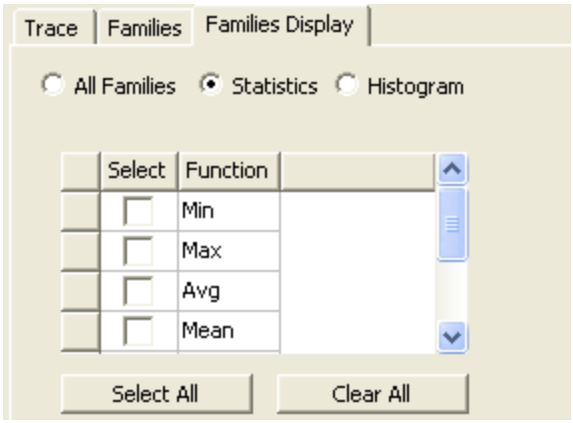


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

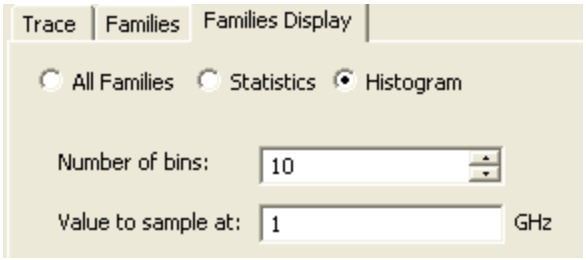
a. To select individual variations, check the select box.
b. To check or clear all variations at once, click the **Select** button at the top of the column.
c. To invert the list order, click the triangle beside the variable name.

The **Families Display** tab has three radio button selections.

a. **All Families**
b. **Statistics** lists a table statistical functions that you can select to apply to the plot. The functions include Min, Max, Avg, Mean, Variance, Std Dev, and Sum. You can use the Select check boxes or the Select All and Clear All button.



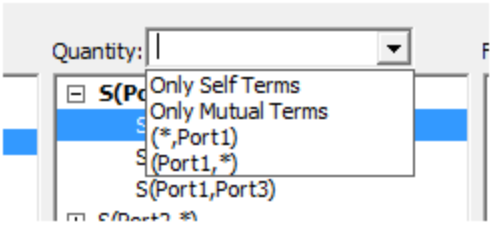
c. **Histogram** lets you select the number of bins to use for a histogram plot, and the sampling frequency to use.



Filtering Quantity Selections for the Reporter

When a two port quantity Category is selected, four predefined filters are added to the combo box. "Port1" is the first matrix element name found in the quantity list.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Only Self Terms – Only display quantities when the first and second port are same.
- Only Mutual Terms – Only display quantities when the first and second port are different.
- (*,Port1) – Only display quantities when the second element name is "Port1". You can edit the element name to display quantities for other elements.
- (Port1,*) – Only display quantities when the first element name is "Port1". You can edit the element name to display quantities for other element.

Deleting Reports

To use **Delete All Reports**:

1. Click **HFSS**> **Results**> **Delete All Reports.** You can also right-click on the Results folder in the Project Manager, to display the shortcut menu, and click **Delete All Reports.**

   All items under the Results folder in the Project Manager are removed.

To use **Delete** for a selected report:

1. Right-click a report icon in the Project Manager to display the shortcut menu.
2. Click **Delete** to remove this report.

> **Warning:**
>
> Solution data that has been deleted cannot be recovered!

Modifying Reports

To modify the data that is plotted in a report:

1. In the Project Manager, click the report you want to modify.
2. Right-click **Modify Report**.

   The **Report** dialog box appears.

3. The **Report** dialog box command buttons permit you create a new report with the settings you provide, or to modify an existing report.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Output Variables** – opens the **Output Variables** dialog box.
- **Add Trace** – this is enabled when you have created or selected a report. Add one or more traces to include in the report.
- **Update Trace** – updates the selected traces in a report based on further processing or changes.
- **New Report -** adds a report to the Project tree under the Results icon. The new Report is displayed in the main window.
- **Options** – opens the **Report Setup Options** dialog box. This contains a check box for using the advanced mode for editing and viewing trace components. This mode is automatic if the trace requires it. It also contains a field for setting the maximum number of significant digits to display for numerical quantities.
- **Close** – closes the **Report** dialog box.

The updated report appears in the view window.

4. **Update Report** setting
   - **Real Time** checked – enable real time updates for all reports while the reports are being edited.
   - **Real Time** unchecked – enables drop-down menu to **Update All Reports** or **Update Report**. Reports will only be updated with one of these user selectable update options or upon exiting the report dialog box. This can be useful if you expect a trace to take time to display. You can then add additional traces without having to wait.
5. In the **Context** section you make selections depending on the design and solution type.
6. The Families tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (for example, for a parametric sweep). If so, the variables other than the one chosen as the **X (Primary sweep)**, are listed under the **Families** tab with columns for the variable, the value, and an Edit column with an ellipsis [**...**] button. See Using Families tab for Reports.
7. In the **Y** Component section of the dialog make selections for the following:
   a. Categories – those depend on the Solution type and the design. For example, Eigenmode quantities include Eigenmodes, variables, output variables, and the design. Driven solutions include such categories as S parameters. Report categories for Transient designs include Spectral and Transient. For a Transient Network design with differential pairs defined, the Reporter interface allows selection of single-ended or differential signals just as for driven terminal. Report categories for SBR+ designs that include an Incident Plane wave and an RCS selection as Monostatic do not require a geometry selection and include a range of Monostatic Quantities when select Monostatic RCS as the Report Category. For SBR+ designs and RCS Monostatic you can choose between Freq, IWaveTheta and IWavePhi variables for specifying sweeps. The selected category provides the default name of the plot, for instance S Parameter Plot *n*. You can edit

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the plot names in the project tree and the plot header text in the report synchronizes.

b. Quantities for Y are relative to the selected category.

> **Note:**
>
> The Quantity text field can be used to filter the Quantity list by typing in text, or by using the four predefined selections. This is useful if the Category selected produces a lengthy Quantities list. See Filtering Quantity Selections for the Reporter.
>
> When the matrix is very large, the number of quantities can be correspondingly huge. Therefore, the Quantities field can optionally use a tree structure to divide matrix quantities into groups by their first element name. The initial display shows groups, without initially listing group members. See Report Setup Options.

c. **Function** list to apply to the Y quantities.

d. Value field displays the currently specified Quantity and Function. You can edit this field directly.

> **Note:**
>
> Color shows valid expression.

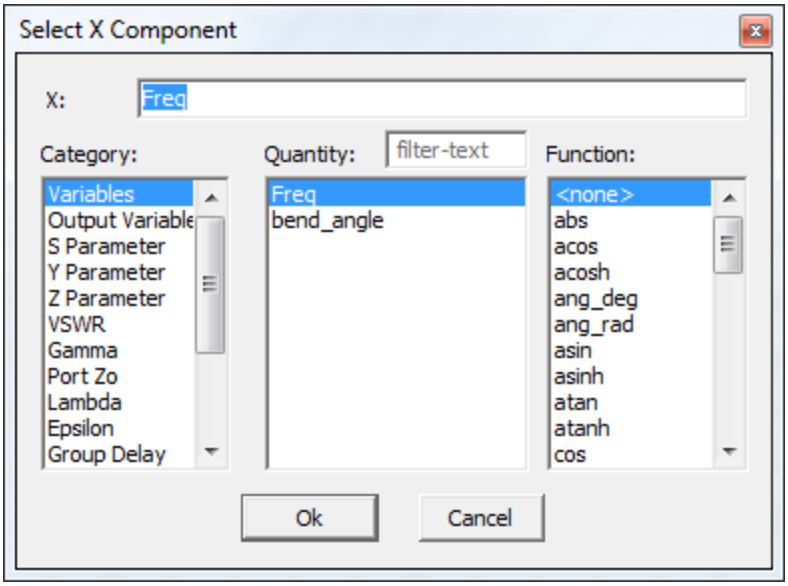
e. **Range Function** button – opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.

8. In the **X (Primary Sweep)** section, make selections for the following:

a. Select the Primary value(s) from the drop-down menu.

To select an X component that is different from the Primary Sweep, uncheck the Default field to enable the X field and **Browse** [...] button. Click **Browse** [...] to display the **Select X Component** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This lets you specify the X component as you do the Y; that is, in terms of Categories which define the selectable Quantities, and Functions to apply. After making selections, **OK** the dialog to assign the X component.

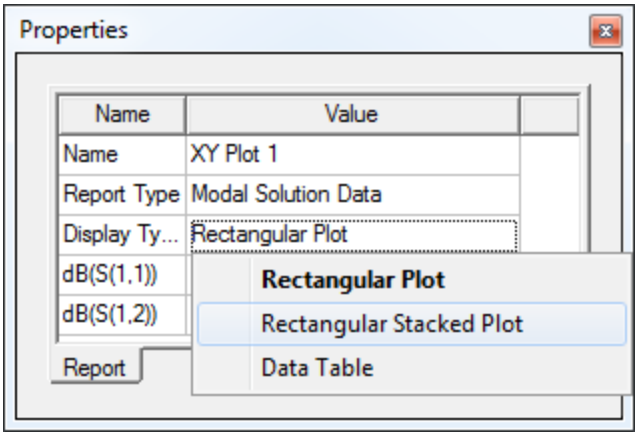
b. If sweeps are available, you can select **Browse** [...] to display a panel that lets you select Use all values, or selected sweep or sweeps, or access an Edit Sweep dialog box with further editing options. Post-Processing variables are Post-Processing sweeps/editable sweeps, so you can use the *Edit Sweep* dialog box to create your own sweep.

c. The **Families** tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (for example, for a parametric sweep). If so, the variables other than the one chosen as the **X (Primary sweep)**, are listed under the **Families** tab with columns for the variable, the value, and an Edit column with an ellipsis [**...**] button. See Using Families tab for Reports.

You can also view and edit the properties of Reports and their traces via their Properties windows. See Modifying the Background Properties of a Report.

You can also modify the display type of an existing plot from the Properties dialog box for that plot. Select the Report icon in the Project Manager to display the **Properties** dialog box. Selecting the Display Type field displays a menu with selections available for that plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

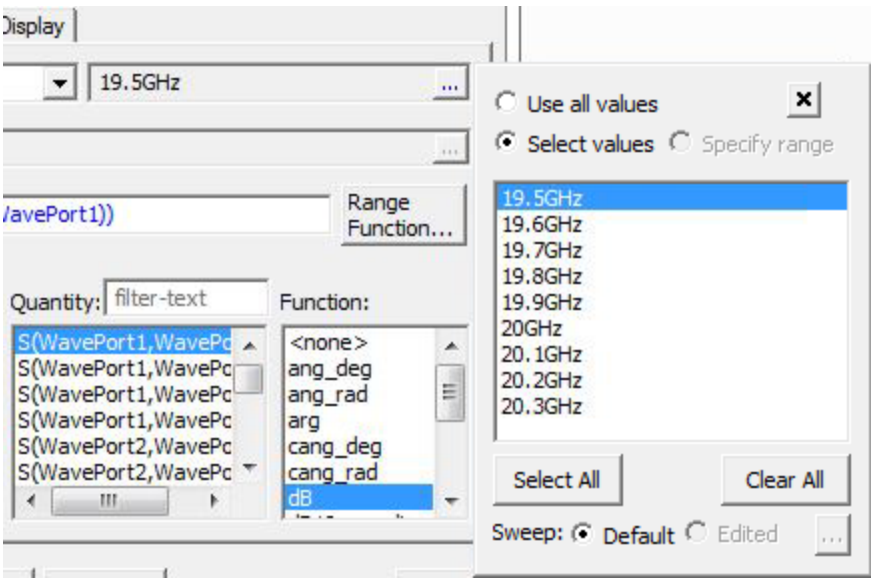


Once you make a selection, the plot display updates for the current selection.

> **Note:**
>
> Remember that for many excitations of interest for plotting, you can control the default base names through the dialog box described here: Setting Default Boundary/Excitation Base Names. This may save you the need to edit individual names in the plots.

Modify Report: Selecting Use All Values or Making Selection

Clicking the browse button on **Primary Sweep** line shows the default selection of **Use all values**. Choose **Select values** to display the sweeps and enable editing, including the **Select All** and **Clear All** buttons.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

With **Select values** selected, you can select one or more by clicking an individual value, dragging to select multiple values, or using Alt+click to select specific values.

You select either the Sweep radio button for Default or Edited selection.

You can also select **Browse** [...] here to display the Edit Sweep dialog for Modify Reports, which includes additional editing features.

Modify Report: Using the Edit Sweep Dialog Box

Clicking **Browse** [...] at the lower right corner of the **Use all values** pane opens the **Edit Sweep** dialog box for Modify Report. This lets you edit the current Primary sweep variable values, including radio button selections for Single value, Linear step or count, and Decade, Octave, or Exponential Counts.

You can specify start, stop and step values and units, and add specific values to the list of current sweep values. The **Add>>** and **Update>>** buttons let you edit the value list.

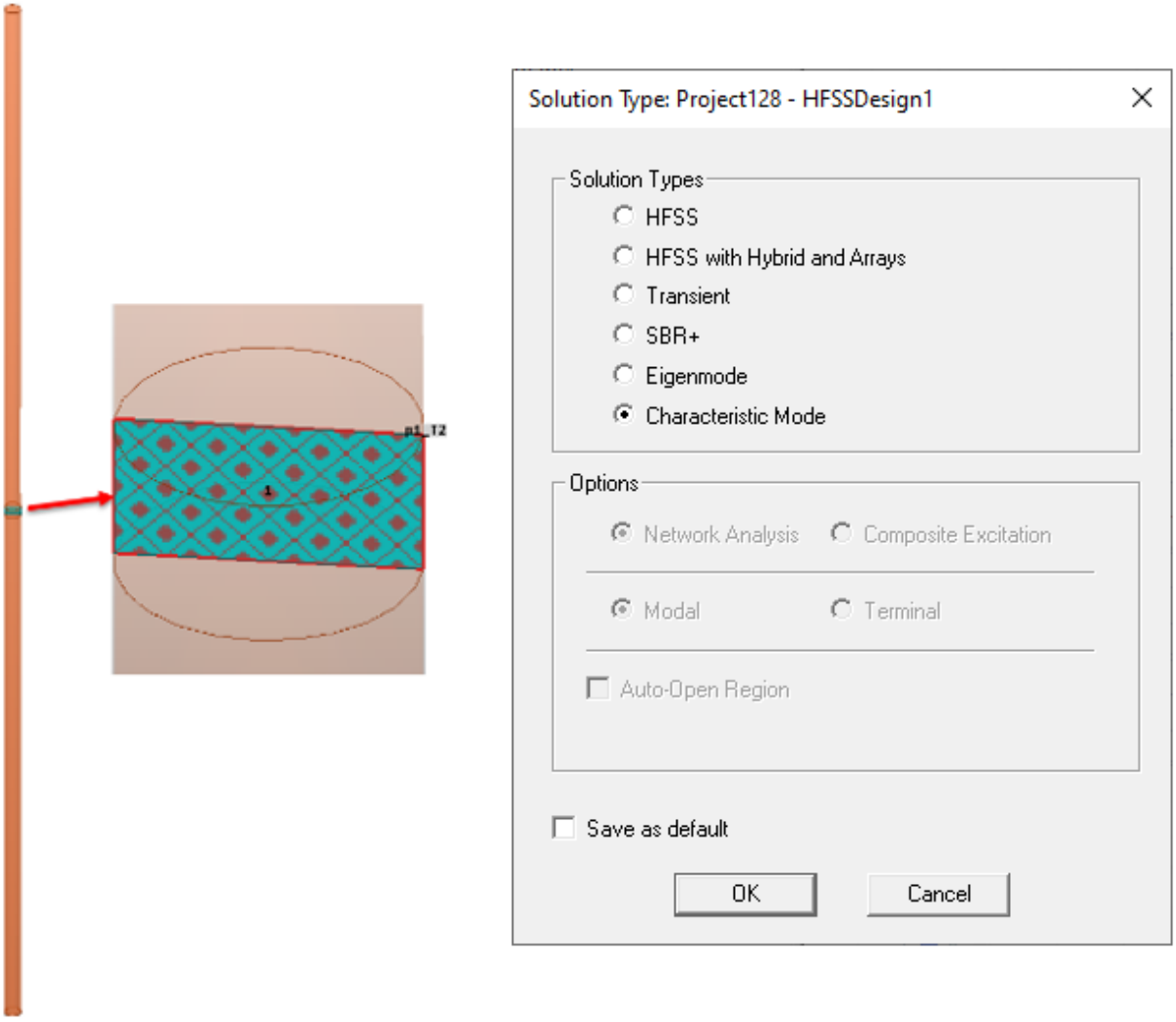
You can use the mouse click, drag, and Alt+click to select values. You can also edit individual values.

Generating Reports For Characteristic Mode Analysis

This section describes how to use the characteristic mode analysis feature in HFSS when designing an antenna and generating characteristic mode analysis reports. The section also

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

covers post processing capabilities available in HFSS when using CMA. To illustrate post processing for characteristic mode analysis, we will use a dipole antenna example. The two conducting arms of the dipole antenna are excited by a lumped port. From the menu item **HFSS > Solution Type**, define **Characteristic Mode** as the solution type.

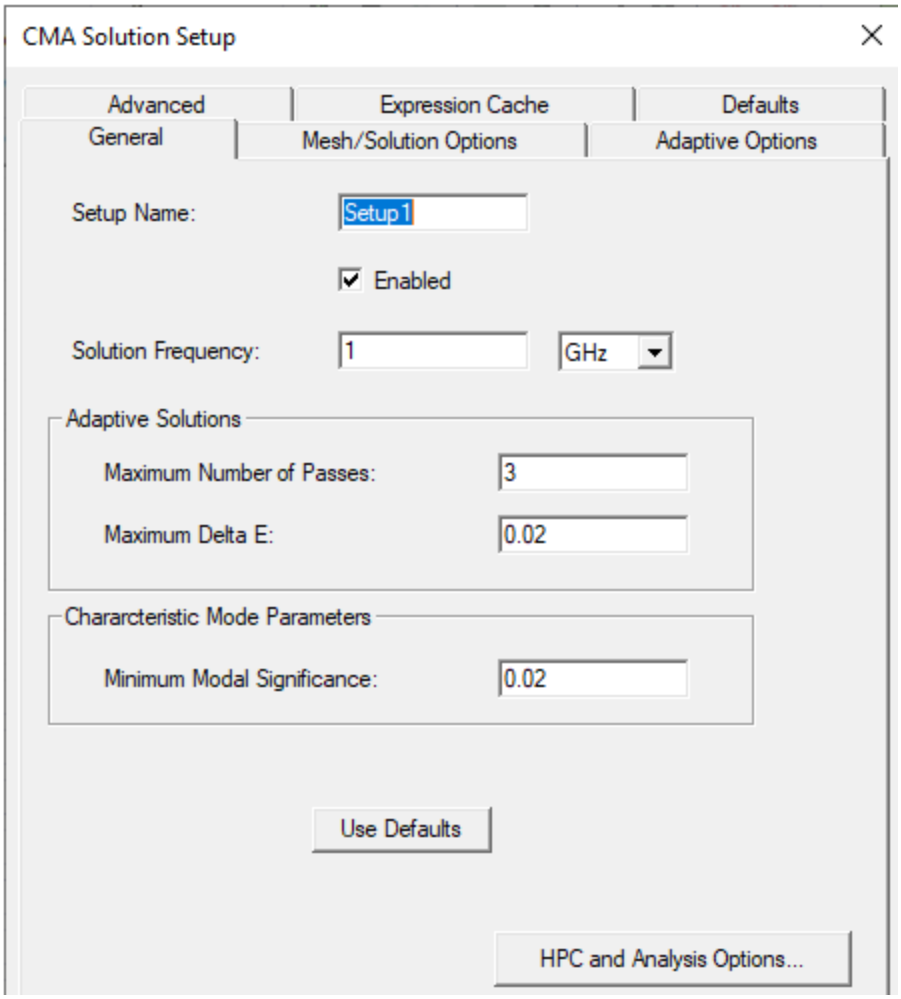


Define the adapt frequency and characteristic mode parameters on the **CMA Solution Setup** dialog box. For Characteristic Modes solution types, these include the following:

a. Solution Frequency
b. Maximum Number of Passes
c. Maximum Delta E
e. Minimum Modal Significance, a measure of how significant a mode is for solving the EFIE at a given frequency. It also determines how hard or how easy it is to excite a

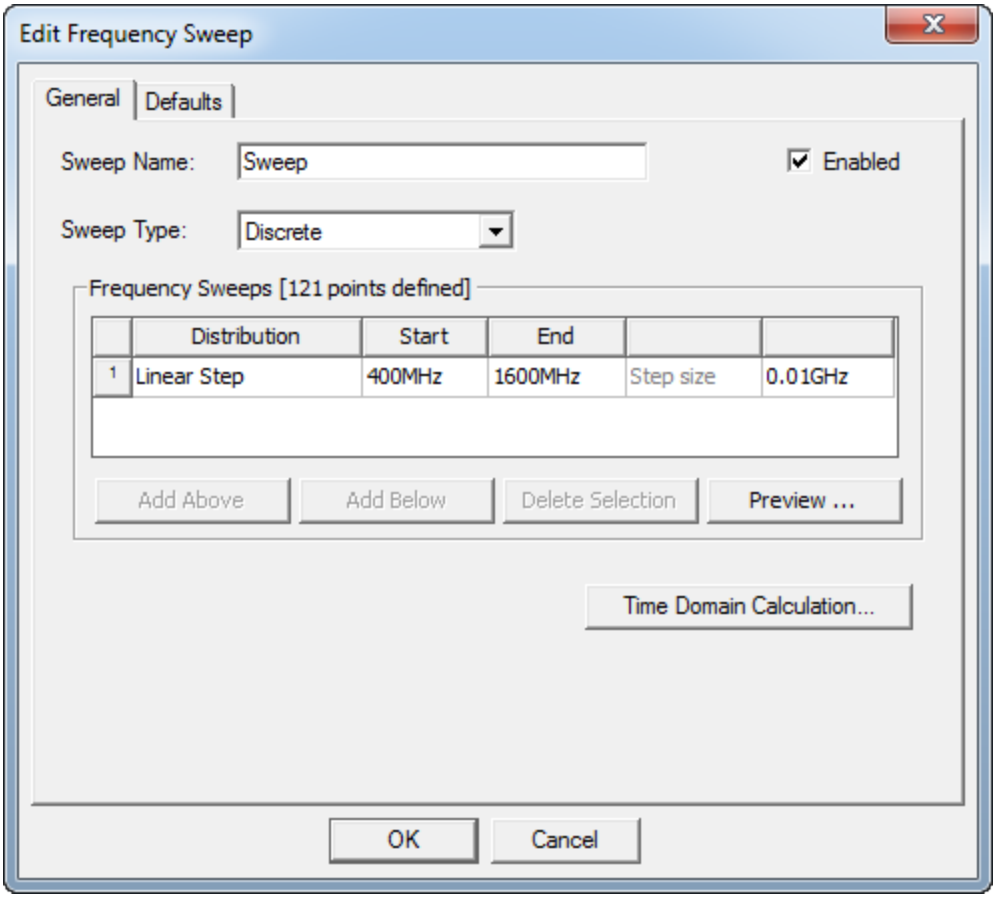
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mode. The default for Minimum Modal Significance is 0.02. All the modes that are above this value 0.02 are calculated.



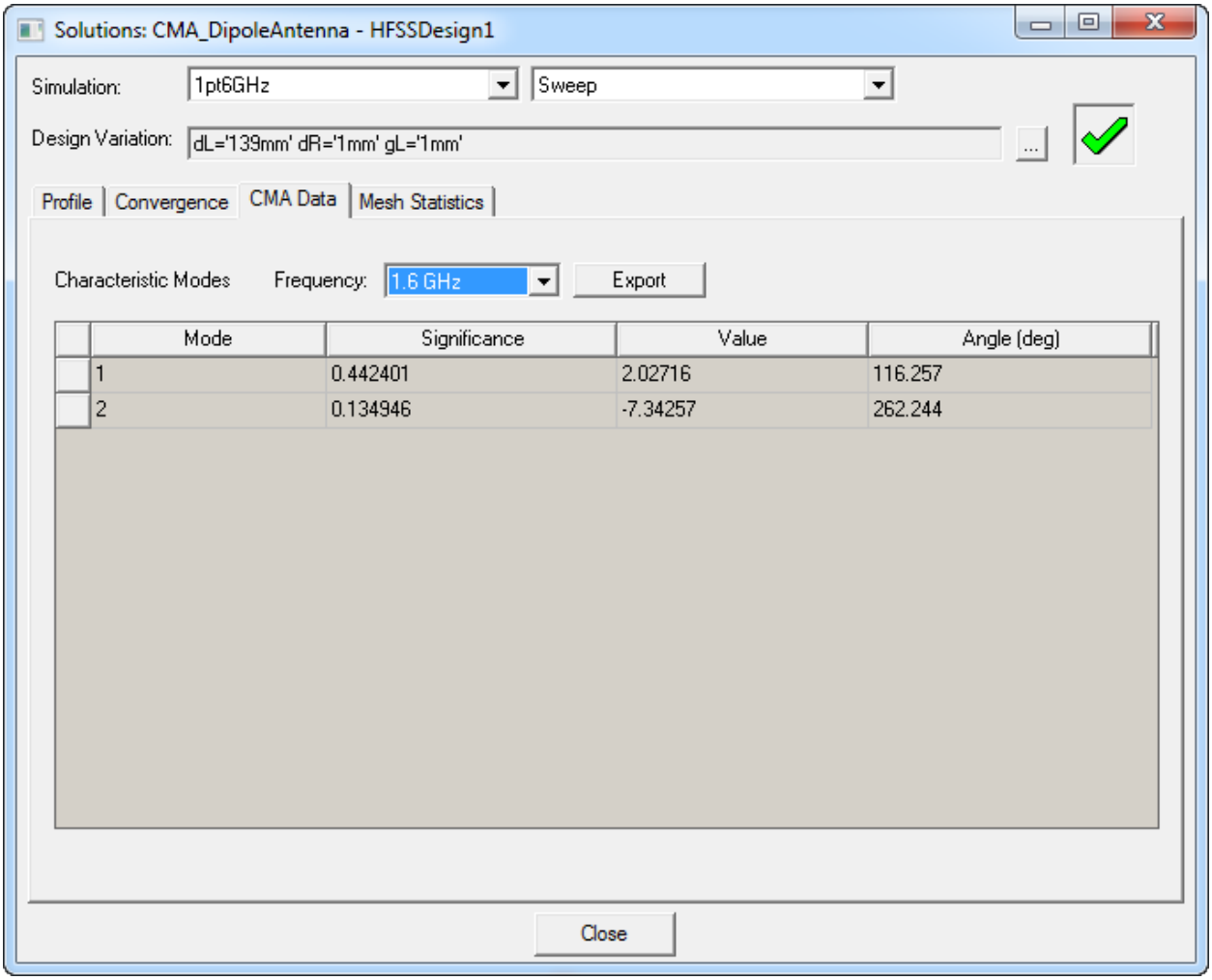
For this design the adapt frequency is set to 1.6 GHz. It's recommended that you define the adapt frequency to be the highest frequency in the discrete sweep.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After the simulation of the dipole antenna is complete, you can view the number of modes above the minimum modal significance, their characteristic angles, and characteristic values at every frequency in the **Solution Data** dialog box. To bring up this dialog box, right-click Results in the project tree and select **Solution Data** from the shortcut menu. The characteristic mode quantities are displayed in the **CMA Data** panel.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



From the Frequency drop-down menu, select the desired frequency view all the quantities for the characteristic modes. As an example, at the adapt frequency of 1.6 GHz, there are two modes in this dipole antenna above the specified minimum modal significance.

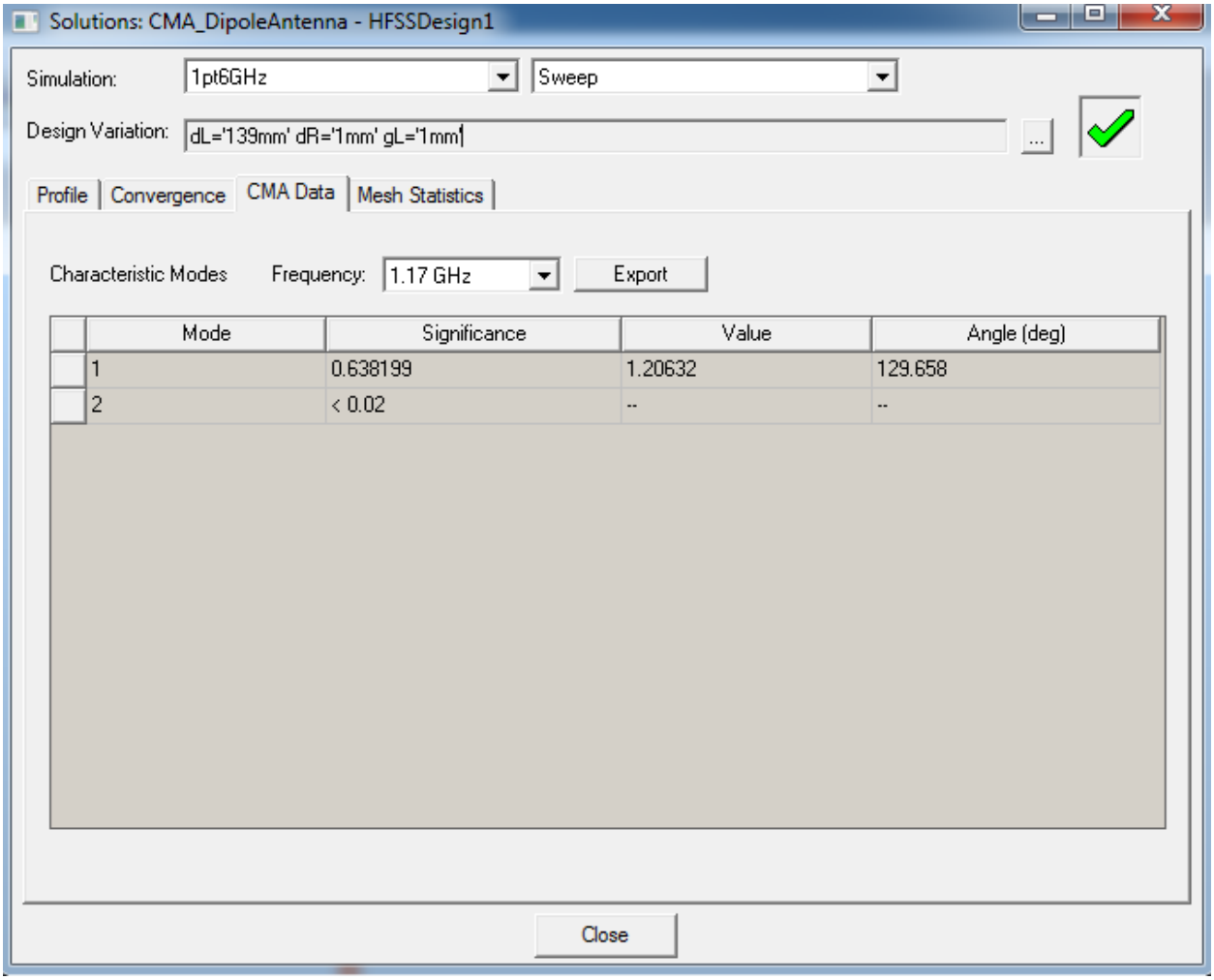
For these two modes:

- Modal Significance MS_1= 0.442401 and MS_2 = 0.134946 at 1.6 GHz
- Characteristic angles α_1= 116.257 degrees and α_2= 262.244 degrees.
- Characteristic values λ_1=+2.02716 and λ_2= -7.34257.

From these values, you can infer that mode 1 is inductive and mode 2 is capacitive.

As you choose lower frequencies, significance of mode 2 reduces and MS_2 approaches a value less than 0.02. At 1.17 GHz, only mode 1 is significant and above 0.02.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

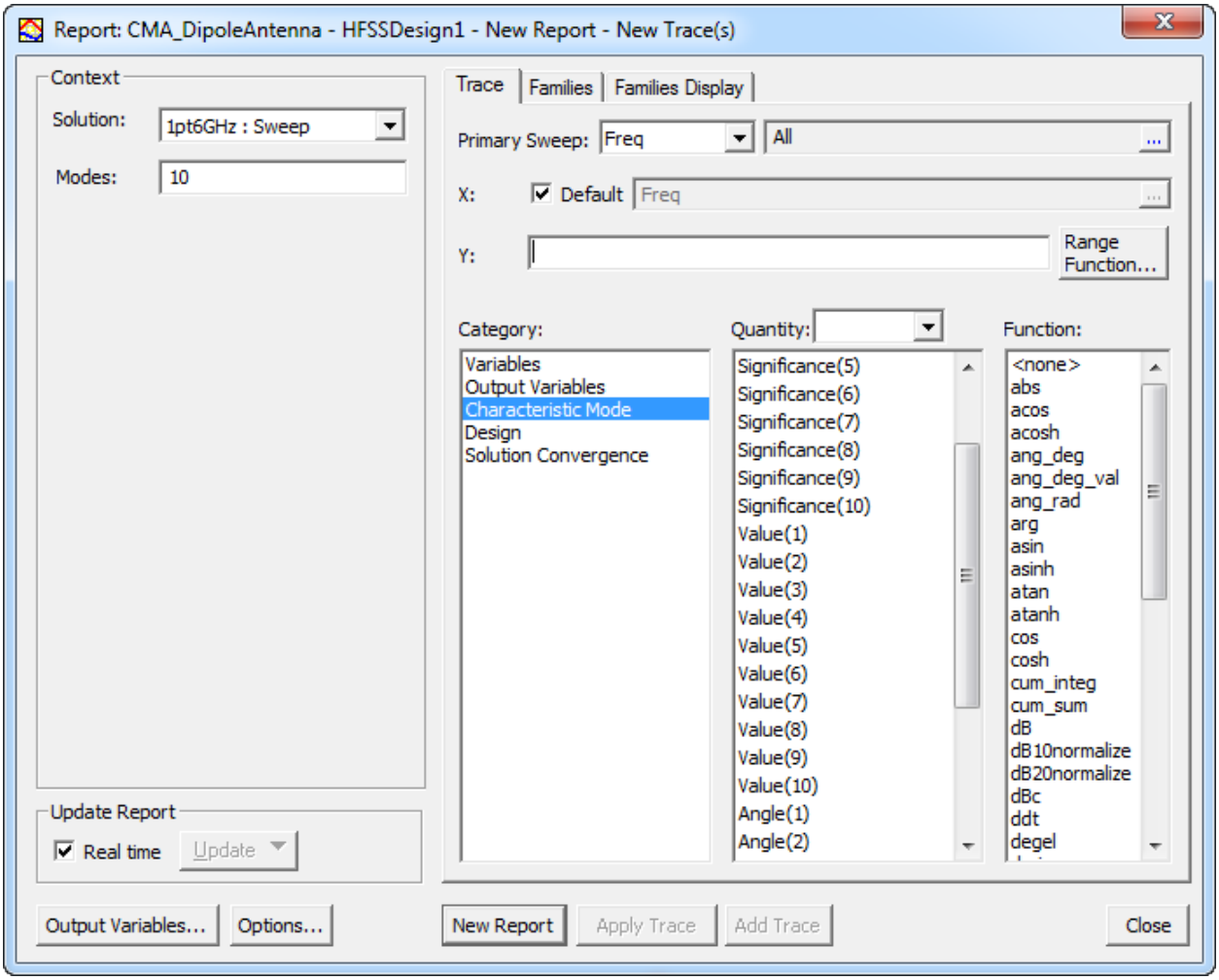


Characteristic Mode Data Report

The process for generating Characteristic Mode Plots for each of these quantities is as follows:

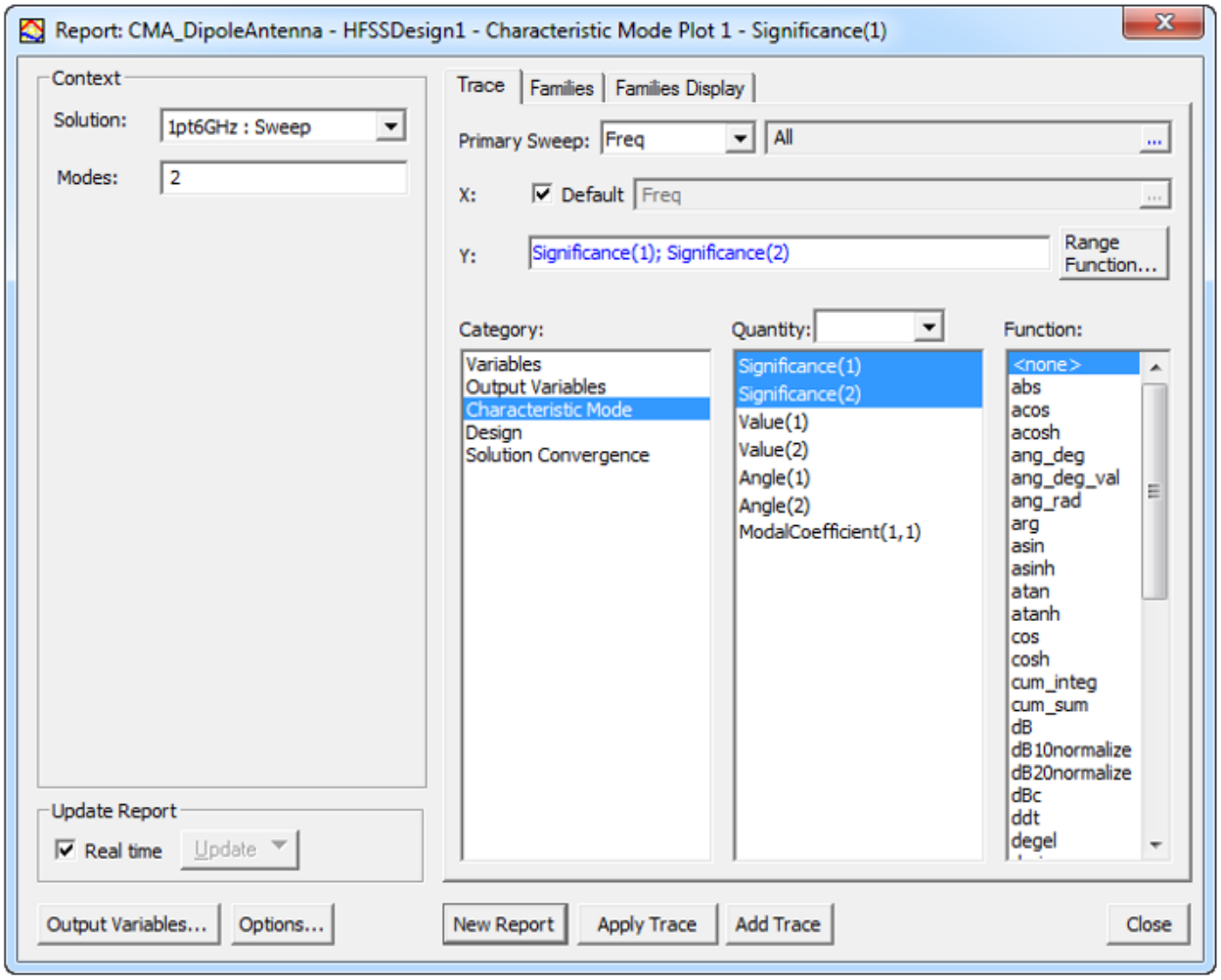
1. Check the maximum number of modes for a design from the **CMA Data** panel at the highest frequency.
2. Right-click **Results > Create Characteristic Mode Data Report > Rectangular Plot**, use the **Results** tab of the ribbon, and select from the **Characteristic Mode Data Report** menu and select the Rectangular Plot icon to open the **Report** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. In the Context panel the number 10 in the Modes field is the default. You can either leave this value as is or change it to the maximum number of modes that are above the minimum modal significance. For this dipole antenna, the Modes field is set to 2 since there are only two modes at the adapt frequency as displayed in the CMA Data panel.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Ensure Characteristic Mode is selected in the **Category** and then select the **Quantity** of interest (Significances, Values, and Angles of the desired modes) and leave the **Function** as none.

   For CMA problems, you can also plot the Modal Coefficients. For Plotting Modal Coefficients, you must select a **Function**. These quantities are named "ModalCoefficient (<port>,<mode>) where <port> contains each of the names of the ports in the design and <mode> runs from the number of modes from 1 through N-1 modes where N is the number of modes specified by the user. The solution from a given port excitation is represented as a linear combination of the modal solutions.

$$J = \sum_{n} \alpha_n J_n$$

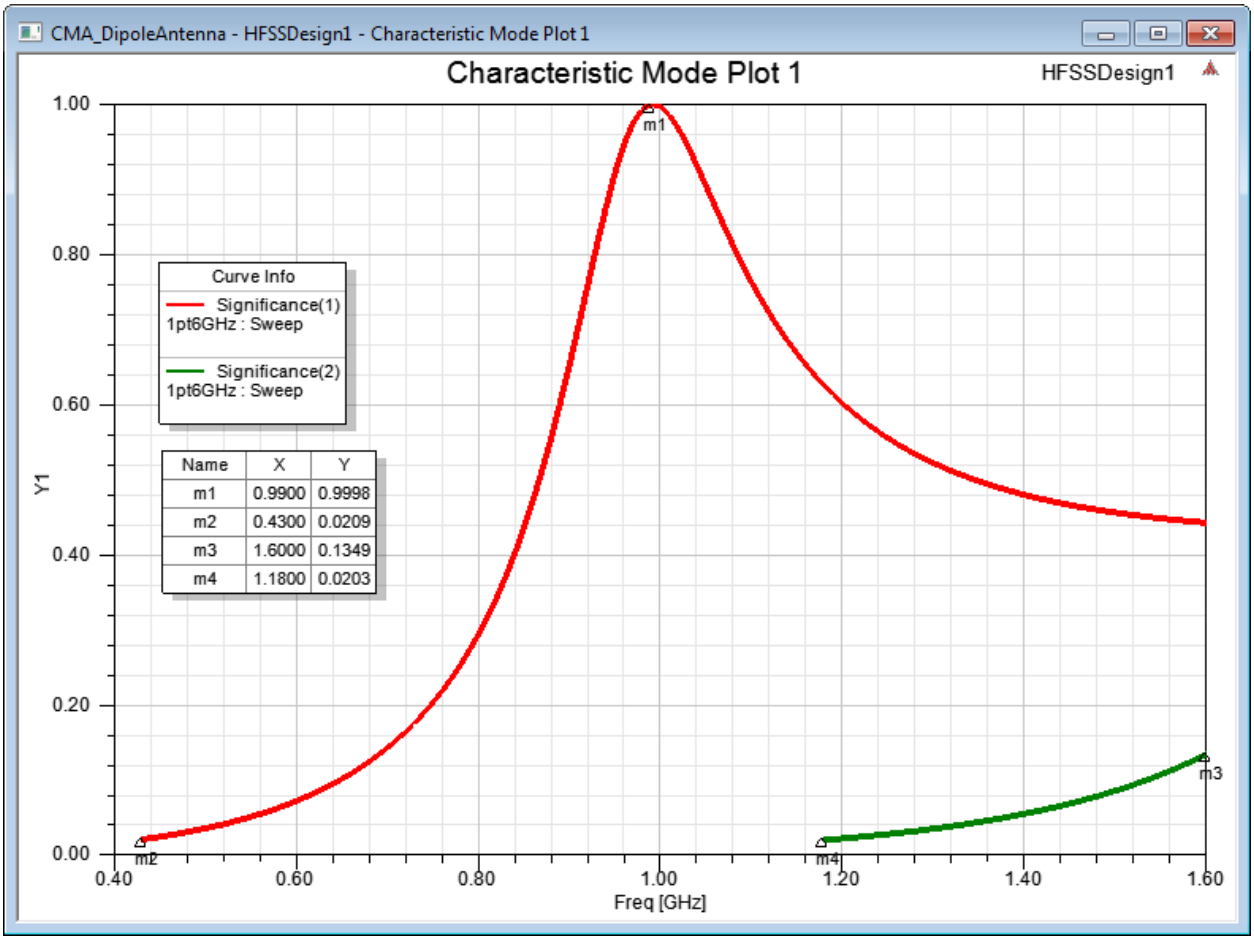
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The modal coefficients are the quantities which we multiply the modal solutions to obtain the port solution.

$$V_{n^i} = \langle J_n, E^i \rangle = \oiint_S Jn \cdot E^i ds.$$

The plots with their corresponding quantities of the characteristic mode are displayed here.

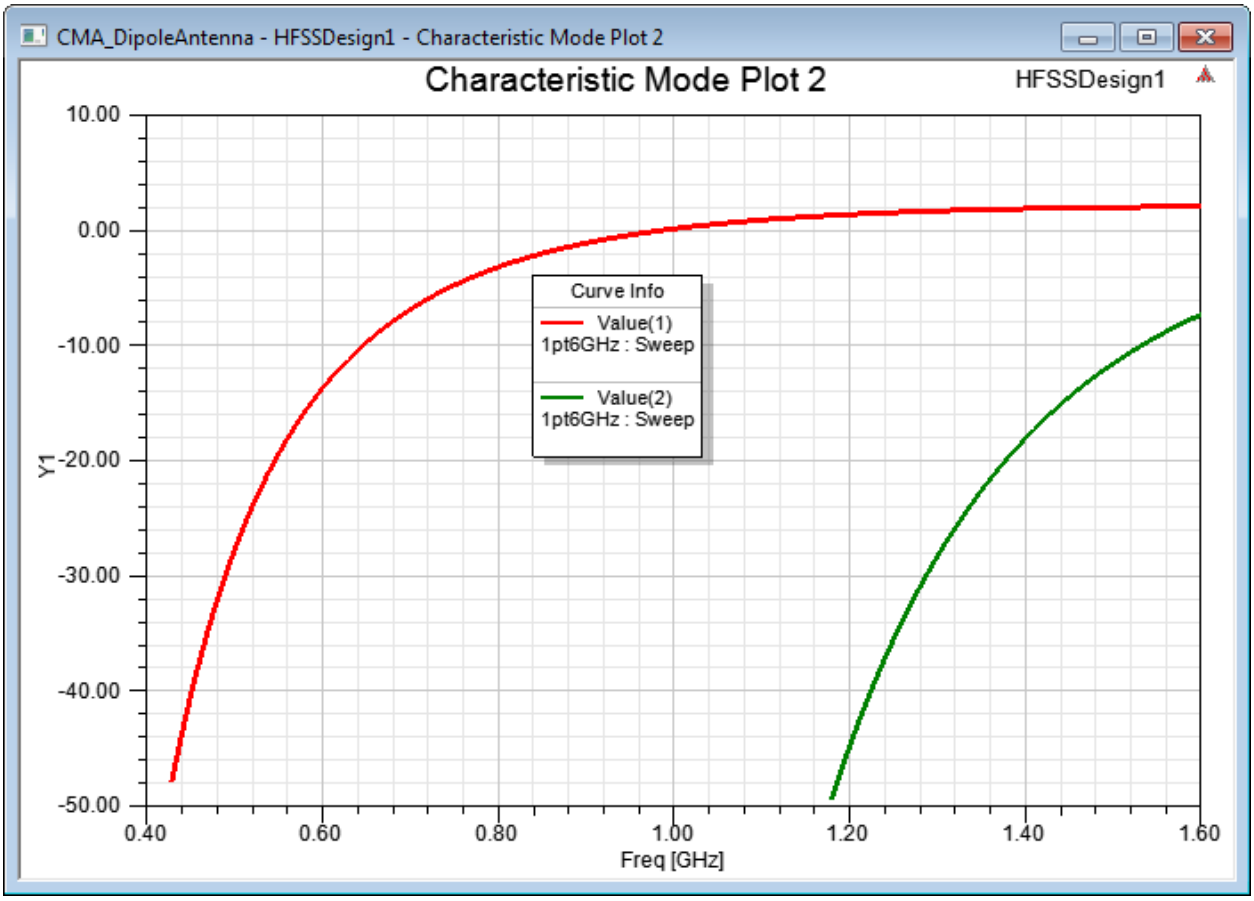
Modal Significance vs Frequency



For this antenna, mode 1 radiates the most at 0.99 GHz and its Significance is 0.9998. Mode 2 radiates most efficiently at 1.6 GHz and its its maximum modal significance 0.1349.

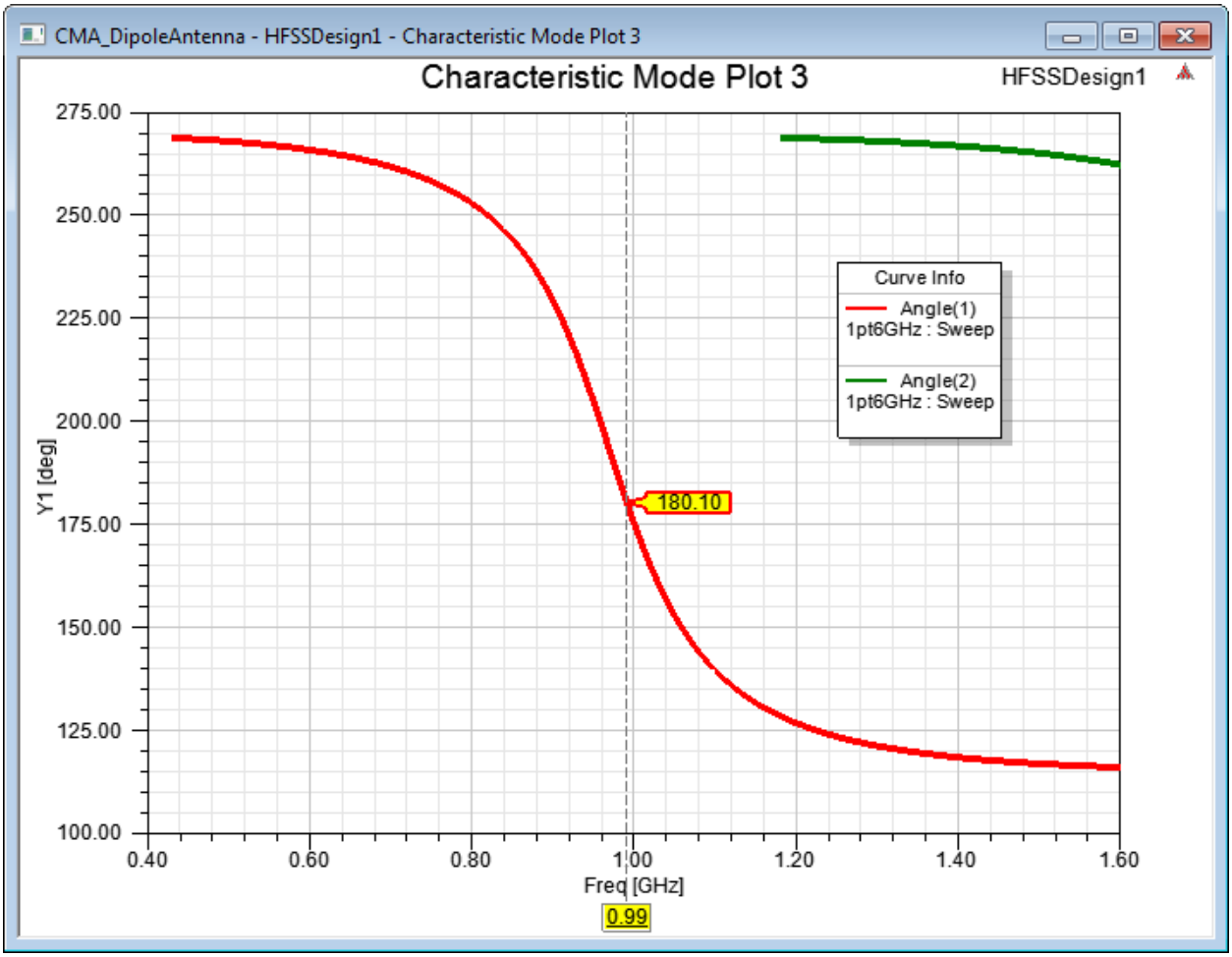
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Characteristic Value vs Frequency



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

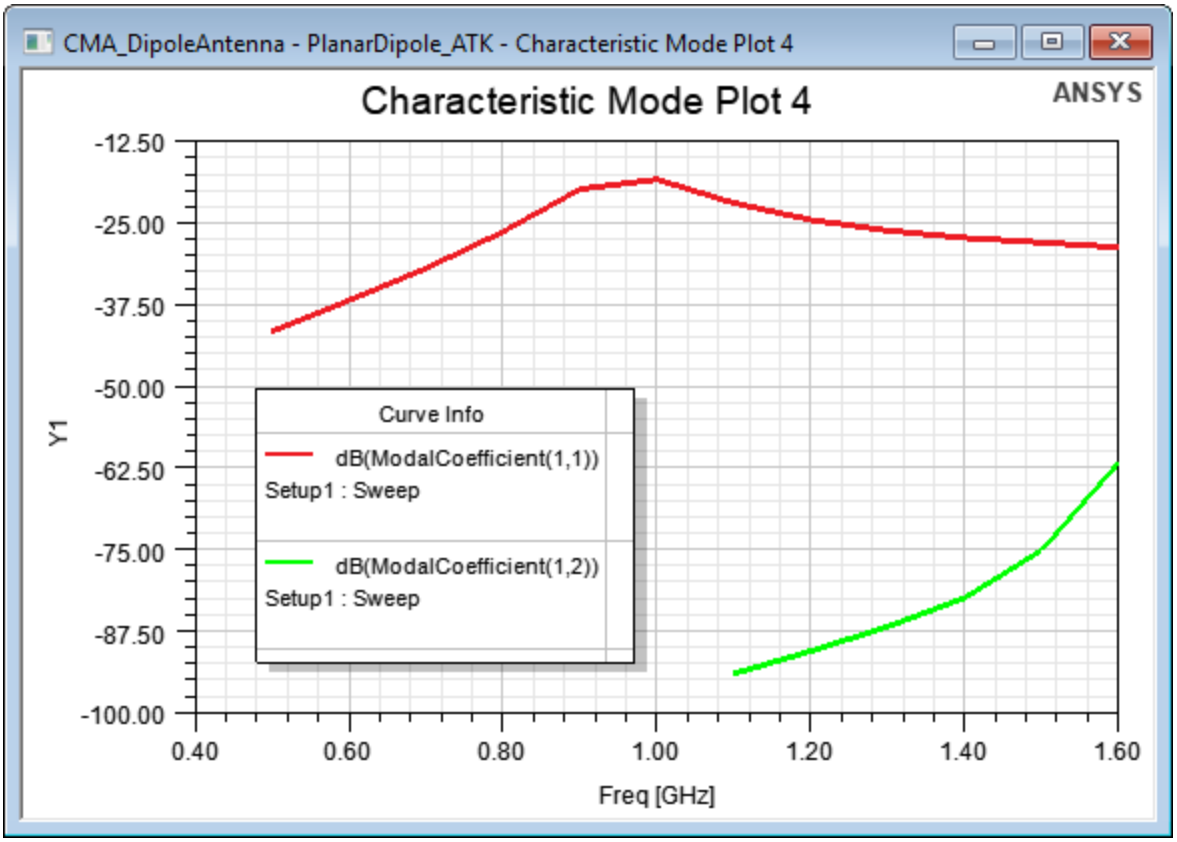
Characteristic Angle vs Frequency



The character angle for Mode 1 is 180 degrees, when it becomes resonant as shown in the above plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

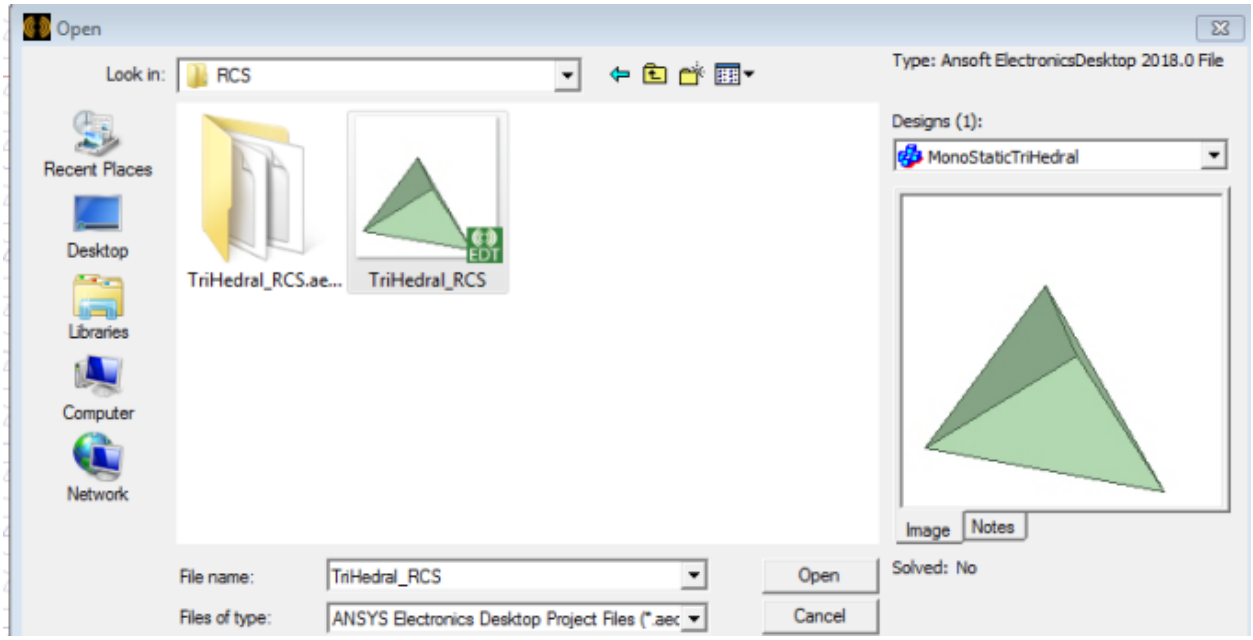
Modal Coefficient Plot



Generating Reports For Monostatic RCS

If you create a design contains both SBR+ Regions and Incident Plane wave you can then create reports for Monostatic RCS. The example projects for HFSS includes an example RCS project that demonstrates this.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

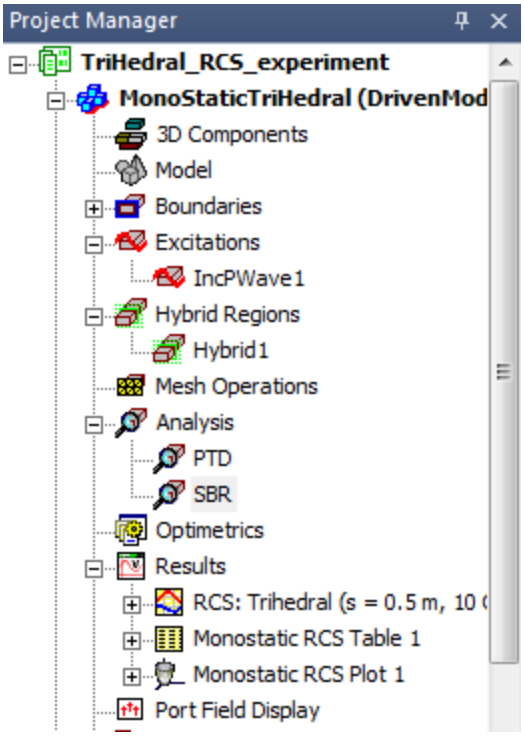


In the case of non-SBR+ projects:

- The new Monostatic report type works with non-SBR+ cases as well.
- For old version projects with Monostatic RCS report already defined it will still work but cannot be modified. The modify dialog will have far fields but not Monostatic RCS in the category column.

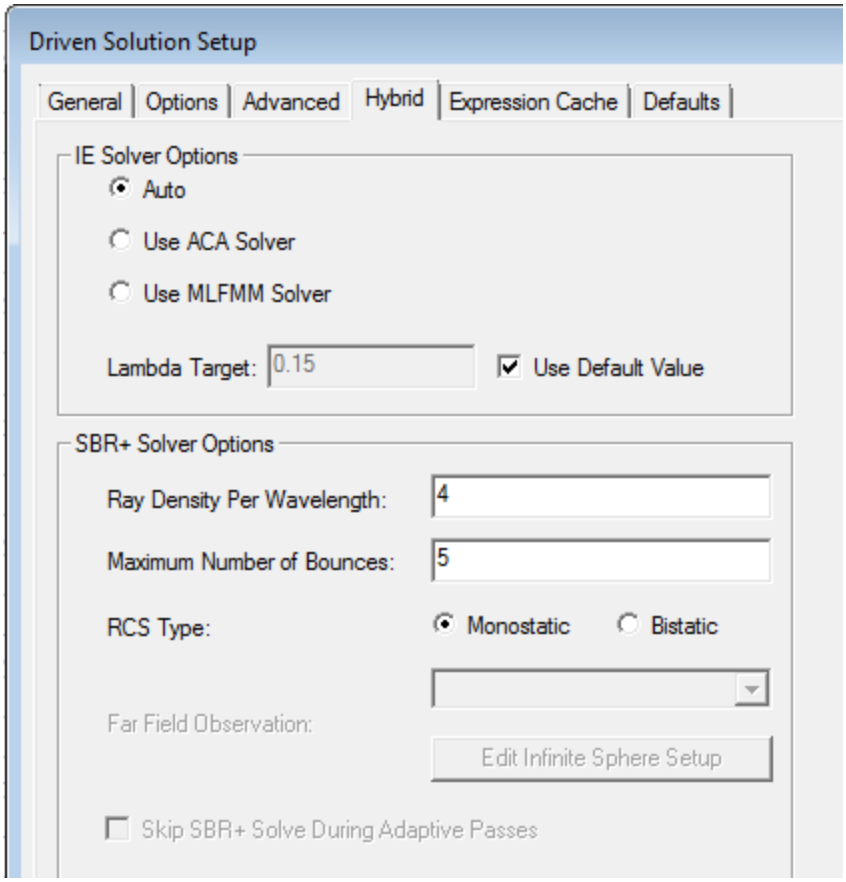
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The RCS example design includes an SBR+ Hybrid Region as well as an incident plane wave:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Designs with these elements include a selection for RCS Type as Monostatic or Bistatic.



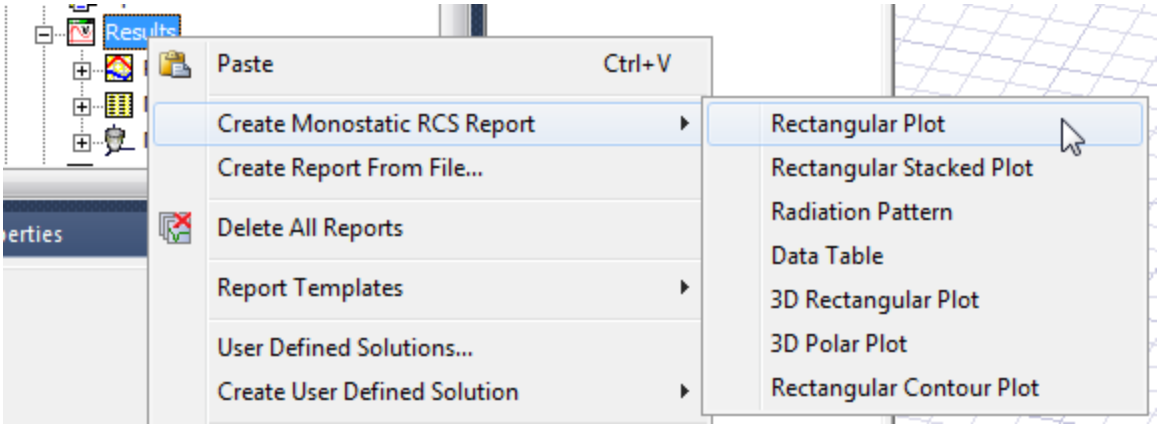
Selecting Monostatic permits you to create Monostatic Reports. In this case, the Infinite Sphere setup is disabled because the solver uses the direction of the Incident Plane wave to calculate the solution.

Monostatic RCS Reports

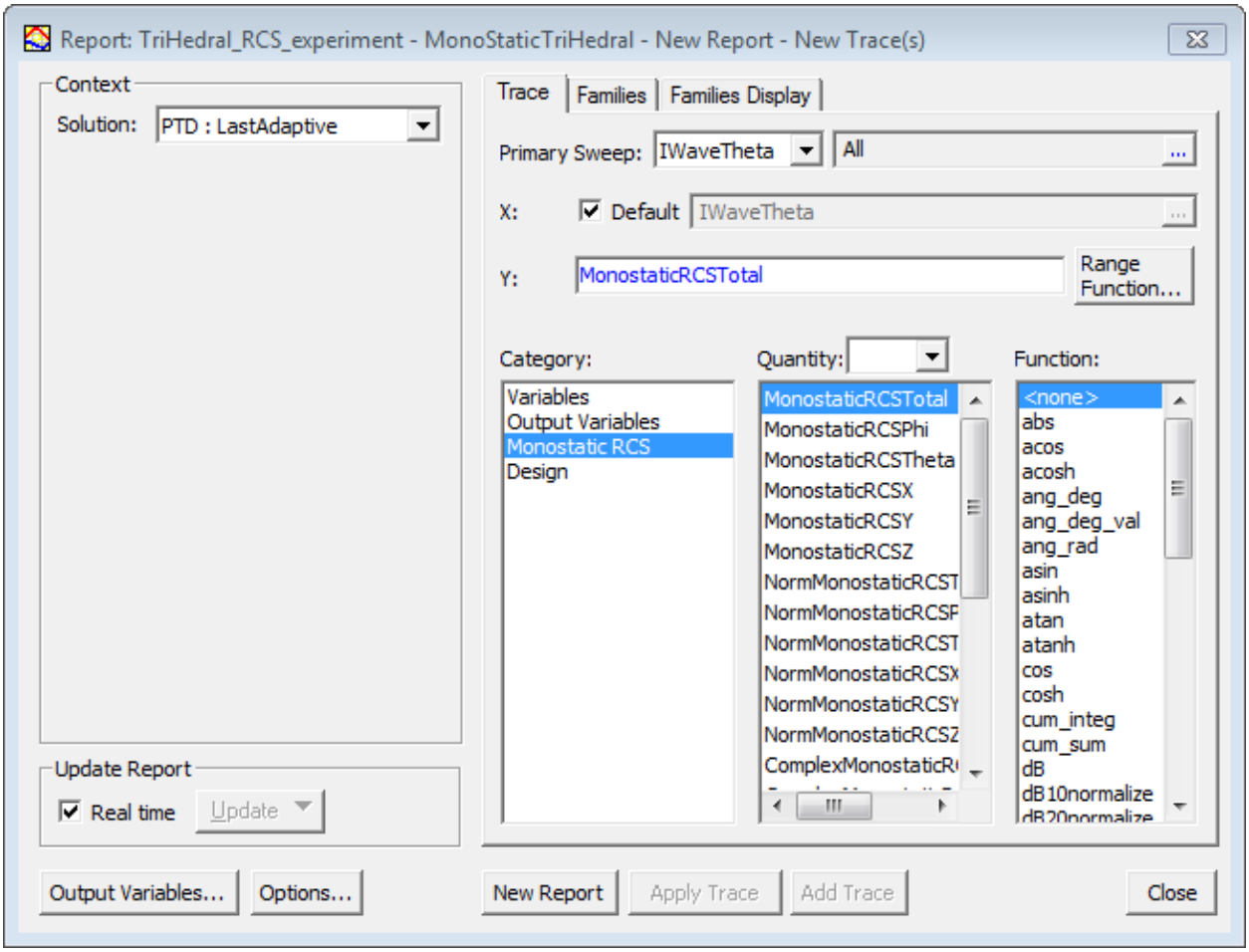
The process for generating Monostatic RCS Plots is as follows--

1. Solve a design containing SBR+ Hybrid Regions and an Incident Plane wave with no other excitations.
2. Right-click **Results > Create Characteristic Mode Data Report > Rectangular Plot** to open the **Report** dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. In the Context panel you can select the solution, but you do not need to select any geometry when using SBR+ and Monostatic RCS. Select the Category as Monostatic RCS to see a Quantity list for Monostatic traces you can create.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. You can choose between Freq, IWaveTheta and IWavePhi variables for specifying sweeps.

Creating a Report from an Ansoft Report Data File

If you have previously saved an Ansoft Report Data Format (using Report2D > Export), you can create a report from that rdat file. This provides a way to reuse the data and/or the format of a previously created report.

1. Right-click the Results icon in the Project Manager to display the short cut menu and select **Create Report From File**, or click **HFSS** or **HFSS-IE** > **Results** > **Create Report From File**.

   A file browser displays.

2. Select an .rdat format file, and click **Open**.

   The report is created. If it contains data, it displays the report with traces. If not, the report uses the format exported to the .rdat file.

Zooming and Fitting Reports

The standard Zoom and Fit commands operate on reports. After clicking in an open report, you can also use a mouse wheel to zoom in and out.

Modifying the Background Properties of a Report

To modify the appearance of a report, or to modify the display properties any object in a report, including traces, axis labels, grids, colors, fonts, legends, color maps, contour color, and so forth:

1. Open the report you want to modify.
2. Select an editable object in the report to be able to edit its properties. Click on an object to select it and to view its properties in the docked **Properties** window. To open a floating **Properties** window, either double-click the selected object, or select **Edit** > **Properties** from the menu bar.

   The **Properties** tabs and options displayed for editable plot objects varies depending on the report type (for example, whether 2D rectangular, 2D polar, Smith, Stacked, or 3D), but can include the following:

   - **Cartesian** – controls the scroll bar and thumb properties for 2D rectangular plots.
   - **Header** – controls the properties for the text displayed at the top of the report, including the Title font, Company Name, Show Design Name, Subtitle Font. The plot title is tied to the report's name and is not a Header property. If you change the report name in the Project tree, plot title synchronizes. The Company Name and the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Show Design Name check box are grouped in the **Properties** dialog box as Subtitle. Edits to the Subtitle Font Property affects both of them.

- **General** – controls the background color (the perimeter around the trace display) for the plot, the contrast color (the trace display background), the Field width, the Precision, and whether to use scientific notation for marker and delta marker displays. (X and Y notation display is set separately, in the Axis property tabs.) An Auto Scale Fonts property is on by default and scales text in plots and colorkey (contour plot, field plots in 3D modeler) for high resolution screens.
- **Legend** – controls the properties for whether to include a Legend Name, Show Trace Name, Solution Name, and Variation Key. At least one of these must be selected. Legend Name is blank by default. When non-empty, a header row for the Legend in plot shows up with that string. You can also specify the File Name Display as Full Path, File Name without Path, or as an Array Index. You can also edit the Font, the background color of the Legend box, the Border Color, the Border Width, Grid Color (for the lines between Trace descriptions), and the Grid line width. See: Modifying the Legend in a Report.
- **Color Key** – for 3D plots, controls the appearance of the color key (colors, transparency, border appearance, fonts, number format, field width and precision.
- **Contour** – for 3D plots, controls the appearance of the color map, including map type, ramp color, spectrum, IsoValType, levels, number of contours, and values shown.
- **Radiation Pattern** – for 2D polar plots, controls whether to show the circular grid and angle lines.
- **Stacked** – for stacked plots, controls properties for X scrollbar, thumb properties, and stack layout, auto fit, and stack height.
- **Smith** – for Smith charts, controls whether to show grids for Imp., Adm., Cir, and angle lines.
- **Traces** – controls the properties for traces, including: Color, Line Style, Line Width, Trace Type, whether to Show a symbol, Symbol Frequency, Symbol style, whether to Fill symbol, symbol color, and whether to show arrows. You can select traces either in the Legend or on the plot. See: Editing the Display Properties of Traces.
- **Axis** for X, Y or Z, or for Phi, Theta or Rho, and circular – the defaults for most of these values (applying to 2D and 3D both) are set in the **Report 2D Options**Axis tab.
    - Display name – check box for whether to display the axis name.
    - Specify name – check box for specifying the Axis name.
    - Name – this describes the axis to which the following properties/options refer. These are selected in the **Report** dialog box.
    - Axis Color – set the color by double-clicking to display the **Set color** dialog box. Select a default or custom color and click OK.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Axis Font – click the cell to display the **Edit Text Font** dialog box. The dialog box lets you select from a list of available fonts, styles, sizes, effects, colors, and script. The dialog box also contains a preview field. OK the selections to apply the font edits and to close the dialog box.
- Display Units – this specifies whether to display the axis units.
- **Window (section):**
  - Window Mode – can be Axis range, Continuous moving window, or Step moving window.
  - Window Width (in) – provide an integer value for the previous selection.
- **Manual Format** (section):
  - Number format – select from the drop down menu, Auto, Decimal, or Scientific notation.
  - Field Width – enter a real value.
  - Field Precision – enter a real value.

■ X or Y or Z **Scaling Tab** – These properties provide control over scaling:

- Axis Scaling – use the drop-down menu to select scaling as Linear or Log. For the Y axis, all zero or negative values are discarded before log scaling is applied. For 3D plots, scaling is on the Axis tabs
- Specify Min – check box
- Min – text entry in same units as axis units. Saved as SI internally.
- Specify Max – check box
- Max – text entry in same units as axis units. Saved as SI internally.
- Specify Spacing – check box
- Spacing – text entry in same units as axis units. Saved as SI internally
- **Manual Units** (section):
  - Auto Units – use the check box compute the correct units for the axis.
  - Units – click on the cell to select from a menu of available units if you have not checked Auto Units.
- **Infinity Visualization** (section):
  - Map Infinity Mode – check box.

    Each axis can be set to treat infinity values in a user defined way. When you check the Map Infinity Mode, any infinity values in the input data get the infinityMap value (negative infinity get the value*-1 and positive infinity the positive value specified). This can be useful if there are zeros, or very small values that HFSStreats as zero, in the data, for example, dB Gain.

  - Map Infinity To – enter a real value for the Map Infinity Mode.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Grid** – properties for grid labels and grid style, appearance, line styles, color, major and minor lines, major and minor circles on polar grids, and scaling. For the 3D rectangular plots, there are separate tabs for the XY, YZ and ZX axes, and for 3D Polar plots, tabs for phi-rho, or theta-rho grids.

3. Edit the properties, and **OK** the dialog box to apply the changes.

Modifying the Legend in a Report

The legend in a report is a list of the curves being plotted. For each curve, the legend gives the name, shows the line color, and lists the setup and the adaptive pass used to generate the curve.

To show or hide a legend in a report:

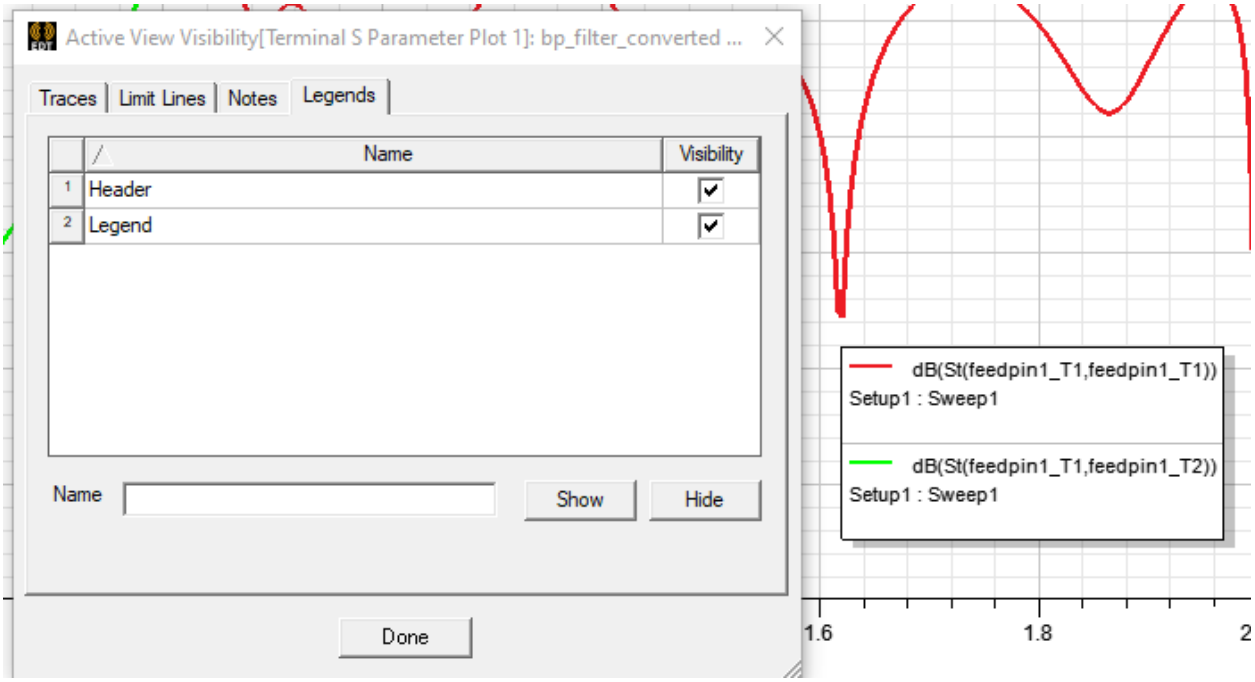
1. Make the report the active view.
2. Use **View** > **Active View Visibility** or the **Show/Hide** icons on the toolbar to display or hide the report.

   Either command displays the **Active View** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Select the **Legend** tab.

   This lists the legend (or legends) in the report.



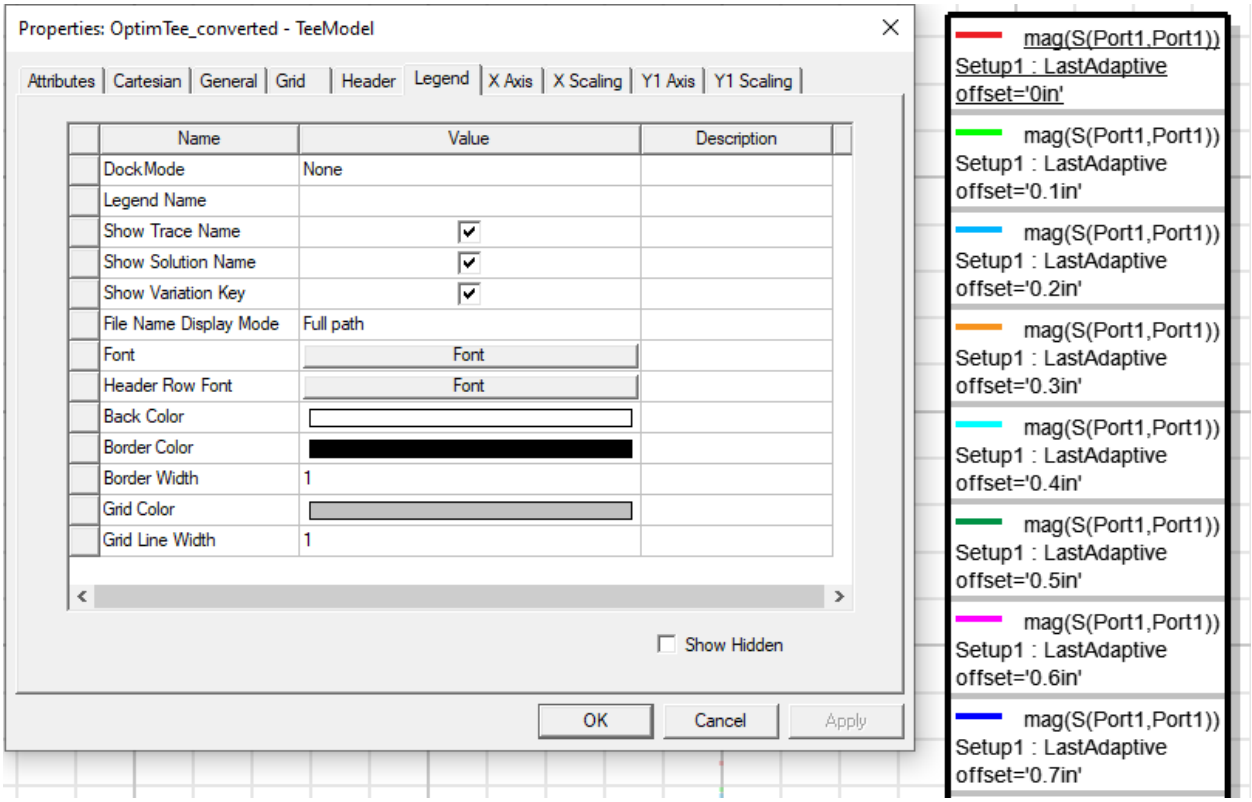
4. Enable the visibility check box, and click **OK** to close the dialog box and apply the change.

To edit the display properties of a legend:

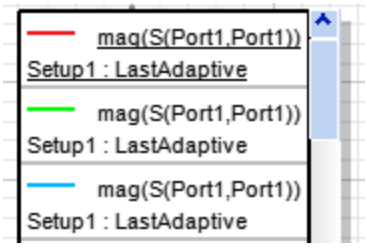
1. Double-click a legend in a report to display a docked properties window, or right-click on the legend and select **Edit** > **Properties** to display the floating **Properties** dialog box.

   This lets you edit the Properties for Dock Mode, Legend Name (default is no name. When non-empty, a header row for the Legend in plot shows up with that string.), whether to Show Trace Name, Solution Name, and Variation Key (which applies to parametric variables, if present). If none of these three are selected, only a trace color shows.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Here is an example with Variation Key off.



You can also edit the Font by clicking the Font cell to display the **Edit Text Font** dialog box. The dialog box lets you select from a list of available fonts, styles, sizes, effects, colors, and script. The dialog also contains a preview field. OK the selections to apply the font edits and to close the dialog box.

You can also edit the background color of the Legend box, the Border Color, the Border Width, Grid Color (for the lines between Trace descriptions), and the Grid line width.

2. Click **OK** to close the **Properties** dialog box and apply the selections.

To change the display name for traces, see: Editing Trace Properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To move a legend in a report:

1. Click and hold on the legend.

   The cursor changes to crossed lines with arrow tips.

2. Still holding, drag the legend to a new location and release.

   The legend is released and the crossed lines change back to a mouse pointer.

To resize a legend in a report:

1. Position the mouse tip over the edge you want to resize.

   The mouse pointer changes to a horizontal or vertical line with arrow tips.

2. Click and drag the horizontal or vertical edge to the desire size.
3. Release.

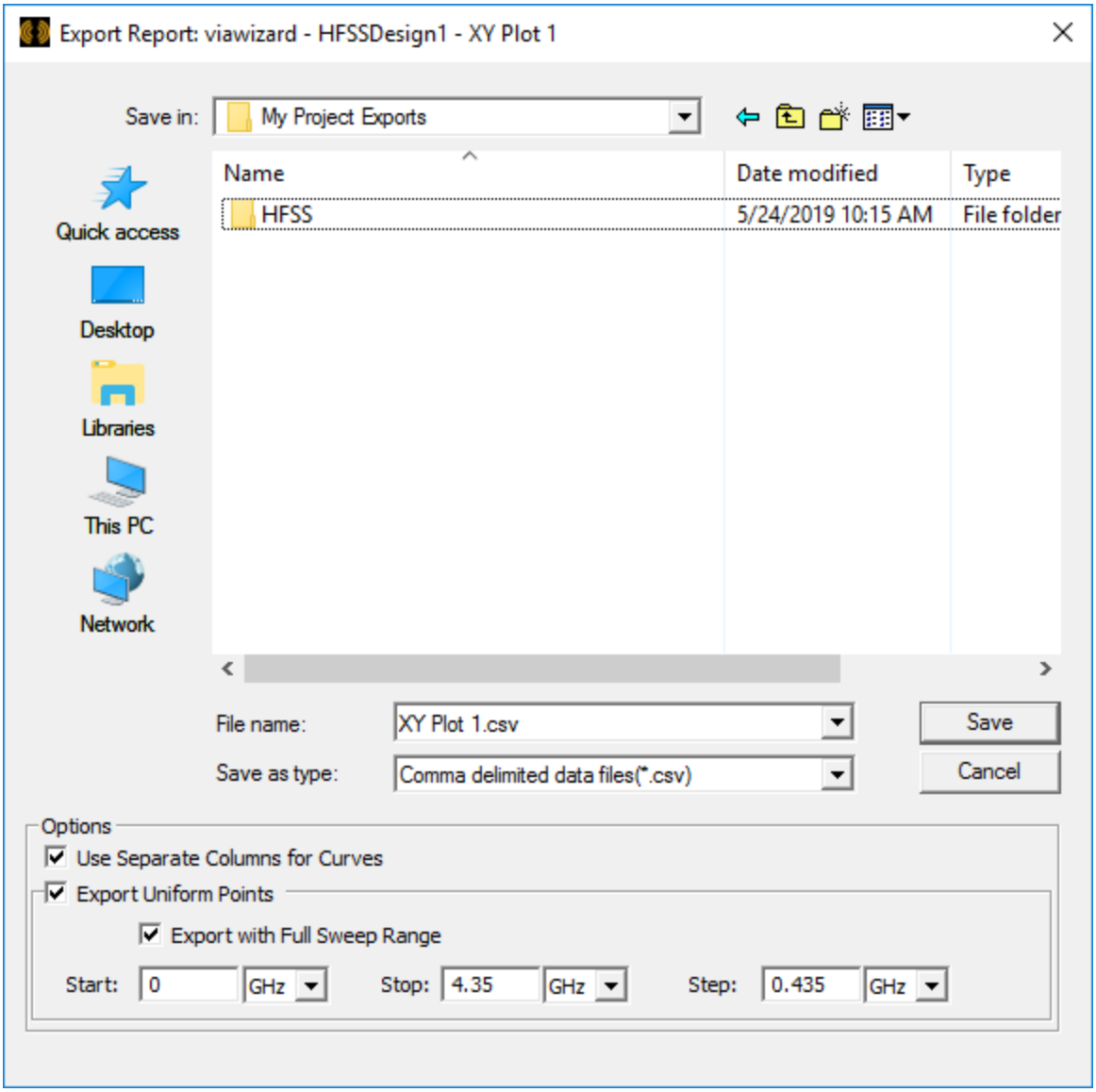
Exporting Ansoft Report Data Format Files

Ansoft Report Data format (*.rdat) files provide a way to export reports or report formats, which you can then import using **Reports> Create Report From File**. This can save repeated editing of properties (for example, the company name or color schemes) when you create other reports. You must have an existing plot open to see the **Report2D** menu.

1. Click **Report2D** > **Export...** or **Report 3D**> **Export...**

   The **Export Report** dialog box appears:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Use the **Save as type** drop-down menu to select **Ansoft Report Data Files (*.rdat)** format.
3. Browse to a destination folder and enter a name for your file in the **File name** field.
4. The **Export Uniform Points** option, if available for your report type, allows the *.rdat file to contain points for the start, stop, and step at the given frequencies. If you do not select this option, the file contains only the current file format, including any modifications you have applied.
5. Click **Save**.

   The file is exported to the specified location as an Ansoft Report Data file (*.rdat).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

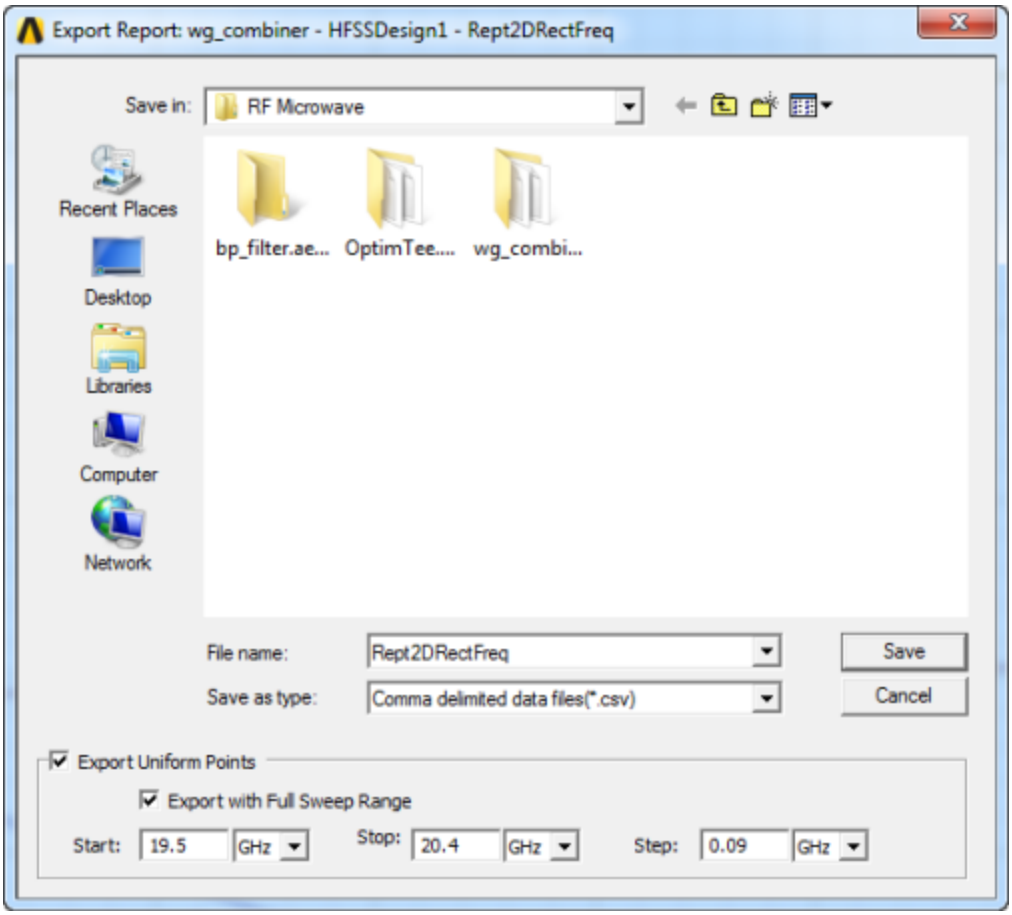
The file will then be available for import using Create Report from file.

Exporting Reports as Graphics

You can export reports as figures in several formats. You must have an existing plot open to see the **Report2D** or **Report 3D** menu.

1. With a report open, click **Report2D** > **Export...** or **Report3D** > **Export...**

   The **Export Report** dialog box appears:



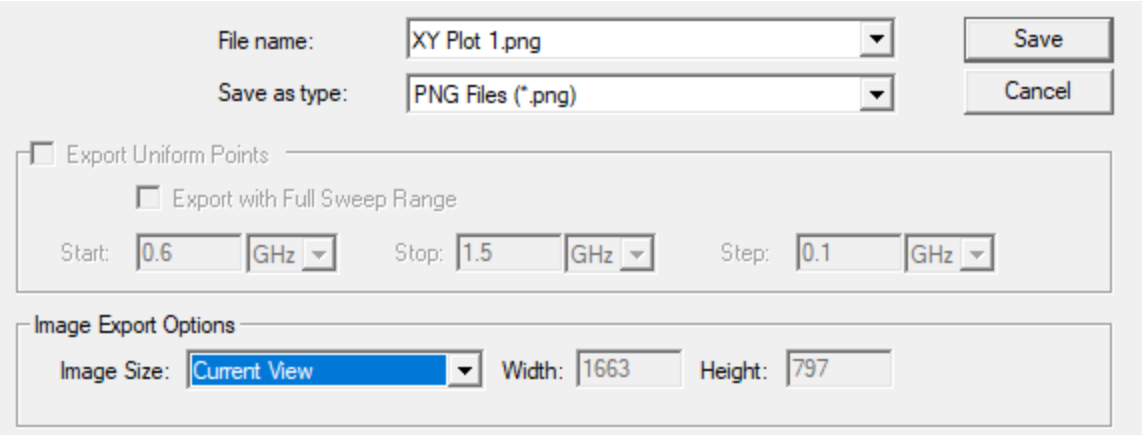
2. Click **Browse...** to open the **Export Report** browser window.
3. Specify the file location and name, and select a graphics format from the drop-down menu:

   - .bmp
   - .gif
   - .jpeg

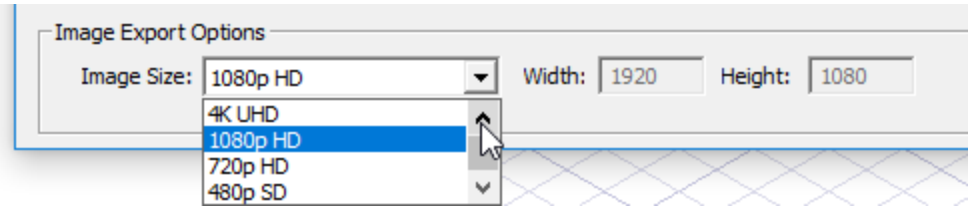
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- .png
- .tiff
- .wrl

4. When you select an image format, the **Image Export Options** appear at the bottom of the dialog box:



5. Specify the **Image Export Options** by selecting from the drop-down menu. You may need to scroll or click the up or down arrows to see all selections. These include: Current View, User Defined (which enables the Width and Height fields), Full Screen, 8K UHD, 4K UHD, 1080p HD, 720p HD, and 480p SD. The resolution for the current selection is displayed in the Width and Height fields.



6. Click **Save** to close the browser window, and then **OK** to close the **Export** window.

To export an image file of a report to specified high resolution, use the scripting commands. The image can be created at a specified resolution. Fonts and line thickness is not scaled. Only the image. You will have to iteratively increase font sizes until you find a suitable output.

Report File Formats

The following table provides information about the file formats that can be imported into plot data reports:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File Extension	Information
csv	• Uses a comma (,) as the separation character. • The x-axis value is in the first column. Each curve's Y-values make up one column, and the curves are in the same order as in the plot legend. • The first row is the X-axis name [unit] and the information for each curve, which is the same as the plot curve legend. • Other rows are the X-axis value and the Y-value for each curve.
tab	• Uses the tab character as the separation character. • The x-axis value is in the first column. Each curve's Y-values make up one column, and the curves are in the same order as in the plot legend. • The first row is the X-axis name [unit] and the information for each curve, which is the same as the plot curve legend. • Other rows are the X-axis value and the Y-value for each curve.
txt	• The txt file must begin with a header that contains the following: • A first line made of 92 equal signs (=) • A second line that begins with the company name and ends with the date in MM/DD/YY format • A third line that begins with a plot name and ends with the time in hh:mm:ss format • A fourth line, which is empty • A fifth line made of 92 hyphens (-) **Note:** Each header line must be 92 characters in length, except the empty line. • Each column has a fixed width, separated by white space. • The x-axis value is in the first column. Each curve's Y-values make up one column, and the curves are in the same order as in the plot legend. • The first row is the X-axis name [unit] and the trace name [unit] for each curve, which is the same as the plot curve legend. • The second row is an empty column and the variable values for each curve, which is the same as the plot curve legend. • Other rows are the X-axis value and the Y-value for each curve.
dat	DAT is an Ansys-specific format. Ansys recommends that you do not generate files using this format. Files imported with these formats should be exported only from Ansys products.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File Extension	Information
rdat	RDAT is an Ansys-specific format. Ansys recommends that you do not generate files using this format. Files imported with these formats should be exported only from Ansys products.

Creating Custom Report Templates and Defaults

You can edit properties from any report type and save it as a template or as the default. This can save repeated editing of properties (for example, the company name, or color schemes and plot attribute settings) when you create other reports. You can prepare a template by copy/pasting plots settings from one plot to another of the same display type. Once you create templates, you can access them from the **Results** > **Report Templates** > menu and the **Report2D** > **Report Templates** > **Apply Settings** menu.

See Modifying the Background Properties of a Report for a discussion of format changes you can make to any report.

To save an edited report as a template:

1. In the Project Tree, right-click the report name of interest to display the shortcut menu and click **Report Templates** > **Save...**. You can also click **Report2D** > **Report templates** > **Save...** or **Report3D** > **Save As Template...**

   This displays the **Save As Report Template** file browser. By default, the directory is your AnsysEM\<*productName*>\userlib\ReportTemplates directory. You can also save to the SysLib directory.

2. Typically, you accept the directory.
3. You must provide a file name, which will be given an *.rpt extension.

   It is good practice to give the template a descriptive name, showing both the kind of format you begin with (such as XY Plot or 3D Plot) and apt description of the distinguishing edits (such as for company name, or color scheme). Once, saved, this name will appear on the *PersonalLib* menu.

   The **Save As Type** field currently supports the Ansoft Report Format (*rpt) format.

4. Click **Save** to save the template to the PersonalLib menu.

All *.rpt templates in the userLib directory appear on the **Results> Report Templates> PersonalLib** menu. Selecting a report from the PersonalLib menu opens a report that you can then Modify to add traces or perform other edits. Templates in the SysLib directory appear on the **Report Templates** menu.

To save an edited report as a default:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the Project Tree, right-click the report name of interest to display the shortcut menu and click **Report Templates** > **Save Settings as default**. You can also click **Report2D** > **Report templates** > **Save Settings as default** or **Report3D** > **Save Settings as default**.

Selecting the Report Type

The **Report Types** available for creating a report depend on the simulation setup. The report category provides the default report name for the project tree and the text displayed in the report header. For example, S Parameters as the Category type causes the default report name to be S Parameter Plot *n*. Depending on the setup, you can make a selection from the following report types.

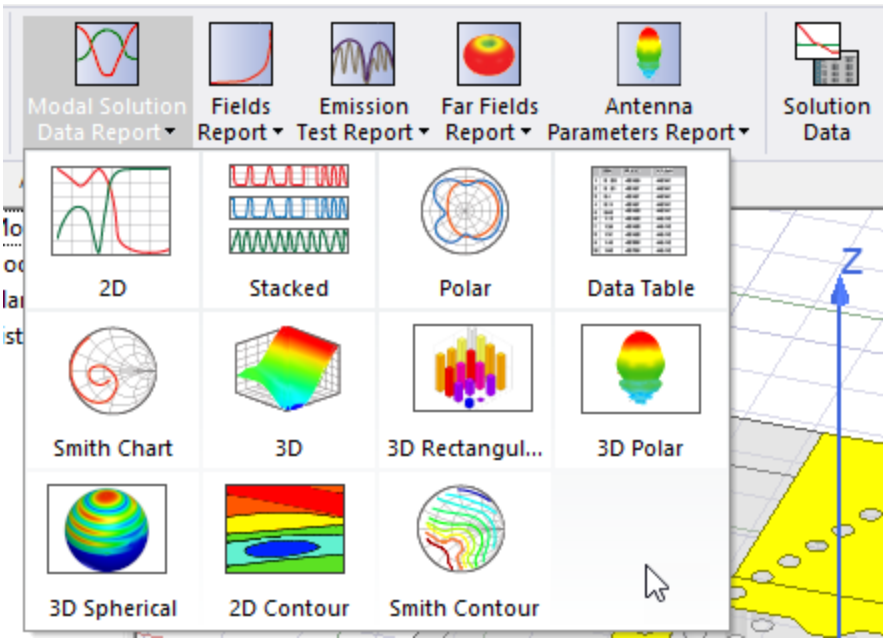
Modal Solution Data	S-, Y-, and Z-parameter data are available to plot, along with propagation constant, characteristic port impedance, reflection/transmission coefficients for HFSS designs, and Voltage Standing Wave Ratio (VSWR) data. For HFSS calculations, phase is currently assigned zero value.
Terminal Solution Data	This solution type results in a terminal-based description in terms of voltages and currents. Some modal data is also available. The terminal-based S-, Y-, and Z- parameters, VSWR, Port Zo, and Active S-, Y-, Z- and VSWR parameters are available to plot.
Eigenmode Parameters	Eigen Modes and Eigen Q data are available to plot.
Characteristic Modes	For HFSS Characteristic Mode solution types, graphing modal significance as a function of frequency lets you easily examine the importance of the modes and understand which of the modes contribute significantly to the radiated power (MS = 1), and the ones that are decaying (MS = 0). This helps toward correct antenna placement on electrically medium-sized structures.
Fields	Basic or derived field quantities calculated on lines or integrated over surfaces or objects are available to plot. Displacement magnitude is available as a predefined modal field result. Predefined thermal field results include temperature, heat flux magnitude, and EM loss densities.
IE Surface Fields	For an HFSS design that uses IE Regions, the solver produces J and Q surface fields for the metallic parts (but not the dielectric parts) of these regions. The Fields Calculator provides access to the IE surface fields quantities.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Far Fields	Radiated fields computed in the far-field region. The following quantities are available to plot: rE, gain, realized gain, beam area, directivity, axial ratio, polarization ratio, and normalized antenna calculated by HFSS. You can do Contour Plots with a Domain of either Theta, Phi, or Sine Space. You must have defined an infinite sphere geometry and at least one radiation or PML boundary to create a far-fields report. If you manually enter Phi or Theta component of a far field quantity when you using Az/El or El/Az far field definition, then the Report uses the Z axis of the Az/El or El/Az coordinate system definition as pole to calculate theta/phi component for rE.
Near Fields	Radiated fields computed in the near-field region. These include: variables, output variables, near E, max near field parameters, and near normalized antenna. You can create Contour Plots with a Domain of either Theta, Phi, or Sine Space. You must have defined a near-field line, box, rectangle, or near-field sphere and at least one radiation or PML boundary to create a near-fields report. Field components and positions in reports are in the local coordinate system of the near field domain, not global coordinates.
Antenna Parameters	Whereas far field reports are computed as points around an infinite sphere, antenna parameters provide one value per quantity for the entire sphere. You must have defined an infinite sphere geometry and at least one radiation or PML boundary to create an antenna parameters report.
Emission Test	You can conduct an emission test under the same conditions as for a near field report except that.an emission test cannot be conducted for a ports-only solution. You must have defined a near-field line or near-field sphere and at least one radiation or PML boundary.

When you click **Product > Results** or right-click **Results** in the Project tree, or select the **Results** tab of the ribbon, one or more of the following report types and displays are available. *Available options depend on the nature of your installation and project(s)*.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



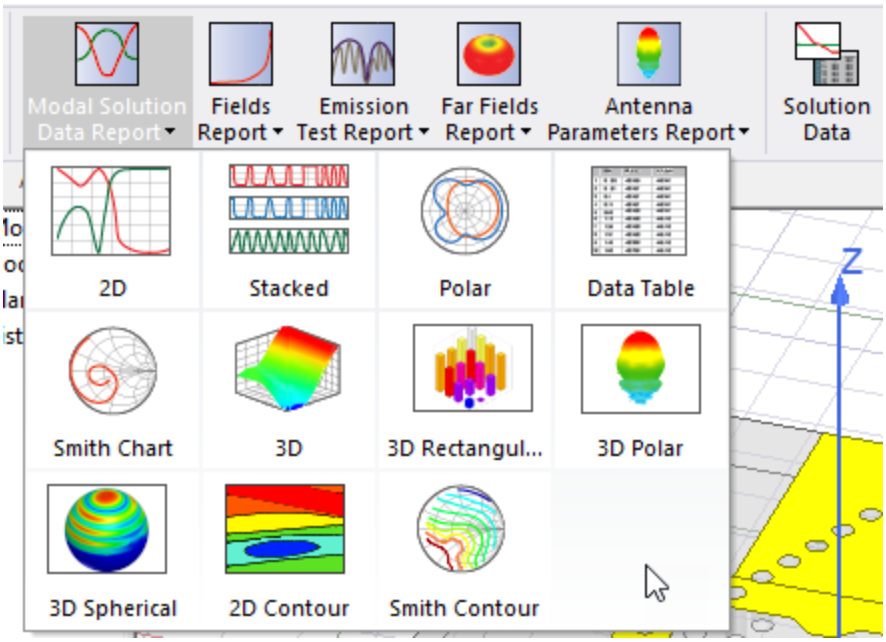
Report Type	Display Type
Create Standard Report	A Standard report can be depicted in any of the following display types: • Rectangular Plot • Data Table • Polar Plot • Smith Chart • Rectangular Stacked Plot • 3D Rectangular Plot • 3D Rectangular Bar Plot • 3D Polar Plot • 3D Spherical Plot • Rectangular Contour Plot • Smith Contour Plot
Create Eye Diagram Report	An Eye Diagram report can be depicted in any of the following display types: • Rectangular Plot • Data Table • Statistical Eye Plot

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Constellation Report	A Constellation report can be depicted in any of the following display types: • Rectangular Plot • Data Table
Create Statistical Eye Report	A Statistical Eye report can be depicted in any of the following display types: • Rectangular Plot • Data Table
Create Report From File	A Data File report can be created in any of the available display types using a previous report that has been saved to a file.

Selecting the Display Type

The information in a report can be displayed in several formats. When you select the **Results** tab in the ribbon, depending on reports available for a given design and Solution type, you see a selection of report type groups (for example, Modal Solution Data, or Fields Report, Emission Test, Antenna Parameters, Near Fields, Terminal Solution Report, Far Fields Report, Characteristic Mode Data, etc.), with drop-down menus that displays the kinds of reports available for the design in that group:



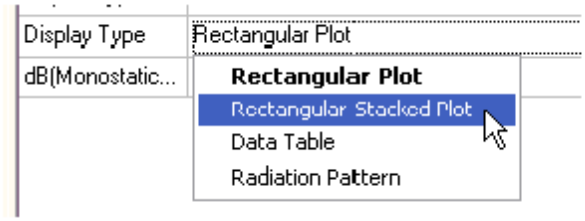
For the initial plot, you can also select from the following **Display Type** formats in the **Create*<type>* Report** submenu:

Rectangular Plot	A 2D rectangular (x-y) graph.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rectangular Stacked Plot	This choice puts each trace into its own 2D rectangular plot, and stacks each plot, rather than overlaying the traces on the same plot.
3D Rectangular Plot	A 3D rectangular (x-y-z) graph.
3D Rectangular Bar Plot	A 3D plot of rectangular bars.
Rectangular Contour Plot	A rectangular (x-y-z) graph. Contour plots are useful to visualize surfaces (e.g., Directivity as a function of phi/theta).
Polar Plot	A 2D circular chart divided by spherical coordinates.
3D Polar Plot	A 3D circular plot divided by spherical coordinates.
3D Spherical Plot	A 3D circular plot where the radius (rho) of the plot at all points is uniform and is equal to the maximum of all rhos at each (theta, phi) points.
Smith Chart	A 2D polar chart of S-parameters upon which a normalized impedance grid has been superimposed.
Smith Contour Plot	A polar chart. Contour plots are useful to visualize surfaces.
Data Table	A grid with rows and columns that displays, in numeric form, selected quantities against a swept variable or another quantity.
Radiation Pattern	A 2D polar plot of radiated fields.

You can also modify the display type of an existing plot from the **Properties** dialog for that plot. Select the Report icon in the Project tree to display the Properties dialog box. Selecting the **Display Type** field displays a menu with selections available for that plot.



Once you make a selection, the plot display updates for the current selection.

Creating 2D Rectangular Plots

A rectangular plot is a 2D, x-y graph of results.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. On the **Results** menu (under the **HFSS** menu, or right-click **Results** in the Project Manager), click **Create < *type* > Report**, and select **Rectangular Plot**.

   The *Report* dialog box appears.

2. In the **Context** section make selections from the following field or fields, depending on the design and solution type:
   a. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
   b. Domain field with a drop down selection list. Whether this field appears, and the domains listed, depend on the Solution type and the report <*type*> selected. For modal and terminal solution data reports, the domain can be **Sweep** or **Time**.

      Before you can examine the time domain, you must perform an Interpolating sweep for a driven solution (Modal or Terminal). If you select **Time**, the **TDR Options** button is enabled. Select it and follow the directions for time-domain plotting.

   c. Geometry field with a drop-down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
3. Under the **Trace** tab, **Y** component section, specify the information to plot along the y-axis:
   a. In the **Category** list, click the type of information to plot. The category selected provides the default plot name.
   b. In the **Quantity** list, click the value to plot.
   c. In the **Function** list, click the mathematical function of the quantity to plot.
   d. Value field displays the currently specified Quantity and Function. You can edit this field directly.

      > **Note:**
      >
      > Color shows valid expression.

   e. **Range Function** button -- opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.
4. On the **Trace** tab, **X** (Primary sweep) line, specify the quantity to plot along the x-axis in one of the following ways:
   - Select the sweep variable to use from the drop-down list.
   - If sweeps are available, you can select the browse button to display a dialog that lets you select particular sweep or sweeps, or all sweeps. The quantity will be plotted against the primary sweep variable listed.
5. On the **Families** tab, confirm or modify the sweep variables that will be plotted.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

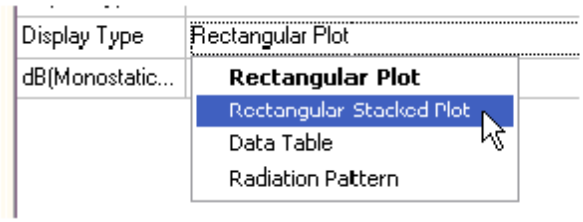
6. Click **New Report**.

   This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog.

   The function of the selected quantity will be plotted against the swept variable values or quantities you specified on an x-y graph. The plot is listed under **Results** in the project tree and the traces are listed under the plot. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plott *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

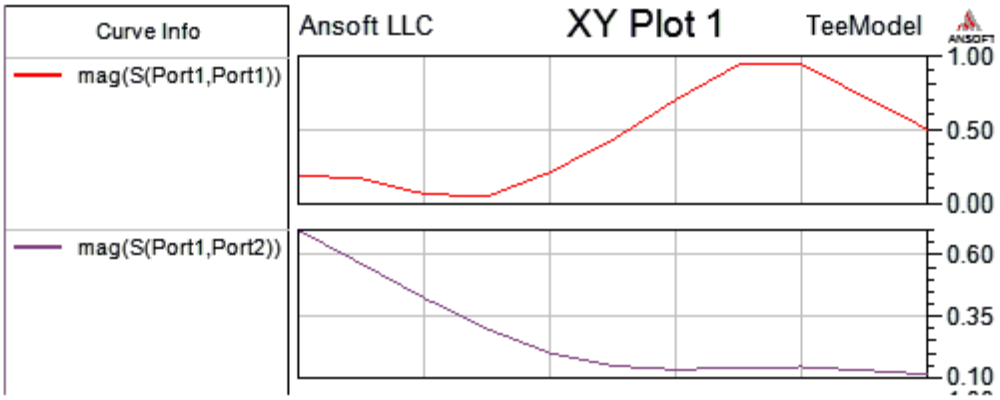
You can also modify the display type of an existing plot from the Properties dialog for that plot. Select the Report icon in the Project tree to display the Properties dialog box. Selecting the Display Type field displays a menu with selections available for that plot.



Once you make a selection, the plot display updates for the current selection.

Creating 2D Rectangular Stacked Plots

A rectangular stacked plot is a 2D, x-y graph of results, with each trace displayed on a separate plot.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. On the **Results** menu (under the **HFSS** menu or right-click **Results** in the Project Manager), click **Create <*type*> Report**, and select **Rectangular Stacked Plot**.

   The *Report* dialog box appears.

2. In the **Context** section make selections from the following field or fields, depending on the design and solution type:
   a. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
   b. Domain field with a drop-down selection list. Whether this field appears, and the domains listed, depend on the Solution type and the report **<*type*>** selected. For modal and terminal solution data reports, the domain can be **Sweep** or **Time**.

      Before you can examine the time domain, you must perform an Interpolating sweep for a driven solution (Modal or Terminal). If you select **Time**, the **TDR Options** button is enabled. Select it and follow the directions for time-domain plotting.

   c. Geometry field with a drop down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
3. Under the **Trace** tab, **Y** component section, specify the information to plot along the y-axis:
   a. In the **Category** list, click the type of information to plot. The category you select provides the default plot name.
      b. In the **Quantity** list, click the value to plot.
   c. In the **Function** list, click the mathematical function of the quantity to plot.
   d. Value field displays the currently specified Quantity and Function. You can edit this field directly.

      > **Note:**
      >
      > Color shows valid expression.

   e. **Range Function** button -- opens the *Set Range Function* dialog box. This applies currently specified Quantity and Function.
4. On the **Trace** tab, **X** (Primary sweep) line, specify the quantity to plot along the x-axis in one of the following ways:
   - Select the sweep variable to use from the drop down list.
   - If sweeps are available, you can select the browse button to display a dialog that lets you select particular sweep or sweeps, or all sweeps. The quantity will be plotted against the primary sweep variable listed.
5. On the **Families** tab, confirm or modify the sweep variables that will be plotted.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

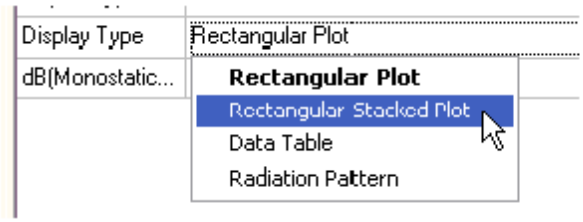
6. Click **New Report.**

   This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes.

   The function of the selected quantity will be plotted against the swept variable values or quantities you specified on an x-y graph. The plot is listed under **Results** in the project tree and the traces are listed under the plot. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

You can also modify the display type of an existing plot from the Properties dialog for that plot. Select the Report icon in the Project tree to display the Properties dialog box. Selecting the Display Type field displays a menu with selections available for that plot.

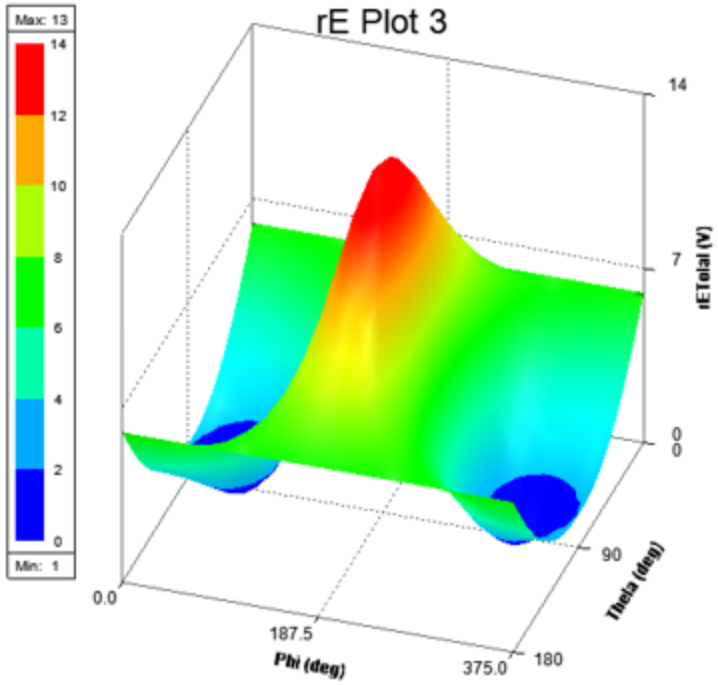


Once you make a selection, the plot display updates for the current selection.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating 3D Rectangular Plots

This is a 3D, x-y-z graph of results.



Working with a 3D Rectangular Plot

You can Rotate, Zoom, and Pan a plot. When you rotate, the Cartesian grid responds so that the curve always remains in front and the grids behind.

Clicking on a plot entity selects it, highlighting the selected entity in bold.

Double-clicking anywhere in the plot brings up the Properties dialog box. The properties are grouped appropriately under various tabs, which correspond to plot entities:

- General: For general plot properties such as Visual Detail level and background color
- Header: Properties related to plot Header/Title.
- Axis [X|Y|Z]: Properties related to the 3 axes
- Grid [XY|YZ|ZX]: Properties related to the 3 grids
- ColorKey: Properties related to ColorKey, including borders, background, Min and Max, as well as number format and precision.
- Contour: Properties related to contouring of all curves/surfaces
- Surface: Properties related to the curve

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting a property also displays its properties in the Property window. You can edit the properties to customize the appearance of the plot. See "Controlling Visual Detail in a 3D Plot" on the facing page.

Creating a 3D Rectangular Plot

1. On the **Results** menu (under the **HFSS** menu or right-click **Results** in the Project Manager), click **Create *<type>* Report**, and select **3D Rectangular plot** from the report type menu.

   The *Report* dialog box appears.

2. In the **Context** section make selections from the following field or fields, depending on the design and solution type.
   a. Solution field with a drop-down selection list. This lists the available solutions, whether sweeps or adaptive passes.
   b. Geometry field with a drop-down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
3. Under the **Trace** tab **Z** Component area, specify the information to plot along the z-axis:
   a. In the **Category** list, click the type of information to plot. The category you select provides the default plot name.
   b. In the **Quantity** list, click the value to plot.
   c. In the **Function** list, click the mathematical function of the quantity to plot.
   d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

      > **Note:**
      >
      > Color shows valid expression.

   e. **Range Function** button -- opens the *Set Range Function* dialog box. This applies currently specified Quantity and Function.
4. On the **Trace** tab **Y** (Secondary sweep) lines, specify the information to plot along the y-axis in one of the following ways:
   - Select the sweep variable to use from the drop-down list.
   - If sweeps are available, you can select the browse button to display a dialog that lets you select particular values. The quantity will be plotted against the primary sweep variable listed.
5. On the **Trace** tab **X** (Primary sweep) lines, specify the information to plot along the x-axis in one of the following ways:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select the sweep variable to use from the drop-down list.
- If sweeps are available, you can select the browse button to display a dialog that lets you select particular values. The quantity will be plotted against the primary sweep variable listed.

6. On the **Families** tab, confirm or modify the sweep variables that will be plotted.
7. Click **New Report.**

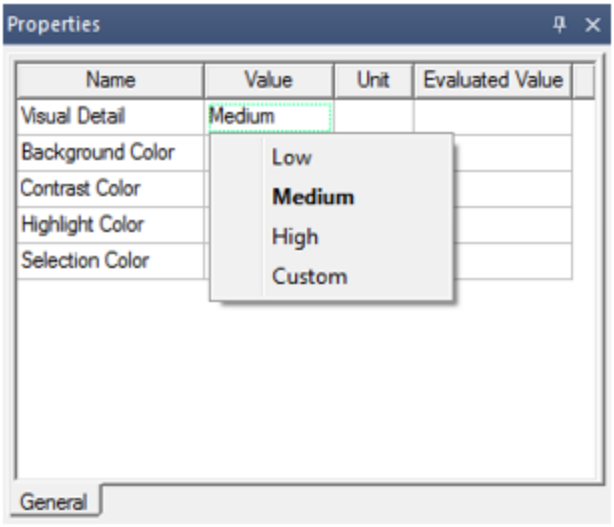
   This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes.

   The function of the selected quantity or quantities will be plotted against the values you specified on an x-y-z graph. The plot is listed under **Results** in the Project Manager. When you select the traces or plots, axis or grid labels, plot header, color key, or variable labels, their properties are displayed in the Properties window. The properties for each plot element can be edited directly to modify the plot content and appearance. See Modifying the Background Properties of a Report.

8. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

Controlling Visual Detail in a 3D Plot

If a particular plot seems busy with information, you can edit plot properties, such as Axis and Grid Attributes for discrete levels of visual detail to improve readability. Double-click anywhere on a plot to display the Properties dialog box. The Visual Detail property on the General tab also provides control suited to different screen and plot sizes.

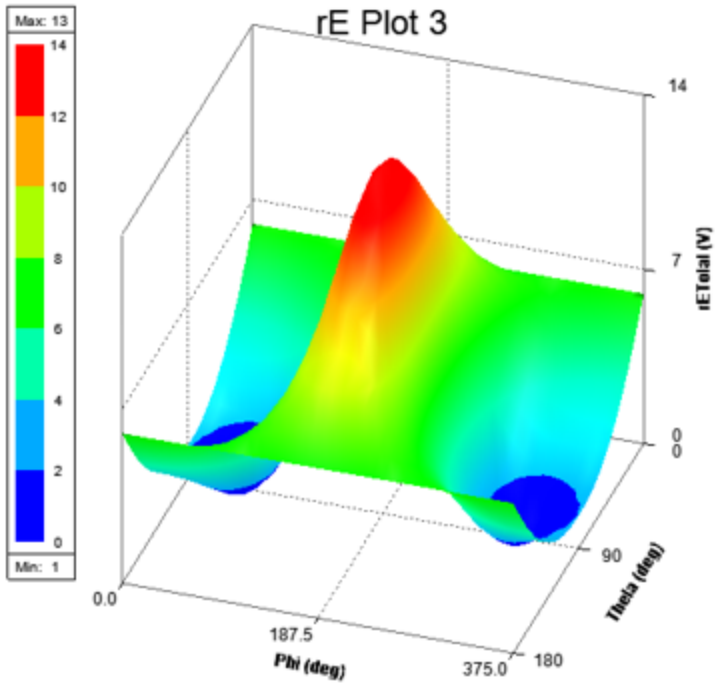


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Visual Detail menu has four options: Low, Medium (the default), High, and Custom. If you select any Visual Detail, the 3D plot is rendered according to the selected Visual Detail level and the properties reflect the values chosen for the selected visual detail level. From this predefined visual detail level, if you modify any properties, Visual Detail is automatically set to Custom (or to another predefined visual detail level if the edits happen to match the settings for that level).

You can also manually set Visual Detail to Custom. In such a case, Custom will inherit property values corresponding to the previous level. This ensures that you can customize settings starting from a baseline provided by the preconfigured Low, Medium or High Visual Detail levels.

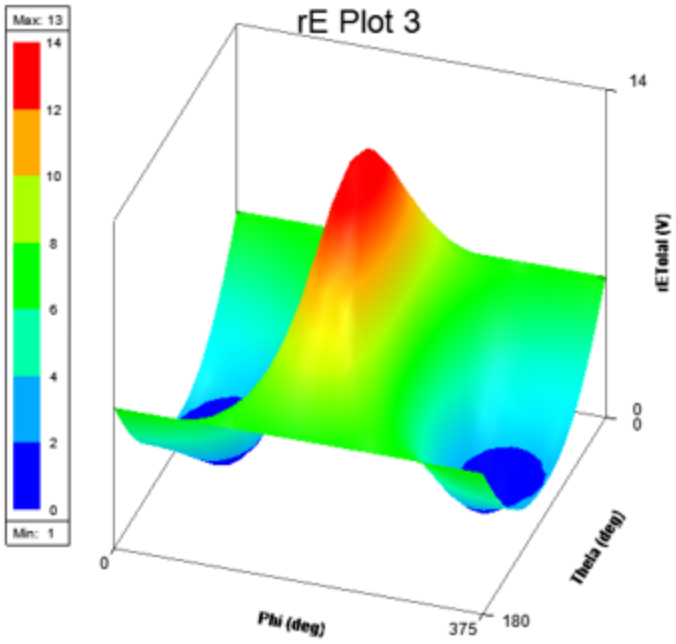
3D Rectangular Plot with Medium Visual Detail



On creation, a 3D Rectangular Plot has Visual Detail set to Medium and looks and feels as shown above. Specifically, under Medium Visual Detail level, a 3D Rectangular Plot has 3 ticks per axis (X, Y, Z axis) which will show min, max and middle value. This setting also shows axes labels.

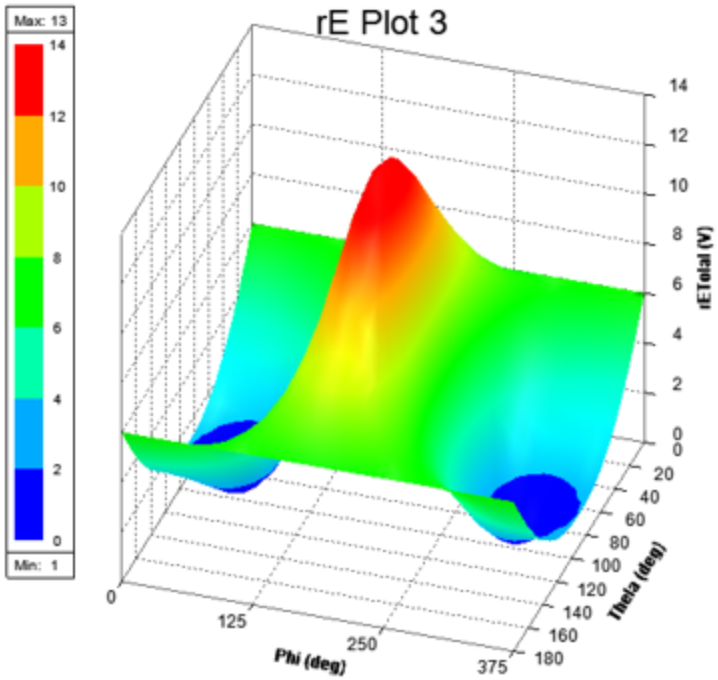
3D Rectangular Plot with Low Visual Detail

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With Visual Detail set to Low, a 3D Rectangular Plot shows axes with 2 ticks corresponding to min and max values. It also shows axes labels and grid borders.

3D Rectangular Plot with High Visual Detail

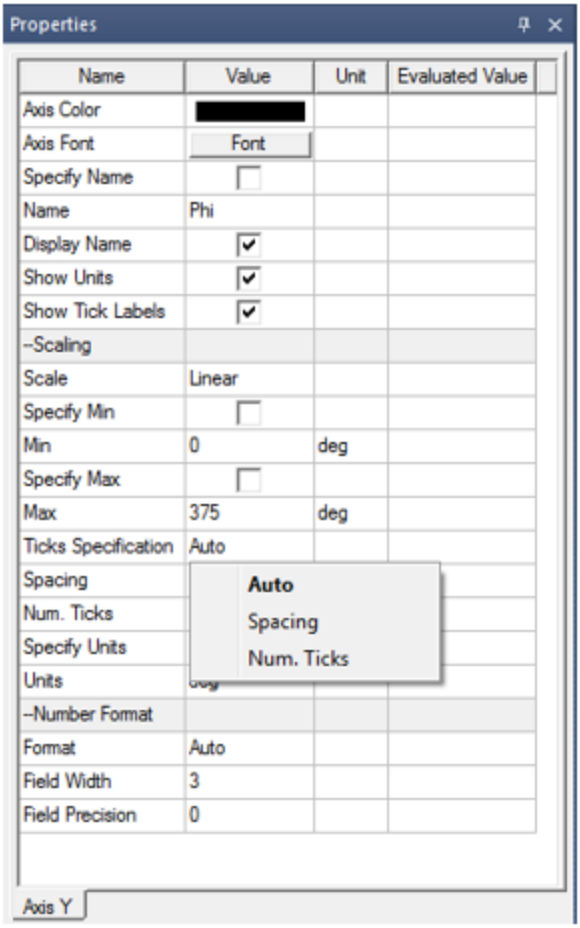


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

With Visual Detail set to High, a 3D Rectangular Plot shows all Cartesian axes and grids together with all ticks and axes labels.

Axis Properties: Ticks Specification and Num. Ticks

Ticks Specification is available on Axis properties, as shown below:



Ticks Specification is a menu with possible values as Auto, Spacing, and Num. Ticks, with Auto being the default value. If Ticks Specification is Auto, then a spacing value is automatically calculated and used to calculate and display the tick labels. **Spacing** shows the calculated value, and Num. Ticks shows the number of ticks based on this spacing value, as shown below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

--Scaling			
Scale	Linear		
Specify Min	☐		
Min	0	deg	
Specify Max	☐		
Max	375	deg	
Ticks Specification	Auto		
Spacing	125	deg	
Num. Ticks	2		

You can edit the **Spacing** field when Ticks Specification is set to Spacing; otherwise, it is read only.

You can edit the **Num. Ticks** field when Ticks Specification is Num. Ticks; otherwise, it is read only.

Valid Num. Ticks are between 0 and 100, including 0 and 100. If you enter an invalid value, an error message is shown. If you enter a spacing value that results in number of ticks greater than 100, then an appropriate value is shown.

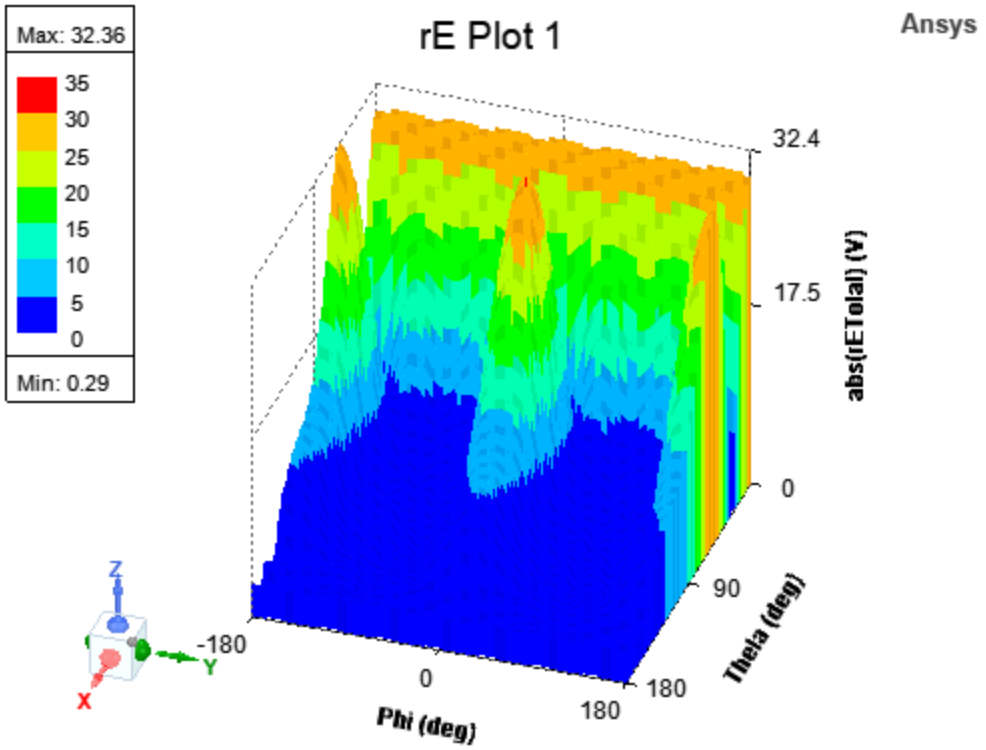
- If Num. Ticks is 0, then no ticks are shown on the axis.
- If Num. Ticks is 1, then only the max value tick is shown on the axis.
- If Num. Ticks is 2, then only the min and max value ticks are shown on the axis.
- If Num. Ticks is greater than 2, then evenly spaced ticks (including min and max) are shown on the axis.

> **Note:**
>
> With the addition of the Ticks Specification property to Axis properties, the **Specify Spacing** property was removed as an Axis property.
>
> - If an R18.0 or R18.1 project is opened with **Specify Spacing** as Unchecked, Ticks Specification is set to Auto.
> - If an R18.0 or R18.1 project is opened with **Specify Spacing** as Checked, Ticks Specification is set to Spacing.

Creating 3D Rectangular Bar Plots

This is a 3D, x-y-z graph of results as rectangular bars.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



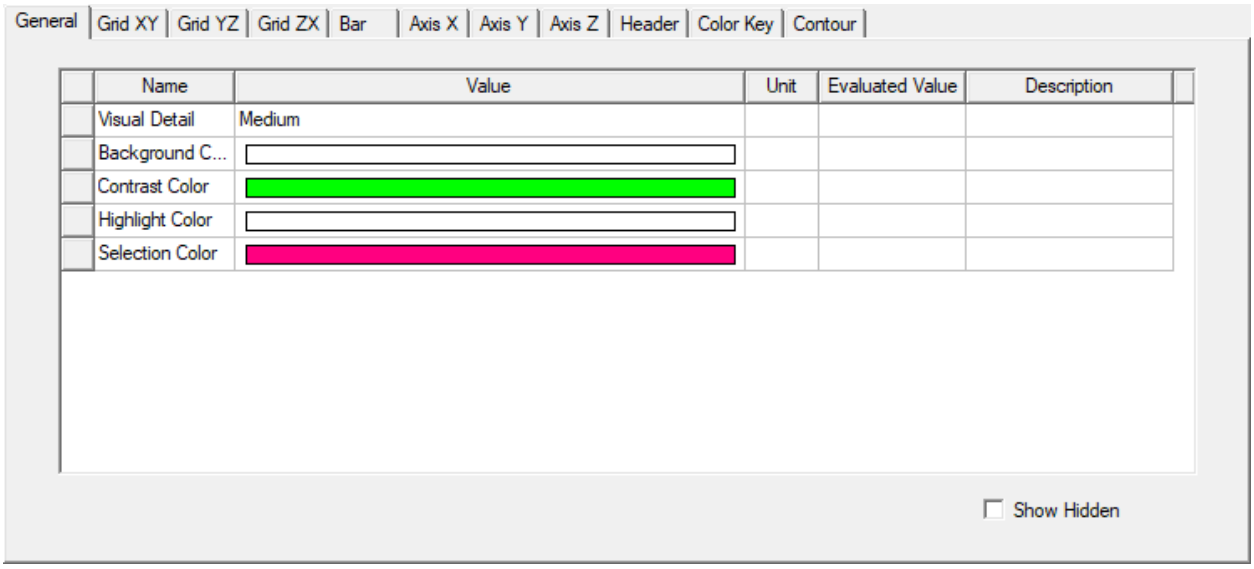
Working with a 3D Rectangular Plot

You can Rotate, Zoom, and Pan a plot. When you rotate, the Cartesian grid responds so that the curve always remains in front and the grids behind.

Clicking on a plot entity selects it, highlighting the selected entity in bold.

Double-clicking anywhere in the plot brings up the **Properties** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The properties tabs for a 3D Rectangular Bar plot are:

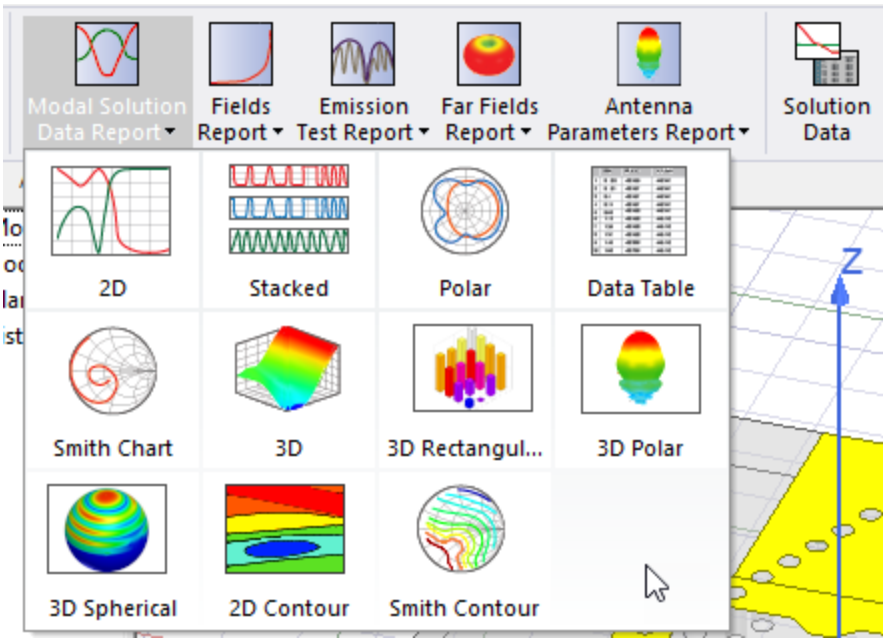
- General: For general plot properties such as Visual Detail level and background color.
- Grid [XY|YZ|ZX]: Properties related to the 3 grids.
- Bar: Properties related to rectangular bars.
- Axis [X|Y|Z]: Properties related to the 3 axes.
- Header: Properties related to plot Header/Title.
- ColorKey: Properties related to ColorKey, including borders, background, Min and Max, as well as number format and precision.
- Contour: Properties related to contouring of all surfaces.

Selecting a property also displays its properties in the **Property** window. You can edit the properties to customize the appearance of the plot. See: "Controlling Visual Detail in a 3D Rectangular Bar Plot" on page 23-104.

Creating a 3D Rectangular Bar Plot

1. Access the Results menu one of two ways:
   - From the **HFSS** menu.
   - From the **Project Manager**, right-click **Results**.
2. From the **Results** menu, click **Create *<type>* Report**, and select **3D Rectangular Bar plot**, or click the **Results** ribbon tab and select the 3D Rectangular Bar icon for the Report type you want to create.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Report** dialog box appears.

3. In the **Context** section make selections from the following field or fields, depending on the design and solution type.
    a. Solution field with a drop-down selection list. This lists the available solutions, whether sweeps or adaptive passes.
    b. Geometry field with a drop-down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
4. Under the **Trace** tab **Z** Component area, specify the information to plot along the z-axis:
    a. In the **Category** list, click the type of information to plot. The category you select provides the default plot name.
    b. In the **Quantity** list, click the value to plot.
    c. In the **Function** list, click the mathematical function of the quantity to plot.
    d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

> **Note:**
>
> Color shows valid expression.

    e. **Range Function** button -- opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. On the **Trace** tab **Y** (Secondary sweep) lines, specify the information to plot along the y-axis in one of the following ways:
   - Select the sweep variable to use from the drop-down list.
   - If sweeps are available, you can select the browse button to display a dialog that lets you select particular values. The quantity will be plotted against the primary sweep variable listed.
6. On the **Trace** tab **X** (Primary sweep) lines, specify the information to plot along the x-axis in one of the following ways:
   - Select the sweep variable to use from the drop-down list.
   - If sweeps are available, you can select the browse button to display a dialog box that lets you select particular values. The quantity will be plotted against the primary sweep variable listed.
7. Click **New Report.**

   This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes.

   The function of the selected quantity or quantities will be plotted against the values you specified on an x-y-z graph. The plot is listed under **Results** in the Project Manager. When you select the traces or plots, axis or grid labels, plot header, color key, or variable labels, their properties are displayed in the Properties window. The properties for each plot element can be edited directly to modify the plot content and appearance. See Modifying the Background Properties of a Report.

8. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

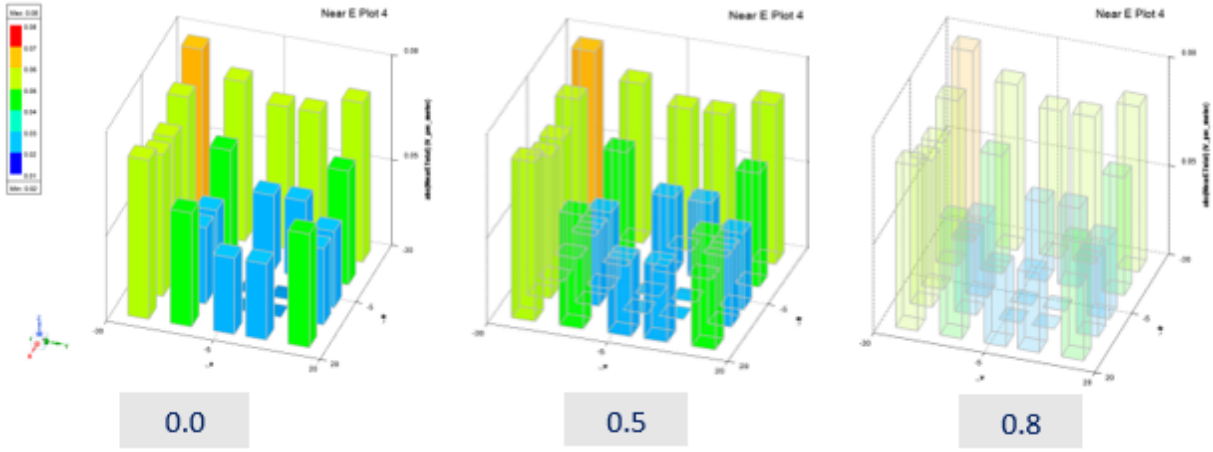
3D Rectangular Bar Customization

Selecting a bar in the plot shows the customizable attributes of bars in a 3D Rectangular Bar plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

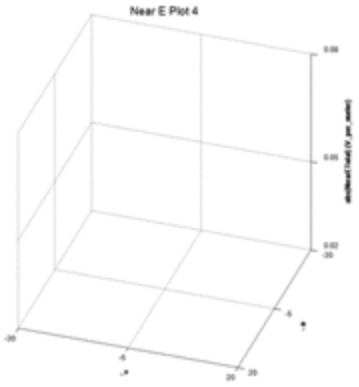
The customizable attributes of the bars in a 3D Rectangular Bar plot include:

- Transparency

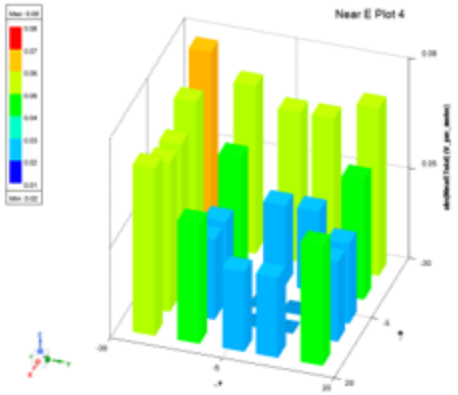


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Filled

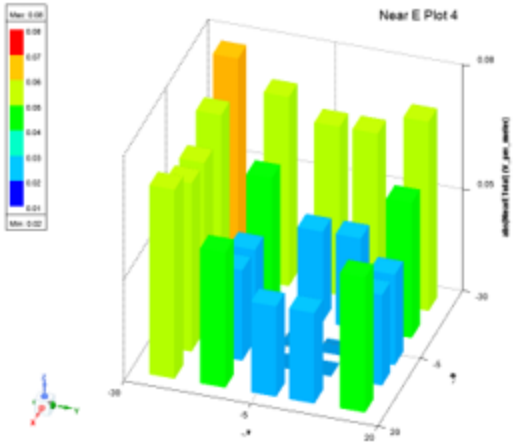


Filled [ ]

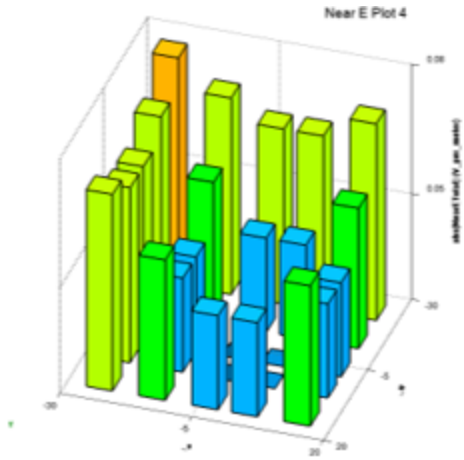


Filled [√]

- Show Outline



Show Outline []

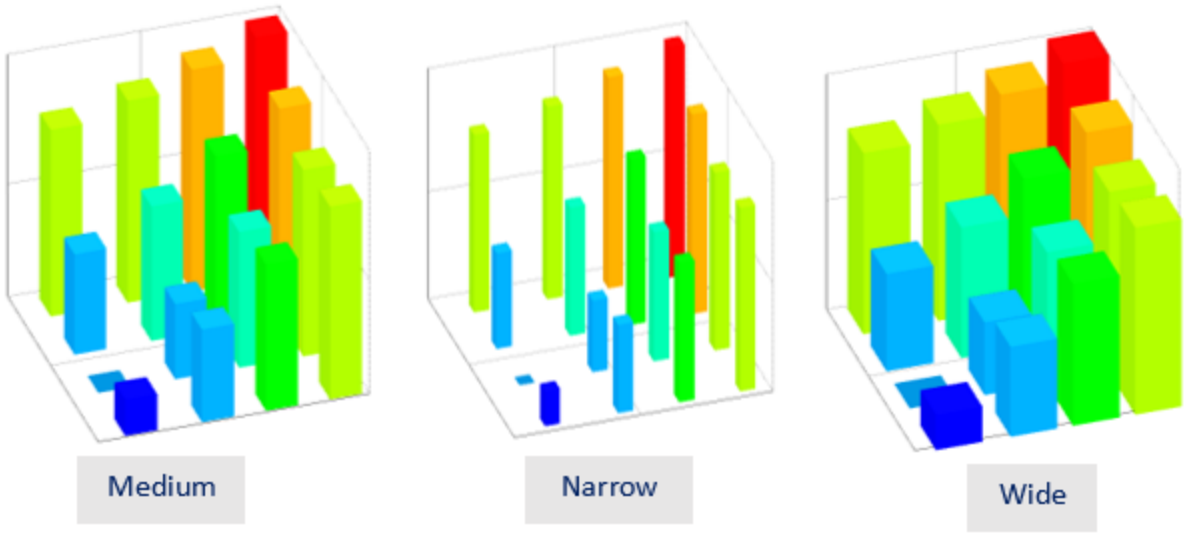


Show Outline [√]

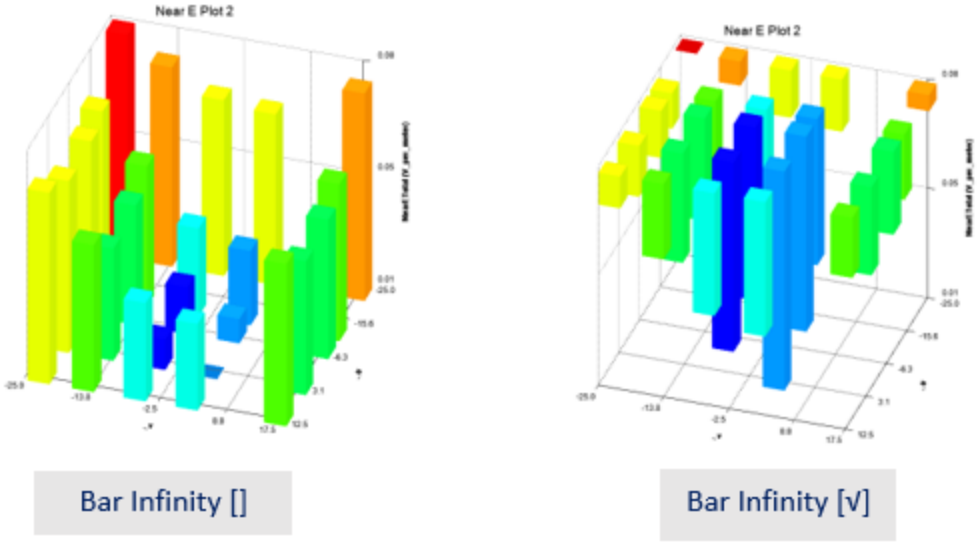
- Outline Color (customizable, see plot above)
- Outline Width (customizable, see plot above)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Bar Width, controls the thickness of the bar. Medium (default), Wide and Narrow are the width that equals to ½, ¾ and ¼ of the min spacing between adjacent points respectively.



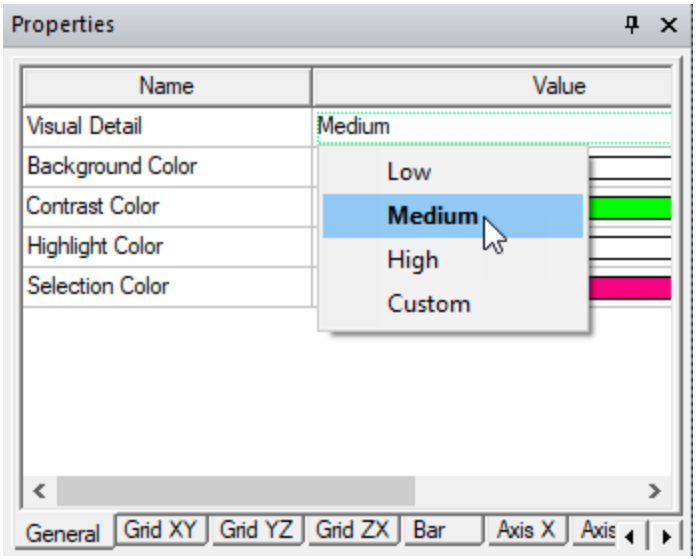
- Bar Infinity represents the third coordinate (z) using the bottom of the bar.



Controlling Visual Detail in a 3D Rectangular Bar Plot

If a particular plot seems busy with information, you can edit plot properties, such as **Axis** and **Grid Attributes** for discrete levels of visual detail to improve readability. Double-click anywhere on a plot to display the **Properties** dialog box. The **Visual Detail** property on the **General** tab also provides control suited to different screen and plot sizes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

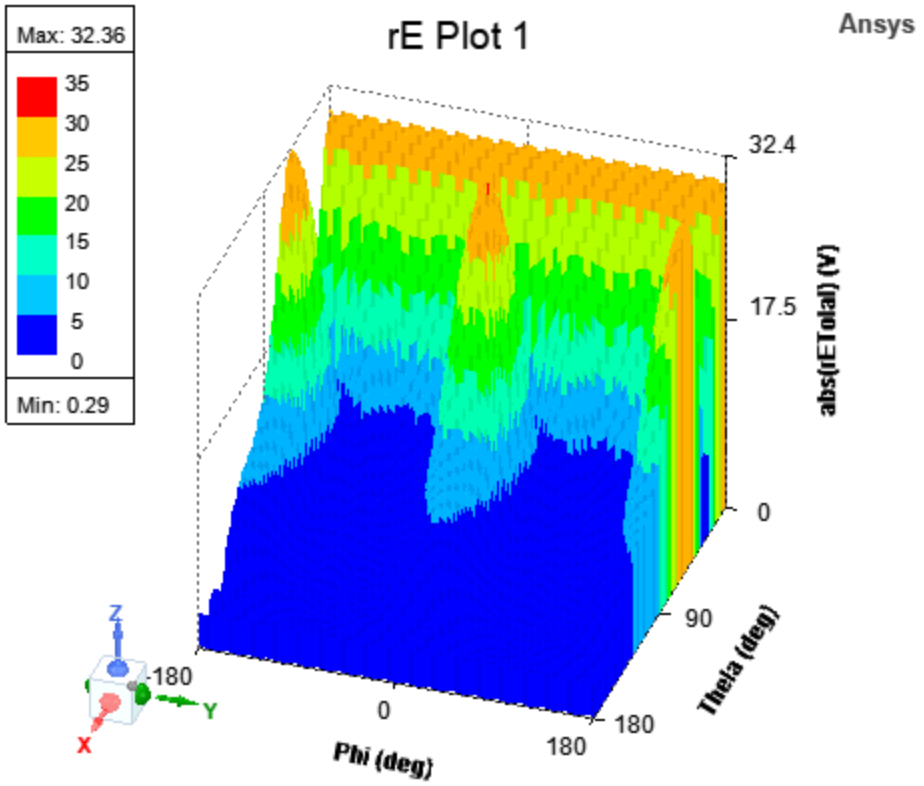


The **Visual Detail** menu has four options: Low, Medium (the default), High, and Custom. If you select any Visual Detail, the 3D plot is rendered according to the selected Visual Detail level and the properties reflect the values chosen for the selected visual detail level. From this predefined visual detail level, if you modify any properties, Visual Detail is automatically set to Custom (or to another predefined visual detail level if the edits happen to match the settings for that level).

You can also manually set Visual Detail to Custom. In such a case, Custom will inherit property values corresponding to the previous level. This ensures that you can customize settings starting from a baseline provided by the preconfigured Low, Medium or High Visual Detail levels.

3D Rectangular Plot with Medium Visual Detail

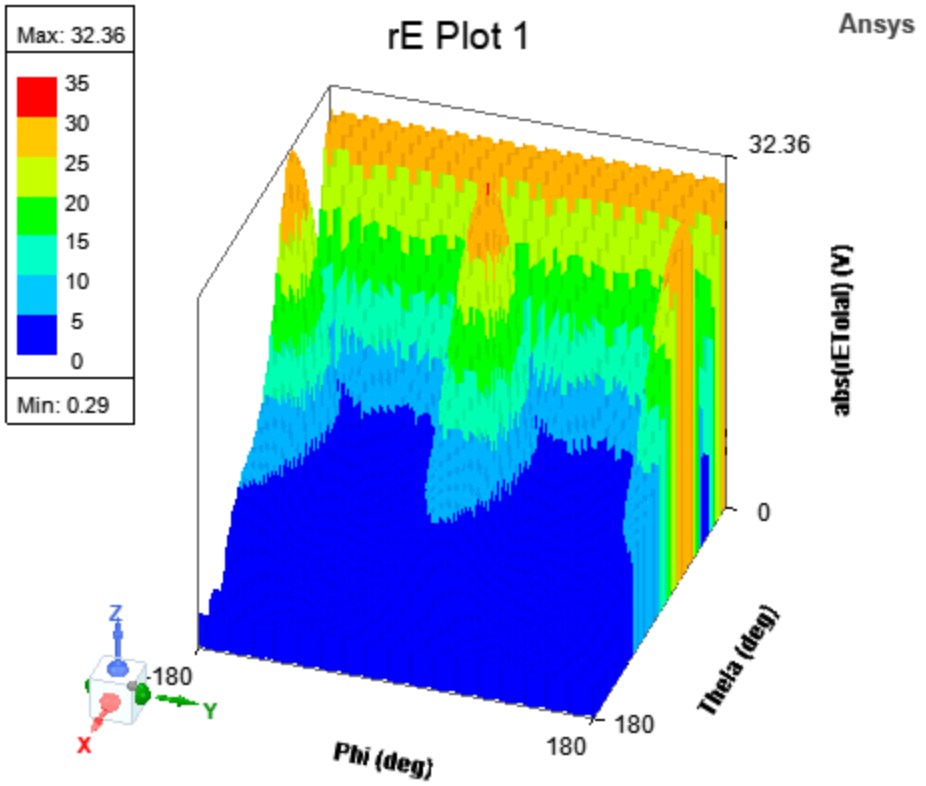
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



On creation, a 3D Rectangular Plot has Visual Detail set to Medium and looks and feels as shown above. Specifically, under Medium Visual Detail level, a 3D Rectangular Plot has 3 ticks per axis (X, Y, Z axis) which will show min, max and middle value. This setting also shows axes labels.

3D Rectangular Bar Plot with Low Visual Detail

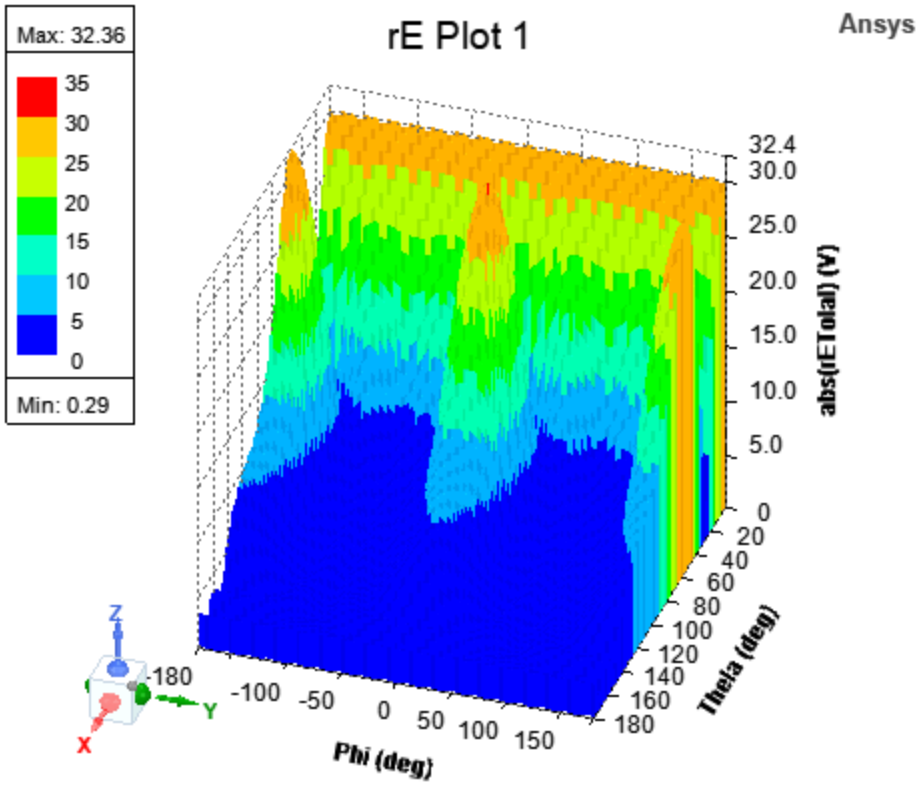
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With Visual Detail set to Low, a 3D Rectangular Plot shows axes with 2 ticks corresponding to min and max values. It also shows axes labels and grid borders.

3D Rectangular Bar Plot with High Visual Detail

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

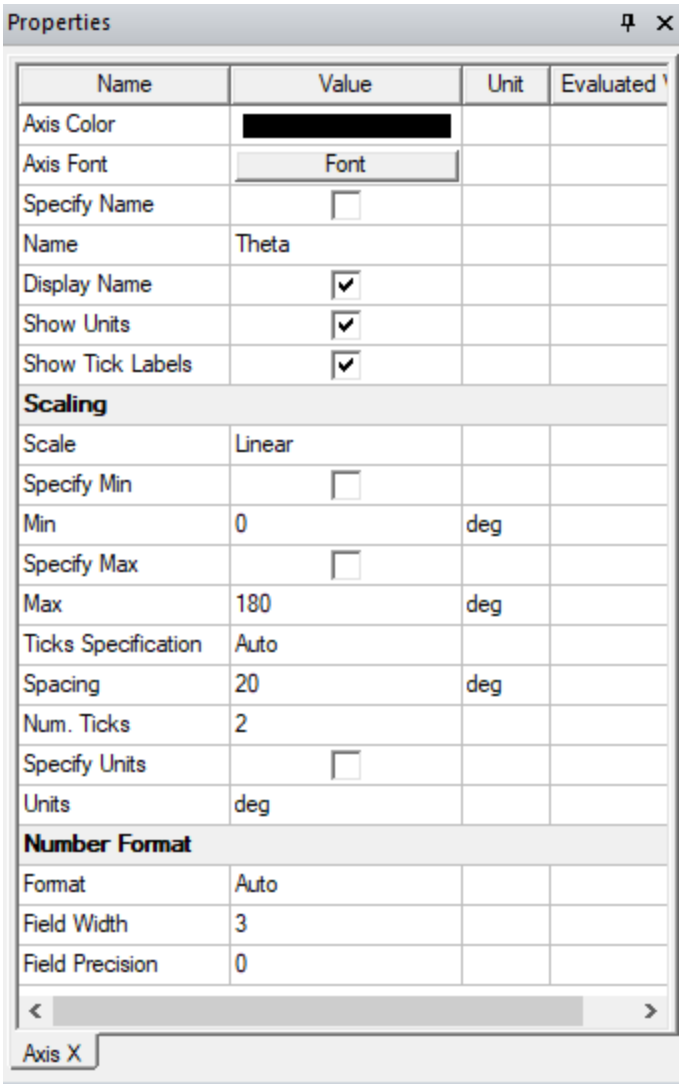


With Visual Detail set to High, a 3D Rectangular Plot shows all Cartesian axes and grids together with all ticks and axes labels.

Axis Properties: Ticks Specification and Num. Ticks

Ticks Specification is available on Axis properties, as shown below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ticks Specification is a menu with possible values as Auto, Spacing, and Num. Ticks, with Auto being the default value. If Ticks Specification is Auto, then a spacing value is automatically calculated and used to calculate and display the tick labels. **Spacing** shows the calculated value, and Num. Ticks shows the number of ticks based on this spacing value, as shown below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

--Scaling			
Scale	Linear		
Specify Min	☐		
Min	0	deg	
Specify Max	☐		
Max	375	deg	
Ticks Specification	Auto		
Spacing	125	deg	
Num. Ticks	2		

You can edit the **Spacing** field when Ticks Specification is set to Spacing; otherwise, it is read only.

You can edit the **Num. Ticks** field when Ticks Specification is Num. Ticks; otherwise, it is read only.

Valid Num. Ticks are between 0 and 100, including 0 and 100. If you enter an invalid value, an error message is shown. If you enter a spacing value that results in number of ticks greater than 100, then an appropriate value is shown.

- If Num. Ticks is 0, then no ticks are shown on the axis.
- If Num. Ticks is 1, then only the max value tick is shown on the axis.
- If Num. Ticks is 2, then only the min and max value ticks are shown on the axis.
- If Num. Ticks is greater than 2, then evenly spaced ticks (including min and max) are shown on the axis.

> **Note:**
>
> With the addition of the Ticks Specification property to Axis properties, the **Specify Spacing** property was removed as an Axis property.
>
> - If an R18.0 or R18.1 project is opened with **Specify Spacing** as Unchecked, Ticks Specification is set to Auto.
> - If an R18.0 or R18.1 project is opened with **Specify Spacing** as Checked, Ticks Specification is set to Spacing.

Creating Rectangular Contour Plots

This is an x-y-z graph of results. Any data that you can currently plot in 3D (as 3D Cartesian or 3D polar) is a candidate for a contour plot.

1. On the **Results** menu (**HFSS** menu or right-click **Results** in the Project Manager), click **Create *<type>* Report**, and select **Rectangular Contour plot** from the report type menu.

   The *Report* dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the **Context** section make selections from the following field or fields, depending on the design and solution type:
    a. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
    b. Geometry field with a drop-down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
    c. Domain field with a drop-down selection list. For Near and Far Field reports, if you have defined the respective Radiation Setups, a Domain field lists Theta, Phi, and Sine Space.

    For details, see Creating Sine Space plots.
3. Under the **Trace** tab **Z** Component area, specify the information to plot as contours:
    a. In the **Category** list, click the type of information to plot. The selected Category provides the default plot name.
    b. In the **Quantity** list, click the value to plot.
    c. In the **Function** list, click the mathematical function of the quantity to plot.
    d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

    > **Note:**
    >
    > Color shows valid expression.

    e. **Range Function** button -- opens the *Set Range Function* dialog box. This applies currently specified Quantity and Function.
4. On the **Trace** tab **Y** (Secondary sweep) lines, specify the information to plot along the y-axis in one of the following ways:
    - Select the sweep variable to use from the drop-down list.
    - If sweeps are available, you can select the browse button to display a dialog that lets you select particular values. The quantity will be plotted against the primary sweep variable listed.
5. On the **Trace** tab **X** (Primary sweep) lines, specify the information to plot along the x-axis in one of the following ways:
    - Select the sweep variable to use from the drop-down list.
    - If sweeps are available, you can select the browse button to display a dialog that lets you select particular values. The quantity will be plotted against the primary sweep variable listed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Click **New Report**.

   This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, for example, S Parameter Plot *n* or Output Variables Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes.

   The function of the selected quantity or quantities will be plotted against the values you specified on an x-y-z graph. The plot is listed under **Results** in the Project Manager. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

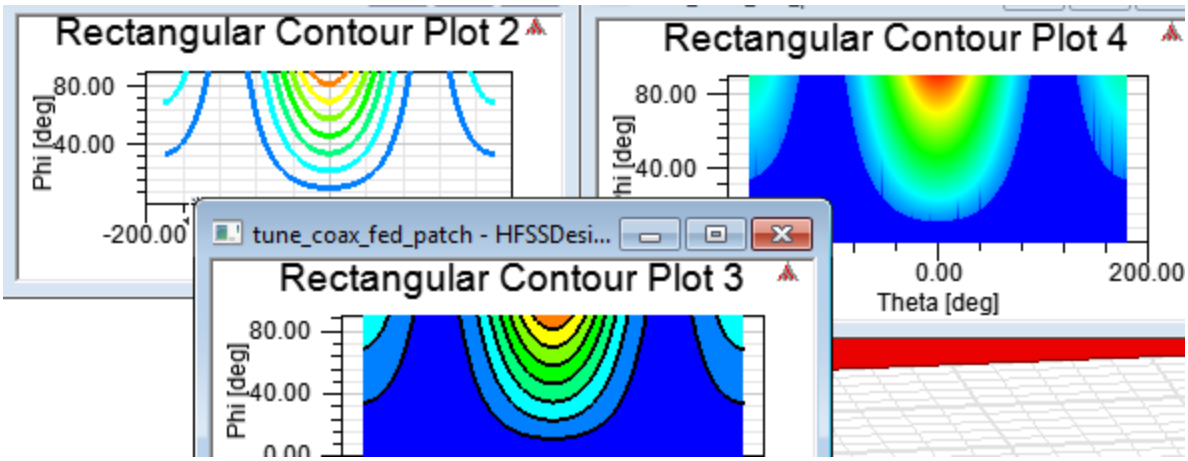
   The Trace Properties for Cartesian plots include tabs for:

   - Attributes tab, including Name, Line Style, Line Width, and Color.
   - Contour tab, including Colormap Type, Color, Spectrum, IsoValType, whether to Overlay Contour Lines (on Fringe or Tone plots), Scaling parameters, and Number of contours and spacing.

   Colormap Type can be Spectrum, Ramp, or Uniform.

   Spectrum can be Rainbow, Temperature, Magenta, or Gray

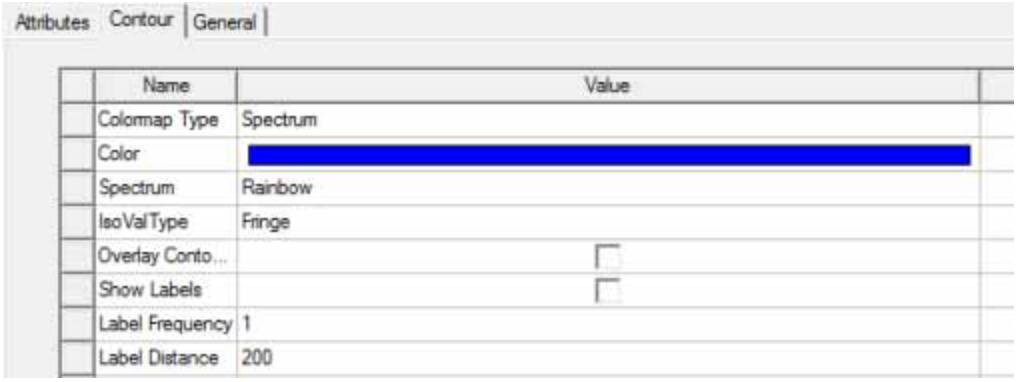
   IsoValType can be Line, Fringe, or Tone.



   General tab, including Back Color, Plot Area Color, Enable Y Axis Stripes, Field Width, Prevision, and whether to Use Scientific Notation.

   You can also access these properties by double-clicking on the Contour plot and viewing the **Contour** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

Creating Sine Space Plots

Sine Space plots provide an alternate way to view radiated field quantities. The transformation from theta, phi space to u, v sine space is defined as follows:

u = sin(theta)*cos(phi)

v = sin(theta)*sin(phi)

Note that this is similar to a polar coordinate system, where the radius is given by sin(theta) and the angle is given by phi. Another way to think about this: Given theta and phi that define a unit sphere, then these u,v coordinates are like viewing the unit sphere from above or below the XY plane. So, a sine space plot is a 2D plot that lies on a unit circle in the u,v space, that shows the values of radiated field quantities projected onto the upper or lower half of a full 3D sphere.

Radiated Fields in HFSS

In HFSS, you create Radiated field setups to specify how radiated fields will be calculated. For a sphere, you provides ranges and step size for theta and phi, or Az over El or El over Az, and can also specify a reference coordinate system and custom radiation surface. The radiated fields are calculated based on these settings, using algorithms that are designed for uniform sampling in theta and phi. If you manually enter Phi or Theta component of a far field quantity when you using Az/El or El/Az far field definition, then the Report uses the Z axis of the Az/El or El/Az coordinate system definition as pole to calculate theta/phi component for rE.

Once radiated field setups have been defined, it possible to create various radiated fields plots in Reporter.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sine Space Plots

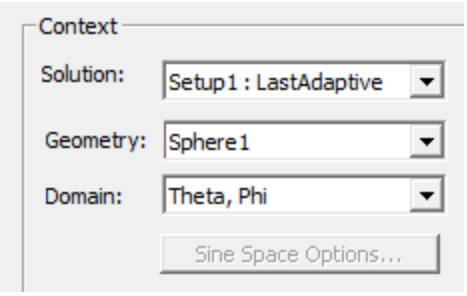
Sine Space plots are implemented as a new Context in Reporter. They will leverage the existing radiated field setups and calculations.

To create a Sine Space plot, after you have specified radiated field Setups.

1. Click **Results> Create Far Fields Report> Rectangular Contour Plot** or **Data Table**

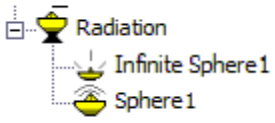
   Results> Create Near Fields Report> Rectangular Contour Plot or **Data Table**.

   The *Report* dialog box displays and shows choices for Domain in the Context field.



   "Theta, Phi" is selected by default and gives the existing functionality.

   Notice that the Context shown above already includes selection of a Geometry, which corresponds to a Radiation setup.



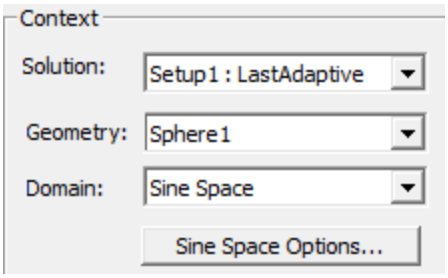
   This radiation setup is used to calculate the fields for the Sine Space plot. Internally, this computes the values of the complex E vector based on the theta, phi sampling in the radiation setup. When data is requested for a sine space plot, it will correspond to some arbitrary theta, phi, and the fields will be interpolated to this location on the sphere. Note that this interpolation will occur on the complex E vector, and can be more detailed than a simple linear interpolation between sample points.

   In order to support sine space plots, the radiation setup must have appropriate sampling in theta and phi. By default, a sphere is defined with phi from 0 to 360 (angle of rotation about Z), and theta from 0 to 180 (angle of rotation away from Z). These settings support a full unit circle representing the upper (theta from 0 to 90) or lower (theta from 90 to 180) half space. If the range of theta is reduced to 0 to 90, it would be possible to support upper half

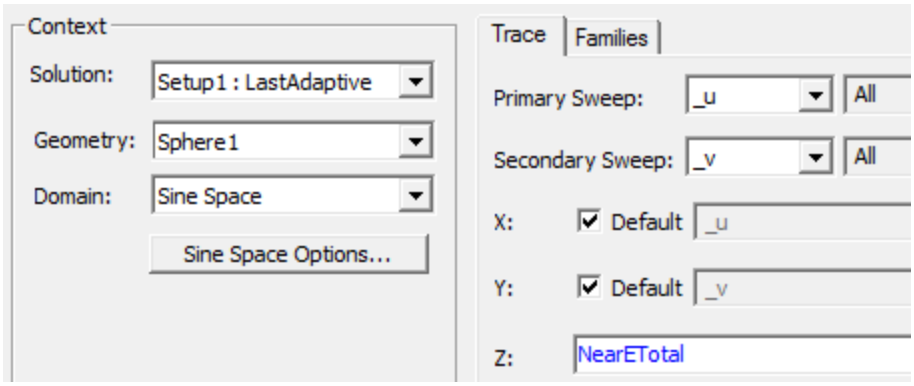
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

space only. Other ranges of theta and phi will likely create odd sine space plots, such as a partial circle, and these are not allowed.

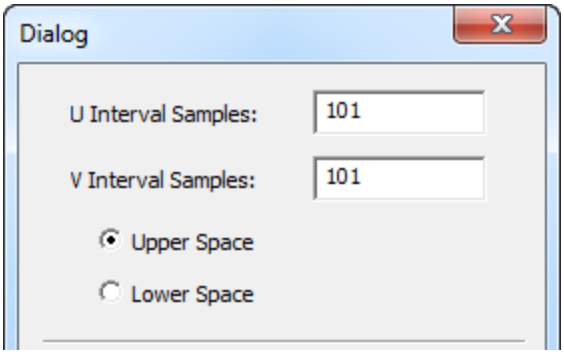
2. Select Sine Space as the Domain. This also enables the **Sine Space** options button.



Selecting "Sine Space" changes the Primary Sweep to U, and the Secondary Sweep to V.



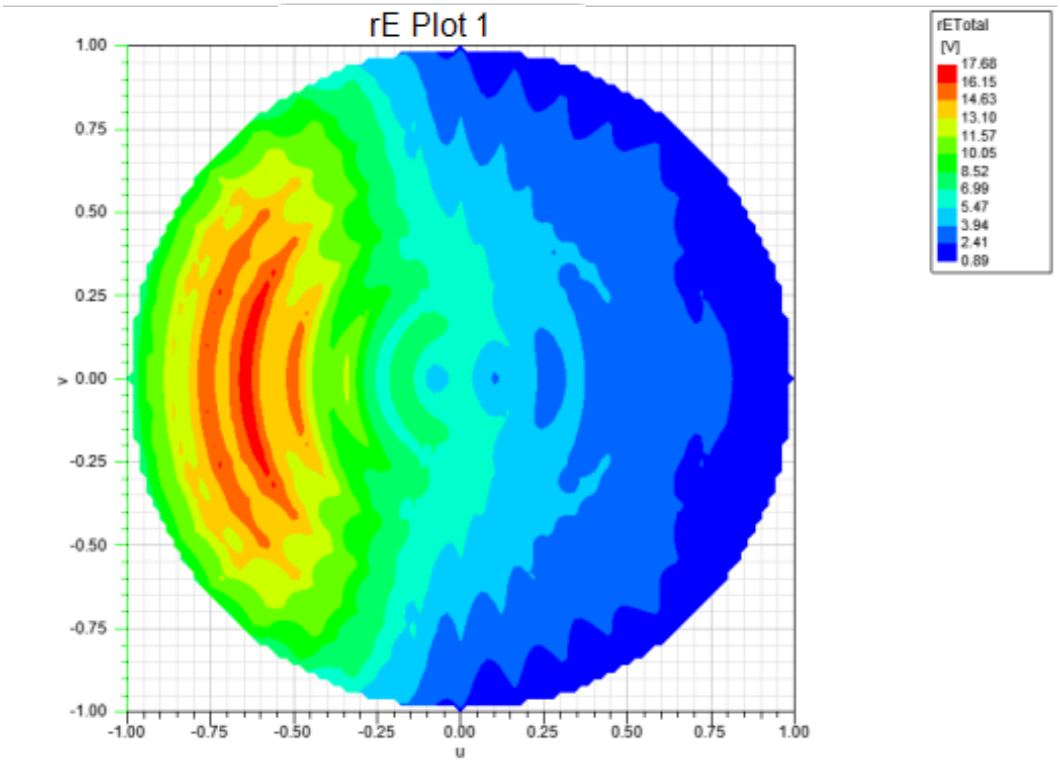
The Sine Space Options provide additional settings, including the sampling interval for the u,v coordinates (default of .05 would provide 40 samples over each axis, -1 to 1) and selection of upper or lower half space (default to upper).



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

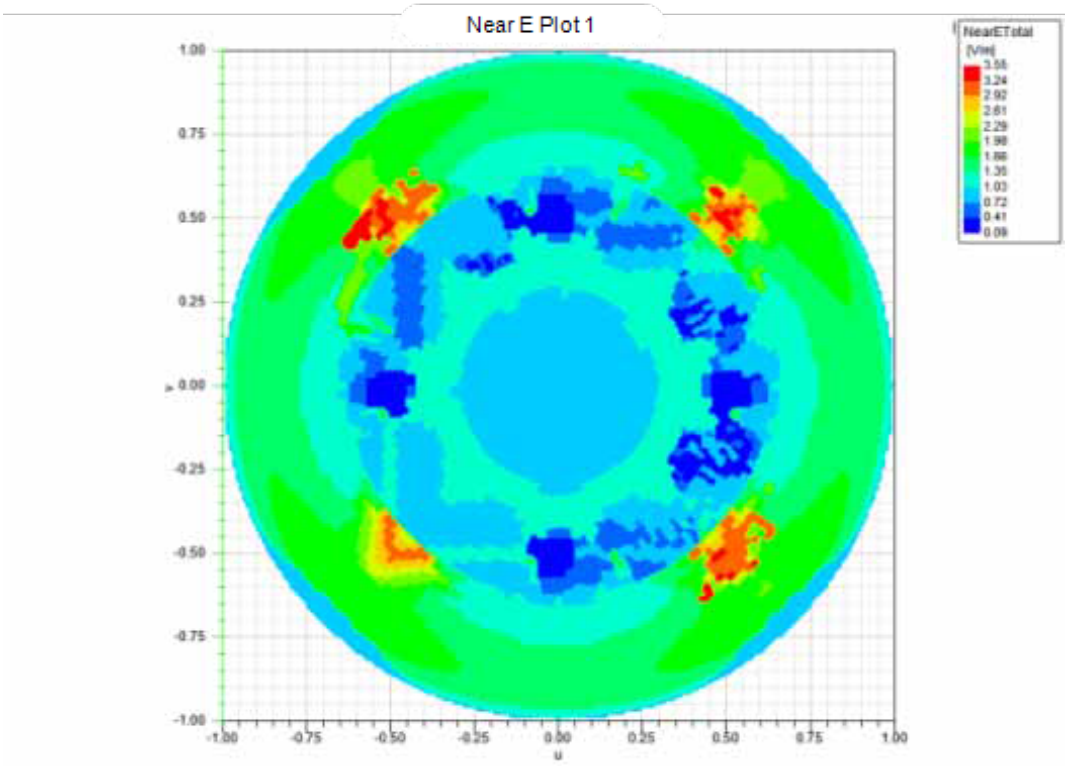
Once you have set up the domain, you define the Sine Space plot, selecting Category, Quantity and Function settings as appropriate. The Report Category you select provides the default report name, for example, S Parameter Plot *n* or Output Variables Plot *n*. Given that the u,v sampling is set in the Sine Space Options, it should not be necessary to modify the sweeps in the Trace area of the dialog box.

Here is an example Sine Space plot of Far Field rETotal.



And here is an example Sine Space plot of Near field E Total.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can use the **Contour** tab of the *Properties* dialog box to modify the appearance of the plot. See Creating Rectangular Contour Plots for discussion and examples.

Creating 2D Polar Plots

In HFSS, a polar plot is a 2D circular chart divided by the spherical coordinates R and theta, where R is the radius, or distance from the origin, and theta is the angle from the x-axis. Following is the general procedure for drawing a polar graph of results:

1. On the **Results** menu (HFSS menu or right-click **Results** on the Project tree), click **Create *<type>* Report**, and select **Polar Plot** from the report type menu.

   The **Report** dialog appears.

2. In the **Context** section make selections from the following field or fields, depending on the design and solution type.
   a. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
   b. Domain field with a drop down selection list. Whether this field appears, and the domains listed depend on the Solution type and the ***<type>*** selected. For modal and terminal solution data reports, the domain can be **Sweep** or **Time**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Before you can examine the time domain, you must perform an Interpolating sweep for a driven solution (Modal or Terminal). If you select **Time**, the **TDR Options** button is enabled. Select it and follow the directions for time-domain plotting.

c. Geometry field with a drop down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.

3. In the **Trace** tab **PolarComponent** area, specify the information to plot:
   a. On the **Category** drop-down menu, click the type of information to plot.
   b. On the **Quantity** list, click the values to plot. Use Ctrl+click to make multiple selections.
   c. In the **Function** list, click the mathematical function to apply to the quantity for the plot.
   d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

> **Note:**
>
> Color shows valid expression.

   e. **Range Function** button -- opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.
4. Click **New Report.**

This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes.

The function of the selected quantity will be plotted against the swept variable values or quantities you specified on an x-y graph. The plot is listed under **Results** in the project tree and the traces are listed under the plot. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

5. Optionally, add another trace to the plot by following the procedure above, using **Add Trace rather than New Report**.

Reviewing 2D Polar Plots

For a polar plot of S-parameters, HFSS displays in the lower-left corner the following derived information about the cursor's location:

MP The magnitude and phase of the point.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RX The normalized resistance (**R**) and reactance (**X**).

GB An alternate view of the normalized resistance and reactance in the form of

$$R + jX = \frac{1}{G + jB}$$

where

- ***G*** = conductance
- ***B*** = susceptance

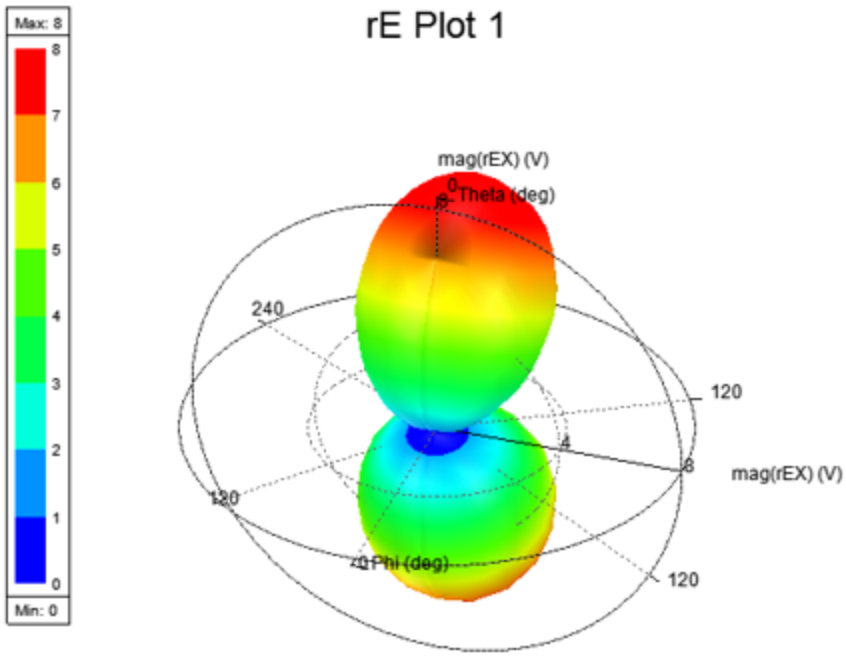
Q The quality factor.

VSWR The voltage standing wave ratio, calculated from the equation $\frac{1 + |S_{ij}|}{1 - |S_{ij}|}$.

A scale below the plot displays the scale of points along the R-axis.

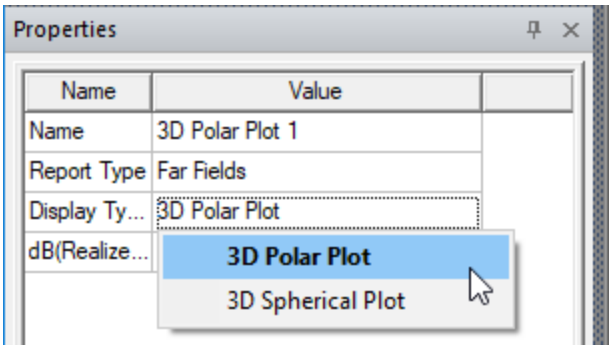
Creating 3D Polar Plots

A 3D polar plot is a 3D circular chart divided by the spherical coordinates R, theta, and phi, where R is the radius, or distance from the origin, theta is the angle from the x-axis, and phi is the angle from the origin in the z direction.

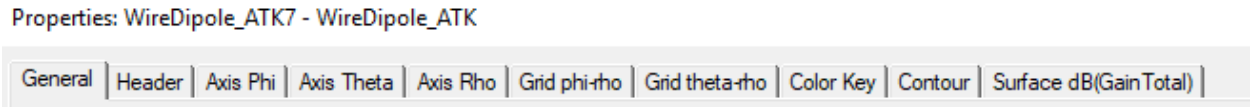


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To convert a 3D Polar plot to a 3D Spherical Plot, select the plot in the Project tree, and select from the Display type menu in the Properties window:



Double-clicking anywhere in the plot brings up the display **Properties** dialog box. The properties are grouped appropriately under various tabs, which correspond to plot entities.



- General: For general plot properties such as Visual Detail level and background color
- Header: Properties related to plot Header/Title
- Axis Phi: Properties related to the circular axis which is in XY plane
- Axis Theta: Properties related to the circular axis which is in YZ plane
- Axis Rho: Properties related to the radial axis
- Grid phi-rho-theta(0): Properties related to phi-rho grid at theta = 0 (XY plane)
- Grid theta-rho-phi(90): Properties related to theta-rho grid at phi = 90 (YZ plane)
- Color Key: Properties related to the color key, including borders, background, Min and Max, as well as number format and precision.
- Contour: Properties related to contouring of all curves/surfaces
- Surface: Properties related to the curve

You can edit the properties on these tabs to customize the appearance of the plot. See "Controlling Visual Detail in a 3D Polar Plot" on page 23-123 below.

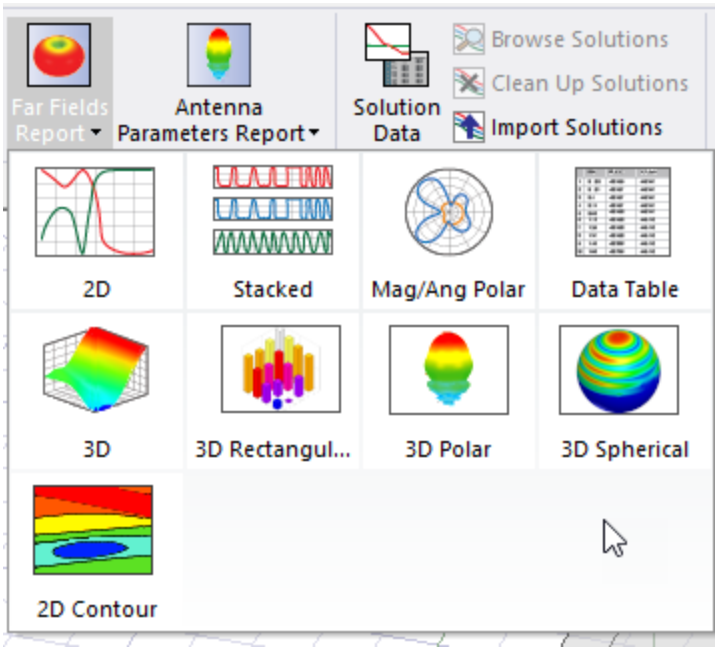
Creating 3D Polar Plots

Following is the general procedure for drawing a 3D polar plot of results:

1. On the **Results** menu (HFSS menu or right-click **Results** on the Project tree), click **Create <*type*> Report**, and select **3D Polar Plot** from the report type menu.

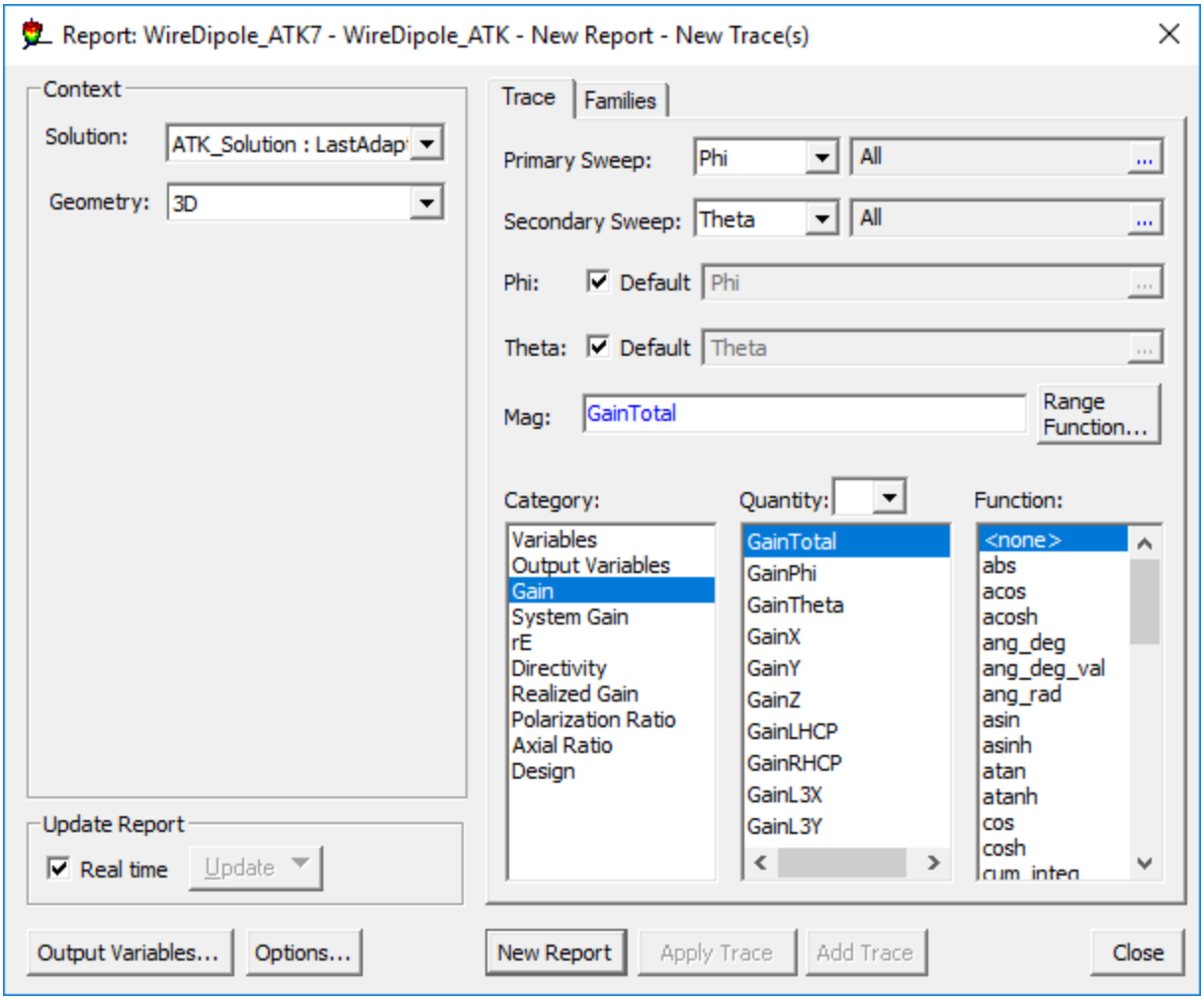
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also select 3D Polar Plots from the **Results** tab of the ribbon if such plots are appropriate for the current design.



The **Report** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. In the **Context** section make selections from the following field or fields, depending on the design and solution type.
    a. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
    b. Geometry field with a drop down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
3. In the **Trace** tab **Mag** area, specify the information to plot along the R-axis, or the axis measuring magnitude:
    a. From the **Category** drop-down menu, select the type of information to plot. The selected category provides the default plot name.
    b. In the **Quantity** list, click the values to plot. Ctrl+click to make multiple selections.
    c. In the **Function** list, click the mathematical function to apply to the quantity for the plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

> **Note:**
>
> Color shows valid expression.

e. **Range Function** button -- opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.

4. On the **Trace** tab **Theta (Secondary Sweep)** line, select the sweep variable from the drop-down menu and specify all values or select values to plot along the theta-axis.
5. On the **Trace** tab **Phi(Primary Sweep)** line, select the sweep variable from the drop-down menu, and specify all values or select values to plot along the phi-axis.
6. Click **New Report.**

   This button creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes.

   The function of the selected quantity or quantities will be plotted against the R-, phi-, and theta-axes on a 3D polar graph. The plot is listed under **Results** in the project tree. When you select the traces or plots, axis or grid labels, plot header, color key, or variable labels, their properties are displayed in the Properties window. The properties for each plot element can be edited directly to modify the plot content and appearance. See Modifying the Background Properties of a Report

7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

Interacting with 3D Polar Plots

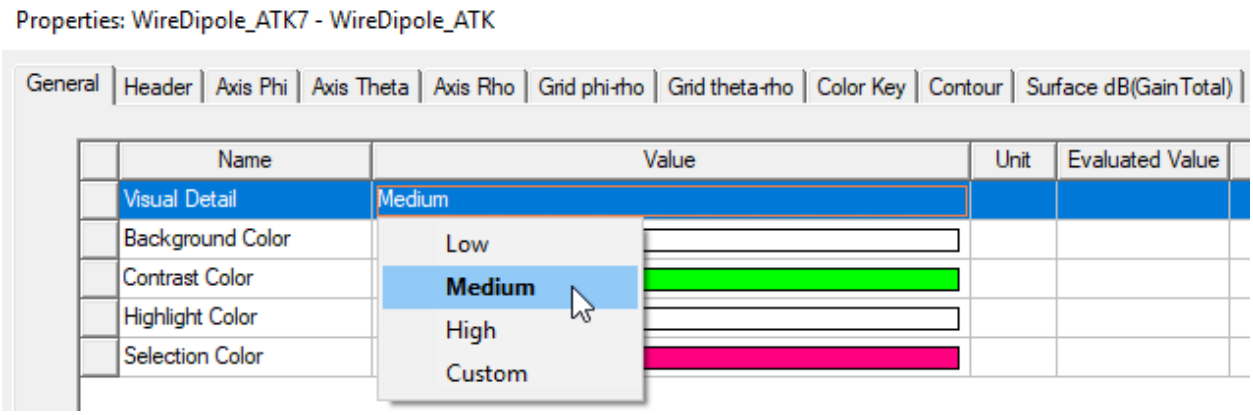
You can Rotate, Zoom, and Pan a plot. When you rotate, the Cartesian grid responds so that the curve always remains in front and the grids behind. Also, see Using the Orientation Gadget.

Controlling Visual Detail in a 3D Polar Plot

If a particular plot seems busy with information, you can edit plot properties, such as Axis and Grid Attributes for discrete levels of visual detail to improve readability. Double-click anywhere on a plot to display the **Properties** dialog box. Clicking on a plot entity selects it, highlighting the selected entity in bold.

The Visual Detail property on the **General** tab also provides control suited to different screen and plot sizes.

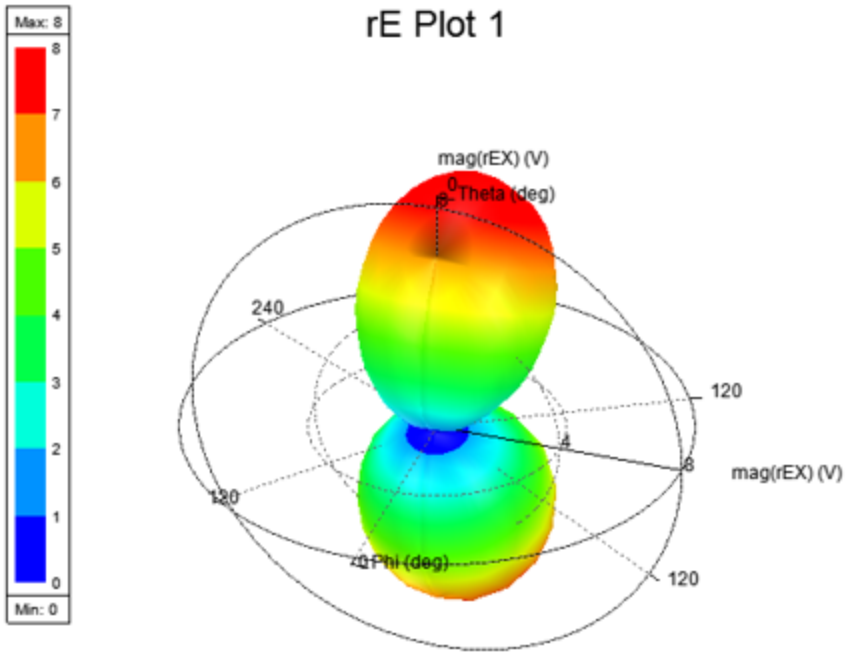
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Visual Detail menu has four options: Low, Medium (the default), High, and Custom. If you select any Visual Detail, the 3D plot is rendered according to the selected Visual Detail level and the properties reflect the values chosen for the selected visual detail level. From this predefined visual detail level, if you modify any properties, Visual Detail is automatically set to Custom (or to another predefined visual detail level if the edits happen to match the settings for that level).

You can also manually set Visual Detail to Custom. In such a case, Custom will inherit property values corresponding to the previous level. This ensures that you can customize settings starting from a baseline provided by the preconfigured Low, Medium or High Visual Detail levels.

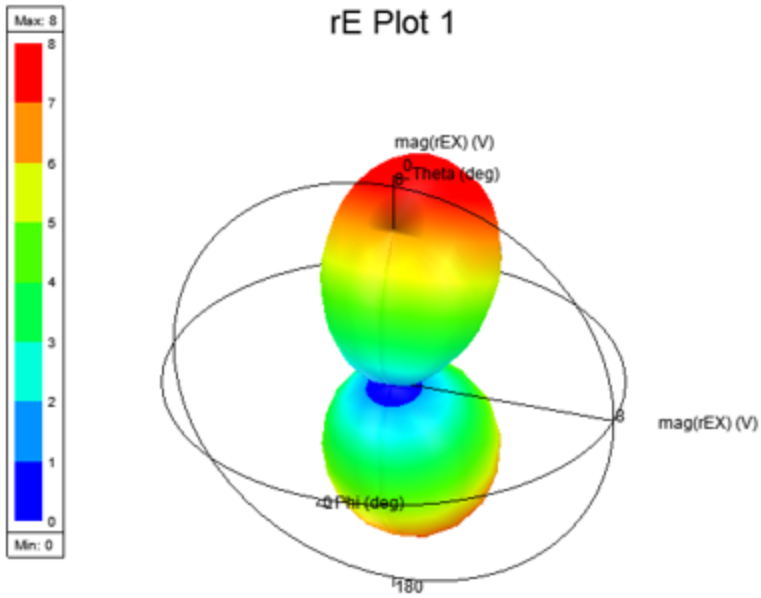
3D Polar Plot with Medium Visual Detail



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

On creation, a 3D Polar Plot has Visual Detail set to Medium and looks and feels as shown above. Specifically, under the Medium Visual Detail level, 3D Polar Plot has 3 ticks per axis (phi, theta and rho axis) which show min, max and middle value. This setting also shows axes labels.

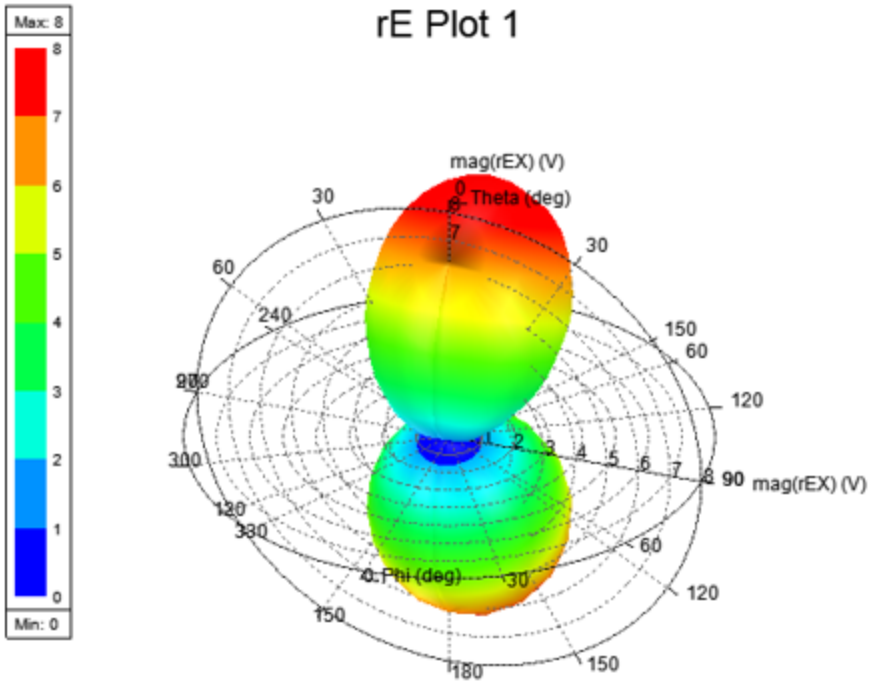
3D Polar Plot with Low Visual Detail



With Visual Detail set to Low, 3D Polar Plot does not show polar grids or grid lines. It only shows axes with 2 ticks corresponding to min and max values. This setting also renders axis labels.

3D Polar Plot with High Visual Detail

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

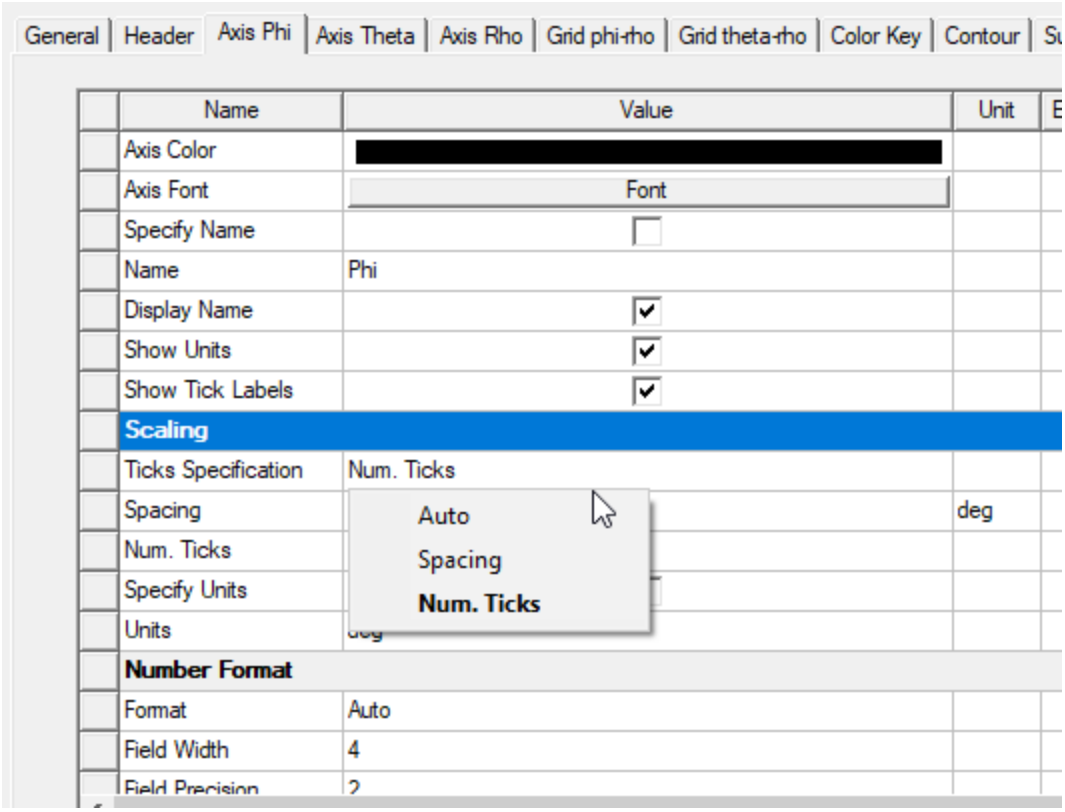


With Visual Detail set to High, a 3D Polar Plot shows all polar axes and grids together with all nice ticks and axes labels. This is ideal for large plot sizes.

Axis Properties: Ticks Specification and Num. Ticks

Ticks Specification is available on Axis properties, as shown below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ticks Specification is a menu with possible values as Auto, Spacing, and Num. Ticks, with Auto being the default value. If Ticks Specification is Auto, then a spacing value is automatically calculated and used to calculate and display the tick labels. **Spacing** shows the calculated value, and Num. Ticks shows the number of ticks based on this spacing value, as shown below:

--Scaling			
Scale	Linear		
Specify Min	☐		
Min	0	deg	
Specify Max	☐		
Max	375	deg	
Ticks Specification	Auto		
Spacing	125	deg	
Num. Ticks	2		

You can edit the **Spacing** field when Ticks Specification is set to Spacing; otherwise, it is read only.

You can edit the **Num. Ticks** field when Ticks Specification is Num. Ticks; otherwise, it is read only.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Valid Num. Ticks are between 0 and 100, including 0 and 100. If you enter an invalid value, an error message is shown. If you enter a spacing value that results in number of ticks greater than 100, then an appropriate value is shown.

- If Num. Ticks is 0, then no ticks are shown on the axis.
- If Num. Ticks is 1, then only the max value tick is shown on the axis.
- If Num. Ticks is 2, then only the min and max value ticks are shown on the axis.
- If Num. Ticks is greater than 2, then evenly spaced ticks (including min and max) are shown on the axis.

> **Note:**
>
> With the addition of the Ticks Specification property to Axis properties, the **Specify Spacing** property was removed as an Axis property.
>
> - If an R18.0 or R18.1 project is opened with **Specify Spacing** Unchecked, Ticks Specification is set to Auto.
> - If an R18.0 or R18.1 project is opened with **Specify Spacing** Checked, Ticks Specification is set to Spacing.

Grid Properties for Phi-Rho and Theta-Rho

You can control a range of display properties for the pho-rho and theta-rho grids, including line styles and colors for major and minor grids.

Overlay 3D Polar Plot on Model Window

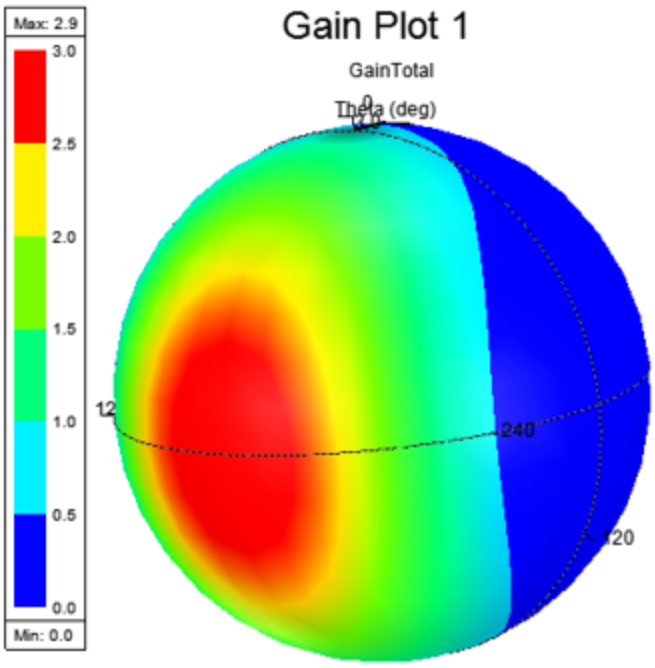
For a 3D polar plot to be eligible for overlay, it must have its primary and secondary sweep from variables Phi and Theta or IWavePhi and IWave Theta in that order. If the plot is unsuitable, the **Overlay** commands are disabled.

Once you create a suitable plot, you can overlay the 3D polar plot on the model window.

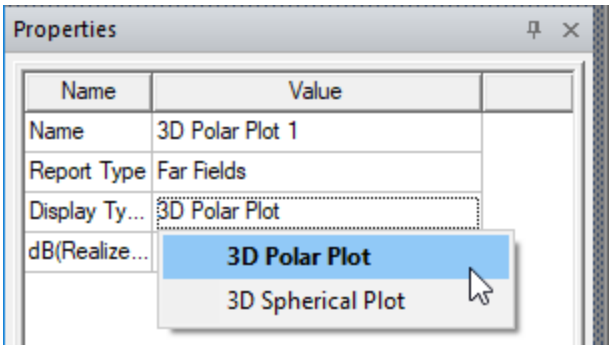
Creating 3D Spherical Plots

A 3D spherical plot resembles a 3D Polar Plot in all aspects except that the radius (rho) of the plot at all points is uniform and is equal to the maximum of all rhos at each (theta, phi) points.

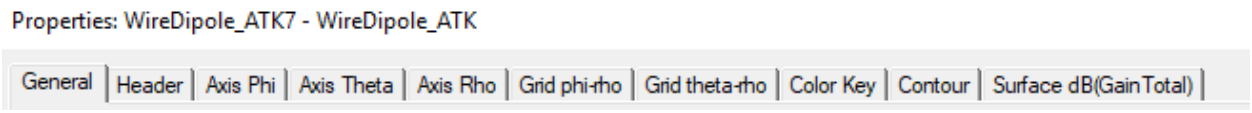
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To convert a 3D Spherical plot to a 3D Polar Plot, select the plot in the Project tree, and select from the Display type menu in the Properties window:



Double-clicking anywhere in the plot brings up the display **Properties** dialog box. The display properties are grouped appropriately under various tabs, which correspond to plot entities.



- General: For general plot properties such as Visual Detail level and background color
- Header: Properties related to plot Header/Title

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Axis Phi: Properties related to the circular axis which is in XY plane
- Axis Theta: Properties related to the circular axis which is in YZ plane
- Axis Rho: Properties related to the radial axis
- Grid phi-rho-theta(0): Properties related to phi-rho grid at theta = 0 (XY plane)
- Grid theta-rho-phi(90): Properties related to theta-rho grid at phi = 90 (YZ plane)
- Color Key: Properties related to the color key, including borders, background, Min and Max, as well as number format and precision.
- Contour: Properties related to contouring of all curves/surfaces
- Surface: Properties related to the curve

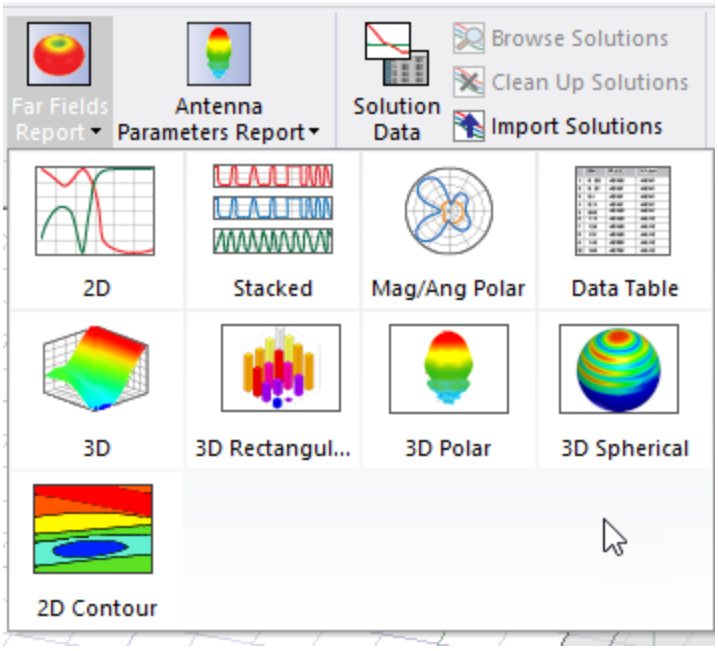
You can edit the properties on these tabs to customize the appearance of the plot. See "Controlling Visual Detail in a 3D Spherical Plot" on page 23-132 below.

Creating 3D Polar Plots

Following is the general procedure for drawing a 3D polar plot of results:

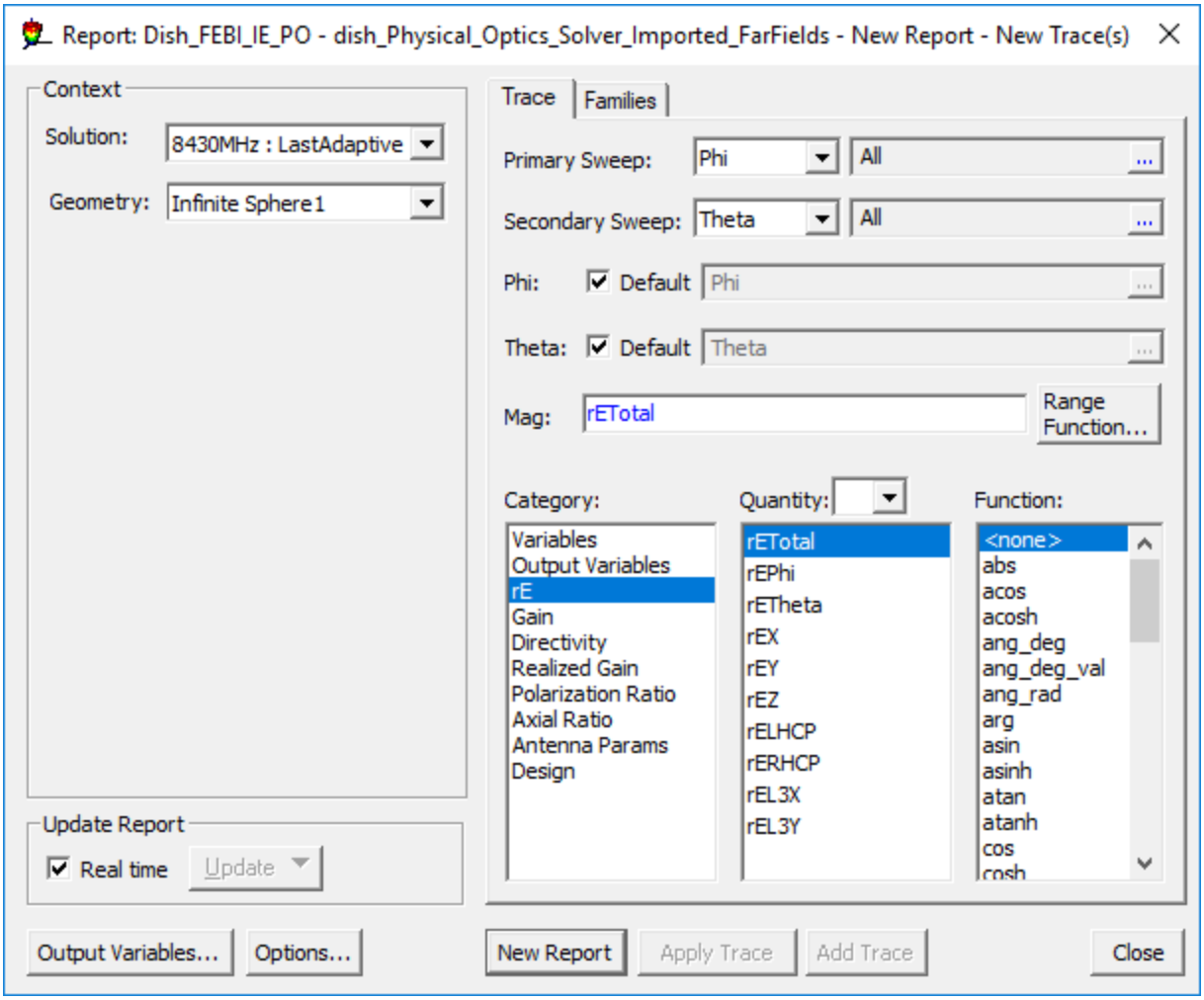
1. On the **Results** menu (HFSS menu or right-click **Results** on the Project tree), click **Create *<type>* Report**, and select **3D Spherical plot** from the report type menu.

   You can also select 3D Polar Plots from the **Results** tab of the ribbon if such plots are appropriate for the current design.



   The **Report** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. In the **Context** section, make selections from the following field or fields, depending on the design and solution type.
    a. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
    b. Geometry field with a drop down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
3. In the **Trace** tab **Mag** area, specify the information to plot along the R-axis, or the axis measuring magnitude:
    a. On the **Category** drop-down menu, click the type of information to plot. The category you select provides the default plot name.
    b. On the **Quantity** list, click the values to plot. Use Ctrl+click to make multiple selections.
    c. In the **Function** list, click the mathematical function to apply to the quantity for the plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

> **Note:**
>
> Color shows valid expression.

e. **Range Function** button -- opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.

4. On the **Trace** tab **Theta(Secondary Sweep)** line, select the sweep variable from the drop-down menu and specify all values or select values to plot along the theta-axis.
5. On the **Trace** tab **Phi (Primary Sweep)** line, select the sweep variable from the drop-down menu, and specify all values or select values to plot along the phi-axis.
6. Click **New Report.**

   This button creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes.

   The function of the selected quantity or quantities will be plotted against the R-, phi-, and theta-axes on a 3D polar graph. The plot is listed under **Results** in the project tree. When you select the traces or plots, axis or grid labels, plot header, color key, or variable labels, their properties are displayed in the Properties window. The properties for each plot element can be edited directly to modify the plot content and appearance. See Modifying the Background Properties of a Report.

7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

Interacting with 3D Spherical Plots

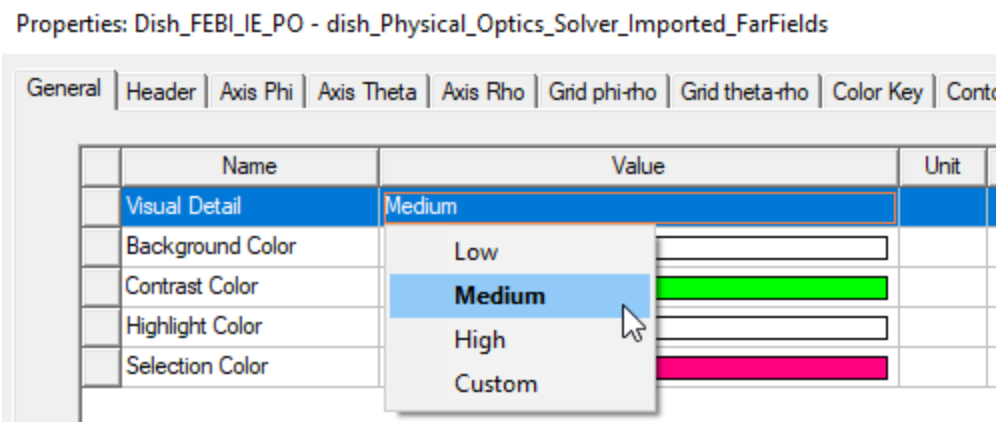
You can Rotate, Zoom and Pan a plot. When you rotate, the Cartesian grid responds so that the curve always remains in front and the grids behind. Also, see Using the Orientation Gadget.

Controlling Visual Detail in a 3D Spherical Plot

If a particular plot seems busy with information, you can edit plot properties, such as Axis and Grid Attributes for discrete levels of visual detail to improve readability. Double-click anywhere on a plot to display the **Properties** dialog box. Clicking on a plot entity selects it, highlighting the selected entity in bold.

The Visual Detail property on the **General** tab also provides control suited to different screen and plot sizes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

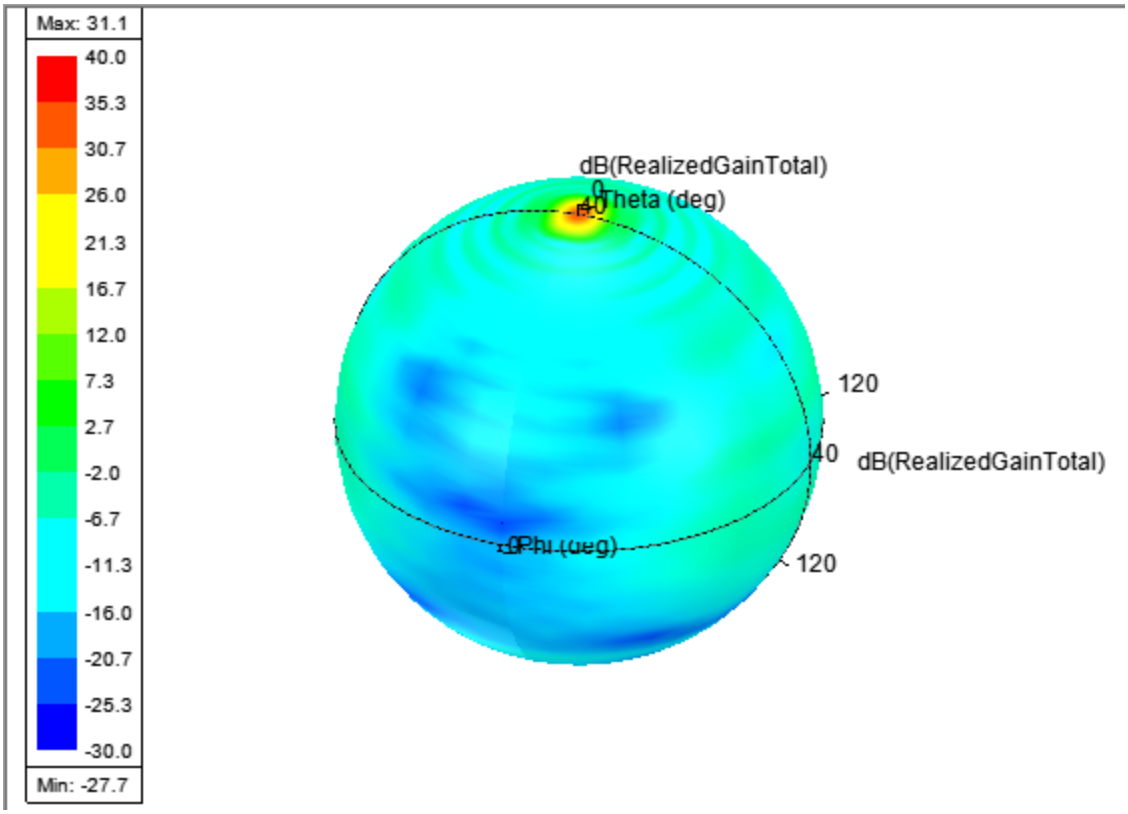


The Visual Detail menu has four options: Low, Medium (the default), High, and Custom. If you select any Visual Detail, the 3D plot is rendered according to the selected Visual Detail level and the properties reflect the values chosen for the selected visual detail level. From this predefined visual detail level, if you modify any properties, Visual Detail is automatically set to Custom (or to another predefined visual detail level if the edits happen to match the settings for that level).

You can also manually set Visual Detail to Custom. In such a case, Custom will inherit property values corresponding to the previous level. This ensures that you can customize settings starting from a baseline provided by the preconfigured Low, Medium or High Visual Detail levels.

3D Spherical Plot with Medium Visual Detail

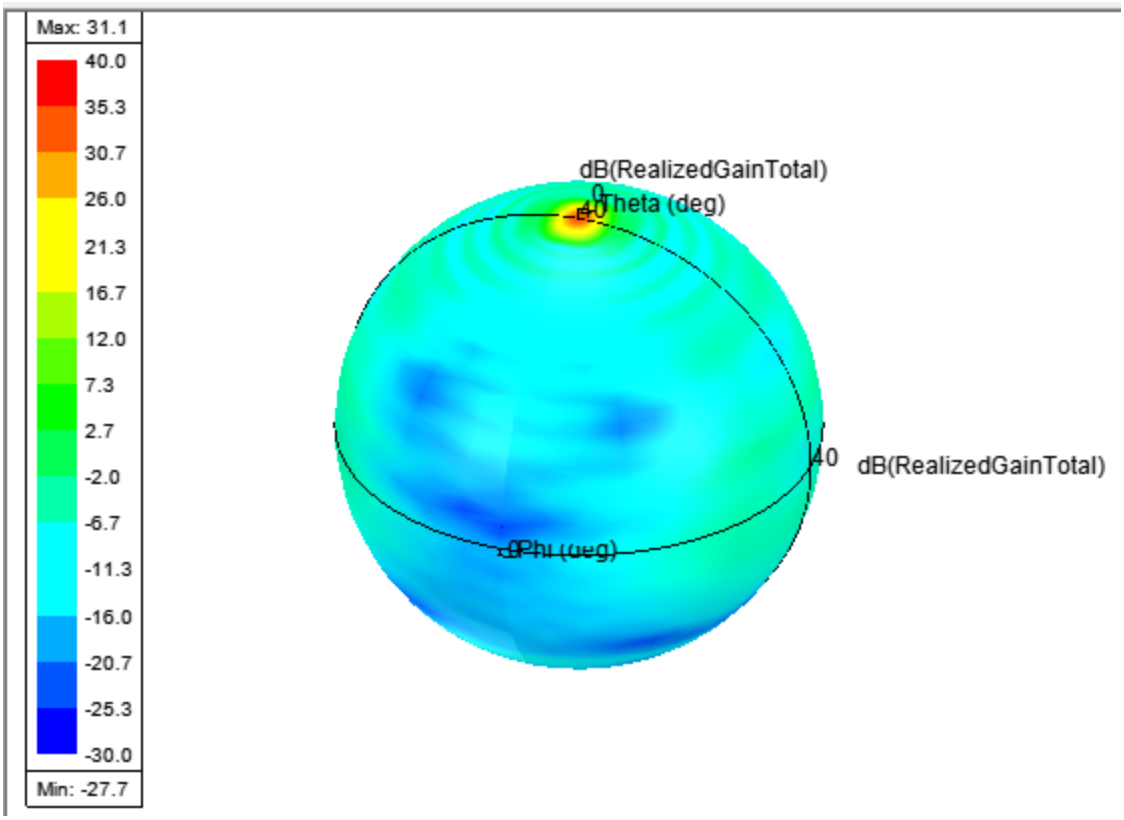
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



On creation, a 3D Polar Plot has Visual Detail set to Medium and looks and feels as shown above. Specifically, under the Medium Visual Detail level, 3D Spherical Plot has 3 ticks per axis (phi, theta and rho axis) which show min, max and middle value. This setting also shows axes labels.

3D Spherical Plot with Low Visual Detail

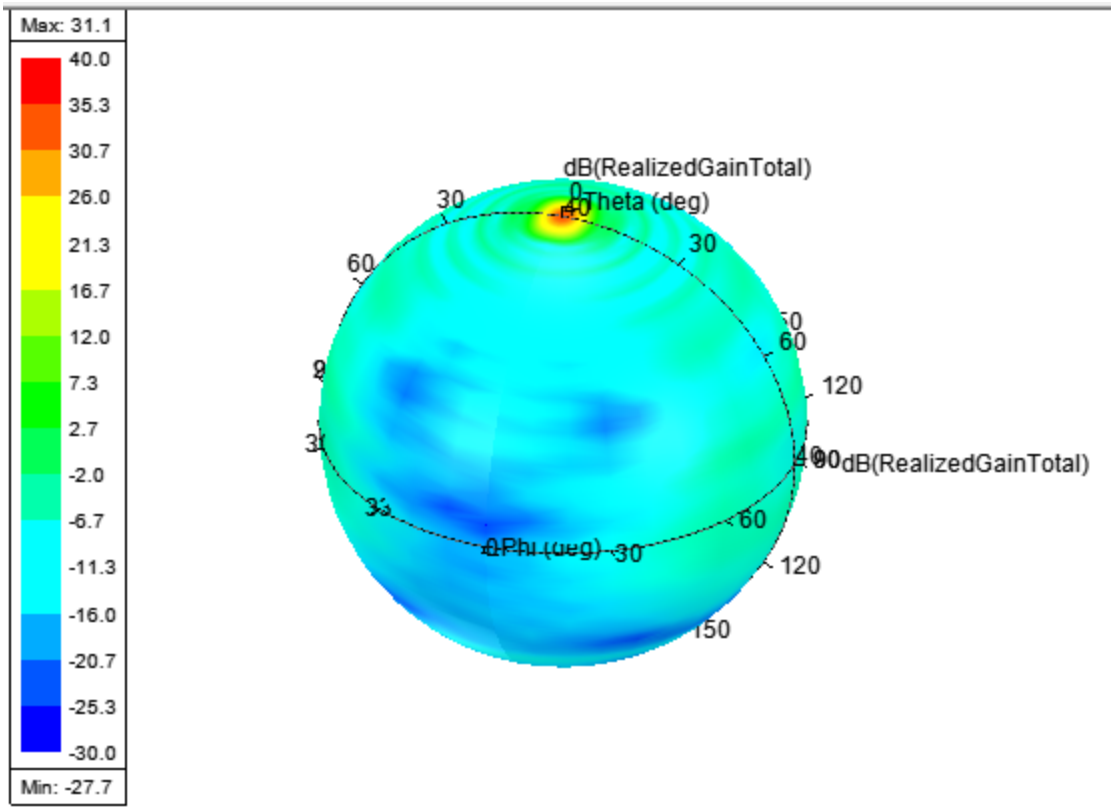
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With Visual Detail set to Low, 3D Spherical Plot does not show polar grids or grid lines. It only shows axes with 2 ticks corresponding to min and max values. This setting also renders axis labels.

3D Spherical Plot with High Visual Detail

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

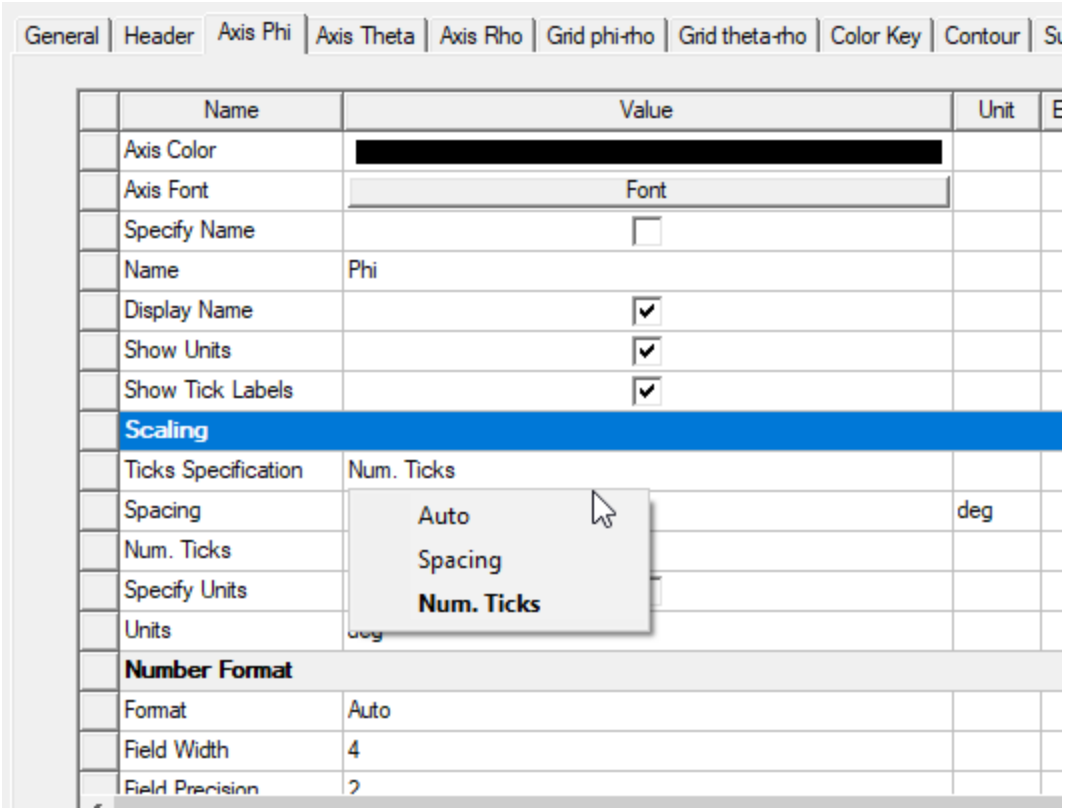


With Visual Detail set to High, a 3D Spherical Plot shows all polar axes and grids together with all nice ticks and axes labels. This is ideal for large plot sizes.

Axis Properties: Ticks Specification and Num. Ticks

Ticks Specification is available on Axis properties, as shown below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ticks Specification is a menu with possible values as Auto, Spacing, and Num. Ticks, with Auto being the default value. If Ticks Specification is Auto, then a spacing value is automatically calculated and used to calculate and display the tick labels. **Spacing** shows the calculated value, and Num. Ticks shows the number of ticks based on this spacing value, as shown below:

--Scaling			
Scale	Linear		
Specify Min			
Min	0	deg	
Specify Max			
Max	375	deg	
Ticks Specification	Auto		
Spacing	125	deg	
Num. Ticks	2		

You can edit the **Spacing** field when Ticks Specification is set to Spacing; otherwise, it is read only.

You can edit the **Num. Ticks** field when Ticks Specification is Num. Ticks; otherwise, it is read only.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Valid Num. Ticks are between 0 and 100, including 0 and 100. If you enter an invalid value, an error message is shown. If you enter a spacing value that results in number of ticks greater than 100, then an appropriate value is shown.

- If Num. Ticks is 0, then no ticks are shown on the axis.
- If Num. Ticks is 1, then only the max value tick is shown on the axis.
- If Num. Ticks is 2, then only the min and max value ticks are shown on the axis.
- If Num. Ticks is greater than 2, then evenly spaced ticks (including min and max) are shown on the axis.

> **Note:**
>
> With the addition of the Ticks Specification property to Axis properties, the **Specify Spacing** property was removed as an Axis property.
>
> - If an R18.0 or R18.1 project is opened with **Specify Spacing** Unchecked, Ticks Specification is set to Auto.
> - If an R18.0 or R18.1 project is opened with **Specify Spacing** Checked, Ticks Specification is set to Spacing.

Grid Properties for Phi-Rho and Theta-Rho

You can control a range of display properties for the pho-rho and theta-rho grids, including line styles and colors for major and minor grids.

Overlay 3D Spherical Plot on Model Window

For a 3D Spherical plot to be eligible for overlay, it must have its primary and secondary sweep from variables Phi and Theta or IWavePhi and IWave Theta in that order. If the plot is unsuitable, the **Overlay** commands are disabled.

Once you create a suitable plot, you can overlay the 3D spherical plot on the model window.

Creating Smith Charts

A Smith chart is a 2D polar plot of S-parameters upon which a normalized impedance grid has been superimposed. Following is the general procedure for creating a Smith chart of results:

1. On the **Results** menu (HFSS menu or right-click **Results** on the Project tree), click **Create <*type*> Report**, and select **Smith Chart** from the report type menu.

   The **Report** dialog appears.

2. In the **Trace** tab **PolarComponent** area, specify the information to plot:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. On the **Category** drop-down menu, click the type of information to plot. The category selected provides the default plot name.
b. On the **Quantity** list, click the values to plot. Use Ctrl+click to make multiple selections.
c. In the **Function** list, click the mathematical function to apply to the quantity for the plot.
d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

> **Note:**
>
> Color shows valid expression.

e. **Range Function** button -- opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.

3. Click **New Report.**

This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes.

The function of the selected quantity will be plotted against the values you specified on a polar plot. In addition, each circle on the plot is labeled with values of *R*, measuring normalized resistance, and each line is labeled with values of *X*, measuring normalized reactance. The plot is listed under **Results** in the project tree and the traces are listed under the plot. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

4. Optionally, add another trace to the plot by following the procedure above, using **Add Trace rather than New Report**.

Creating Smith Contour Charts

A Smith contour chart is a polar plot of S-parameters upon which a normalized impedance grid has been superimposed. Following is the general procedure for creating a Smith chart of results:

1. On the **Results** menu (HFSS menu or right-click **Results** on the Project tree), click **Create <*type*> Report**, and select **Smith Chart** from the report type menu.

The **Report** dialog appears.

2. In the **Trace** tab **Mag** area, specify the information to plot:
   a. On the **Category** drop-down menu, click the type of information to plot. The Category selection provides the default plot name.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. On the **Quantity** list, click the values to plot. Use Ctrl+click to make multiple selections.
c. In the **Function** list, click the mathematical function to apply to the quantity for the plot.
d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

> **Note:**
>
> Color shows valid expression.

e. **Range Function** button -- opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.

3. On the **Trace** tab **(Secondary Sweep)** line,select the sweep variable from the drop-down menu and specify all values or select values to plot along the theta-axis:

To select an Secondary sweep component that is different from the default, uncheck the Default field to enable the X field and **Browse** [...] button. Click **Browse** [...] to display the **Select X Component** dialog box. This lets you specify the X component as you do the Y; that is, in terms of Categories which define the selectable Quantities, and Functions to apply. After making selections, **OK** the dialog to assign the X component.

a. If sweeps are available, you can select **Browse** [...] to display a dialog that lets you select particular sweep or sweeps, or all sweeps.
b. The **Families** tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (for example, for a parametric sweep). If so, the variables other than the one chosen as the **X (Primary sweep)**, are listed under the **Families** tab with columns for the variable, the value, and an Edit column with an ellipsis **[...]** button. See Using Families tab for Reports.

4. On the **Trace** tab **(Primary Sweep)** line,select the sweep variable from the drop-down menu, and specify all values or select values to plot along the phi-axis:

To select an X component that is different from the default, uncheck the Default field to enable the X field and browse [...] button. Click **Browse** [...] to display the **Select X Component** dialog box. This lets you specify the X component as you do the Y; that is, in terms of Categories which define the selectable Quantities, and Functions to apply. After making selections, **OK** the dialog to assign the X component.

a. If sweeps are available, you can select **Browse** [...] to display a dialog that lets you select particular sweep or sweeps, or all sweeps.
b. The **Families** tab provides a way to select from valid solutions for sweeps where a simulation has multiple variables defined (for example, for a parametric sweep). If so, the variables other than the one chosen as the **X (Primary sweep)**, are listed

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

under the **Families** tab with columns for the variable, the value, and an Edit column with an ellipsis [**...**] button. See Using Families tab for Reports.

5. Click **New Report.**

This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plot *n*). You can edit the plot names in the project tree and the plot header text in the report synchronizes.

The function of the selected quantity will be plotted against the values you specified on a polar plot. In addition, each circle on the plot is labeled with values of *R*, measuring normalized resistance, and each line is labeled with values of *X*, measuring normalized reactance. The plot is listed under **Results** in the project tree and the traces are listed under the plot. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

6. Optionally, add another trace to the plot by following the procedure above, using **Add Trace rather than New Report**.

Creating Data Tables

A data table is a grid with rows and columns that displays, in numeric form, selected quantities against a swept variable or other quantities.

1. Click **HFSS**> **Results**> **Create *<type>* Report** or right-click **Results** in the Project Manager and click **Create *<type>* Report**.
2. In the **display type** menu, click **Data Table**.

   The *Report* dialog box appears.

3. In the **Context** section make selections from the following field or fields, depending on the design and solution type.
   a. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
   b. Domain field with a drop down selection list. Whether this field appears, and the domains listed depend on the Solution type and the report ***<type>*** selected. For modal and terminal solution data reports, the domain can be **Sweep** or **Time**.

      Before you can examine the time domain, you must perform an Interpolating sweep for a driven solution (Modal or Terminal). If you select **Time**, the **TDR Options** button is enabled. Select it and follow the directions for time-domain plotting.

   c. Geometry field with a drop down selection list. For field and radiated field reports, this applies the quantity to a geometry or radiated field setup.
4. Under the **Trace** tab **Y** component section, select the quantity you are interested in and its associated function:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. On the **Category** drop-down menu, click the type of information to plot. The category selected provides the default name for the plot.
b. On the **Quantity** list, click the values to plot. Use Ctrl+click to make multiple selections.
c. In the **Function** list, click the mathematical function to apply to the quantity for the plot.
d. The **Value** field displays the currently specified Quantity and Function. You can edit this field directly.

> **Note:**
>
> Color shows valid expression.

e. **Range Function** button -- opens the *Set Range Function* dialog box. This applies currently specified Quantity and Function.

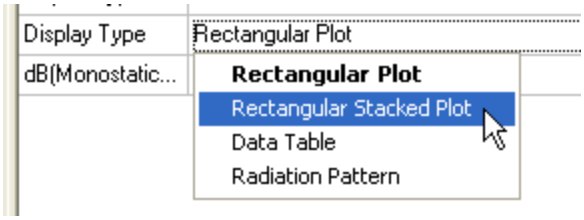
5. On the **Trace** tab **X (Primary sweep)** line, select the sweep variable from the drop-down menu and specify all values or select values.
6. Click **New Report**.

   This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, for example, S Parameter Plot *n* or Output Variables Plot *n*. You can edit the plot names in the project tree and the plot header text in the report synchronizes.

   The Y quantity will be listed at each variable value or additional quantity value you specified. The data table is listed under **Results** in the project tree. The plot is listed under **Results** in the project tree and the traces are listed under the plot. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

You can also modify the display type of an existing plot from the Properties dialog for that plot. Select the Report icon in the Project tree to display the Properties dialog box. Selecting the Display Type field displays a menu with selections available for that plot.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once you make a selection, the plot display updates for the current selection.

If you choose to print a data table:

- Selecting print "All" prints the whole table for current data page (if there is more than one data page).
- Selecting print "Pages" prints user-specified pages.
- If the table is bigger than the screen view (that is, it has scroll bar), printing first scrolls right, prints until no more scrolling and then scroll down.
- The Page number appears at the bottom of the page, aligned at center.
- The table layout of each page follows the screen, but with no scroll bar will be printed, and no data page bar as on screen.

Creating Radiation Patterns

A radiation pattern is a 2D polar plot displaying the intensity of near- or far-field radiation patterns. It is divided by the spherical coordinates R and theta, where R is the radius, or distance from the origin, and theta is the angle from the x-axis. Following is the general procedure for drawing a radiation pattern of results:

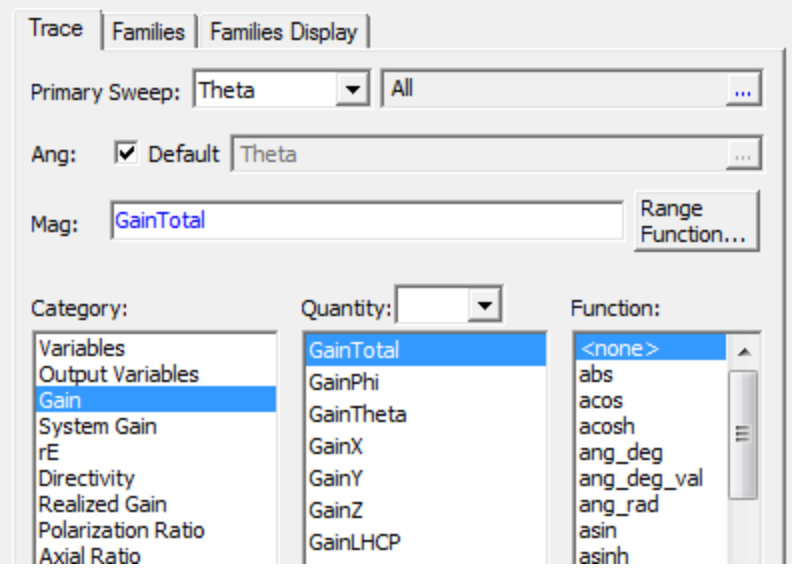
1. Click **HFSS>Results>Create Far Fields Report>Radiation Pattern**, or right-click the **Results** icon in the **Project tree** and click **Create Far Fields Report>Radiation Pattern**.

The **Report** dialog box appears, and a Radiation Pattern Plots icon appears under **Results** in the Project tree.

3. In the **Context** section make selections from the following field or fields, depending on the design and solution type.
    a. Solution field with a drop down selection list. This lists the available solutions, whether sweeps or adaptive passes.
4. In the **Trace** tab area, specify the Primary Sweep and Ang information to plot along the R-axis, or the axis measuring magnitude. The Primary Sweep field has a drop-down menu of current choices for Theta, Phi, and defined variables. Click the ellipsis button if you want to select from available values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. In the **Ang**, if you want to change the default, uncheck to enable the field and ellipsis button.



   a. On the **Category** drop-down menu, click the type of information to plot. The selected category also provides the default plot name.
   b. On the **Quantity** list, click the values to plot. Use Ctrl+click to make multiple selections.
   c. In the **Function** list, click the mathematical function to apply to the quantity for the plot.
   d. The **Mag** field displays the currently specified Quantity and Function. You can edit this field directly.

> **Note:**
>
> Color shows valid expression.

   e. **Range Function** button -- opens the **Set Range Function** dialog box. This applies currently specified Quantity and Function.

5. The **Families** tab is helpful if you want to use the plot as an overlay and you need to restrict the Phi (default) or Theta values to make the plot appropriate for overlay. See Overlaying 2D Radiation Field Plots on Models.
6. Click **New Report.**

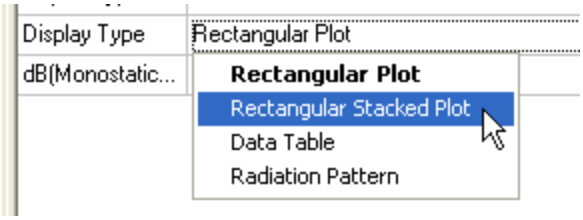
This creates a new report in Project tree, displays the report with the defined trace, and enables **Add Trace** on the **Report** dialog box. The default name is based on the Report Category you selected, for example, System Gain Plot *n* or Directivity Plot *n*. You can edit the plot names in the project tree and the plot header text in the report synchronizes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The function of the selected quantity or quantities will be plotted against the values you specified on a 2D polar plot. The plot is listed under **Results** in the project tree and the traces are listed under the plot. When you select the traces or plots, their properties are displayed in the Properties window. These properties can be edited directly to modify the plot.

7. Optionally, add another trace to the plot by following the procedure above, using **Add Trace** rather than **New Report**.

You can also modify the display type of an existing plot from the Properties dialog for that plot. Select the Report icon in the Project tree to display the Properties dialog box. Selecting the Display Type field displays a menu with selections available for that plot.



Once you make a selection, the plot display updates for the current selection.

Delta Markers in 2D Reports

To view the difference between any two marker points in a report:

1. Set the first marker by clicking and holding the mouse button.
2. Move the mouse without releasing left button to another position, and then release the left button to create second marker.

In the marker text window, you see the difference between the two markers instead of the X, Y value of marker.

Plotting in the Time Domain

The idea behind Time-Domain Reflectometry (TDR) is to excite a structure with a step function, and inspect the reflections as a function of time. Before you can examine the time domain, you must perform an Interpolating sweep for a driven solution (Modal or Terminal or Transient). You can then select **Time** from the **Domain** list in the **Report** dialog box. You also need to specify the input signal, whether step or impulse.

With **Time** selected as the domain, you can select from several Categories and associated Quantities to plot, for example mag(S_{11}). When you plot in the Time domain, every frequency domain quantity is first converted to the time-domain before the formula is evaluated. For example, if you type in

$S_{11} / (1 - S_{11})$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and plot it in the time domain the reporter will plot

$IFFT(S_{11} * input) / (1 - IFFT(S_{11} * input))$

It will NOT plot

$IFFT(S_{11}/ (1 - S_{11}) * input)$

The two expressions are not equivalent.

If you select Time Domain Impedance as the Category, you can select the TDRZ quantity. This is defined as

$TDRZ(t) = Zref * (1 + IFFT(S_{11} * input)) / (1 - IFFT(S_{11} * input))$

where "input" denotes the Fourier transform of the input signal (step or impulse) and "IFFT(.)" denotes the inverse FFT.

This equation is the instantaneous ratio of the time-domain voltage v(t) to the time-domain current i(t). That is because voltage and current are defined (in the frequency domain) in terms of the incident and reflected waves a and b, respectively, as

$V = sqrt(Zo) * (a + b) = sqrt(Zo) * (1 + S_{ii}) * a$

$I = 1/sqrt(Zo) * (a - b) = 1/sqrt(Zo) * (1 - S_{ii}) * a$

This lets the incident wave be the input step signal, and so when we take the inverse FFT of V and I, we get v(t) and i(t) in the time domain. Taking their ratio as a function of time then yields TDRZ(t). By default, Zo is equal to 50 Ohm.

For HFSS, HFSS 3D or Layout Linear Network Analysis on Circuit, to create a plot in the **Time Domain**:

1. For a design with an existing sweep setup, follow steps 1 - 4 for creating a report for design.
2. In the **Report** dialog box, in the **Domain** list, click **Time**.

   This enables the **TDR Options** button and for terminal solution data reports includes the Terminal TDR Impedance in the Category list.

3. Click the **TDR Options button**.

   The **TDR Options** dialog box appears.

4. Select the input signal type, **Step** or **Impulse**.

   A **Step** describes a sustained change in the signal, whereas the **Impulse** is a brief excitation. **Impulse** is a very narrow rectangular pulse, with zero rise and fall time, width of 1 time step, and height of 1/(time step).

   Selecting **Step** enables the **Rise Time** field, and **Impulse** disables it.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. If you selected **Step**, enter the rise time of the pulse in the **Rise Time** text box.

The rise time should be appropriate for the frequency context.

With a band width from DC to f_{max}, the best time resolution that can be achieved is $1/(2f_{max})$.

A rise time of $1/(2f_{max})$ is the shortest rise time that can be resolved. However, a rise time of 0 s gives equally valuable information, so 0 is the default in this panel. See the example plot.

6. Enter the total time on the plot in the **Maximum Plot Time** text box.

The default maximum plot time in the **TDR Options** dialog is related to the delta frequency df in the frequency sweep: it is 1/2df, since that is the extent of time for which the IFFT gives information. This is often very long relative to the time delay that corresponds to the length of your device under test, so you may want to reduce this value. Alternatively, you can adjust the time axis of your TDR plot after it has been created.

7. Set the number of time points to plot in the **Delta Time** text box. By default, this is set to the number of points in the frequency sweep.

The delta time is based on the bandwidth of the sweep: with a frequency sweep from DC to f_{max}, the smallest time resolution you can obtain is given by $1/(2f_{max})$. The IFFT algorithm provides data points as a spacing of $1/(2f_{max})$, but you can smoothly interpolate between points by setting a finer resolution, e.g., $1/(10f_{max})$, at the expense of extra computation time.

8. Optionally, under **TDR Window**, modify the window type and width.
9. You can use the **Save as Default** to set the current values as a default, and the **Use Defaults** button to use previously saved options. Note that when you select a trace, the initial displayed values are those of the selected trace.
10. Click **OK**.

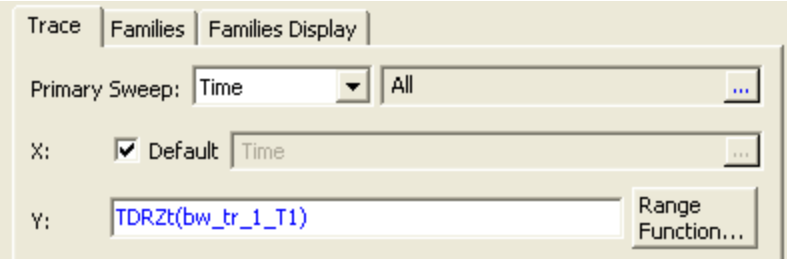
Optionally, to plot Terminal TDR impedance (that is, rather than calculate the S-parameter for waveport1 versus frequency, instead calculate the delay versus time at a particular impedance), do the following:

a. In the **Category** list, click **Terminal TDR Impedance**.
b. In the **Quantity** list, click a quantity to plot.

The default impedance (Zo) for the TDRZ quantity is 50 Ohms, unless you specified differently when you Set Renormalizing Impedance for Terminals when you created the terminals in the model. If you need a different impedance value, you can either edit the value in the **Report** dialog (as shown below), or you can create an Output Variable representing $Zo \times (1+S_{ii})/(1-S_{ii})$ with the Zo of your choice. To edit the Zo value in the **Report** dialog:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. For the Category, select Terminal TDR Impedance, and the Port and Function of interest.



2. Edit the value by placing the cursor in the Value field.

   In this example, the value for Zo is changed from the default to 75 Ohms by typing ',Zo=75ohm' in the Y-column field.

   c. In the **Function** list, click the mathematical function of the quantity to plot.

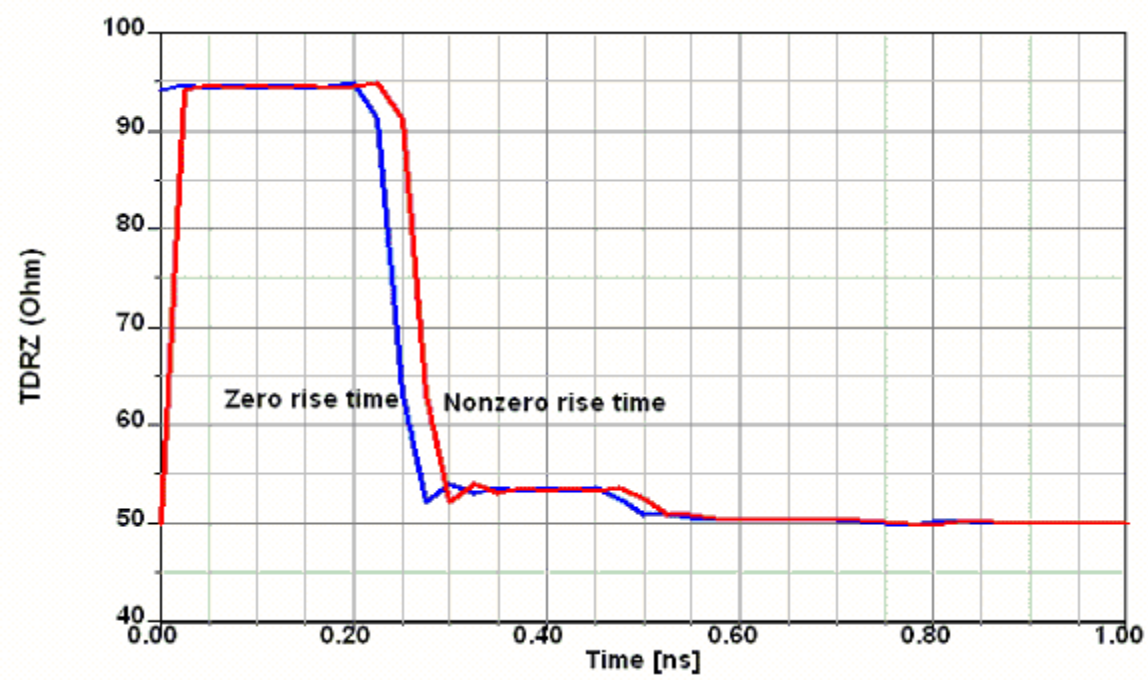
3. Click **Done**.

The report appears in the view window. It will be listed in the project tree.

If S_{11} = 0 at DC, the time-domain step response will settle to zero and the TDRZ step response settles to Zref. If S_{11} is nonzero at DC, the time-domain step response will settle to a nonzero value and TDRZ will settle to a value different from Zref. The time-domain impulse response will always settle to zero, since it can be seen as the derivative of the step response. The TDRZ impulse response will always settle to Zref.

The plot below shows the difference between a short nonzero rise time and zero rise time for a transmission line segment of 94 Ohm. Note that the trace with zero rise time starts at the correct line impedance while the other starts at the renormalizing impedance. Other than that, one trace is a shifted version of the other. The reason the plot with finite rise time starts at 50 ohms is that the time-domain voltage and current are still at their steady state values, so v = Zref * i. As the pulse arrives, the TDRZ response changes from the steady-state behavior because there's a reflection from the transmission line back to the exciting source, which has a different renormalizing impedance from the characteristic impedance of the transmission line.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Some things to keep in mind with TDR:

$$Spatial resolution \Delta x = c/(2B) \quad (1)$$

where c is the speed of light in the medium and B is the bandwidth of the signal. Since TDR is usually based on a frequency band that starts at DC, the spatial resolution becomes

$$\Delta x = c/(2F_{max}) \quad (2)$$

where F_{max} is the highest frequency in the frequency sweep. For example, if F_{max} = 15 GHz and the medium has ε_r=4, the spatial resolution will be (1.5E8 m/s)/(3E10 /s) = 5 mm.

A spatial resolution of $c/(2F_{max})$ corresponds to a resolution in time

$$\Delta t = 1/(2F_{max}) \quad (3)$$

Let N be the number of points in the IFFT. N equals the number of time samples, and it also equals twice the number of frequency samples. The density of frequency samples in the frequency sweep influences the total time T as follows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$2F_{max}/(\Delta f) = N(\text{number of points in IFFT}) = T/\Delta t \quad (4)$$

So increasing the density of the frequency samples leads to an increase in total time T. In practical case, this often leads to a long tail in the TDR plot with little useful information. Therefore, the TDR Options interface lets you set the maximum plot time to a smaller value.

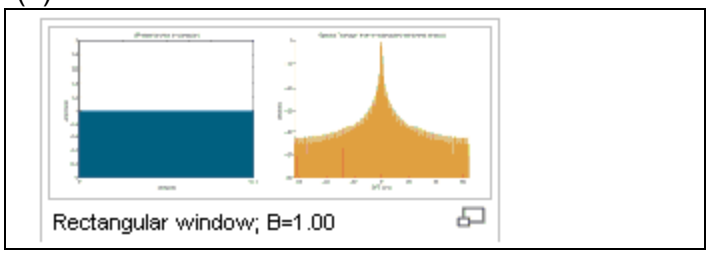
The TDR Options interface also lets you choose a smaller Δt than given by equation (3) above. When you choose a smaller Δt, you increase F_{max} by "zero padding" (adding zero values for S_{11} beyond the calculated frequency sweep). Whether this is justified depends on your judgment. It leads in practice to a smoother TDR signal.

HFSS also lets you set the rise time of your input signal. The rise time should be at least $1/(2F_{max})$. Even this rise time is a bit short for comfort, as it equals the duration of only one time sample. An input signal with a longer rise time has a smaller high-frequency content and will lead to reduced "ringing" in the TDR response.

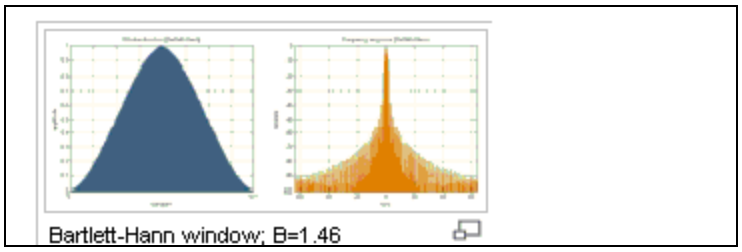
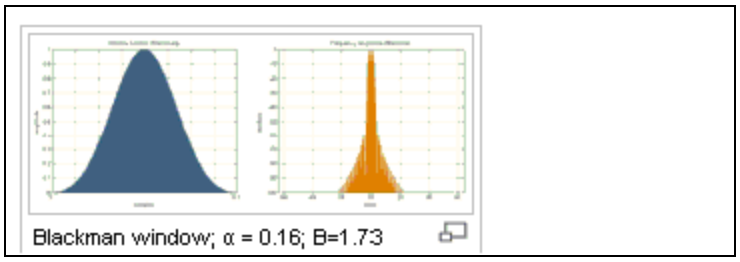
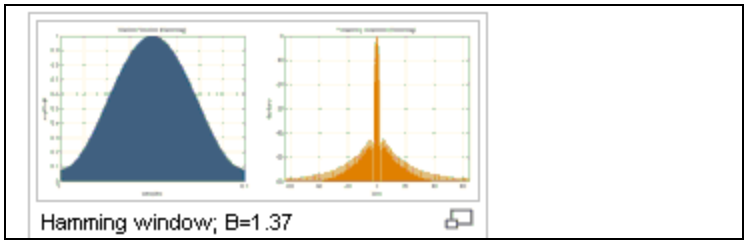
A Hamming or Hann filter will also reduce the high-frequency content and tends to lead to a smoother TDR response. With these filters, one can select a width. A width of 100% is often a good choice.

TDR Windowing Functions

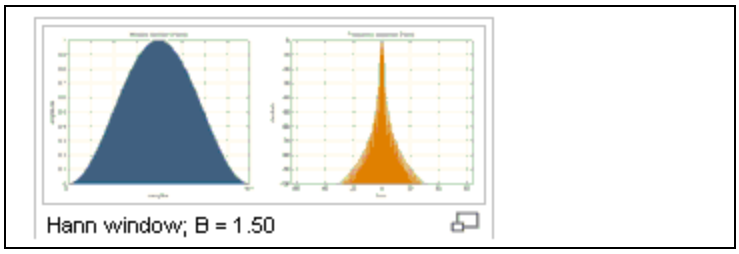
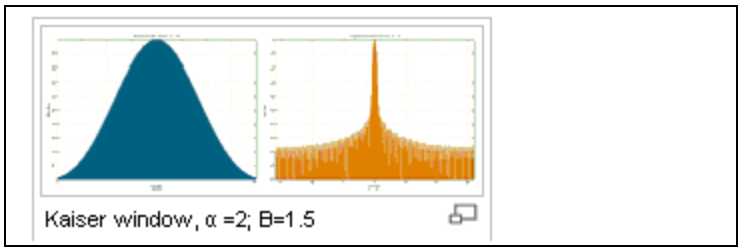
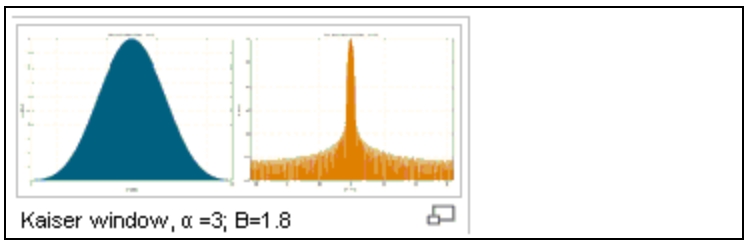
Windowing functions cause the FFT of the signal to have non-zero values away from ω. Each window function trades off the ability to resolve comparable signals and frequencies versus the ability to resolve signals of different strengths and frequencies. The window type list includes the following:

Window Function	Preferred Use
Rectangular	A low dynamic range function offering good resolution for signals of comparable strength. Poor when signals have very different amplitudes. $w(n)=1$. 
Bartlett	A high dynamic range function, with lower resolution, designed for wide band applications. $$w(n) = a_0 - a_1\left\lvert\frac{n}{N-1} - \frac{1}{2}\right\rvert - a_2\cos\left(\frac{2\pi n}{N-1}\right)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Window Function	Preferred Use
	where a_0=0.62; a_1=0.48; a_2=0.38
	 Bartlett-Hann window; B=1.46
Blackman	A high dynamic range function, with lower resolution, designed for wide band applications. $w(n) = a_0 - a_1 \cos\left(\frac{2\pi n}{N-1}\right) + a_2 \cos\left(\frac{4\pi n}{N-1}\right)$ where a_0 =(1-α)/2; α_1=1/2; α_2=α/2
	 Blackman window; α = 0.16; B=1.73
Hamming	A moderate dynamic range function, designed for narrow band applications. It minimizes the maximum sidelobe. $w(n) = 0.54 - 0.46 \cos\left(\frac{2\pi n}{N-1}\right)$
	 Hamming window; B=1.37
Hanning (default)	A moderate dynamic range function, designed for narrow band applications. $w(n) = 0.5\left(1 - \cos\frac{2\pi n}{N-1}\right)$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Window Function	Preferred Use
	
Kaiser	Selecting the Kaiser plot also enables a field to specify an associated Kaiser parameter. The larger the Kaiser parameter, the wider the window. The parameter controls the trade off between width of the central lobe and the area of the side lobes.  
Welch	This approach applies a parabola-shaped window to the frequency domain data. It is based on the Bartlett method but splits the signal into overlapping segments, which are then windowed. The intent is to balance the influence of data in the center of the function.

You can use the **Save as Default** to set the current values as a default.

Working with Traces

A trace in a 2D or 3D report defines one or more curves on a graph. A trace in a data table defines part of the displayed matrix of text values.

The values used for a plot's axes (which may be X, Y, Z, phi, theta, or R depending on the display type) can be variables in the design, such as frequency, or functions and expressions based on the design's solutions. If you have solved one or more variables at several values, you can "sweep" over some or all of those values, resulting in a curve in 2D or 3D space.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A report can include any number of traces and, for rectangular graphs, up to 20 independent y-axes. Traces appear in the Project tree under their report. They can be selected, copied and pasted.

When you move a cursor over a trace in a report, the cursor changes to show that you can make a selection:

- For PC systems, the cursor changes to the color of the selectable trace.
- For Linux systems, the cursor changes to a solid black arrow, rather than the default black outline.

In general, to add a trace to a report:

1. Select a report in the Project window and right-click and select **Modify Report**.
2. In the *Report* dialog box, specify the Y component information.
    a. Specify the Category of information you want to plot from the drop-down menu.

       The Category drop-down menu lists the available categories for the Solution type and the current design. Selecting a category changes the Quantity and Function lists to represent what is available for that category.

    b. Specify the Quantity you want to plot by selecting from the Quantity list.

       The selected quantity appears in the Value field, operated on any selected function.

    c. Select the Function to apply to the specified quantity.
    d. The Value field shows the trace being readied for plotting on the Y-axis. This field is editable when the text cursor is present. You can modify the information to be plotted by typing the name of the quantity or sweep variable to plot along an axis directly in the text boxes.

       > **Note:**
       >
       > Color shows valid expression.

    e. **Range Function** button -- opens the *Set Range Function* dialog box. This applies the currently specified Quantity and Function.
3. In the *Report* dialog box, specify the X axis information (for example Primary Sweep).
4. Click **Add Trace**.

   A trace is added to the traces list under its report icon in the Project tree. The trace represents the function of the quantity you selected and will be plotted against other quantities or swept variable values. Selecting a Trace in the Project tree displays the Properties window for that Trace. Selecting a trace in the report or legend displays the display Properties window for that trace.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Trace icons can be selected, copied, and pasted for their definitions or their data. They can be selected and deleted from the Project Manager.

By the default, the Trace name is the definition (the category, quantity and function). The trace will be visible in the report when you click **Add Trace**.

Trace properties can be edited directly in the respective Properties windows or edited in the *Report* dialog box. To change the name or definition of a trace, see Editing Trace Properties. To edit other display properties of a trace, see Editing the Display Properties of Traces

Editing Trace Properties

To edit trace properties such as the name, Y Axis association, the component definition, the context, or the variables select the trace in the Project tree.

To edit a **trace name**:

1. Select the trace in the Project Manager.

   This displays a docked Properties window for the Trace.

2. Check the Specify Name box.

   This enables editing of either the Name field in the docked properties dialog box, or the Trace label text in the Project tree. Editing this name changes the display in the Legend and in the Project tree, but not the underlying Y-component definition.

> **Note:**
>
> To control the display of the Solution Name and Variation Key in the Legend, see Report 2D: Legend Tab.

To edit the Y Axis associated with the trace (2D Rectangular and Rectangular Contour plots):

1. Select the trace in the Project Manager.
2. In the docked properties window for the trace, select the Y Axis to be associated with the trace from the drop-down menu. Up to twenty independent Y axes can be added to a plot.

To edit a **trace component definition**:

1. Select the trace in the Project Manager.
2. In the docked Properties window for the trace, select the component field of interest, and select **Edit...** from the drop-down menu.

   This displays the edit Component field window from which you can edit the category, quantity and function.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **OK** to apply the changes and close the dialog box.

To edit a **trace Context**:

1. Select the trace in the Project Manager to display the docked properties window.
2. In properties window, click the Solution field or the Domain field. If other selections are possible, they can be selected from the drop-down menu.

To edit a **variable** for a trace:

1. Select the trace in the Project Manager to display the docked properties window.
2. Under the -Variables category, on the Families line, click the Edit button to display the Edit families dialog box.

   From this dialog box, you can select the Sweeps or Variations radio buttons. If other nominal values are available you can click the ellipsis button to select from a list.

Editing the Display Properties of Traces

Editing the display properties of traces differs for 2D and 3D reports. To edit the display properties of a trace for a 2D report:

1. Select a trace in an open **Report** window.
2. Click once on the trace to view settings in the docked *Properties* window, or double-click to open *Properties* dialog box.

   The display properties window for a 2D trace includes a **General** tab and an **Attributes** tab.

   The *General* tab properties apply to the general appearance of the plot. They include the Background color, Contrast color, Field width, and Whether to use Scientific notation for marker and delta marker displays. (X and Y notation display is set separately, in the Axis property tabs.)

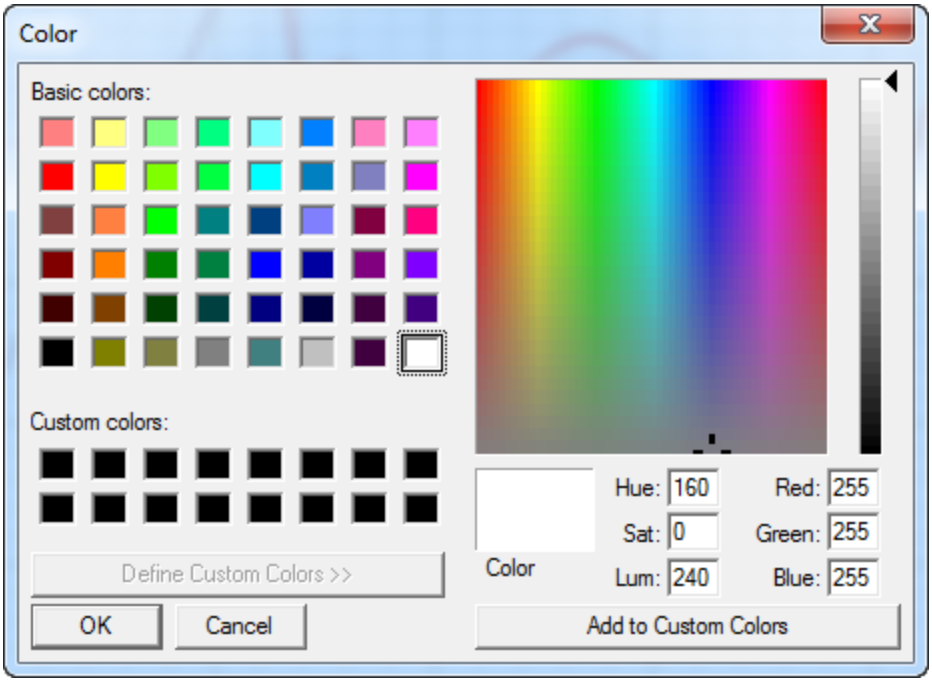
   The *Attributes* tab properties apply specifically to the trace. The defaults are set in the Report2D options. They include:

   - Name -- not editable by selecting the trace from the Report. It shows the characteristics of the trace as defined in the *Report* dialog box.

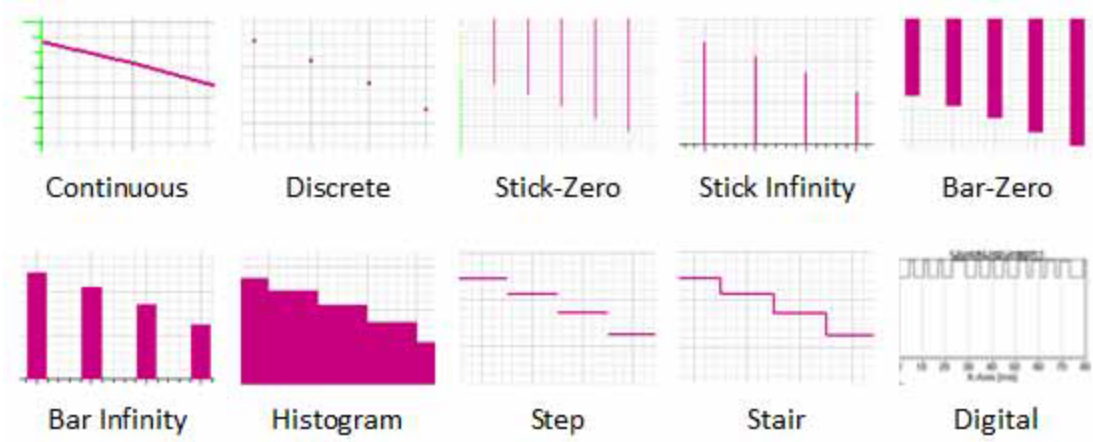
     To edit a trace name, see Editing Trace Properties.

   - Color -- shows the Trace color. Double-click to open a Color dialog box. You can select from Basic colors, or custom colors. You can define up to 16 custom colors by selecting or by editing the values for Hue, Saturation, Luminescence, and the Red, Green, and Blue.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



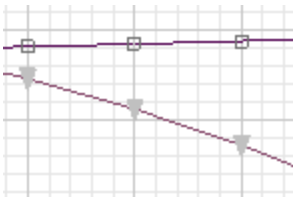
- Line style -- a drop-down menu lets you select Solid, Dot, Dash, or Dot-dash.
- Line width -- a text field lets you edit the numeric value.
- Trace type -- the drop-down menu contains entries for Continuous, Discrete, Bar-Zero, Bar Infinity, Stick Zero, Stick Infinity, Histogram, Step, and Stair.



Notice the difference between Stair and Digital is that each Stair centers on a data point with transitions halfway between points, and Digital transitions from each data point to the next value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The next four properties work together to define whether to show a symbol on data points, the symbol frequency, the symbol style, and whether to display the symbol as solid or hollow.



- Show Symbol -- whether to show a symbol at the data points on the line.
- Symbol Frequency -- how often to show symbols on the trace, based on the number of data points per symbol used. For example, specify 1 for one symbol per data point. Specify 10 for one symbol for every 10 data points.
- Symbol Style -- use a drop-down menu to select from box, circle, vertical ellipse, horizontal ellipse, vertical up triangle, vertical down triangle, horizontal left triangle, horizontal right triangle.
- Fill Symbol -- use the check box to set the symbol display as a solid or as hollow.
- Symbol Arrows -- use the check box to use arrows on the curve ends.

> **Note:**
>
> So that curves with single points always appear, Box is the default symbol.

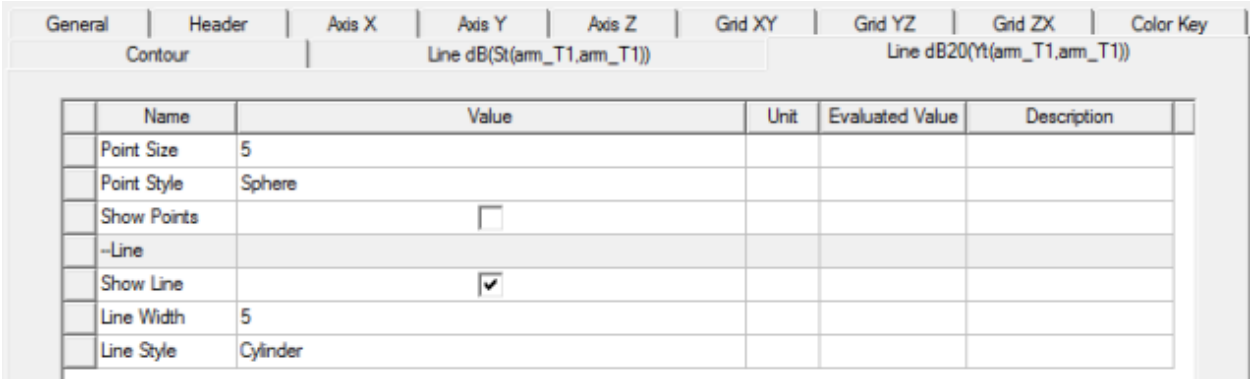
3. Edit the properties, if needed. Click OK to apply the changes and close the window.

To edit the display properties of a trace in a 3D report:

1. Click on the trace. This opens a Properties dialog for the plot with a tab named for the trace selected.
2. The editable properties include Point size, Point Style, whether to show points, whether to

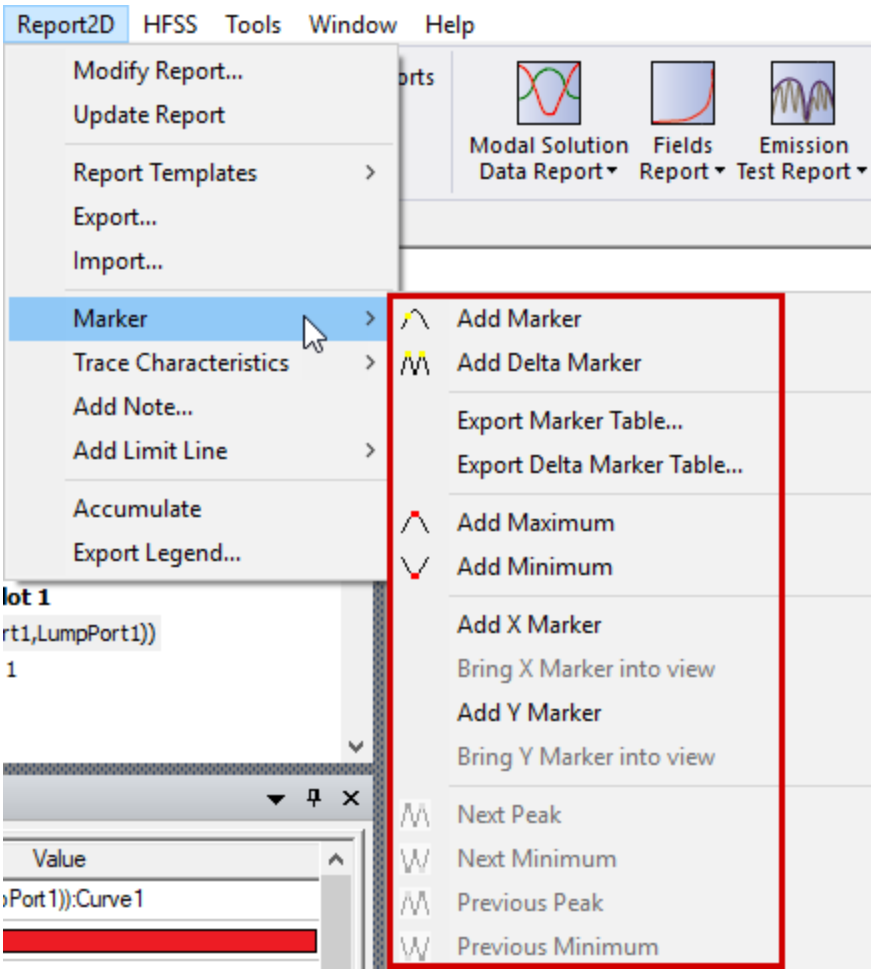
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

show line, line width, and line style.



Adding Data Markers to Traces

The Reporter includes **Report 2D** > **Marker** > menu commands and icons:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also access these commands from the shortcut menu that appears when you right-click inside a report plot window.

These command let you add markers to traces. A marker appears as "m*N*" at the marked point, where *N* increments from 1 as you place additional markers. Each marker can be selected and has editable properties including name, font, background and color. As you place markers, one or more marker legends may be displayed, depending on the **View> Active View Visibility** settings for the legends. The main marker legend appears in the upper left of the plot, and lists the marker names and their X and Y values in a table. You can control the number format for the table values via the properties window, general tab. Under *Marker/Other Number Format*, you can specify field width, precision, and whether to use scientific notation. This value is independent of the Axis tab number properties. A separate marker legend appears for Delta Markers, as described for the **Delta Marker** command.

When you enter Marker mode, the cursor arrow is accompanied by an "m" while a circle on the selected trace shows the current position for a potential marker.

To end Marker mode, right-click to display the shortcut menu, and select **End Marker Mode**.

The available Marker mode commands and associated icons are the following:

- **Marker** – this command lets you place a marker at an arbitrary point on a selected trace.
- **X Marker** – this command adds up to 10 movable markers at the origin of the plot with a vertical line rising from the X axis. Each added marker has its own color and editable properties. To move an X marker, click on the X label and drag it to the desired location. The label at the bottom of the line gives the X coordinate, and flag on the vertical line identifies the Y coordinate on the trace. A trace property lets you lock the drag feature to leave the marker in place. The X markers are cleared by the **Clear All** command.
- **Bring X Marker into view** – this command is enabled if an X Marker is not visible in the plot. It allows you to select from a list of existing X Markers to bring into view.
- **Y Marker** – this commands adds up to 10 Y Markers with a horizontal line extending from the Y axis. For more detail on Y Markers and their use, see Y Markers in stacked XY plots.
- **Bring Y Marker into view** – this command is enabled if a Y Marker is not visible in the plot. It allows you to select from a list of existing Y Markers to bring into view.
- **Maximum** – places a marker at the Maximum value on the selected trace.
- **Minimum** – places a marker at the Minimum value on the selected trace.
- **Delta Marker** – enters delta marker mode, placing a circle on the selected trace. Clicking on the trace sets an initial point and subsequent clicks on arbitrary points on the trace place additional markers until you leave marker mode. These markers have their own legend, which includes the following information for each pair of markers specified:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Delta(X)	Delta(Y)	Slope(Y)	InvSlope(Y)
d(m2,m3)	0.4700	1.8319	3.8976	0.2566

- **Next Peak** – moves a selected marker on the next peak on a trace. You must exit marker mode and select a marker to enable this command.
- **Next Minimum** – moves a selected marker to the next minimum on a selected trace. You must exit marker mode and select a marker to enable this command.
- **Previous Peak** – moves a selected marker on the previous peak on a selected trace. You must exit marker mode and select a marker to enable this command.
- **Previous Minimum** – places a marker on the previous minimum on a selected trace. You must exit marker mode and select a marker to enable this command.
- **Go to Start** (Right arrow) – moves a selected trace marker to the first data point. Enabled by leaving marker mode and selecting a marker.
- **Go to Previous** (Left arrow) – moves a selected trace marker to the previous data point.
- **Go to Next** – moves a selected trace marker to the next data point.
- **Go to End** – moves a selected trace marker to the last data point.
- **Next Curve** – selects the next curve in the report, based on the order in the trace legend.
- **Previous Curve** – selects the previous curve in the report, based on the order in the trace legend.
- **Clear All** – clears all markers on a report.

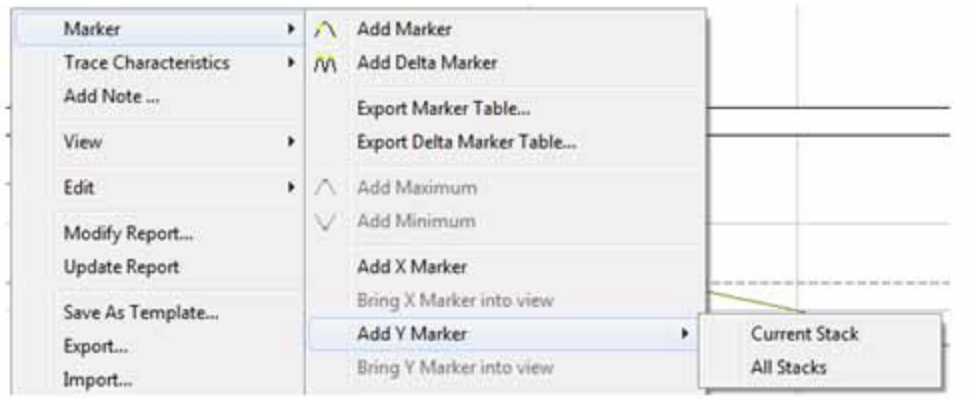
Y Markers in Stacked XY Plots

Y Markers allows for easy analysis and comparison of curves at a particular y-coordinate. Y Markers can be used to compare stacked curves.

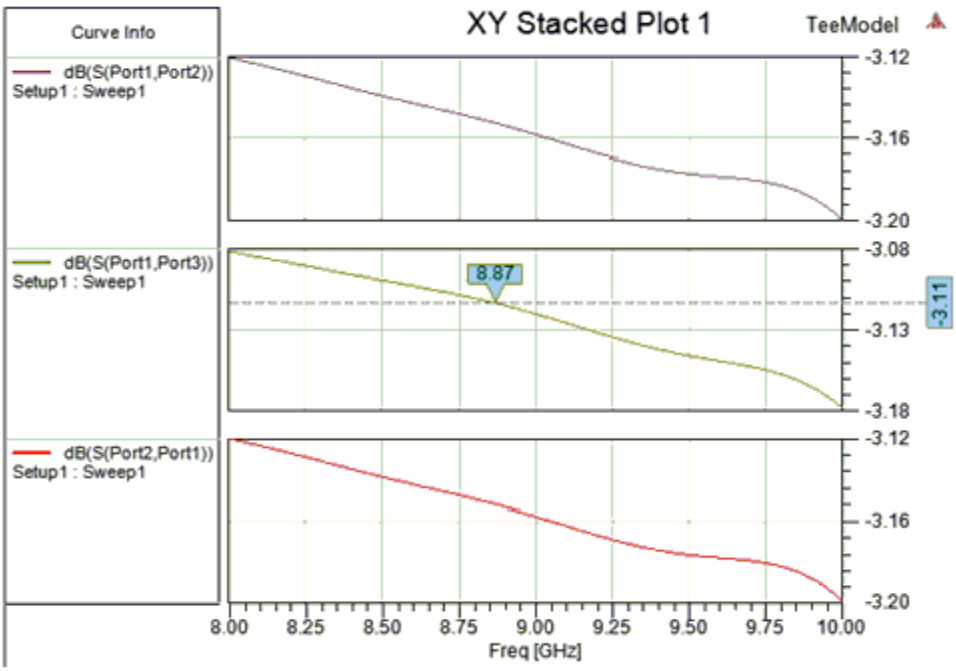
Creating Y Markers in Stacked Plots

There are two ways to create Y Markers in Stacked Plots. You can create a Y Marker for a particular stack or for all stacks. Right-clicking on any stack shows the following shortcut menu:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

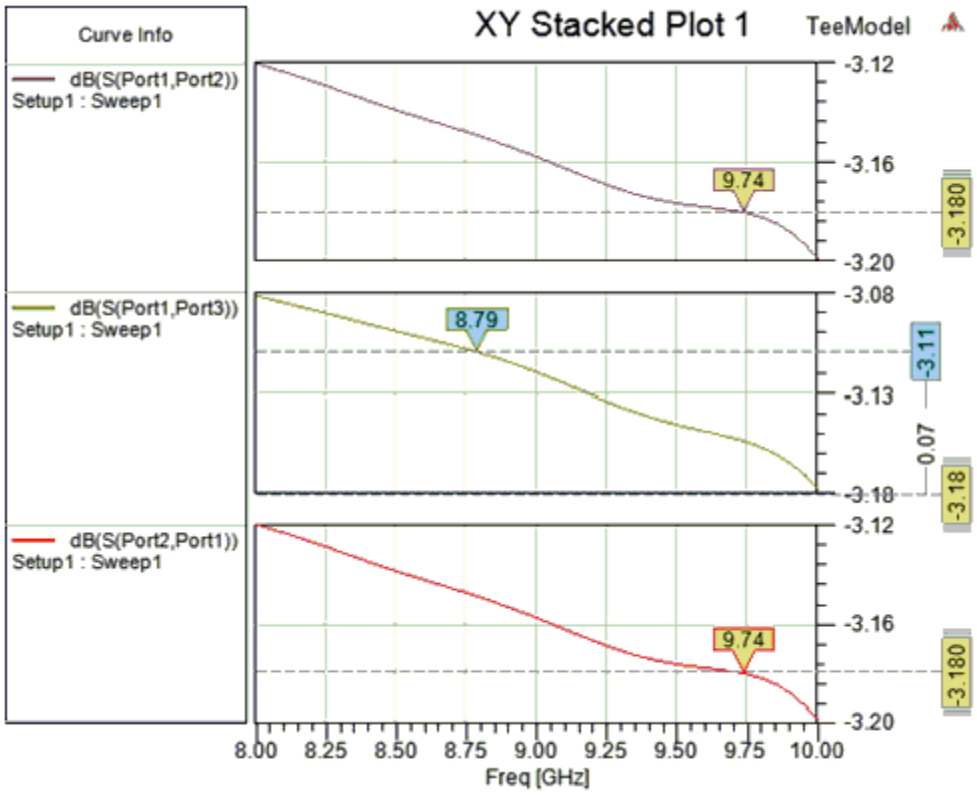


Add Y Marker > **Current Stack** creates a Y marker for the stack on which user performed right mouse button click. The following figure shows that a Y Marker was added to second stack only:



Add Y Marker > **All Stacks** creates one Y marker in each stack with same value. Initially this value is the minimum Y value of the Y ranges in all the stacks. This is shown is figure below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

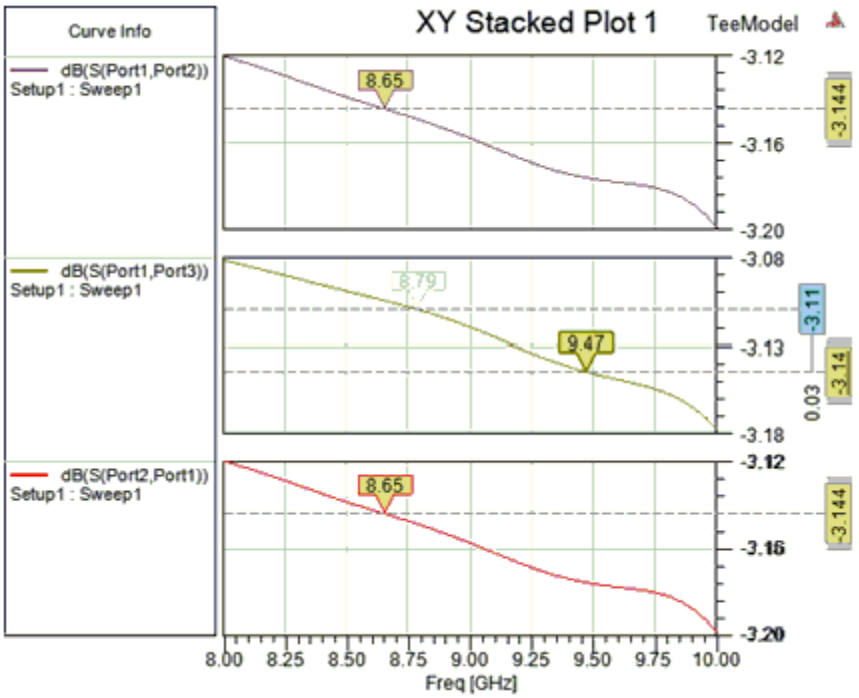


Notice that the Y Marker for All Stacks has a different appearance than the Y Marker for a particular stack, that is, it has double parallel lines above and below the Y Marker textbox.

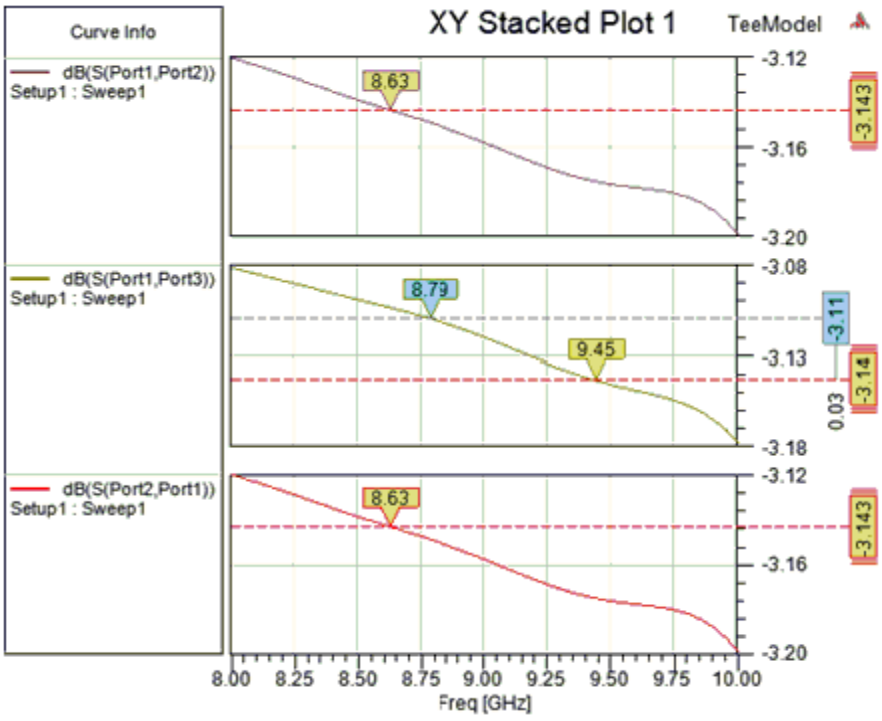
Synchronized Y Markers

All the "same" Y markers for all stacks are synchronized, that is to say that if one Y marker is dragged or it's value is changed, all the "same" Y markers in all the stacks will change their position too. The following figure shows that when Y marker in bottom stack was dragged, Y marker in top stacks moved as well:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



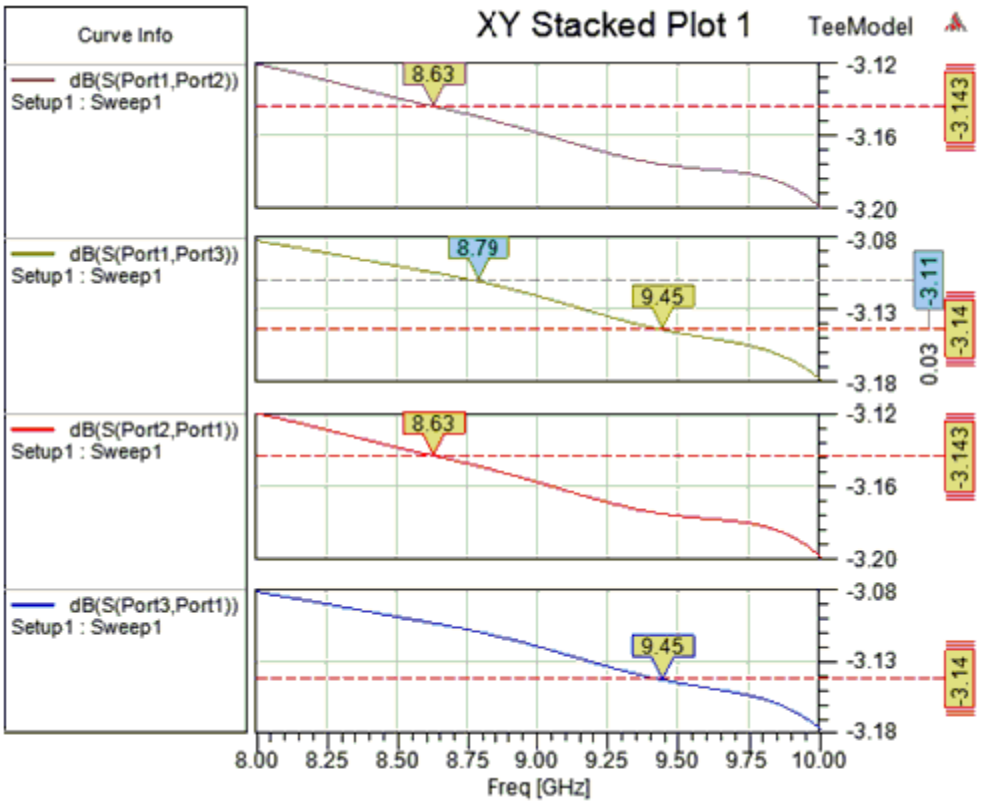
Also if a property of any one Y Marker is changed, all the "same" Y Markers show the change in property as well. For example the following figure shows that when the line color of a Y Marker in the top stack was changed to red color, a Y Marker in bottom stack show the same line color as well:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automatic Y Markers for the New Stack

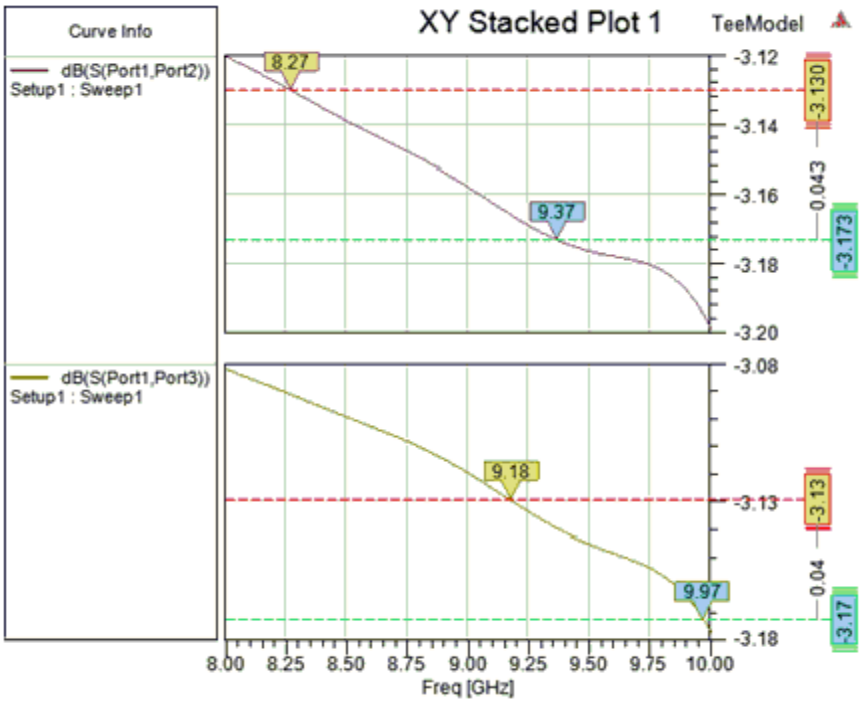
When a new curve is added to the plot, it gets all the Y Markers for all stacks in other stacks, excluding the Y Marker for particular stacks. The following figure shows that when the new curve "dB(S(Port3, Port1)" was added, a Y Marker was added to it with value -3.14 and it has all the same properties as other "same" Y Markers in other stacks:



Y Marker Delta Annotations

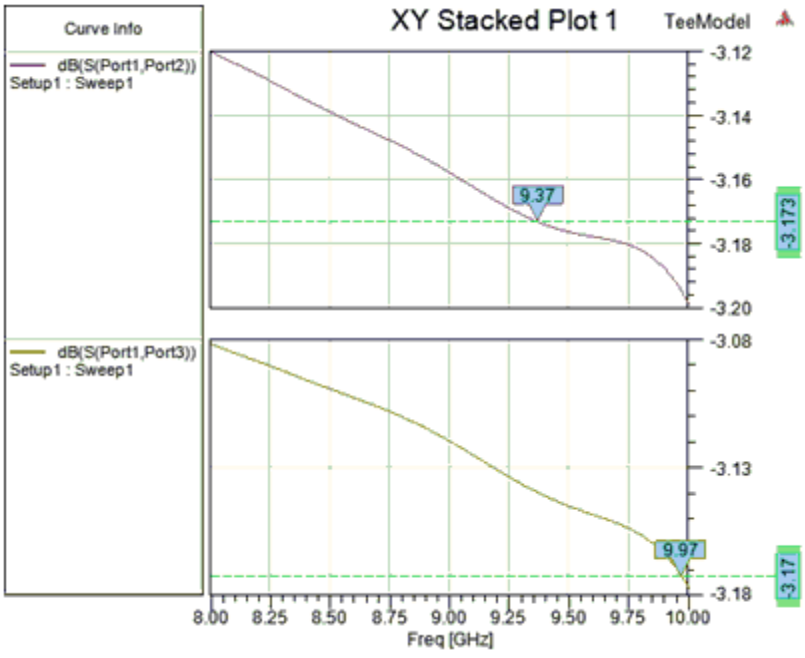
When two more Y Markers are present in a Stacked Eye Diagram, then delta annotations are shown between a pair of adjacent Y Markers in all the stacks, as shown in figure below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Deleting a Y Marker

To delete a Y Marker, select a Y Marker in any stack and press the Delete key. This action will also delete all the corresponding Y Markers in all the stacks. For example, when the Y Marker with value -3.13 (red Y Marker) was deleted from the bottom most stack, all of the corresponding Y Markers were also deleted:

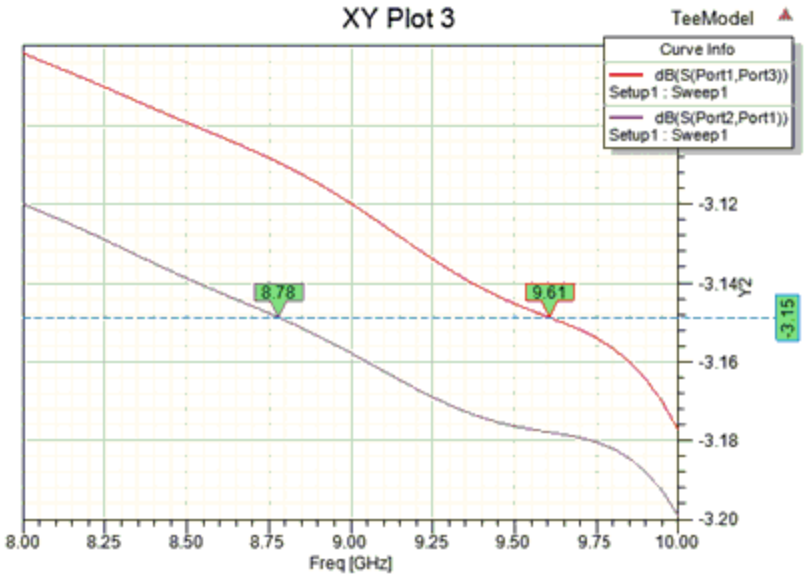


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

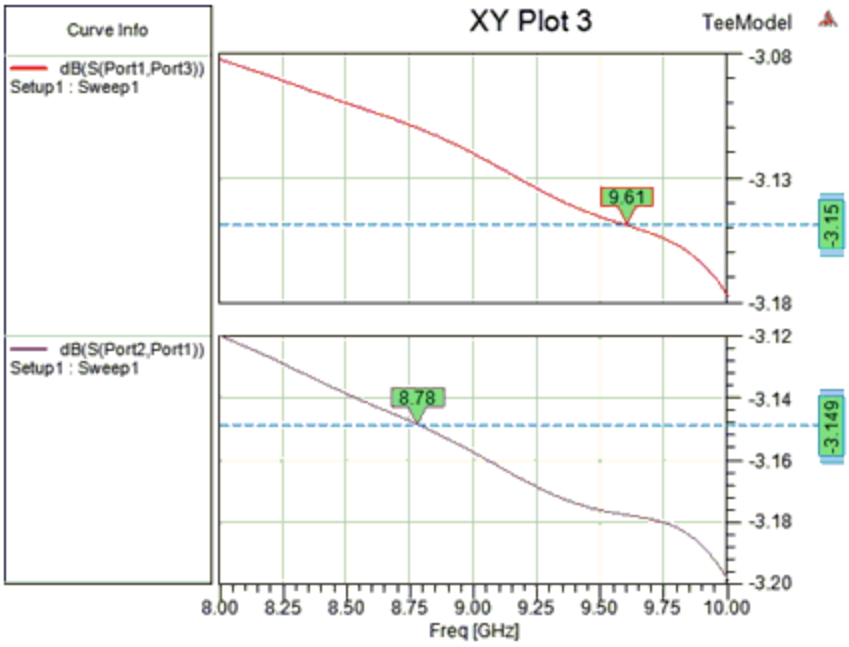
Note that on deleting a stack, Y Markers in other stacks are not affected.

Converting Rectangular XY Plot to Rectangular Stacked XY Plot

The following figure shows a Rectangular XY Plot with two curves and a Y Marker with value -3.15 (blue Y Marker):



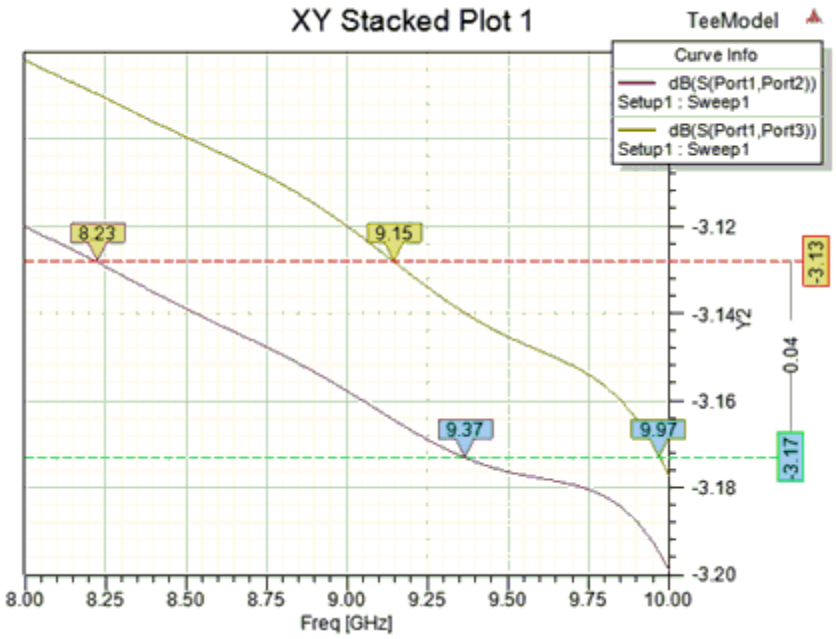
If you change the Display Type property of this plot to Rectangular Stacked Plot then a Rectangular Stacked XY Plot is created with each curve in its own stack and a Y Marker is shown in each stack with value -3.15 (blue Y Marker):



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Similarly when you change a Rectangular Stacked XY Plot to a Rectangular XY Plot then all the "same" Y Markers in all the stacks are shown as a single Y Marker in Rectangular XY Plot as shown in following figures:

The Rectangular Stacked XY Plot in the previous figure, when converted to Rectangular XY Plot, looks like the following figure:



Discarding Report Values Below a Specified Threshold

To prevent real small numbers from skewing a plot, you can discard small values (below a specifiable threshold).

1. Double-click the X or Y axis of interest on an open plot display.

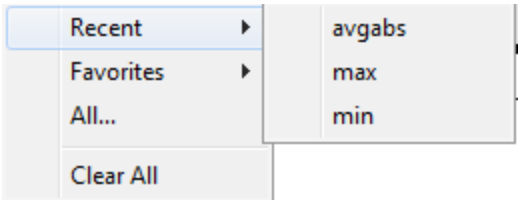
   This opens the **Properties** window for the Axis.

2. Under the **Axis** tab, use the scroll bar to find the **Specify Discard Values** property.
3. Click the check box to enable the property.
4. Enter a value in the **Discard Below** field. Units specified elsewhere in the Axis property are applied to this value. The Discard Below text box is inactive if the Specify Discard Values check box is not enabled.
5. Click **OK** to apply the Discard Values to the report.

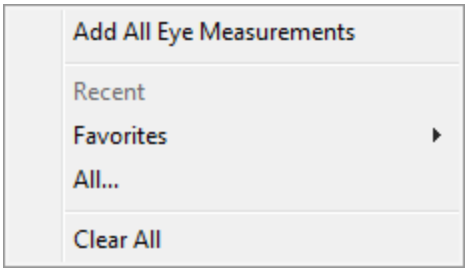
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adding Characteristics to a Trace

There are several options for adding characteristics to a trace. When you click **Report2D** > **Trace Characteristics**, or right-click a selected trace, the shortcut menu is displayed. The following example shows the menu with expanded **Recent** selections.



The following shows the shortcut menu for Eye Measurements, which includes the **Add All Eye Measurements** option. You can use this option to add all eye measurements at once.



Adding a Recently Used Trace Characteristic

If you recently used a characteristic, you can add it to a selected trace by selecting from a list of recently used characteristics. A maximum of 10 is displayed in the menu, and they are sorted alphabetically.

To add a recently used characteristic to a selected trace:

1. Select a trace in a report plot or legend.
2. Click **Report2D** > **Trace Characteristics**, or right-click the selected trace to display the short cut menu.
3. Select **Recent**, and then select the function you want. The specified characteristic is added to the trace.

Adding a Trace Characteristic from Favorites

You can add a trace characteristic to a selected trace by selecting from a list of favorites. A maximum of 10 is displayed in the menu, and they are sorted alphabetically.

To add a favorite characteristic to a selected trace:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select a trace in a report plot or legend.
2. Click **Report2D** > **Trace Characteristics**, or right-click the selected trace to display the short cut menu.
3. Select **Favorites** and then select the function you want. The specified characteristic is added to the trace.

Adding Trace Characteristics to your Favorites

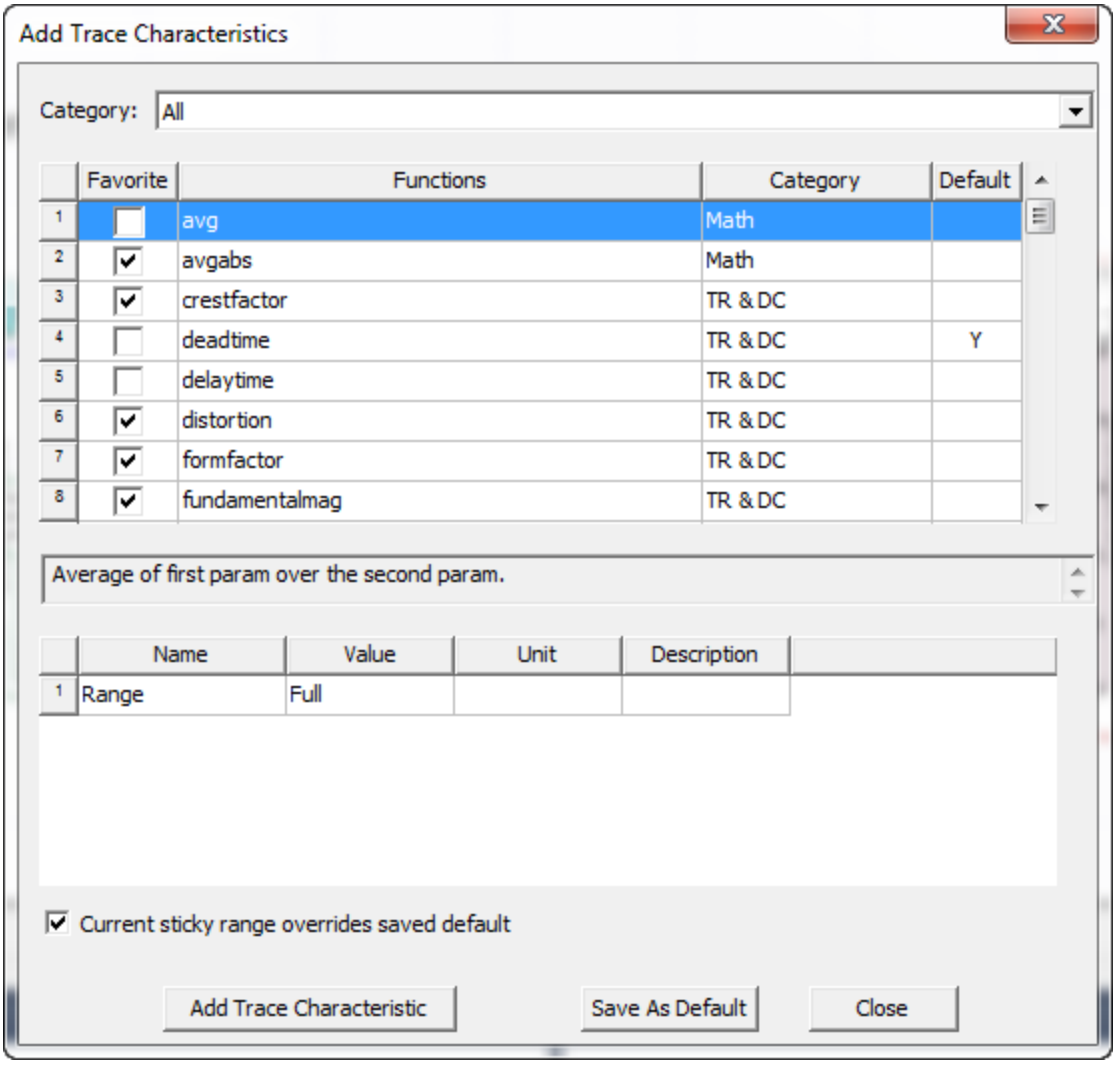
You can add trace characteristics to your list of favorites.

To add characteristics to your list of favorites:

1. Select a trace in a report plot or legend.
2. Click **Report2D** > **Trace Characteristics** or right-click the selected trace to display the short cut menu.
3. Select **All**.

   The *Add Trace Characteristics* dialog box displays.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click the **Favorite** check box in front of any function you want to add to your Favorites. You can define as many favorites as you need, but no more than ten are displayed in the menu, and they are displayed in alphabetical order.
5. Click **Close**. You can view the current favorites by selecting Favorites in the Category drop-down menu.

> **Note:**
>
> You can remove favorites by clearing the **Favorite** check box for one or more functions, and clicking **Close**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

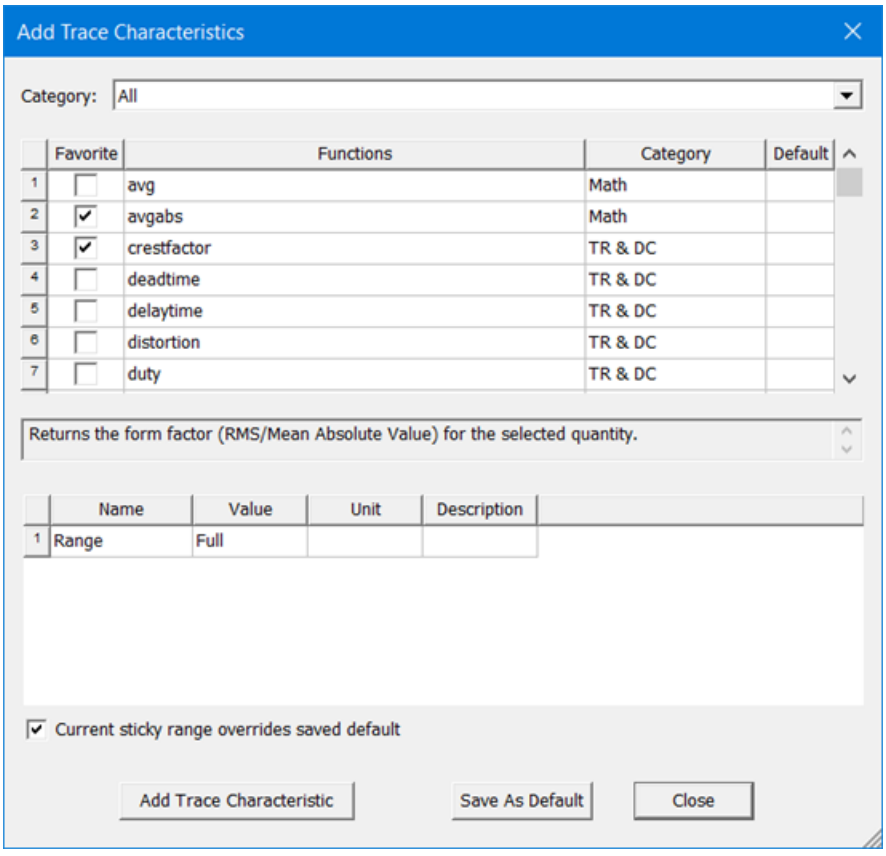
Adding Characteristics Using the Add Trace Characteristics Window

You can add characteristics to a selected trace by selecting from the **Add Trace Characteristics** window.

To add additional characteristics to a selected trace:

1. Select a trace in a report plot or legend.
2. Click **Report2D** > **Trace Characteristics**, or right-click the selected trace to display the short cut menu.
3. Select **All**.

   The **Add Trace Characteristics** window displays:



4. Select the desired **Category**.

   The available categories depend on the plot, and the selecting of a category displays its associated functions.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note: Categories** and **Functions** only appear if they are available for that plot.

Category	Functions
Recent	Displays the most recent functions used, sorted by the time they were added.
Favorites	Displays all favorites. The defaults are avg, max, min, and pk2pk.
All	Displays all available functions.
Math	avg, avgabs, integ, integabs, max, mean, min, pk2pk, pkavg, ripple, rms, rmsAC, stddev, sum, variance, XatYMax, XatYMin, XatYVal, XWidthAtYVal, YatXMax, YatXMin, YatXVal
PulseWidth	pulsefall9010, pulsefront1090, pulsefront3090, pulsemax, pulsemaxtime, pulsemin, pulsemintime, pulsetail50, pulsewidth5050, pw_minus, pw_minus_avg, pw_minus_max, pw_minus_min, pw_minus_rms, pw_plus, pw_plus_avg, pw_plus_max, pw_plus_min, pw_plus_rms
Overshoot/ Undershoot	overshoot, overshootAbs, undershoot, undershootAbs
TR & DC	crestfactor, deadtime, delaytime, distortion, formfactor, fundamentalmag, risetime, settlingtime
Error	iae, ise, itae, itse
Period	per, pmax, pmin, prms
AC	gainmargin, phasemargin, gaincrossover, phasecrossover, lowercutoff, uppercutoff, bandwidth, peakgain, peakgainfreq
Radiation	lSidelobeY, rSidelobeY, lSidelobeX, rSidelobeX, xdb10Beamwidth, xdb20Beamwidth
Eye Measurements	EyeLevelZero, EyeLevelOne, EyeAmplitude, EyeHeight, EyeSignalToNoise, EyeOpeningFactor, EyeWidth, EyeJitterP2P, EyeJitterRMS, EyeRiseTime, EyeFallTime, MinEyeWidth, MinEyeHeight
TDR	Shunt_C_in_pF, Series_L_in_nH

For a selected function, the *Add Trace Characteristics* dialog box displays the function's purpose in a text field. For a list of functions and their definitions, see the table in Defining Traces Using Range Functions.

5. Some categories and functions call for you to specify one or two additional values in a table. You can save these values using the **Save as Default** button. The Default column shows a Y if there is a saved default value for the function.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Select the **Current sticky range overrides saved default** check box if you do not want the range value in the table to be changed when the function selection is changed: the current range value becomes the "sticky range." If the check box is not checked, the range value is updated from the saved default values and becomes a new sticky range.
7. Click the **AddTrace Characteristic** button to add the specified characteristics to the trace.
8. Click **Close**.

Removing All Trace Characteristics

1. Select a trace in a report plot or legend.
2. Click **Report2D** > **Trace Characteristics** or right-click the selected trace to display the shortcut menu.
3. Select **Trace Characteristics** > **Clear All**.

Trace characteristics are cleared from the selected trace.

Removing Traces

You can remove traces from the traces list in the following ways:

To *remove one trace* from the report:

- Select the trace you want to remove from the Project tree, and then click **Delete**.

To *remove all traces* from the report:

- Select all the traces and click **Delete**.

Copy and Paste of Report and Trace Definitions

You can copy and paste report and individual trace definitions within a single design or across designs. The report or trace definition will be evaluated within the context of the target design or report.

> **Note:**
>
> - If the report or trace definition contains properties that do not exist in the target design (for example, a port name) an error will be posted that indicates a solution does not exist for this trace
> - You must copy and paste trace definitions between the same report types. For example, you cannot copy a trace from a Modal Solution Data report and paste it in a Far Fields report.

To copy a Report Definition:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Right-click the report name in the Project Manager and select **Copy Definition** from the shortcut menu.

To paste the Report Definition:

Right-click Results in the Project Manager of the target design and select **Paste**.

A new report is created and it contains the copied definitions.

To copy an individual Trace Definition(s):

Right-click the trace or traces under a report name in the project tree and select **Copy Definition**.

To paste the Trace Definition(s):

Right-click the report in the target design to which you would like to copy the trace or traces and select **Paste**.

A new trace(s) is added to the report and it contains the copied trace definition(s).

> **Note:**
>
> If you copy and paste a report or trace definition to a design that contains a definition with the same name, then an incremented number is appended to the pasted report or trace name.

Copy and Paste of Report and Trace Data

You can copy and paste report and individual trace data within a single design or across designs. The report and trace definitions and all underlying data within the report or trace are copied and pasted to the target design or report.

To copy all data from a report:

Right-click the report name in the project tree and select **Copy Data**, or use the menu bar **Edit** > **Copy Data**, or right-click within a plot to display a shortcut menu with **Copy Data**.

To paste copied report data:

Right-click Results in the project tree of the target design and select **Paste**.

To copy data from an individual trace(s) in a report:

Right-click the trace or traces under a report name in the project tree and select **Copy Data**.

To paste copied trace data:

Right-click the report in the target design to which you would like to copy the trace data and select **Paste**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

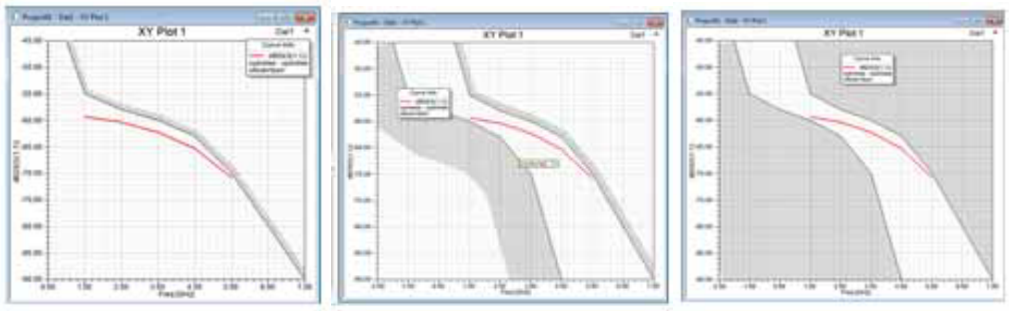
Note:

If you copy and paste report or trace data which contains the same name definition as a report or trace in the target design then an incremented number will be appended to the pasted name.

Limit Lines in Cartesian Plots

Limit lines are simple graphical representation of constraints on XY plots. These are modeled as a sequence of XY point pairs, or as offsets from a selected curve. You can designate a single limit line to delineate an upper limit, or two lines to delineate upper and lower limits, or upper and lower offset lines simultaneously.

You can control the display properties of the line including color and hatch width in pixels. These lines are available only on XY plots (and not on the XY-like plots: bode, stacked etc)

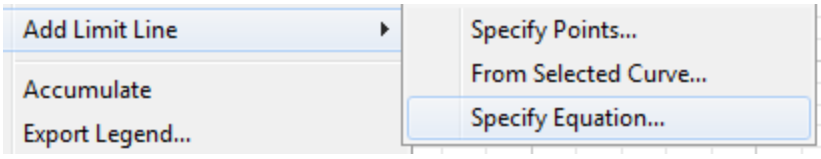


Note:

Limit lines are available only on Rectangular (XY) plots, not on XY-like plots such as Bode or Rectangular Stacked. On 2D Plots, the axes extents are based on the extents of the curves.

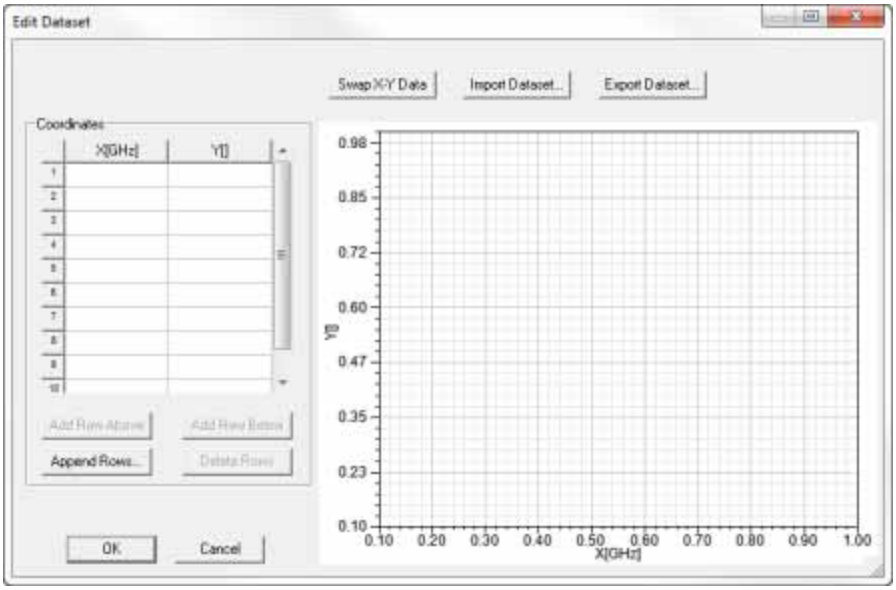
To create a limit line:

1. Click **Report2D** > **Add Limit Line** or right-click an XY plot and select **Add Limit Line...** from the Context menu. You then select whether to **Specify Points**, or **From a Selected Curve**, or **Specify Equation**.

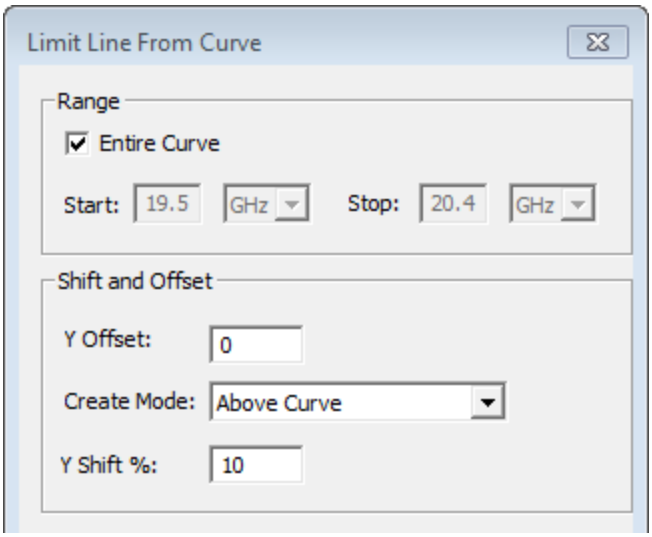


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select **Specify Points** to open an *Edit Dataset* dialog box so that you can specify points.

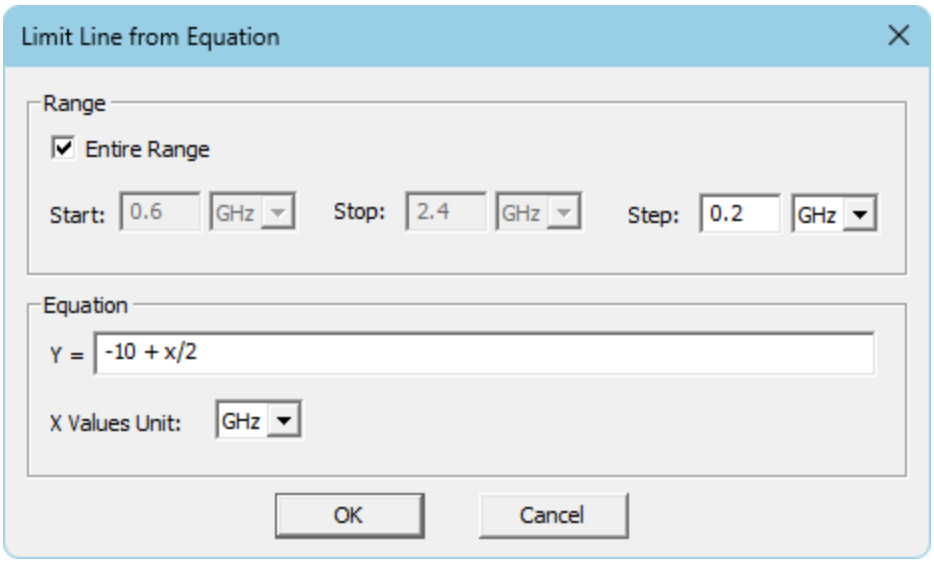


Select **From Selected Curve** to open the *Limit Line From Curve* dialog box.



Select **Specify Equation** to open the *Limit Line from Equation* dialog box. Specify the desired **Equation** (y as a function of x). Choose the **X Values Unit** from the drop-down menu. The available units depend on the type of sweep quantity (frequency, distance, and so on). The default choice is the current X axis unit for the plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. You can use the *Edit Dataset* dialog box to
   - enter the EY values directly.
   - import XY values from a .tab file.
   - export Dataset to a file.

   If you require additional data points, you can use the buttons to **Append Rows** to the Coordinates table. If you select a row in the Coordinates table, you can then use the buttons to **Add Row Above**, **Add Row Below** the selected rows, or **Delete Rows**.

   You can use Shift+click to select multiple adjacent rows, or Ctrl+click to select any rows for deletion.

   > **Note:**
   >
   > Each limit line is associated with a particular Y axis (because it has to be scaled the same way as all the curves associated with the axis, follow its log/linear scale and so on). This Y axis association defaults to the first available Y axis when the limit line is created. However, if the plot contains multiple Y axes, it can be associated with a different Y axis later via its properties tab.

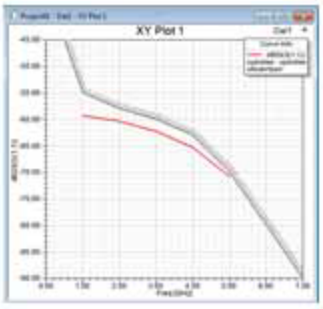
3. You can use the *Limit Line From Curve* dialog box to create a limit line with a:
   - Range using the Entire Curve, or a specified Start and Stop.

   If you uncheck “Entire Curve,” the Start and Stop fields are enabled and initialized based on the zoom level.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

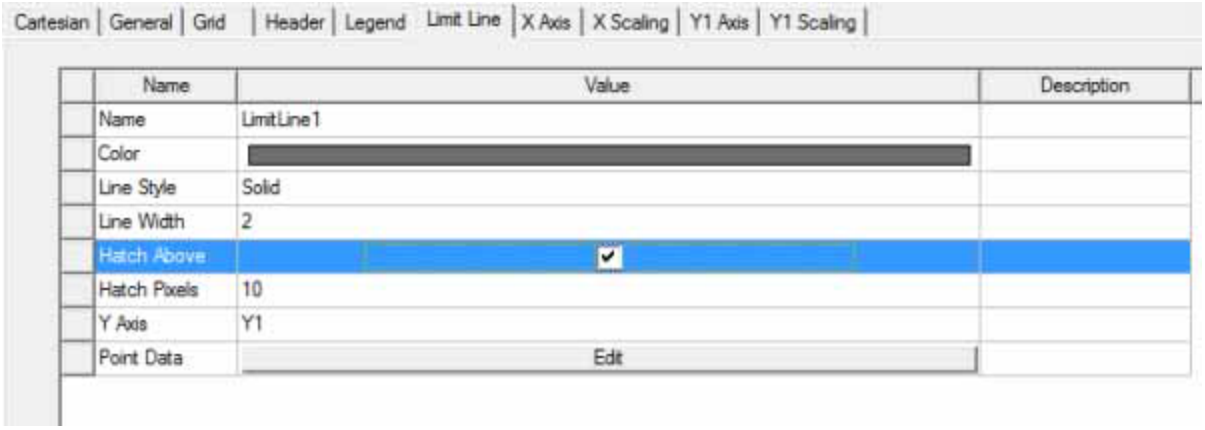
- Shift and Offset relative to the Y, either as a Y Offset value or as a Y Shift %.
- Create Mode as Above Curve, Below Curve, or Above and Below Curve.

4. You can use the *Limit Line from Equation* dialog box to create a limit line based on an equation in the form Y = f(x).
    - Only x variable is allowed
    - x value is linearly sampled and f(x) is evaluated to get y. By default, the entire range for plot is covered. You can uncheck **Entire Range** to specify Start and Stop. You can always edit Step and unit values for x.
    - If y results in NaN, an error is indicated and limit line is shown.
    - If y results in Inf value, then it maps to the MapInfValue specified on its associated y axis.
    - Selecting the limit line added in such a way should show: Start, Stop, Step, and Equation in limit line property window. You can edit these values in the Properties window.
5. Once you click **OK**, the limit line, or lines, you define are added to the plot. Each limit line divides the plot into two regions within the context of its length. By default, the upper region is hatched to designate constraint violation.



6. You can select the limit line in the plot to edit its properties via the **Limit Line** tab of the plot properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

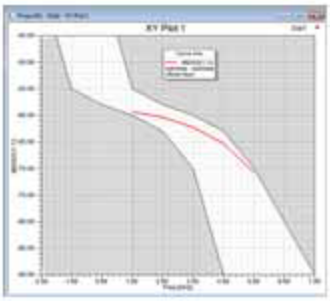


- Line properties: Color, Style and Width
- Y axis association
- Hatch properties
- Hatch width in pixels
- Hatch direction (hatch above or below the Limit line)
- The point definition of the limit line itself.

For a limit line specified as an Equation, the Properties also include:

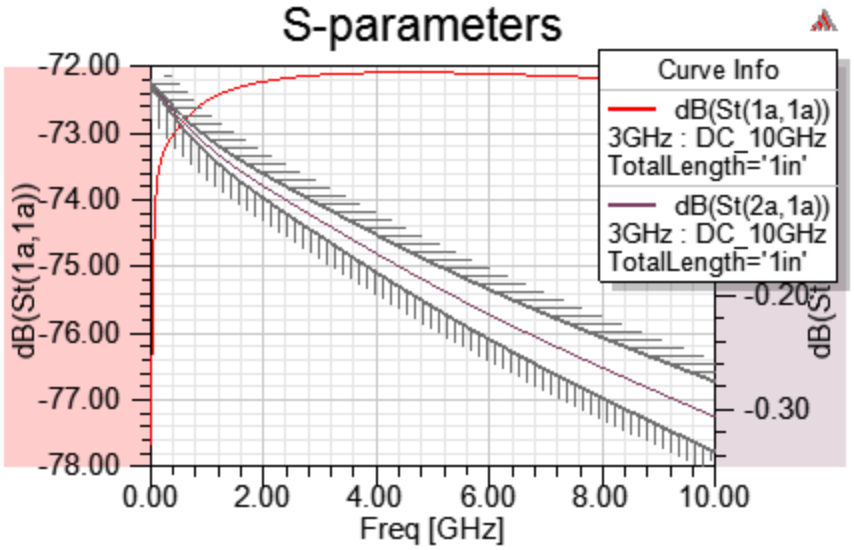
- Start, Stop, Step, and Equation values.

7. If you add a second limit line, you can designate it as hatch below by unchecking **Hatch Above** to produce a tunnel marking upper and lower constraints.



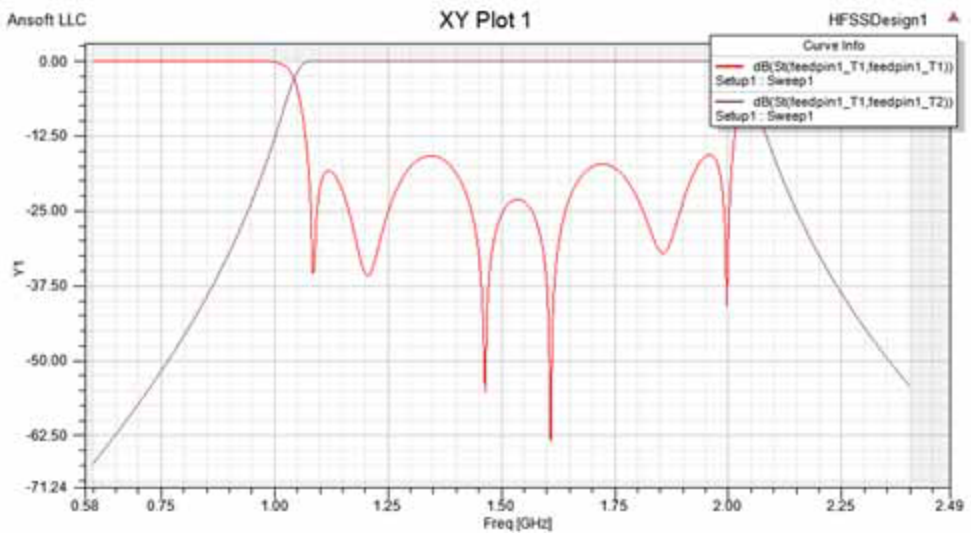
The following example shows the a limit line from curve plot, where the Hatch Above property for the lower limit line has been unchecked.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



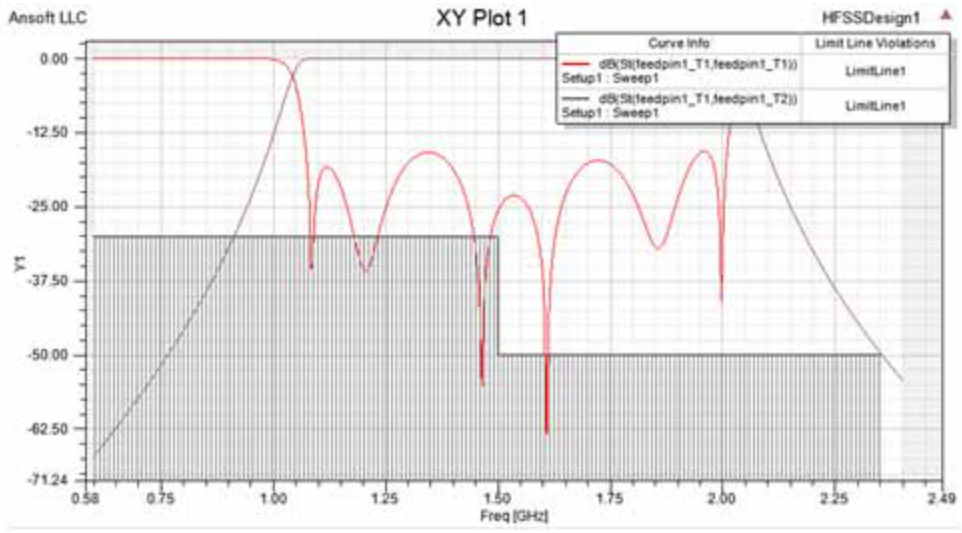
Limit Line Violations

You can use a plotting feature to help you discern whether a curve violates a limit line or not. Consider following plot which shows two curves:

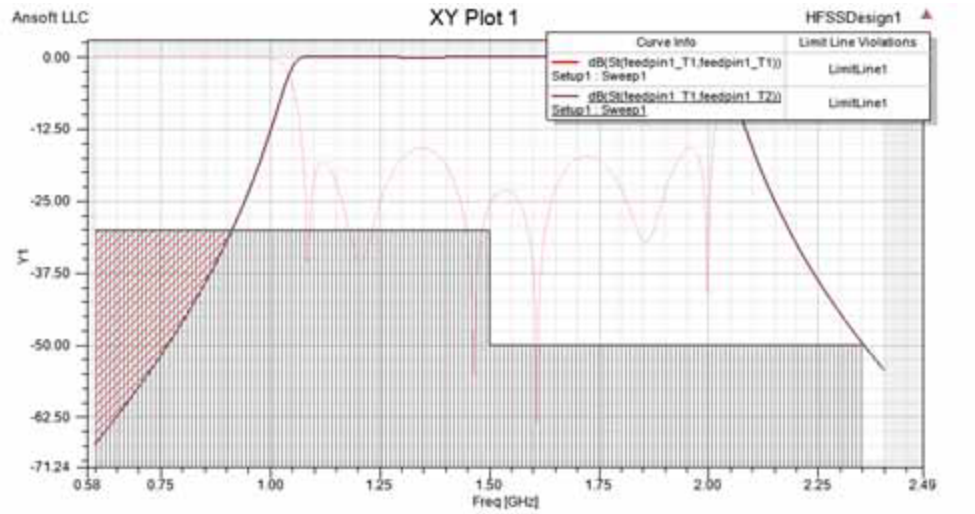


Suppose that the response cannot be below -30 dB until 1.5 GHz, and cannot be below -50 dB at 1.5 GHz or higher. You add a limit line for this requirement in the plot using 'Add a Limit Line' functionality. The plot automatically calculates whether a curve violates this requirement, that is, the limit line and show it in the legends window, as shown in following figure:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



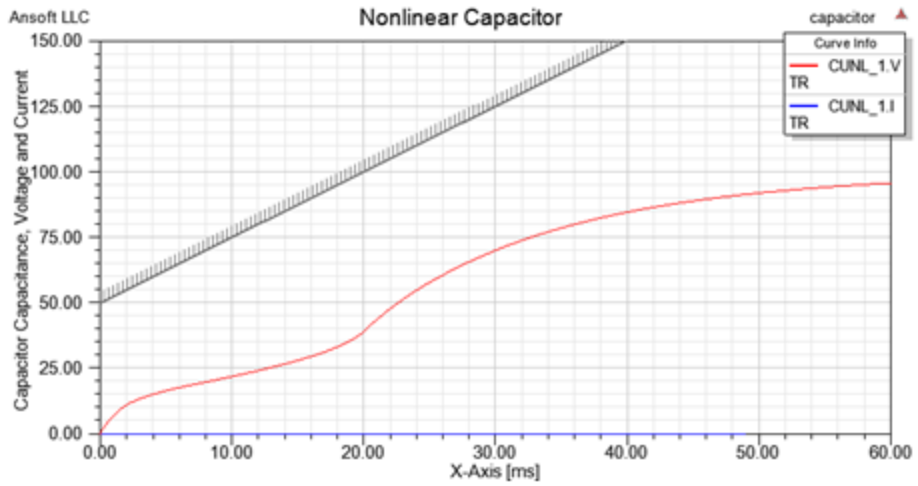
If a curve is selected, then the plot shows the region of the curve that violates the limit line (shaded with slanted red lines), as shown in following figures:



Note:

As shown in the following figure, if no curve violates a limit line, then the limit line hatching is restrained to 10 pixels in length. The minimal hatching keeps the focus on the curve traces rather than the limit lines.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Error Handling

If an error is encountered while calculating Limit Line Violations for a particular plot trace, the *Curve Info* window shows "NaN (*<Limit Line name>*)" under the *Limit Line Violations* column for that curve. For example, if a value is not available or goes to infinity at a particular point, the program is unable to evaluate that point. One of the coordinates is "Not-a-Number" (NaN).

Sweeping a Variable in a Report

In HFSS, a swept variable is a variable that typically has more than one value. You can plot any calculated or derived quantity against one or more of the swept variable's values.

For large projects or projects with many variables, you may obtain faster post processing before generating a solution by selecting which variables function as Sweep variables. Only the variables with Sweep enabled are indexed for post processing. See Adding a Design Variable and Adding a Project variable.

To specify the swept variable values to plot a selected quantity against:

1. In the **Report** dialog box, select the variable from the X (Primary Sweep) drop-down menu.
2. To modify the values that will be plotted for a variable:
    a. Click the ellipsis **[...]** button on the **X (Primary Sweep)** line of the **Report** dialog to displays a pop-up list of the possible values.
    b. Select **Use all values** or click the Edited button to display a dialog that lets you specify the sweeps to use.

All of the selected variable's values will be plotted.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sweeping Values Across a Distance

1. If you are plotting a field quantity along a line, define a polyline object in the problem region.

   If you are plotting a near-field quantity along a line, set up a near-field line.

2. In the *Report* dialog box, click the line geometry of interest in the **Geometry** list.
3. Specify the quantities you want to plot along the axes.
4. For the **X (Primary Sweep)**, select the **Distance** variable.

   The values at which the selected quantity or quantities will be plotted are listed to the right. By default, a post-processing polyline object is divided into 100 equally spaced points.

5. For Near field, to plot the selected quantity or quantities at every point on the line, select **All Values**.

   For Near field, to plot the selected quantity, or quantities, at specific points on the line, clear the **All Values** option, and then select the point values on which you want to plot.

> **Note:**
>
> All maximum near-field data calculated by HFSS is at their maximum over the selected line object; if you plot the parameter over a sweep of values, the parameter will have the same value at each point on the plot.

Sweeping Values Across a Sphere

1. Set up a near-field sphere or a far-field infinite sphere.
2. In the **Report** dialog box, click the sphere geometry of interest in the **Geometry** list.
3. For the **Sweeps** variable corresponding to **phi**, select the ellipsis [...] button.

This displays a small dialog.

4. Clear the Use all values check box to enable selection and editing of the sweep values.

All of the possible values for the phi variable are listed in the dialog box. The values are the result of the range of phi you specified during the infinite sphere's setup. To modify the values of phi to be plotted across the sphere, do the following:

a. Click **Edit Sweep**.
b. Specify the following information:

Step or Count	Whether to sweep by steps, or by linear count, decade count, octave count, or exponential count.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Start Value	The point where the rotation of phi begins.
End Value	The point where the rotation of phi ends.
Step or Count	The number of values between the start value and the end value.

c. Click **Update Values**, and then click **OK**.

The values listed are updated to reflect the new number of points.

5. To plot the selected quantity or quantities at every value of phi, select **All Values**.

To plot the selected quantity or quantities at specific values of phi, clear the **All Values** option, and then select the phi values at which you want to plot.

6. For the **Sweeps** variable corresponding to **theta**, follow steps 4 and 5 for modifying the values of theta, if necessary, and specifying the theta values at which to plot the selected quantity or quantities.

> **Note:**
>
> All antenna parameters and maximum far-field data calculated by HFSS is at their maximum over the selected object; if you plot the parameter over a sweep of values, the parameter will have the same value at each point on the plot.

Selecting a Function for a Plot

The value of a quantity being plotted depends upon its mathematical function, which you select from the **Trace** tab **Function** list in the *Report* dialog box. The available, valid functions depend on the type of quantity (real or complex) that is being plotted. The function is applied to the quantity which is implicitly defined by all the swept and current variables. For example, "S(11)" is the value of the S-parameter for every swept combination of variables (e.g., "height", "frequency"). (A smaller set of functions appears for the Function list in the Output Variables dialog.)

These functions can also be applied to previously specified Quantities and Functions as **Range Functions** when using the Set Range Function dialog box.

Some of these functions can operate along an entire curve. These are: min, max, integ, avg, rms, pk2pk, cang_deg and cang_rad.

You can select from the following functions in the **Trace** tab **Function** list or type them directly into the Yor X field, if necessary.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

abs	Absolute value of the simulation quantity which results in a number that is always positive.
acos	Arc cosine (the inverse function of a cosine).
acosh	Inverse hyperbolic arc cosine.
ang	Magnitude of an angle.
ang_deg	Angle (phase) of a complex number, cut at +/-180. Returns angular values in degree units, and not suitable for use in Optimetrics, which works in SI values and evaluates ang_deg expressions in radians. See ang_deg_val.
ang_deg_val	Angle (phase) of a complex number in unitless degree values, and suitable for use in Optimetrics, which works in SI values. Returns simple numbers.
ang_rad	Angle in radians.
arg	Argument of a complex number. It is the angle the complex number makes with the positive x axis. Same as **ang_deg**.
asin	Arc sine (the inverse function of sine).
asinh	Inverse hyperbolic sine.
atan	Arc tangent (the inverse function of a tan).
atanh	Inverse hyperbolic tan.
atan2	Two argument function. For non-0 x,y, the function returns the angle between the + x-axis and the given x,y coordinates.
avg	Returns the average of the values of the selected quantity. avg = (Area between the curve and the X-axis) / (X length of the curve)
avgabs	Returns the mean of the absolute value of the selected quantity.
bandwidth	Returns the 3dB bandwidth of the selected simulation quantity. For bandwidth, the calculation is based on 3dB below the maximum peak.
cang_deg	Cumulative angle (phase) of the first parameter (a complex number) in degrees, along the second parameter (typically sweep variable). Returns a double precision value cut at +/-180. Returns angular values in degree units, and not suitable for use in Optimetrics, which works in SI values and evaluates cang_deg expressions in radians. See cang_deg_val.
cang_deg_val	Cumulative angle (phase) of the first parameter of the selected simulation quantity in unitless degree values and suitable for use in Optimetrics, which works in SI values. . Returns simple numbers.
cang_rad	Cumulative angle of the first parameter in radians along a second parameter (typically a sweep variable). Returns a double precision value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

cmplx(*re*, *im*)	A complex number, where *re* is the real part and *im* is the imaginary part.
conjg	Conjugate of the complex number.
cos	Cosine.
cosh	Hyperbolic cosine.
crestfactor	Returns the crest factor (peak/RMS) for the selected quantity.
cum_integ	The cumulative integral function returns a set of values that have the same length as the original set of points (the first element will always be zero). Element *I* of the set returned by cum_integ is the integral of elements 1 through *I* of the original data set.
cum_sum	The cumulative sum function returns a data set that has the same length as the original set of points. Element *I* of the set returned by cum_sum is the sum of elements 1 through *I* of the original data set.
dB(x)	20*log10(|x|) to base 10.
dBc	Decibels relative to the carrier. It is the power ratio of the signal to a carrier signal. Gives the relative signal strength.
dBm(x)	10*log10(|x|) +30.
dBm	(for electric field quantities) is computed as: 20.0*log10(x) + 60.0
dBu	(for electric field quantities) is computed as: 20.0*log10(x) + 120.0
dBW(x)	10*log10(|x|).
dB10	10*log(|x|) to base 10.
dB10normalize	10*log [normalize(mag(x))].
dB20	20*log(x) to base 10.
dB20normalize	20*log [normalize(mag(x))].
deadtime	Obtains the latest time when the qtyl is within a tolerance of zero.
delaytime	Obtains the time from zero to 50% of the target point.
degel	Conversion from degrees electrical to seconds with respect to Hz.
deriv	Derivative of a given parameter.
distortion	Returns the total distortion for the selected simulation quantity and an additional argument frequency, which is the frequency in Hz at which to calculate the fundamental RMS of the simulation quantity.
even	Returns 1 if integer part of the number is even; returns 0 otherwise.
exp	Exponential function (the natural anti-logarithm) of the simulation quantity.
fmod	nReturns the double precision remainder of x/y.
formfactor	Returns the form factor (RMS/Mean Absolute Value) for the selected quantity.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fundamentalmag	Returns the RMS value of the fundamental frequency for the selected quantity, and an additional argument, Frequency, which specifies the fundamental frequency.
gaincrossover	Returns the gain crossover frequency (where the gain is 0 dB) of the selected simulation quantity in Hz.
gainmargin	Returns the gain margin in dB at the phase crossover frequency of the selected simulation quantity. It also requires a reference simulation quantity to which the measured quantity is compared and the AC magnitude and phase angle of the reference quantity. These are entered as the arguments Reference Channel, Base Source Magnitude, and Base Source Angle.
iae	Returns the integral of the absolute deviation of the selected quantity from a target value that is entered via the additional argument.
if	if(cond_exp,true_exp, false_exp).
im	Imaginary part of the complex number.
int	Truncated integer function.
integ	Integral of the selected quantity. Uses trapezoidal area.
integabs	Absolute value of integral.
ise	Returns the integral of the squared deviation of the selected quantity from a target value that is entered via an additional argument.
itae	Returns the time-weighted squared deviation of the selected quantity from a target value that is entered via an additional argument.
itse	Returns the time-weighted squared deviation of the selected quantity from a target value that is entered via an additional argument. To use this function, you need to open the Add Trace Characteristics dialog and select the Error category.
j0	Bessel function of the first kind (0^{th} order).
j1	Bessel function of the first kind (1^{st} order).
jn	Bessel function of the first kind (nth order).
ln	Natural logarithm.
log	Natural logarithm (same as ln).
log10	Logarithm base 10.
lowercutoff	Returns the lower 3dB frequency of the selected simulation channel in Hertz.
lsidelobeX	The 'x' value for the left side lobe: the next highest value to the left of the max value.
lsidelobeY	The 'y' value for the left side lobe: the next highest value to the left of the max value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mag	Magnitude of the complex number.
max	Returns maximum value of the simulation quantity.
max_swp	Returns maximum value of a sweep.
max2	Maximum value of the two simulation quantities. For example, **max2 (a,b)** will plot maximum of **a** and **b** for a particular instance.
mean	Returns the average in the set of quantities selected. mean = sum(all y-value) / (number of y-values)
min	Returns the minimum value of the simulation quantity.
min_swp	Returns the minimum value of a sweep.
min2	Minimum value of the two simulation quantities. For example, **min2 (a,b)** will plot minimum of **a** and **b** for a particular instance.
mod	Returns the double precision modulus (x - floor(x/y)*y). Not supported in solvers.
nint	Nearest integer.
none	Returns null value.
normalize	Divides each value within a trace by the maximum value of the trace. ex. normalize(mag(x)).
odd	Returns 1 if integer part of the number is odd; returns 0 otherwise.
overshoot	Calculates peak overshoot given a threshold value and number of evenly spaced points over entire time range.
peakgain	Returns the peak value of gain of the selected simulation quantity in dB.
peakgainfreq	Returns the frequency in Hz at which the peak gain of the selected simulation quantity occurs.
polar	Coverts the complex number in rectangular co-ordinates to polar co-ordinates.
per	Returns the period of a simulation quantity.
phasecrossover	Returns the phase crossover frequency, at which the phase is -180 degrees, in Hz for the selected simulation quantity.
phasemargin	Returns the phase angle in degrees at the gain crossover frequency of the selected simulation quantity.
pk2pk	Peak to peak. Difference between max and min of the first parameter over the second parameter. Returns the peak-to-peak value for the selected simulation quantity.
pkavg	Returns the ratio of the peak to peak-to-average for the selected quantity.
pmax	Maximum period of the selected simulation quantity.
pmin	Minimum period of the selected simulation quantity.
pow	Raises x to the power of y; pow(x,y).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

prms	Period Root Mean Square.
pulsefall9010	Returns the pulse fall time of the selected quantity according to the 90%-10% estimate.
pulsefront1090	Returns the pulse front time of the selected quantity according to the 10%-90% estimate.
pulsefront3090	Returns the pulse front time of the selected quantity according to the 30%-90% estimate.
pulsemax	Returns the pulse maximum from the front and tail estimates for the selected quantity.
pulsemaxtime	Returns the time at which the maximum pulse value of the selected quantity is reached.
pulsemin	Returns the pulse minimum from the front and tail estimates for the selected quantity.
pulsemintime	Returns the time at which the minimum pulse value of the selected quantity is reached.
pulsetail50	Returns the pulse tail time of the selected quantity from the virtual peak to 50%.
pulsewidth5050	Returns the pulse width of the selected quantity as measured from the 50% points on the pulse front and pulse tail.
pwl	Piecewise Linear.
pwl_periodic	Piecewise Linear for periodic extrapolation on x.
pwlx	Piecewise Linear x with linear extrapolation on x.
pw_minus	Pulse width of the first negative pulse.
pw_minus_avg	Returns the average of the negative pulse width input stream.
pw_minus_max	Returns the maximum pulse width of the negative pulse of input stream.
pw_minus_min	Returns the minimum pulse width of the negative pulse of input stream.
pw_minus_rms	RMS of the negative pulse width input stream.
pw_plus	Pulse width of the first positive pulse.
pw_plus_avg	Average of the positive pulse width input stream.
pw_plus_max	Max. Pulse width of the positive pulse of input stream.
pw_plus_min	Min. Pulse width of the positive pulse of input stream.
pw_plus_rms	RMS of the positive pulse width input stream.
re	Real part of the complex number.
rect	Converts the complex number in polar to rectangular co-ordinates.
rem	Returns the fractional part of a decimal number such that rem(x) = x-int (x) Syntax: rem(x)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ripple	Returns the ripple factor (AC RMS/Mean) for the selected quantity.
risetime	Obtains the time taken to go from 10% to 90% of target point.
rms	Returns the root mean square value of the selected quantity.
rmsAC	Returns the AC RMS for the selected quantity.
root	nth root function.
rSidelobeX	Returns the X value of right side-lobe occurrence.
rSidelobeY	Returns the Y value of right side-lobe occurrence.
settlingtime	Returns the latest time at which the value of the selected simulation quantity fell outside its tolerance band. The target value of the quantity and the +/- bandwidth of the tolerance band are the additional args.
sgn	Sign extraction.
sin	Sine.
sinh	Hyperbolic sine.
slidingmean	Returns the moving average value of the selected simulation quantity (specified by the first argument). The average is calculated over a period (specified by the second argument).
slidingrms	Returns the moving RMS value of the selected simulation quantity (specified by the first argument). The RMS value is calculated over a period (specified by the second argument).
sqr	Square of the selected simulation quantity.
sqrt	Square root of the selected simulation quantity.
stddev	Returns the standard deviation of given values.
sum	Returns the sum of the given values.
tan	Tangent.
tanh	Hyperbolic tangent.
undershoot	Calculates peak undershoot given a threshold value and number of evenly spaced points over entire time range.
uppercutoff	Returns the upper 3dB frequency of the selected simulation channel in Hz.
variance	Calculates the variance of the given values.
XAtYMax	Threshold crossing time: report first time (x value) at which an output quantity crosses YMax.
XAtYMin	Threshold crossing time: report first time (x value) at which an output quantity crosses a user definable threshold.
XAtYVal	Returns the X value at the first occurrence of Y value.
XWidthAtYVal	Returns the X width between the first 2 occurrence of Y value.
xdb10beamdwidth	Width between left and right occurrences of values 'x' db10 from max. Takes 'x' as argument (3.0 default). To use this function, you need to

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	open the Add Trace Characteristics dialog and select the Radiation category.
xdb20beamwidth	Width between left and right occurrences of values 'x' db20 from max. Takes 'x' as argument (3.0 default). To use this function, you need to open the Add Trace Characteristics dialog and select the Radiation category.
YAtXMax	Returns the X value at maximum value of Y.
YAtXMin	Returns the Y value at minimum value of X.
YAtXVal	Returns the Y value at the first occurrence of X value.
y0	Bessel function of the second kind (0^{th} order).
y1	Bessel function of the second kind (1^{st} order).
yn	Bessel function of the second kind (nth order).

Selecting Solution Quantities to Plot

When you create a report, each trace in the report includes a quantity that is plotted along an axis. The quantity being plotted can be a value that was calculated by HFSS, a value from a calculated expression, or an intrinsic variable value (such as frequency or theta).

> **Important:**
>
> Available solution quantity categories depend on the design type, solution type, setup parameters, and plot domain. The selected category provides the default name for the plot, such as "S Parameter Plot *n*". You can edit plot names in the project tree and plot header text in the Properties window.

To select solution quantities to plot:

1. Click **Results** > **Standard Report** and select the type of report you would like to generate.

   The **Report** window appears.

2. On the **Trace** tab, select a **Category** from the list.

Category	Description
Variables	Intrinsic variables (such as frequency or theta), or user-defined project variables (such as the length of a quarter-wave transformer).
Output Variables	User-defined expressions applied to derive quantities from the original field solution.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Description
S-Parameter	S-parameters from the S-matrix. For designs which include a Frequency Selective Surface (FSS)-referenced radiation boundary, S_{11} and S_{21} represent the extracted reflection and transmission coefficients, respectively.
Y-Parameter	Admittance matrix parameters computed from the S-parameters and port impedances.
Z-Parameter	Impedance matrix parameters computed from the S-parameters and port impedances.
Group Delay	Quantity calculated as rate of change of the total phase shift with respect to angular frequency, $\frac{d(\Phi)}{d(\omega)}$
VSWR	Voltage standing wave ratio, calculated from the equation: $\frac{1+\lvert S_{ij}\rvert}{1-\lvert S_{ij}\rvert}$
Gamma	Propagation constants for the S-parameters.
Port Z_o	Characteristic port impedances.
Matched S Parameter	Matched S is the S-matrix produced by solver. It becomes useful to view when passive ports exist and the matched S-matrix is rectangular instead of square. When passive ports exist, S is the square matrix of the active ports and the passive ports represent a loss mechanism. Matched S-matrix provides coupling information to the passive ports. In Transient, we show Matched S entries while the simulation is in process which is not clean for regular S when port post processing exists.
Short Circuit Z	Short circuit impedance, calculated as: $\mathrm{ShortCircuitZ}\,(\mathrm{P1}) = \mathrm{mag}\left(\frac{1}{\mathrm{Y(P1,\ P1)}}\right)$
Loop Inductance	LoopL computes with all other ports shorted, even ports on other nets. These short circuits permit induced currents to flow in nearby nets, meaning LoopL will always be lower than a more rigorous loop inductance computation. $\mathrm{LoopL}\,(\mathrm{P1}) = \frac{\mathrm{im}\left(\frac{1}{\mathrm{Y(P1,\ P1)}}\right)}{(2 \times \pi \times \mathrm{freq})}$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Description
Loop Resistance	Calculated as: $\mathrm{LoopR}\left(\mathrm{P1}\right) = \mathrm{re}\left(\frac{1}{\mathrm{Y(P1,\ P1)}}\right)$
Capacitance	This capacitance calculation leaves other nets open-circuited, whereas a rigorous capacitance calculation would short them to ground, meaning "Capacitance" computed here will always be too low. $\mathrm{C}\left(\mathrm{P1}\right) = \frac{\mathrm{im}\left(\frac{1}{\mathrm{Z(P1,\ P1)}}\right)}{\left(2 \times \pi \times \mathrm{freq}\right)}$
Lambda	Guided wavelength.
Epsilon	Effective permittivity.
TDR Impedance	Time-Domain Reflectometry (TDR) impedance for non-terminal problems. The idea is to excite a structure with a step function, and inspect the reflections as a function of time. This quantity is available only when the **Time** domain is selected. Selecting the TDR Impedance category displays the TDRZ of every terminal or mode in the ports. The list of available Functions includes those that can operate on the TDRZ values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Description
Active S-Parameter	Active S parameters represent the reflection coefficient of one port when another port is excited. HFSS offers passive S-parameters as well as active S-Parameters. If you plot Active S-Parameters, HFSS includes the effects of mutual coupling with other ports or elements. One use is in Optimization, to view how S-parameters change as design parameters change. In this case you may plot Active S Parameters. In such cases they allow you to forecast design behavior in the presence of an ideal feed network. A 0w value is a matched load, but 1w values take that into account. Active S-parameters are given as a linear combination of the regular S-parameters, with some weights 'a', based on the matrix equation $\mathbf{b} = \mathbf{S} \times \mathbf{a}$. In the post-processor, the active S-parameter for port*n* is displayed to the user as ActiveS (P*n*:*m*) where "P*n*" indicates the port number and ":*m*" indicates the mode number. Active S, Y, Z, VSWR are supported only for driven modal projects. Power is proportional to the square of the fields. As a consequence, active S-parameters, which are field related, are proportional to the square root of what you set under Edit Sources. You can check this by setting all power to 0 except for one port of interest where you want to see the Active S-parameter. Then for a neighboring port, change the input power in steps, and monitor how the active S-parameter changes. Keep in mind that this involves complex numbers which complicates things. Given a driven model project with a total of *N*port modes, let a_k denote the complex excitation for the *k*-th mode specified in the Edit Sources dialog box. Also let S denote the computed *N* x *N*scattering matrix. If $a_m \neq 0$ define active-S_m, *m* = 1..., *N* by $$active\text{-}S_m \equiv \sum_{n=1}^{N} S_{mn} \frac{a_n}{a_m} \qquad m = 1, \ldots, N$$ If *S* is renormalized or deembedded, the interpretation is that the stimulations are applied to the renormalized external transmission lines at the plane of deembedding. The other relative active quantities are simply transformations on the active S_m.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Description
Active Y-Parameter	$Y_0^{(M)}$ is the port admittance of the *m*-th mode. $$active\text{-}Y_m \equiv Y_0{}^{(m)} \frac{1 - active\text{-}S_m}{1 + active\text{-}S_m} \qquad m = 1, ..., N$$
Active Z-Parameter	$Z_0^{(m)}$ is the impedance of the *m*-th mode. $$active\text{-}Z_m \equiv Z_0{}^{(m)} \frac{1 + active\text{-}S_m}{1 - active\text{-}S_m} \qquad m = 1, ..., N$$
Active VSWR	Active Voltage standing wave ratio is supported only for driven modal projects. $$active\text{-}VSWR_m \equiv \frac{1 + \lvert active\text{-}S_m \rvert}{1 - \lvert active\text{-}S_m \rvert} \qquad m = 1, ..., N$$
Passivity	A scalar quantity based on the matrix Q = I - conjugate (transpose(S)) * S For every frequency, the value must be no larger than 1. A uniform renormalization of 50 ohms is performed on the solution data for Passivity checking.
Power	For Composite Excitation Solution types, selecting Power as Category lists Quantities for Accepted (Pacc), Reflected (Pref), and Incident (Pinc) powers/port based on your Edit Sources settings.
Design	Enables you to plot or tabulate properties of objects, such as their volumes.
Expression Cache	The values of expressions listed in the Expression cache of the Solution setup can be plotted, for example, as a function of adaptive pass to monitor their convergence.
Expression Convergence	This is intended to plot convergence, as a function of adaptive pass, of expressions in the Expression cache of the Solution setup. In defining the report, for "Context", select Solution: SetupN:Adaptive Pass. ExprDelta will show the change in the value of the expression as a function of adaptive pass, while ExprGoal will show, for comparison, the convergence goal for this expression, as defined in the Analysis Setup under the panel Expression Cache.

3. Select a **Quantity** from the list.
4. If desired, select a **Function**.
5. Click **Close**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting a Field Quantity to Plot

When plotting field quantities, the quantity can be a value that was automatically calculated by HFSS such as the magnitude of S_{11}, a value from a calculated expression, or an intrinsic (inherent) variable value such as frequency or phase.

To select a field quantity to plot:

1. When you create the report, specify the Report Type as "Fields" and the plot type (for example, radiation pattern.)
2. In the *Report* dialog box, select Geometry for the Context, unless you are plotting scalar (for example, integration). For example, to plot near-field values across a sphere, you select the sphere object from the **Geometry** list in the *Traces* dialog box when you create a report.
3. In the *Report* dialog box, select one of the following categories. The selected category provides the default plot name. You can edit the plot names in the project tree and the plot header text in the report synchronizes.

Variables	Intrinsic variables, such as frequency or phase, or user-defined project variables, such as the length of a quarter-wave transformer.
Output Variables	User defined expressions applied to derive quantities from the original field solution.
Calculator Expressions	Includes scalar and vector field quantities automatically calculated by HFSS, as well as derived field quantities that are defined by calculated expressions you set up in the Fields Calculator.

4. Select a quantity to plot from the **Quantity** list. The available quantities will depend upon the selected category and the setup of the design. See Field Quantities list for definitions.

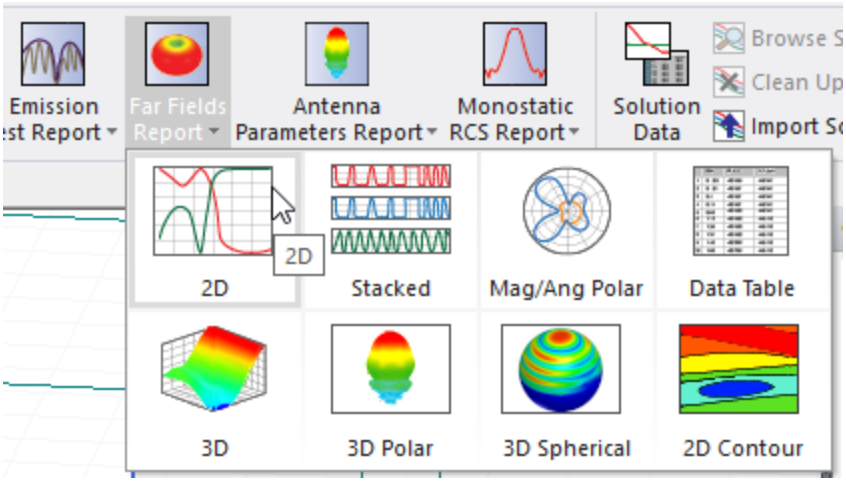
Selecting a Far-Field Quantity to Plot

When plotting far-field quantities, the quantity can be a value that was calculated by HFSS such as antenna gain, a value from a calculated expression, or an intrinsic (inherent) variable value such as frequency or theta.

Prerequisites

For the **Far Field Report** icons to appear on the ribbon, or for the **Create Far Field Reports** command to appear on the **Results** menus, the design must contain radiation boundaries, PMLs or Hybrid regions, and must contain a Far Field Setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

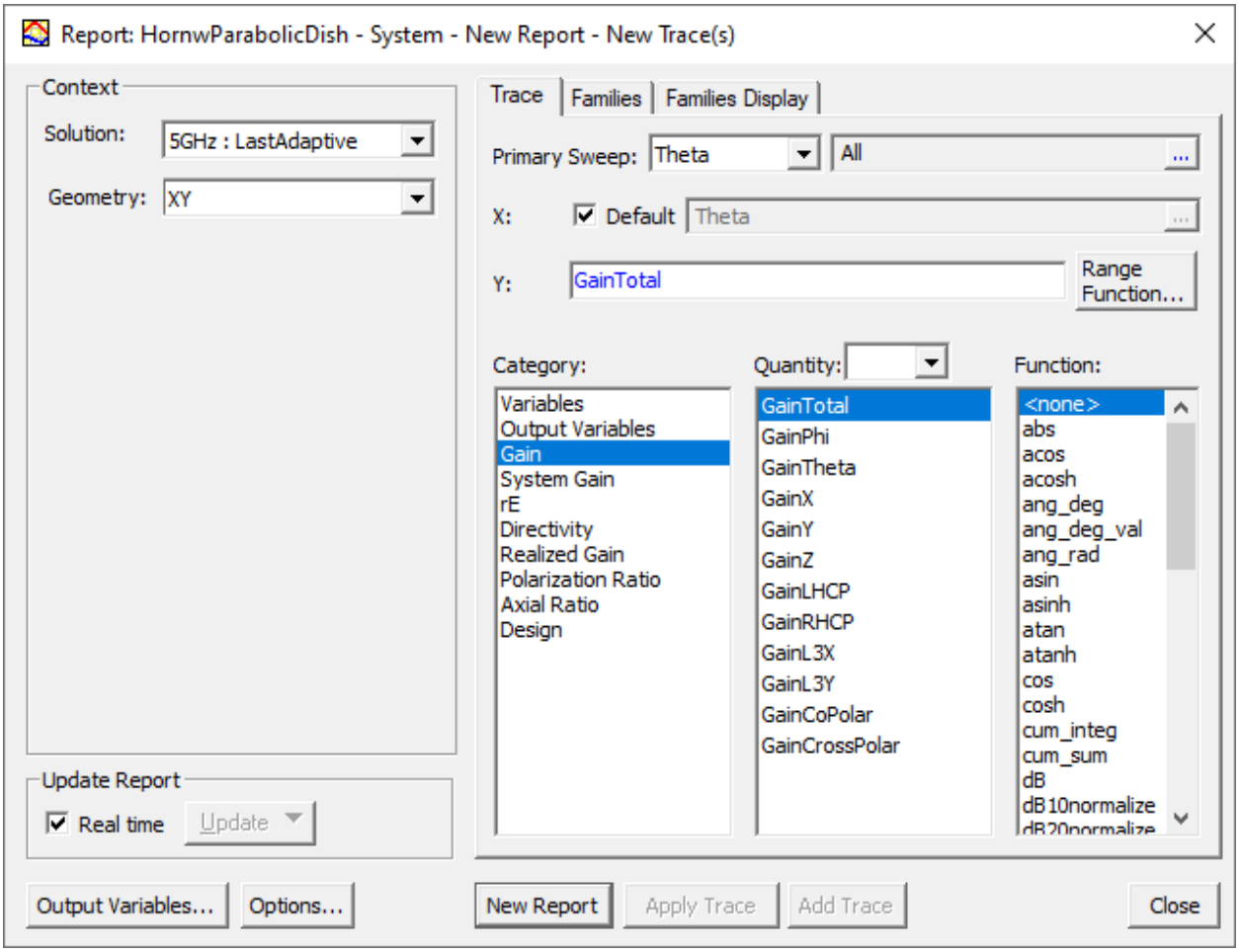


To tune Far Field plots based on partial quantities, the design must include a variable from which derivatives can be calculated. See: Derivative Tuning for Reports.

To plot a far-field quantity:

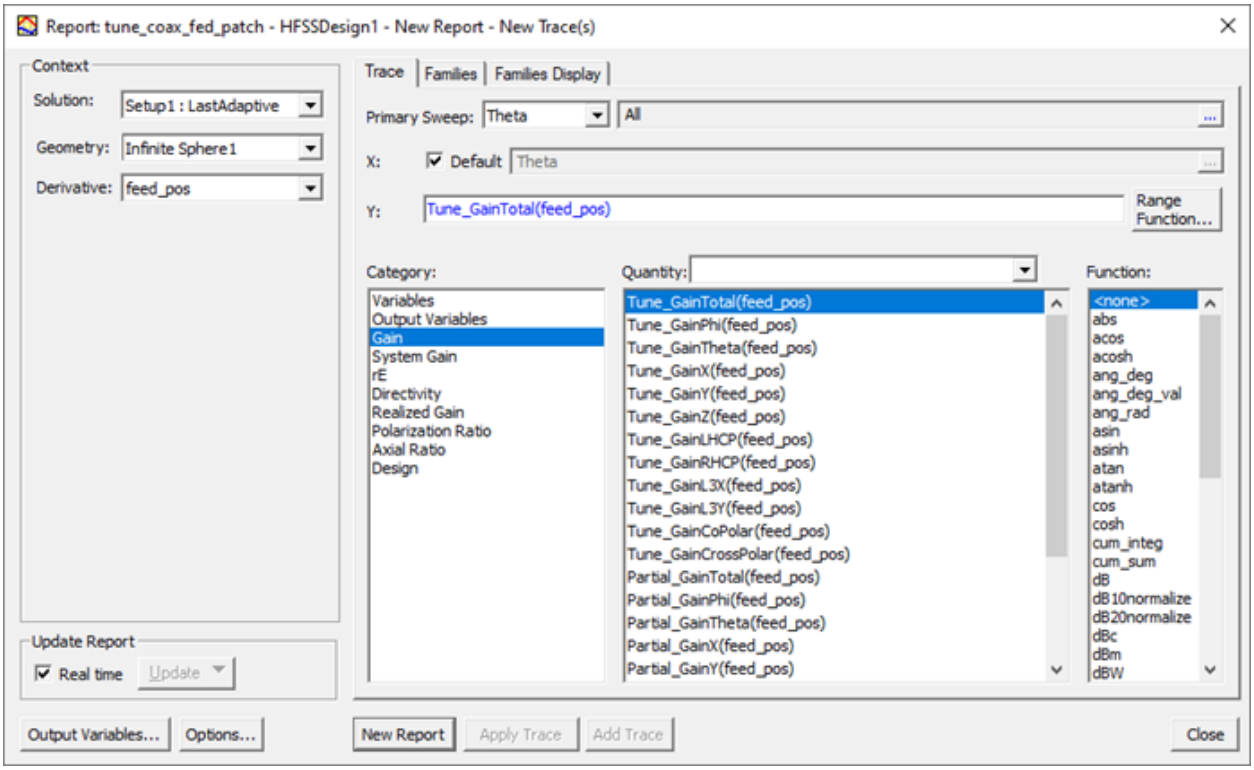
1. From the **Results** ribbon, select the **Far Fields Report** icon, or select **Results** > **Create Far Fields Report** to open the **Report** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If your design includes one or more variables for derivatives, you can select one or all derivatives. Selecting a **Category** will list tunable **Quantities** available:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. In the **Context** area, select the **Solution**, **Geometry**, and **Source**.

> **Note:**
>
> In the case of finite array, select a Source group.

3. Make a selection from the **Category** list. The available options may differ depending on previous choices. Selecting one or more Derivatives results in Tunable variations of these quantities being listed:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

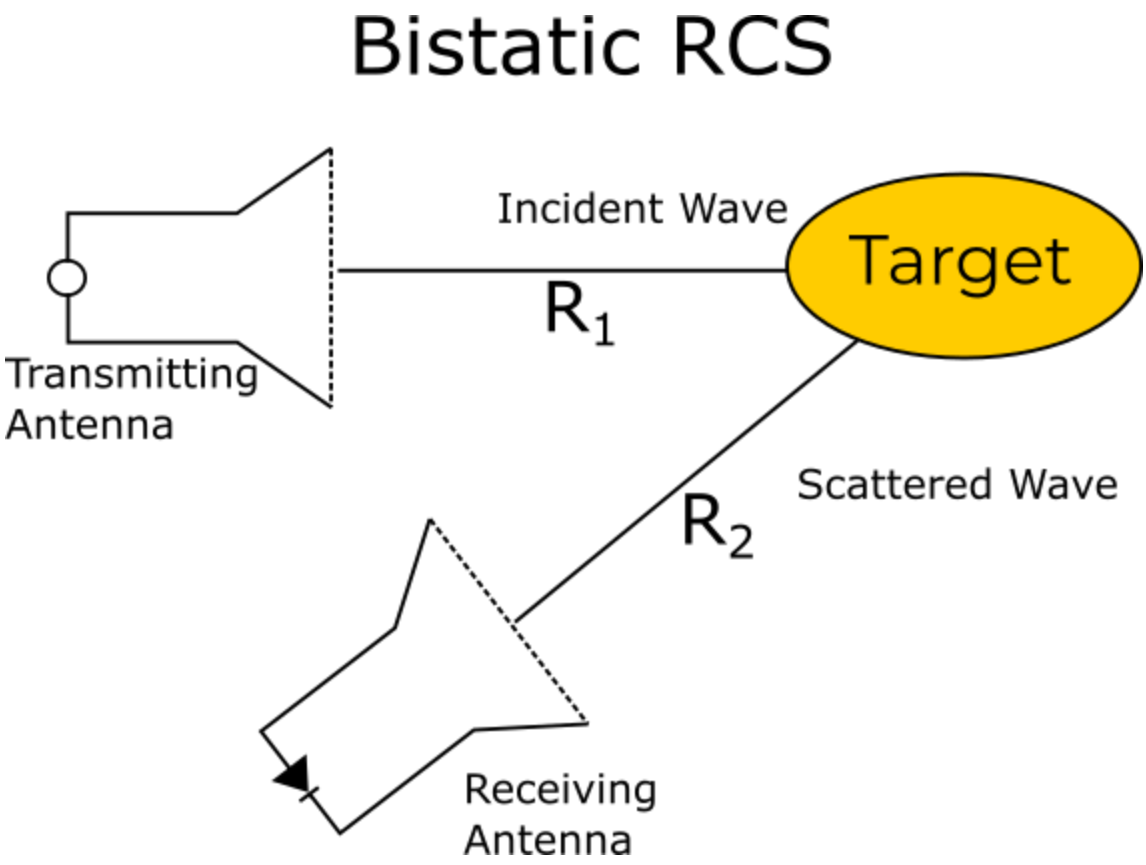
Variables	Intrinsic variables, such as frequency or theta, or user-defined project variables, such as the length of a quarter-wave transformer.
Output Variables	User-defined expressions applied to derive quantities from the original field solution.
Gain	Gain is 4π times the ratio of an antenna's radiation intensity in a given direction to the total power accepted by the antenna.
System Gain	System gain appears in the Quantity list if Edit Sources was used to specify system power for gain calculations. System gain is 4π times the ratio of an antenna's radiation intensity in a given direction to the user-specified power.
rE	The selected component of the radiated electric field, which is multiplied by the radial distance, *r*.
Directivity	Directivity of the antenna.
Realized Gain	Realized gain is 4π times the ratio of an antenna's radiation intensity in a given direction to the total power incident upon the antenna port(s).
Polarization Ratio	Polarization ratio of the electric field.
Axial Ratio	Axial ratio of the electric field.
Normalized Bistatic RCS	The normalized radar cross-section: $\frac{\sigma}{\lambda_0^2}$ where $\lambda_0$is the wavelength of free space. *RCS is supported for designs with Plane Incident Waves. RCS is not supported for other types of incident waves.*
Radar Cross-Section (Bistatic RCS)	The radar cross-section (RCS) or echo area (σ) is measured in meters squared and represented for a bistatic arrangement (e.g., when the transmitter and receiver are in different locations as shown in the figure below). This is represented by: $\sigma = \frac{4\pi r^2 \lvert E_{\text{scat}} \rvert^2}{\lvert E_{\text{inc}} \rvert^2}$ where:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	• E_{scat} is the scattered E-field. • E_{inc} is the incident E-field. *RCS is supported for designs with Plane Incident Waves. RCS is not supported for other types of incident waves.*
Complex (Bistatic) RCS	The equation for complex (bistatic) RCS is calculated as: $\sqrt{\sigma} = 2\sqrt{\pi \mathrm{R}} \frac{E_{\mathrm{scat}}}{\lvert E_{\mathrm{inc}} \rvert}$ where: • E_{scat} is the scattered E-field. • E_{inc} is the incident E-field. This form retains the phase information. *RCS is supported for designs with Plane Incident Waves. RCS is not supported for other types of incident waves.*
Monostatic RCS	The radar cross-section (RCS) or echo area when the transmitter and receiver are at the same location. For Monostatic RCS, you need not be concerned with the Theta and Phi values defined in the radiation sphere. Only the incident wave Theta and incident wave Phi are used in calculating a Monostatic RCS plot. *RCS is supported for designs with Plane Incident Waves. RCS is not supported for other types of incident waves.* *A proper incident angle sweep should exist at the incident wave source setup before HFSS can plot Monostatic RCS*

The following diagram shows the bistatic RCS concept, with separate transmitting and receiving antennas.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Each **Category** item selected updates the **Quantity** list to offer quantities appropriate to selected category. **Category** selection for a Variable of an Output Variable lists those available in each case. Selecting Antenna Parameters as Category causes the Quantity list to show Antenna parameters. The selected Category provides the default report name, such as rE Plot *n*.

4. Select the **Quantity** to apply to the selected **Category**.

   If the **Category** item is rE, Gain, Directivity, or Realized Gain, you will need to specify the polarization of the electric field by selecting from the **Quantity** list. If you manually enter Phi or Theta component of a far field quantity when using Az/El or El/Az far field definition, the Report uses the Z axis of the Az/El or El/Az coordinate system definition as pole to calculate theta/phi component for rE.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Total	The combined magnitude of the electric field components.
Phi	The phi component (φ).
Theta	The theta component (θ).
X	The x component.
Y	The y component.
Z	The z component.
LHCP	The dominant component for a left-hand, circularly polarized field.
RHCP	The dominant component for a right-hand, circularly polarized field.
CircularLHCP	The polarization ratio for a predominantly left-hand, circularly polarized antenna.
CircularRHCP	The polarization ratio for a predominantly right-hand, circularly polarized antenna.
SphericalPhi	The polarization ratio for a predominantly φ-polarized antenna.
SphericalTheta	The polarization ratio for a predominantly θ-polarized antenna.
L3X	The dominant component for an x-polarized aperture using Ludwig's third definition of cross polarization.
L3Y	The dominant component for a y-polarized aperture using Ludwig's third definition of cross polarization.
CoPolar	Reports fixed polarization rotated around Z axis by “slant” angle from X axis of coordinate system of Infinite sphere definition: $rE_{CoPolar} = \cos(\varphi - a)\hat{\theta} - \sin(\varphi - a)\hat{\varphi}$ where *a* is the slanted angle.
CrossPolar	$rE_{CrossPolar} = -\sin(\varphi - a)\hat{\theta} - \cos(\varphi - a)\hat{\varphi}$ where *a* is the slanted angle.

The plot's Y axis field shows the combined selections.

For example, if you select Gain as the Category and RHCP as the Quantity, HFSS evaluates the equation as:

$$Gain_{RHCP} = \frac{4\pi U_{RHCP}(\theta,\varphi)}{P_{acc}} = \frac{4\pi |rE_{RHCP}(\theta,\varphi)|^2}{2\eta P_{acc}}$$

$$= \frac{4\pi \left| r\vec{E}(\theta,\varphi) \bullet \left(\frac{\hat{\theta} - j\hat{\varphi}}{\sqrt{2}} \right) \right|^2}{2\eta P_{acc}}$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. You can also select a function to apply to the your selections for the **Category** and **Quantity** (e.g., mag).

   As you make selections for **Category**, **Quantity**, and **Function**, the **Y** field shows the combined calculation they describe.

6. When you are finished, click **New Report** to create the report.

   The new report based on your selections is displayed.

Plotting Vertical Cross-Sections of Far Fields

When plotting far fields, a vertical cross-section plot results from holding phi fixed and sweeping theta through a range of values.

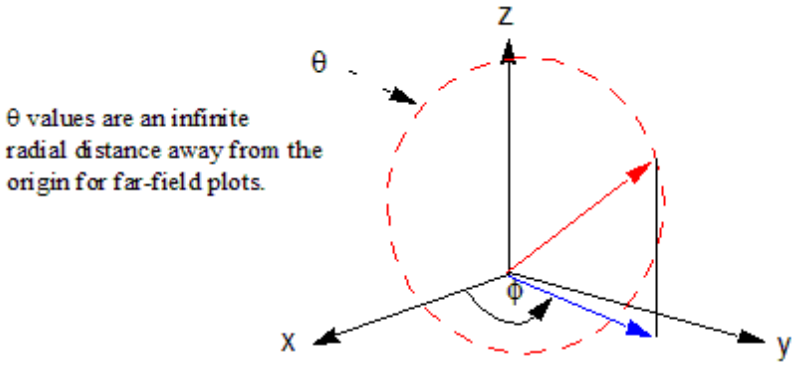
1. Open the **Report** dialog box.
2. Click the ellipsis **[...]** button for the **sweep variable corresponding to phi.**

This displays a dialog listing all values for the phi variable. The values are the result of the range of phi you specified during the infinite sphere's setup.

3. Select the fixed value that phi should take in the plot.

HFSS will display values for the vertical cross-section at selected phi cuts of the problem region at a set of theta rotations.

The figure shown below demonstrates the orientation of the vertical cross-section when ϕ is the fixed variable:



Plotting Horizontal Cross-Sections of Far Fields

When plotting far fields, a horizontal cross-section results from holding theta fixed and sweeping phi through a range of values.

1. Click the ellipsis **[...]** button for the **sweep variable corresponding to theta.**

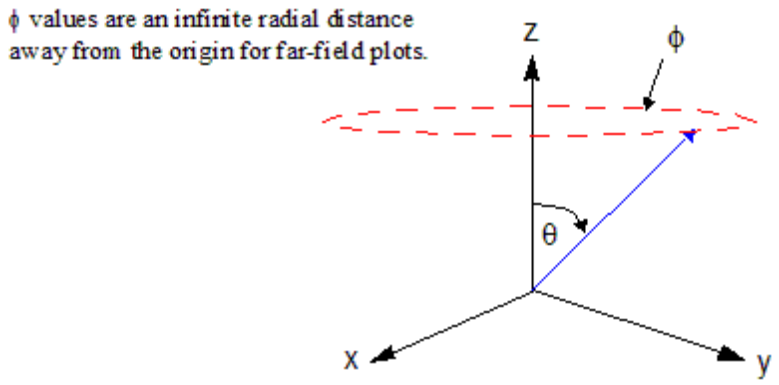
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To the right, all of the possible values for the theta variable are listed. The values are the result of the range of theta you specified during the infinite sphere's setup.

2. Select the fixed value that theta should take in the plot.

HFSS will display values for the horizontal cross-section at selected theta cuts of the problem region at a set of phi rotations.

The figure shown below demonstrates the orientation of the sphere on which the field is computed when θ is the fixed variable:



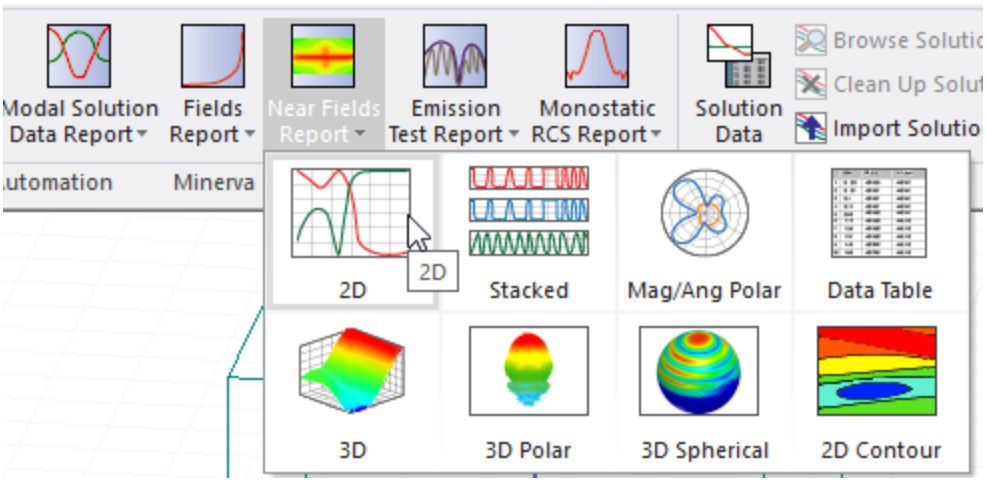
Selecting a Near-Field Quantity to Plot

When plotting near-field quantities, the quantity can be a value that was calculated by HFSS, a value from a calculated expression, or an intrinsic (inherent) variable value such as frequency or theta.

Prerequisites

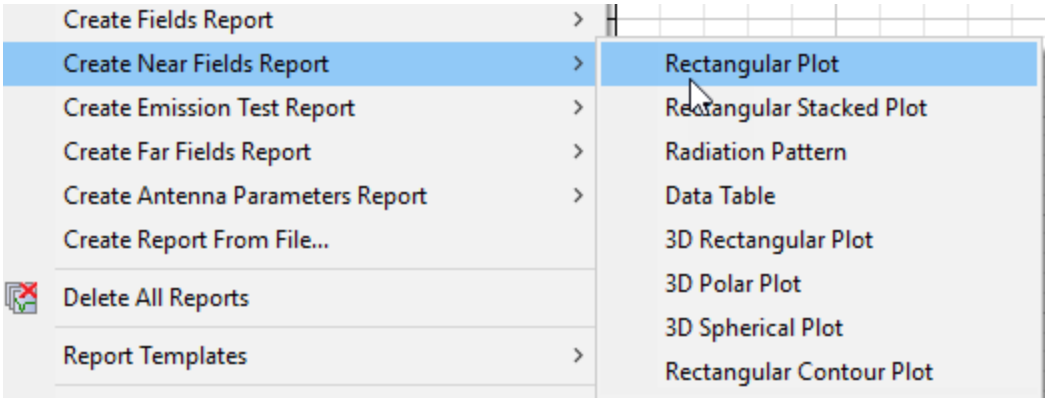
For the **Near Field Report** icons to appear on the ribbon, or for the **Create Near Field Reports** command to appear on the **Results** menus, the design must contain radiation boundaries, PMLs or Hybrid regions, and must contain a Near Field Setup for sphere, line or rectangle.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



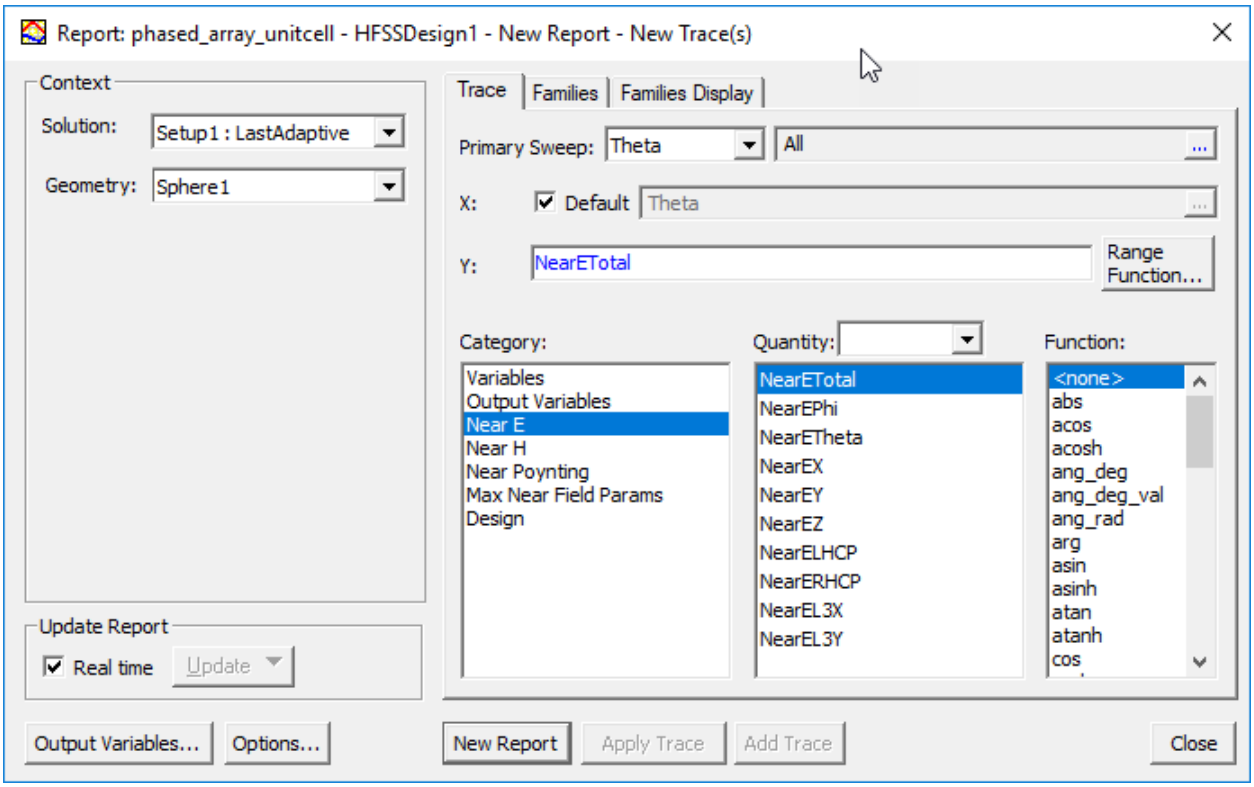
To select a near-field quantity to plot:

1. When you create the report, use the **Results** tab of the ribbon and select the **Near Fields Report** icon to select from the available report types, or if you use the **Results** menus, select **Create Near Fields Report** and select the plot type from the menu.



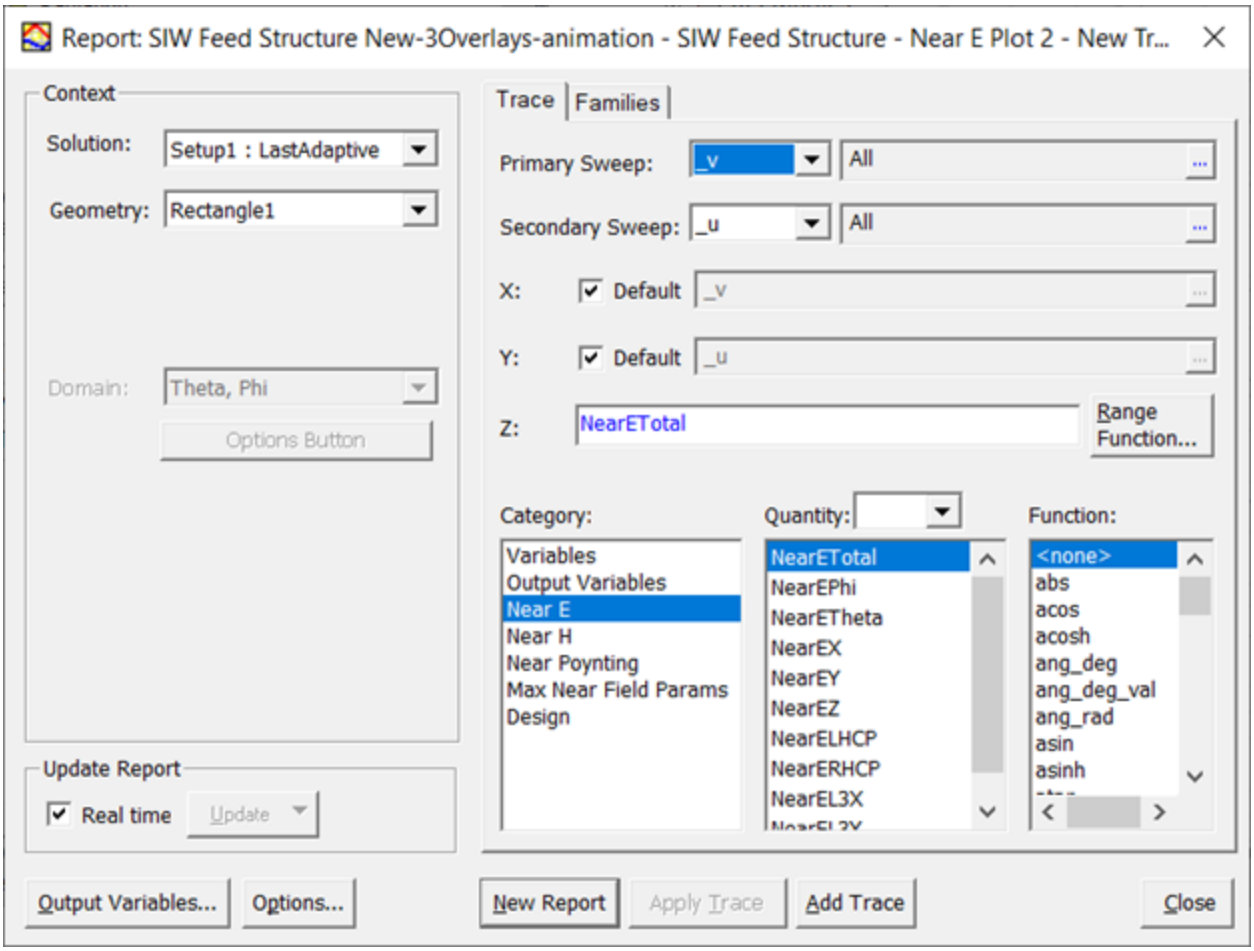
The **Report** dialog appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When you create a Near Field Rectangular Contour plot, if the Context Geometry is a Rectangle radiation setup and the Primary and Secondary sweep is the combination of _u and _v, then the plot creation dialog is as follows.

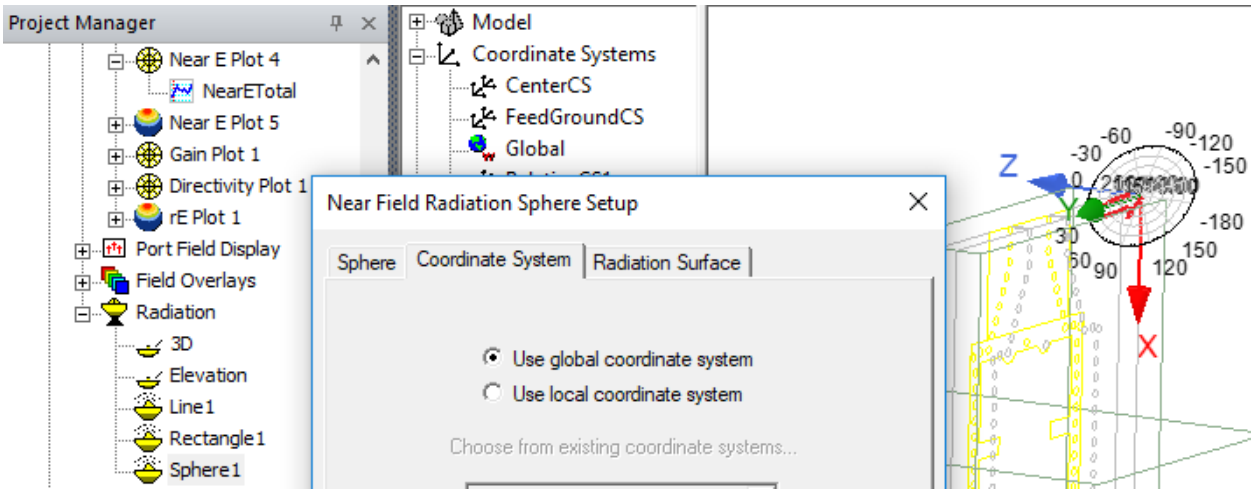
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



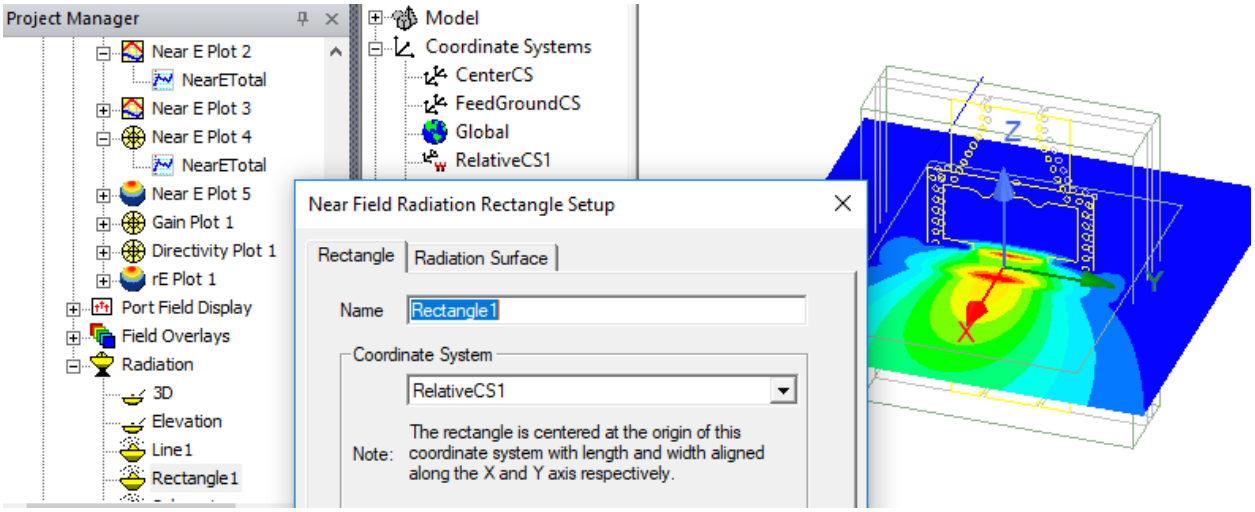
This kind of near field rectangular contour plot can be overlaid in its actual defined rectangle in 3D space in the Modeler window.

2. In the **Geometry** portion, select a predefined Sphere, Rectangle or Line. The geometry selected affects where the plot data is selected and if, where and how the plot will be overlaid on the model, if you choose to do so, after the plot is created. In the following figure, the Near E Polar plot overlaid on the model uses Sphere1 as the Geometry and that geometry uses the Global CS.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

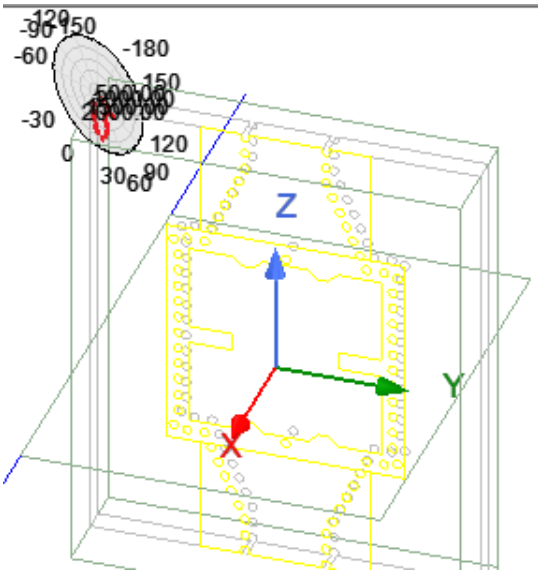


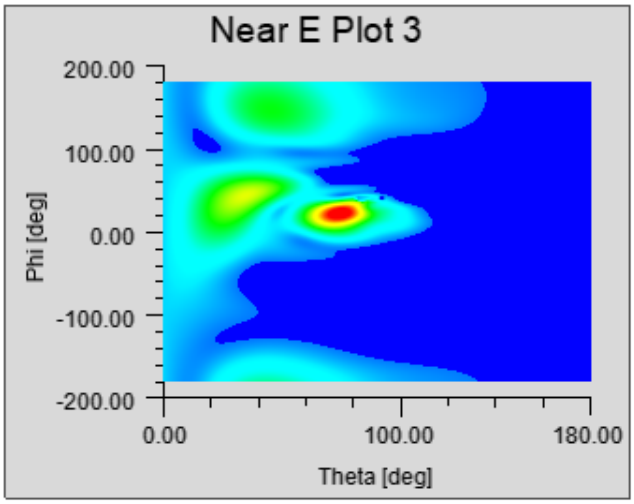
In the following case, Near E Plot 2 Uses Rectangle1, which uses RelativeCS1.



In the case of a plot generated on a line, a plot can be created but not overlaid.
In the case of a 2D Contour Near Field Plot generated on a Sphere, the plot can be overlaid as a plot in the upper right of the model window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





3. In the **Report** dialog box, select one of the following categories. The selected category provides the default name of the report.

Variables	Intrinsic variables, such as frequency or theta, or user-defined project variables, such as the length of a quarter-wave transformer.
Output Variables	User defined expressions applied to derive quantities from the original field solution.
Near E	The radiated electric field in the near field region.
Near H	The radiated H field in the near region
Near Poynting	The Poynting field in the near region. This is 0.5 * E *conj(H).
Max Near Field Params	The maximum radiated electric field in the near region.
Near Normalized Antenna	The resultant plot is: field quantity / (maximum field quantity value over the entire infinite sphere).

4. If you selected the **Near E** category, specify the polarization of the electric field by selecting one of the following types of quantities from the **Quantity** list. The corresponding options appear for Near H and Near Poynting. If you select Max Near Field Params, there are additional **Quantities** for E, H and Poynting.

NearETotal	The combined magnitude of the electric field components.
NearEPhi	The phi component of the electric field.
NearETheta	The theta component of the electric field.
NearEX	The x-component of the electric field.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NearEY	The y-component of the electric field.
NearEZ	The z-component of the electric field.
NearELHCP	The dominant component for a left-hand, circularly polarized electric field.
NearERHCP	The dominant component for a right-hand, circularly polarized electric field.
NearECircularLHCP	The polarization ratio for a predominantly left-hand, circularly polarized antenna.
NearECircularRHCP	The polarization ratio for a predominantly right-hand, circularly polarized antenna.
NearEL3X	The dominant component for an x-polarized aperture using Ludwig's third definition of cross polarization.
NearEL3Y	The dominant component for a y-polarized aperture using Ludwig's third definition of cross polarization.

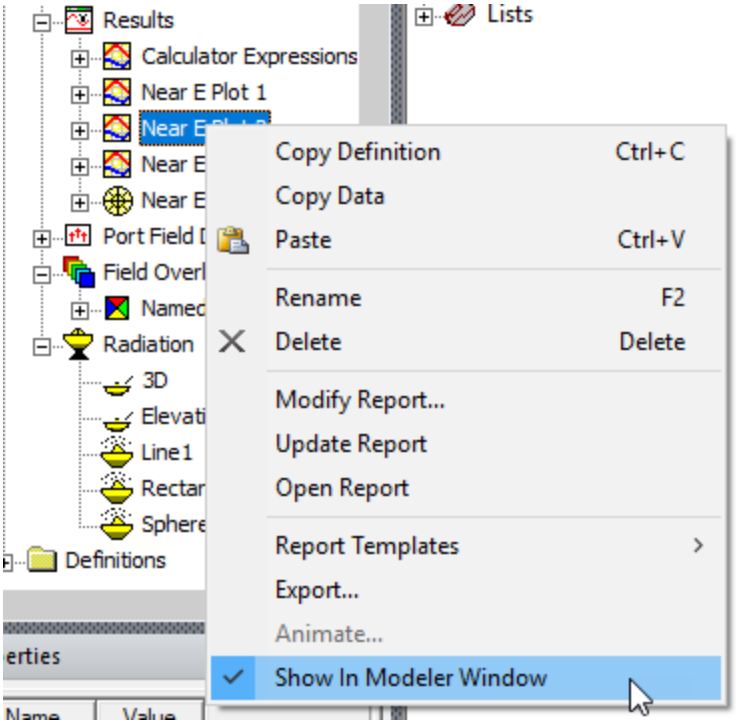
If a Near-field plot takes a long time to plot, be sure to perform **File>Save** when the plot is displayed. This saves the calculated data and permits fast display on subsequent viewings of the plot.

Reports appear under Results, and can be opened as reports. 3D Reports can be overlaid on the model, based on the selected geometry. 2D Near Field Reports on Rectangles can be overlaid as 2D plots.

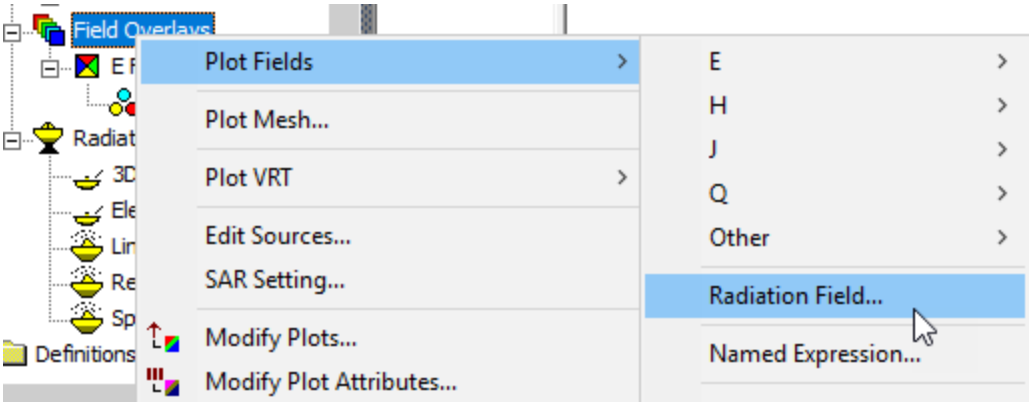
Overlaying Near Field Reports

Depending on the geometry selection and type of plot, some near field plots be shown in the model window. If a plot is appropriate for this purpose, the **Show in Modeler Window** option appears on the **Project** window short cut menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

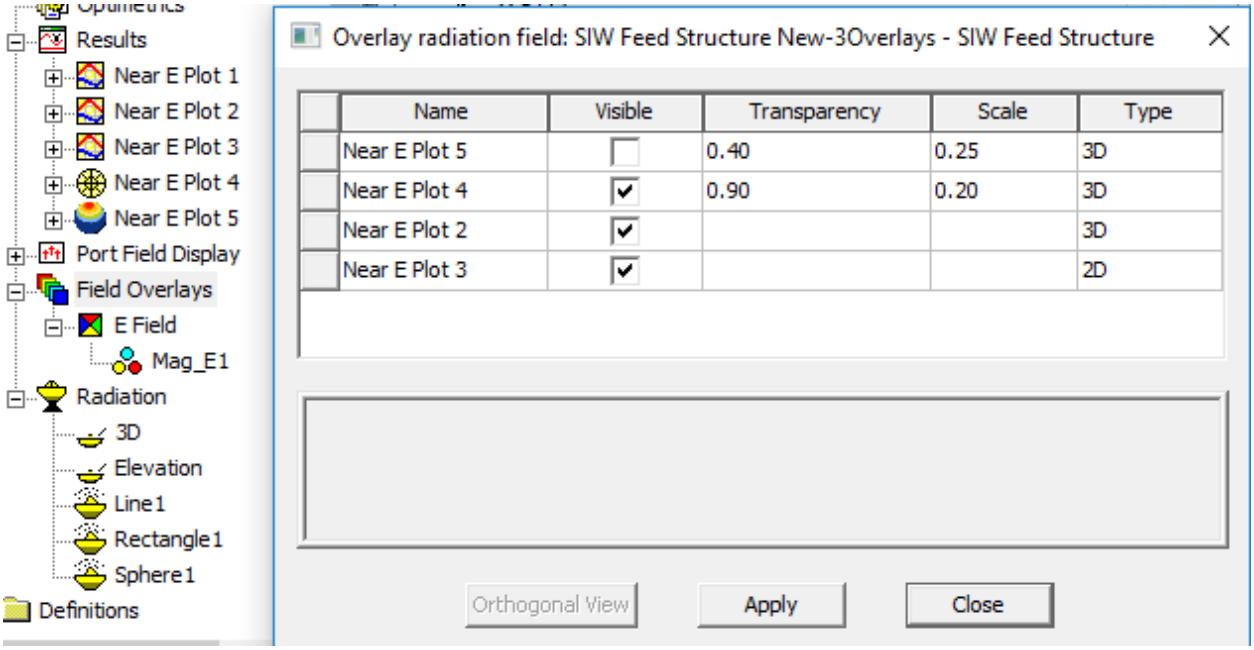


You can also see these plots listed by right clicking on **Field Overlays** in the **Project** tree and selecting **Plot Fields>Radiation Field**.

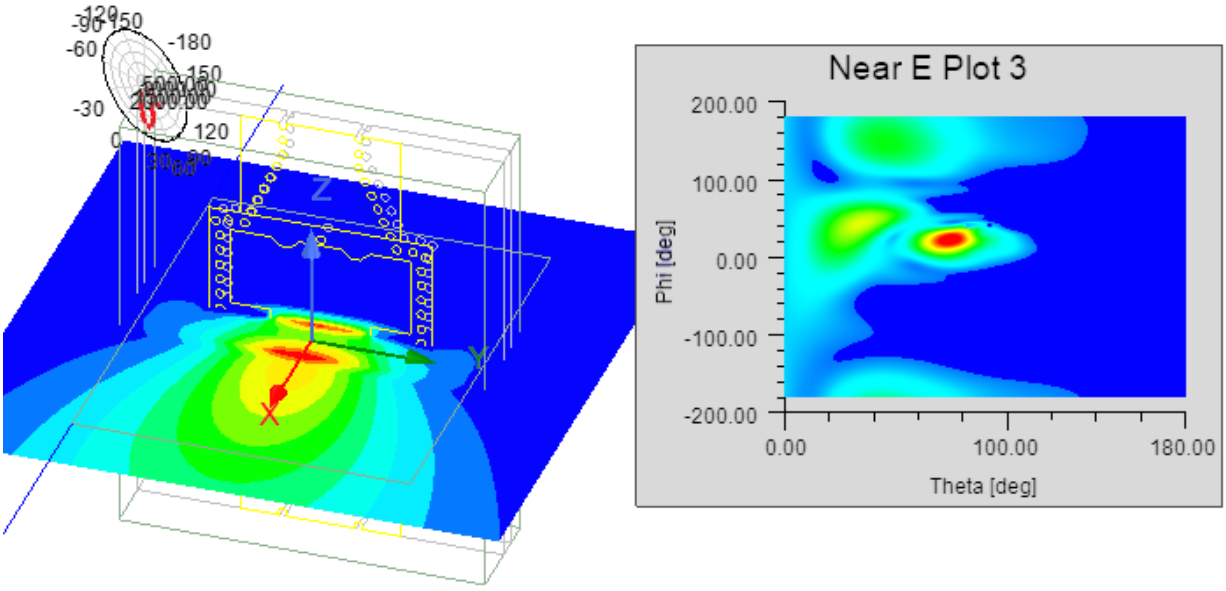


This displays the **Overlay radiation field** dialog box. Notice that it lists **Results** plots that can be overlaid, and not those listed under **Field Overlays**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can display multiple kinds of Plot overlays and Field Overlays at the same time.

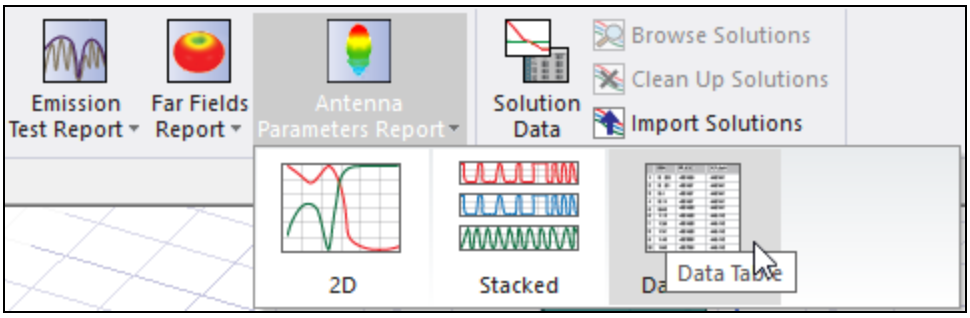


Selecting Antenna Parameters to Plot

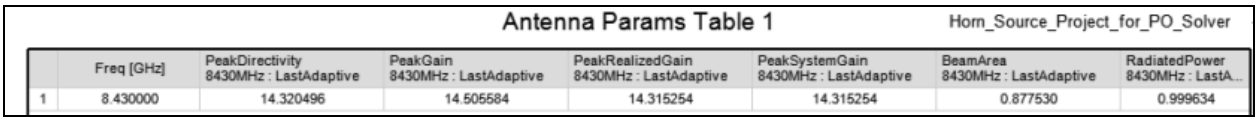
Complete these steps to select and plot antenna parameters.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. When you create the report, select Antenna Parameters, using either **Results>Create Antenna Parameters Report><*reportType*>**, or using the ribbon:



The format can be Data Table, Rectangular Plot, or Rectangular Stacked Plot, but because each trace includes a single quantity, you may prefer Data Table as the most appropriate display.



Antenna Params Table 1 — Horn_Source_Project_for_PO_Solver

	Freq [GHz]	PeakDirectivity 8430MHz : LastAdaptive	PeakGain 8430MHz : LastAdaptive	PeakRealizedGain 8430MHz : LastAdaptive	PeakSystemGain 8430MHz : LastAdaptive	BeamArea 8430MHz : LastAdaptive	RadiatedPower 8430MHz : LastA...
1	8.430000	14.320496	14.505584	14.315254	14.315254	0.877530	0.999634

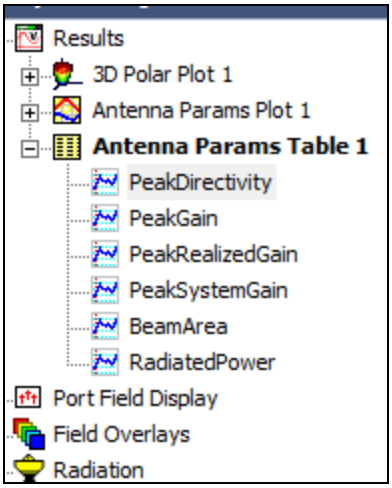
2. In the **Report** dialog box, select Antenna Params as the Category and apply an appropriate Quantity from the list. HFSS-calculated quantities that include peak directivity, peak gain, peak realized gain, beam area*, radiated power, accepted power, radiation efficiency including the total and component values at Phi and Theta, max U, and array factors. For descriptions of CoPolar and Cross Polar, see the table in Step 4 of Selecting a Far Field Quanity to Plot. For far-field setups, the decay factor for lossy materials is calculated as a constant for all far fields. *Beam area is the calculated as the beam solid angle. In its simplest form it is = 4*pi/D; where D is the max directivity.

PeakDirectivity	DecayFactor	MaxrEZCompAtTheta	MaxrELudwig3XComp
PeakGain	FrontToBackRatio	MaxrEThetaComp	MaxrELudwig3XCompAtPhi
PeakRealizedGain	MaxrETotal	MaxrEThetaCompAtPhi	MaxrELudwig3XCompAtTheta
PeakSystemGain	MaxrETotalAtPhi	MaxrEThetaCompAtTheta	MaxrELudwig3YComp
BeamArea	MaxrETotalAtTheta	MaxrEPhiComp	MaxrELudwig3YCompAtPhi
RadiatedPower	MaxrEXComp	MaxrEPhiCompAtPhi	MaxrELudwig3YCompAtTheta
AcceptedPower	MaxrEXCompAtPhi	MaxrEPhiCompAtTheta	MaxErCoPolar
IncidentPower	MaxrEXCompAtTheta	MaxrELHCPComp	MaxErCoPolarAtPhi

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RadiationEfficiency	MaxrEYComp	MaxrELHCPCompAtPhi	MaxErCoPolarAtTheta
TotalEfficiency	MaxrEYCompAtPhi	MaxrELHCPCompAtTheta	MaxErCrossPolar
SystemEfficiency	MaxrEYCompAtTheta	MaxrERHCPComp	MaxErCrossPolarAtPhi
MaxU	MaxrEZComp	MaxrERHCPCompAtPhi	MaxErCrossPolarAtTheta
ArrayFactor	MaxrEZCompAtPhi	MaxrERHCPCompAtTheta	

3. If appropriate, select one or more options from the **Function** list.
4. Make any other selections, as necessary.
5. Click **New Report**.
6. If necessary, return to the **Report** window to select additional options from the **Quantity** and **Function** lists. Then click **Add Trace**. Each additional trace appears in the report and in the **Project Tree**.



7. Click **OK** to accept the specified values or **Cancel**, **Use Defaults**, or **Save As Default** as appropriate.

Selecting an Emission Test Quantity to Plot

1. When you create the report, select Emission Test.
2. In the **Report** dialog box, select one of the following categories and apply an appropriate Quantity. The selected category provides the default name for the report.

Variables	Intrinsic variables, such as frequency or theta, or user-defined project variables, such as the length of a quarter-wave transformer.
Output	User defined expressions applied to derive quantities from the original field

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Variables	solution.
Sphere	A sphere of 1, 3, 10, or 30 meters, or of the same dimensions and RBS Simple or RBS exact (where RBS is random binary [bit] sequence).
Cylinder	A cylinder of 3 or 10 meters, or of the same dimensions and PRBS Simple or PRBS exact.

3. Select a **Function** for the quantity from the function list.
4. For Emission Test, the **Report** dialog also contains a button for specifying the digital signal options. The default values are a rise time of 0 seconds, and a hold time of 1 second. To specify other values, click **Digital Signal...**.

This displays the **Digital Signal Options** dialog box. It contains fields for the rise time and hold time.

5. **OK** the specified values or Cancel, Use Defaults, or Save As Default as appropriate.

Plotting Imported Solution Data

1. In the **Solution** drop-down menu of the *Report* dialog box, click the imported data you want to plot.
2. Follow the procedure for creating a report.

Setting a Range Function

To apply a range function to the Y, Z, or Mag component of a trace:

1. Click **Range Function** in the *Report* dialog box.

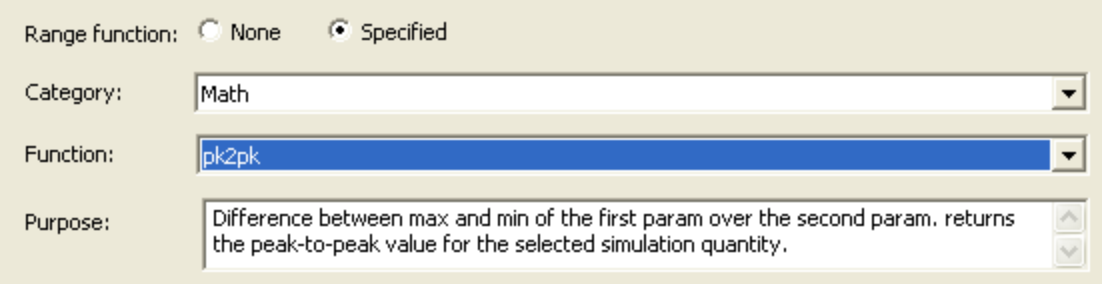
   This opens the *Set Range Function* dialog box. The functions available are the same as described in the Selecting a Function section, with the exception of those for the Eye Measurements category.

2. To enable Range function selection, click the Specified radio button.

   Selecting the None radio button disables the Range Function fields.

3. Select the **Category**, and then an associated Function to apply. The available categories depend on the plot, and Category enables the display of associated functions.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Given a selected Function, and Category, the **Set Range Function** dialog displays a text field that explains the Purpose of the function. See figure above.

Selecting a function causes the display of a description in the **Purpose** field. If the function requires a value (such as the XatYVal Math function or the pw_minus_max Pulse Width function), the following table the function field displays the name, editable value field, unit, and description.

4. Use the **Over Sweep** drop-down menu to select from available sweeps.
5. To select from available Sweeps, or to edit them, use the ellipsis [**...**] button and uncheck **Use All Sweeps**.

   This enables a list of the sweeps. The sweep(s) you select is displayed on the Over Sweep line. You can use the buttons to **Clear All Selections** or **Select All** sweeps.
6. Select the Sweeps **Default** or **Edited** radio buttons to specify whether to accept the default or edited sweeps.
7. To edit the sweeps further, select the ellipsis button to display an *Edit Sweep* dialog box.

   For frequency variables, this lets you specify a single value, linear step, linear count, decade count, octave count, or exponential count. You can **Add** legal values to the list of sweep values, **Update** the list for changes, or **Delete** selected entries.
8. Click **OK** to apply the range function.

Range Functions

The following table shows the **Functions** according to their **Categories**. The most commonly used range categories are **Math** and **Radiation**. Other functions could be used if needed. Use the category links to navigate to tables with definitions of functions.

Category	Functions
Math	max, min, pk2pk, rms, sum, mean, variance, stddev, integabs, avgabs, rmsAC, ripple, pkavg, XatYMin, XatYMax, YAtXMin, YAtXMax, XAtYVal, YAtXVal, XWidthAtYVal

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PulseWidth	pulsemin, pulsemax, pulsemintime, pulsemaxtime, pulsefall9010, pulsefront1090, pulsefront3090, pulsetail50, pulsewidth5050, pw_plus, pw_minus, pw_plus_avg, pw_minus_avg, pw_plus_max, pw_minus_max, pw_plus_min, pw_minus_min, pw_plus_rms, pw_minus_rms
Overshoot, Undershoot	overshoot, undershoot.
TR & DC	crestfactor, formfactor, distortion, fundamentalmag, delaytime, risetime, deadtime, settlingtime
Error	iae, ise, itae, itse
Period	per, pmax, pmin, prms
AC	gainmargin, phasemargin, gaincrossover, phasecrossover, lowercutoff, uppercutoff, bandwidth, peakgain, peakgainfreq.
Radiation	xdb10bandwidth, xdb20bandwidth, lSidelobeX, lSidelobeY, rSidelobeX, rSidelobeY
Eye Measurements	EyeLevelZero, EyeLevelOne, EyeAmplitude, EyeHeight, EyeSignalToNoise, EyeOpeningFactor, EyeWidth, EyeJitterP2P, EyeJitterRMS, EyeRiseTime, EyeFallTime, MinEyeWidth, MinEyeHeight

> **Note:**
>
> Refer to the SI Wave or Nexxim help for more information. The Purpose field offers brief descriptions of each.

Math Functions	
***avg**	Returns the average of the values of the selected quantity. avg = (Area between the curve and the X-axis) / (X length of the curve)
avgabs	Returns the mean of the absolute value of the selected quantity.
integabs	Absolute value of integral.
max	Returns maximum value of the simulation quantity.
mean	Returns the average in the set of quantities selected. mean = sum(all y-value) / (number of y-values)
min	Returns the minimum value of the simulation quantity.
rms	Returns the root mean square value of the selected quantity.
rmsAC	Returns the AC RMS for the selected quantity.
ripple	Returns the ripple factor (AC RMS/Mean) for the selected quantity.
pkavg	Returns the ratio of the peak to peak-to-average for the selected quantity.
pkp2pk	Peak to peak. Difference between max and min of the first parameter over the second parameter. Returns the peak-to-peak value for the selected

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Math Functions	
	simulation quantity.
sum	Returns the sum of the given values.
stddev	Returns the standard deviation of given values.
variance	Calculates the variance of the given values.
XAtYMax	Threshold crossing time: report first time (x value) at which an output quantity crosses YMax.
XAtYMin	Threshold crossing time: report first time (x value) at which an output quantity crosses a user definable threshold.
XAtYVal	Returns the X value at the first occurrence of Y value.
XWidthAtYVal	Returns the X width between the first 2 occurrences of Y value.
YAtXMax	Returns the X value at maximum value of Y.
YAtXMin	Returns the Y value at minimum value of X.
YAtXVal	Returns the Y value at the first occurrence of X value.

Radiation Functions	
lsidelobeX	The 'x' value for the left side lobe: the next highest value to the left of the max value.
lsidelobeY	The 'y' value for the left side lobe: the next highest value to the left of the max value.
rSidelobeX	Returns the X value of right side-lobe occurrence.
rSidelobeY	Returns the Y value of right side-lobe occurrence.
xdb10beamdwidth	Width between left and right occurrences of values 'x' db10 from max. Takes 'x' as argument (3.0 default). To use this function, you need to open the Add Trace Characteristics dialog and select the Radiation category.
xdb20beamwidth	Width between left and right occurrences of values 'x' db20 from max. Takes 'x' as argument (3.0 default) To use this function, you need to open the Add Trace Characteristics dialog and select the Radiation category.

Pulse Width Functions

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

In this table, the functions with the asterisk (*) do not appear on the Range Function drop-down menu. They can still be used via text entry.

pulsefall9010	Returns the pulse fall time of the selected quantity according to the 90%-10% estimate.
pulsefront1090	Returns the pulse front time of the selected quantity according to the 10%-90% estimate.
pulsefront3090	Returns the pulse front time of the selected quantity according to the 30%-90% estimate.
pulsemax	Returns the pulse maximum from the front and tail estimates for the selected quantity.
pulsemaxtime	Returns the time at which the maximum pulse value of the selected quantity is reached.
pulsemin	Returns the pulse minimum from the front and tail estimates for the selected quantity.
pulsemintime	Returns the time at which the minimum pulse value of the selected quantity is reached.
pulsetail50	Returns the pulse tail time of the selected quantity from the virtual peak to 50%.
pulsewidth5050	Returns the pulse width of the selected quantity as measured from the 50% points on the pulse front and pulse tail.
***pwl**	Piecewise Linear.
***pwl_periodic**	Piecewise Linear for periodic extrapolation on x.
***pwlx**	Piecewise Linear x with linear extrapolation on x.
pw_minus	Pulse width of the first negative pulse.
pw_minus_avg	Returns the average of the negative pulse width input stream.
pw_minus_max	Returns the maximum pulse width of the negative pulse of input stream.
pw_minus_min	Returns the minimum pulse width of the negative pulse of input stream.
pw_minus_rms	Returns the rms of the negative pulse width input stream.
pw_plus	Returns the pulse width of the first postive pulse.
pw_plus_avg	Returns the average of the positive pulse width input stream.
pw_plus_max	Returns the maximum pulse width of the positive pulse of input stream.
pw_plus_min	Returns the minimum pulse width of the positive pulse of input stream.
pw_plus_rms	Returns the rms of the positive pulse width input stream.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Overshoot/Undershoot	
Overshoot	Calculates peak overshoot given a threshold value and number of evenly spaced points over entire time range.
Undershoot	Calculates peak undershoot given a threshold value and number of evenly spaced points over entire time range.

TR & DC Functions	
crestfactor	Returns the crest factor (peak/RMS) for the selected simulation quantity.
formfactor	Returns the form factor (RMS/Mean Absolute Value) for the selected quantity.
distortion	Returns the total distortion for the selected simulation quantity and an additional argument frequency, which is the frequency in Hz at which to calculate the fundamental RMS of the simulation quantity.
fundamentalmag	Returns the RMS value of the fundamental frequency for the selected quantity, and an additional argument, Frequency, which specifies the fundamental frequency.
delaytime	Obtains the time from zero to 50% of the target point.
risetime	Obtains the time taken to go from 10% to 90% of target point.
deadtime	Obtains the latest time when the qtyl is within a tolerance of zero.
settlingtime	Returns the latest time at which the value of the selected simulation quantity fell outside its tolerance band. The target value of the quantity and the +/- bandwidth of the tolerance band are the additional arguments.

Error Functions	
iae	Returns the integral of the absolute deviation of the selected quantity from a target value that is entered via the additional argument.
ise	Returns the integral of the squared deviation of the selected quantity from a target value that is entered via an additional argument.
itae	Returns the time-weighted squared deviation of the selected quantity from a target value that is entered via an additional argument.
itse	Returns the time-weighted squared deviation of the selected quantity from a target value that is entered via an additional argument. To use this function, you need to open the Add Trace Characteristics dialog and select the Error category.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

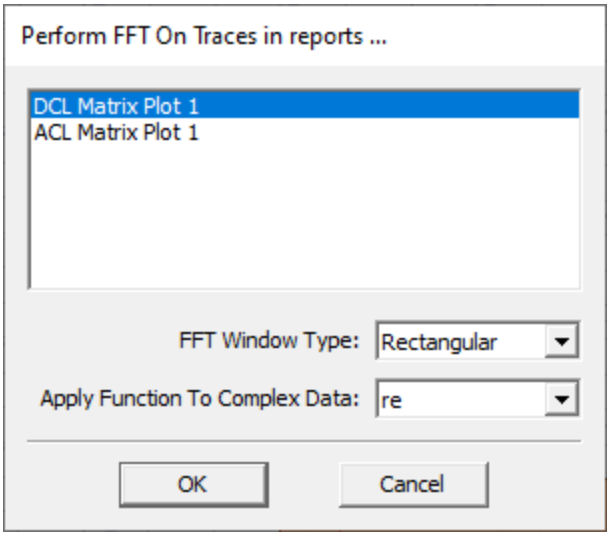
Periodic Functions	
per	Returns the period of a simulation quantity.
pmax	Max period of the selected simulation quantity.
pmin	Minimum period of the selected simulation quantity.
prms	Period Root Mean Square.

AC Functions	
gainmargin	Returns the gain margin in dB at the phase crossover frequency of the selected simulation quantity. It also requires a reference simulation quantity to which the measured quantity is compared and the AC magnitude and phase angle of the reference quantity. These are entered as the arguments Reference Channel, Base Source Magnitude, and Base Source Angle.
gaincrossover	Returns the gain crossover frequency (where the gain is 0 dB) of the selected simulation quantity in Hz.
phasecrossover	Returns the phase crossover frequency, at which the phase is -180 degrees, in Hz for the selected simulation quantity.
phasemargin	Returns the phase angle in degrees at the gain crossover frequency of the selected simulation quantity.
lowercutoff	Returns the lower 3dB frequency of the selected simulation channel in Hz.
uppercutoff	Returns the upper 3dB frequency of the selected simulation channel in Hz.
bandwidth	Returns the 3dB bandwidth of the selected simulation quantity. For bandwidth, the calculation is based on 3dB below the maximum peak.
peakgain	Returns the peak value of gain of the selected simulation quantity in dB.
peakgainfreq	Returns the frequency in Hz at which the peak gain of the selected simulation quantity occurs.

Perform FFT on a Report

You can perform FFT on an existing 2D plot by using the **Results** > **Perform FFT** command. You can perform TDR on an existing 3D plot by using **HFSS** > **Results** > **Perform FFT**. This opens the **Perform FFT on Traces in Reports** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

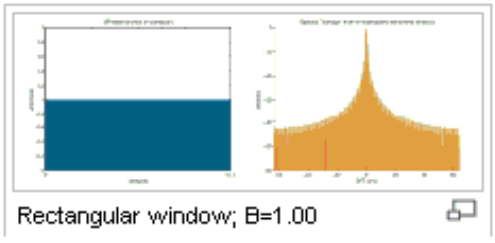


1. Select the report you want from the list.
2. Select the FFT window type to apply. Winding functions cause the FFT of the signal to have non-zero values away from ω. Each window function trades off the ability to resolve comparable signals and frequencies versus the ability to resolve signals of different strengths and frequencies.
3. Select the function to apply to complex data.

   The new report appears under **Results** in the **Project Manager**. The new report name prefixes FFT to the name of the original report. Trace names are also prefixed with FFT.

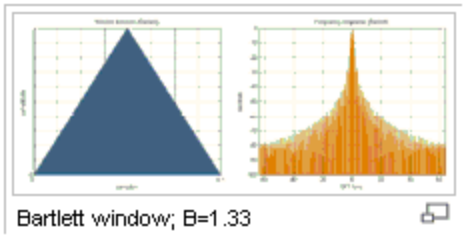
FFT Window Functions

The window type list for **Perform FFT on Report** includes:

Window Function	Preferred Use
Rectangular	A low dynamic range function offering good resolution for signals of comparable strength. Poor when signals have very different amplitudes. $w(n)=1$ 
Tri	A Bartlett window with the endpoints valued at zero.

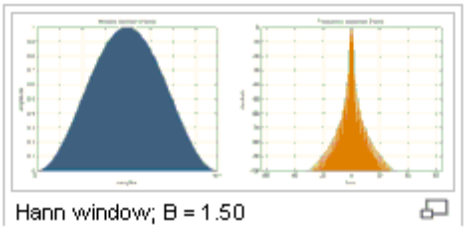
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Window Function	Preferred Use


Bartlett window; B=1.33

Van Hann — A moderate dynamic range function, designed for narrow band applications.

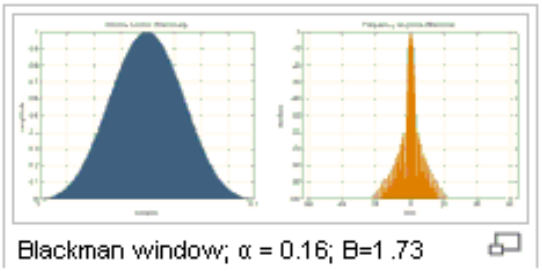
$$w(n) = 0.5(1 - cos\frac{2\pi n}{N-1})$$


Hann window; B = 1.50

Blackman — A high dynamic range function, with lower resolution, designed for wide band applications.

$$w(n) = \alpha_0 - \alpha_1 cos(\frac{2\pi n}{N-1}) + \alpha_2 cos(\frac{4\pi n}{N-1})$$

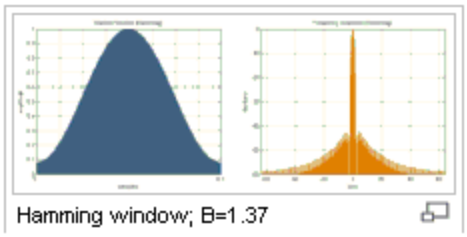
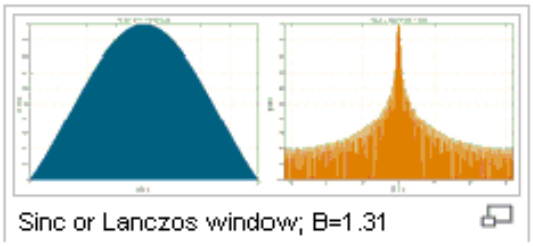
where a_0 =(1-α)/2; α_1=1/2; α_2=α/2


Blackman window; α = 0.16; B=1.73

Hamming — A moderate dynamic range function, designed for narrow band applications. It minimizes the maximum sidelobe.

$$w(n) = 0.54 - 0.46cos\left(\frac{2\pi n}{N-1}\right)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Window Function	Preferred Use
	 Hamming window; B=1.37
Lanczos	The Lanczos window offers a windowed form of the infinite sinc filter, providing the central lobe of a horizontally stretched sinc, sinc(x/a) for -a £ x £ a.
	 Sinc or Lanczos window; B=1.31
Weber	
Welch	This approach applies a parabola-shaped window to the frequency domain data. It is based on the Bartlett method but splits the signal into overlapping segments, which are then windowed. The intent is to balance the influence of data in the center of the function.

Apply FFT to Report Functions

The choices in the Perform FFT on Traces in Reports dialog include:

ang_deg	**Angle (phase) of a complex number, cut at +/- 180**
ang_deg_val	**Angle (phase of a complex number in unitless degree values. Returns simple numbers.**
ang_rad	**Angle in radians**
arg	
cang_deg	**Cumulative angle (phase) of the first parameter (a complex number) in degrees, along the second parameter (typically sweep**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	variable). Returns a double precision value cut at +/-180.
cang_deg_val	**Cumulative angle (phase) of the first parameter of the selected simulation quantity in unitless degree values. Returns simple numbers.**
cang_rad	**Cumulative angle of the first parameter in radians along a second parameter (typically a sweep variable) Returns a double precision value.**
dB(x)	**20*log10(|x|)**
dB 10normalize	**10*log [normalize(mag(x))]**
dB 20normalize	**20*log [normalize(mag(x))]**
dBc	
GetGroupDelay	
im	**Imaginary part of the complex number**
mag	**Magnitude of the complex number**
normalize	**Divides each value within a trace by the maximum value of the trace. ex. normalize (mag(x))**
re	**Real part of the complex number**

Note:

The evaluated value of an expression is always interpreted in SI units. However, when an angle quantity is plotted in a report, you have the option to plot values in units other than SI. If you want to plot the polar angle of a complex simulation result, S_{11} say, you can choose between ang_deg(S_{11}) and ang_rad(S_{11}). Both of these return the exact same angle quantity but in degree and radian units respectively.

Note that when used in expressions, some surprising outcomes might result. For example, the expression "1+ang_deg(S_{11})" represents an ‘angle’ and the number "1" is treated as "1 rad". The angle SI unit is attached to any unitless number that is added/subtracted from an angle value. If you want to treat "1" as degrees, make it explicit and use "1deg + ang_deg(S_{11})" instead.

If you are interested in unitless degree values, two additional functions exist: ang_deg_val(S_{11}) and cang_deg_val(S_{11}). These return simple numbers and are treated as such by any expression. If the complex S_{11} lies on the positive Y axis say, ang_deg_val (S_{11}) would be 90 and "1 + ang_deg_val(S_{11})" will be 91.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Perform TDR on Report

You can perform TDR on an existing 2D plot by using the **Results** > **Perform TDR on Report** command. You can perform TDR on an existing 3D plot by using the **HFSS 3D Layout** > **Results** > **Perform TDR on Report** command. This opens a **Perform TDR on Traces in reports** dialog box.

> **Note:**
>
> The accuracy of the early time TDR results depends on the highest frequencies simulated in the frequency sweep. Use the **Time Domain Calculation** dialog box to set up the frequency sweep. Set the maximum simulation frequency for your desired rise time.

1. Select the report you want from the list in the dialog box.
2. Specify the input signal as **Step** or **Impulse**.
3. Set the **Rise Time**.
4. In the **Type** drop-down menu, select which window type to apply.

   Windowing functions cause the FFT of the signal to have non-zero values away from ω. Each window function trades off the ability to resolve comparable signals and frequencies versus the ability to resolve signals of different strengths and frequencies.

5. To specify a window width, enter a percentage into the **Width (%)** field.
6. If you select the Kaiser function, specify a number in the **Kaiser Parameter**.

The new report displays and appears in the Project tree. The new report name prefixes TDR to the name of the original report. Trace names are also prefixed with TDR.

Window Functions and Time Domain Plotting

HFSS and SIwave allow for time-domain plotting of S-parameters. Often, this feature is used to calculate a step response or time-domain reflectometry (TDR) plot of the structure being simulated. Fourier analysis provides the mathematical mechanism for transforming frequency sweep data to a time-domain plot, but two approximations are involved. First, the transform is between two sets of discrete data points, as opposed to continuous waveforms. Second, the frequency sweep data cannot have infinite bandwidth, but must truncate at some upper limit. This section discusses the implications of these approximations, and provide information for successful time-domain plotting.

Note: Frequency sweep data consists only of positive frequencies, but the negative frequencies are simply the complex conjugate of the positive: $S(-f) = S^*(f)$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This is true for any frequency-domain function when the corresponding time-domain waveform is real-valued.

Transforming Frequency- To Time-Domain

It is easier to make generalizations about the effect of finite bandwidth if we have continuous functions. Consequently, we will initially assume our frequency- and time- domain data is continuous, and defer discussion of the effects of discretization until later. With a continuous-time sweep over an infinite bandwidth, we could – at least in principle – calculate a time-domain response by multiplying our sweep data $S(f)$ with the spectrum of a time-domain excitation function and evaluating the inverse Fourier integral:

$$f(t) = \int_{-\infty}^{\infty} S(f)E(f)e^{2\pi jft}df$$

In practice, however, sweep data does not extend to infinite frequencies and is restricted to a bandwidth b. If we simply assume that the spectrum is zero-valued outside of the bandwidth, we can interpret the data as an infinite sweep that has been multiplied by a rectangular “window” function $W(f)$, with a value of 1 within the bandwidth and a value of 0 otherwise.

$$f(t) = \int_{-\infty}^{\infty} W(f)S(f)E(f)e^{2\pi jft}df = \int_{-b}^{b} S(f)E(f)e^{2\pi jft}df$$

This process is illustrated in Fig. 1, assuming that $S(f)$ $E(f)$ corresponds to an ideal unit step function in the time domain. In Fig 1.a, the frequency spectrum is truncated beyond a certain upper limit. Since multiplication in the frequency domain corresponds to convolution in the time domain, this has the effect of convolving the time-domain step with a sinc function – the inverse Fourier transform of the rectangle (Fig 1.b). The final result is an edge with a finite rise time and some oscillation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

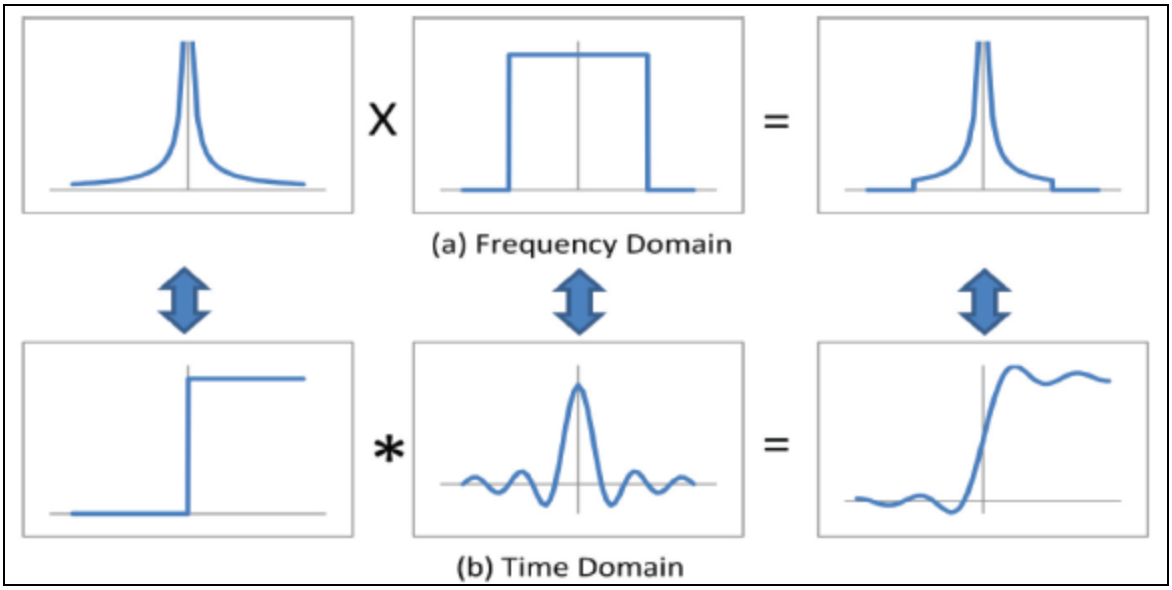


Figure 1. Multiplying the spectrum of a step function with a rectangular window produces a finite edge in the time domain

If the sweep is extended to higher frequencies – making the window function wider – the corresponding sinc pulse more closely approaches an impulse, and the time-domain edge becomes sharper. However, the oscillation never disappears for any finite sweep. Fig. 2 shows a step response for increasingly wider bandwidths.

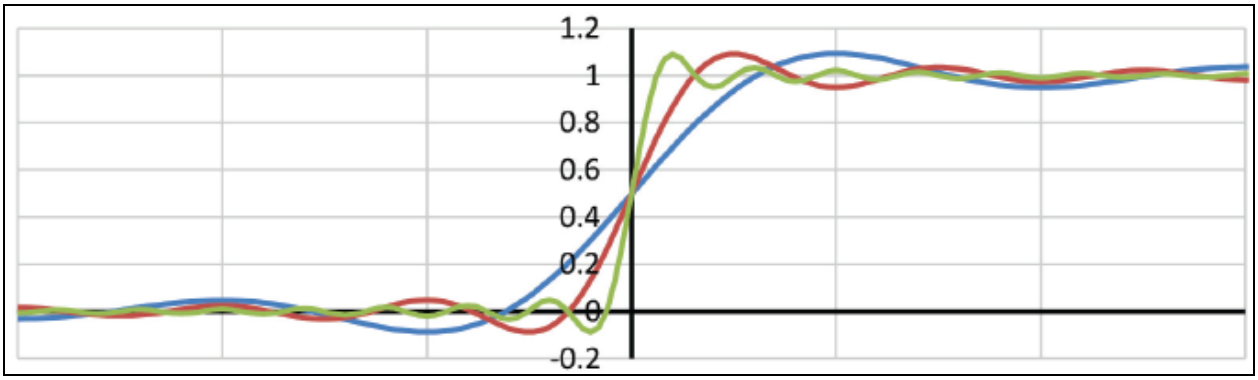


Figure 2. Increasing the width of the rectangular window makes the time-domain edge sharper, but does not eliminate the oscillation

Some distortion of the true time-domain waveform is unavoidable if the frequency sweep does not include the entire bandwidth of the signal, but there are other window functions besides the rectangle which distort the time waveform in ways which may be more desirable. In particular, it

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

would be nice to reduce the spurious oscillation. The next sections will describe the window functions available and discuss their effects.

Window Functions

The window functions available are plotted in Figs. 3 and 4, and their expressions are given in the Appendix. All the window functions have a spectral width *w* and are zero-valued for I*f* I > *w*/2. In addition to truncating the data outside of the bandwidth, the non-rectangular windows filter the spectrum inside. The windows differ from each other in how strongly they attenuate the spectrum as the frequency approaches the upper limit. The Kaiser window has a parameter, which controls how sharply it decays. For $\alpha = 0$, the Kaiser window is equivalent to the rectangular window; for $\alpha = 5.4414$, it is equivalent to the Hamming window; and for $\alpha = 8.885$, the Blackman window.

Although the TDR Options dialog allows for windows that are narrower than the bandwidth of the simulation, it is generally best to set the window width to 100% and take full advantage of the available bandwidth.

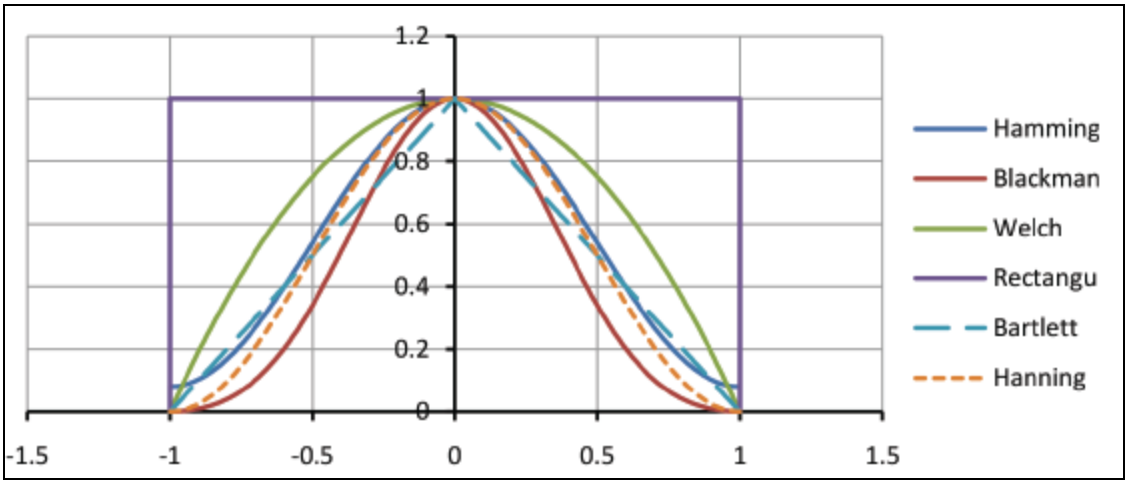


Figure 3. Window functions with a width of *w*=2

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

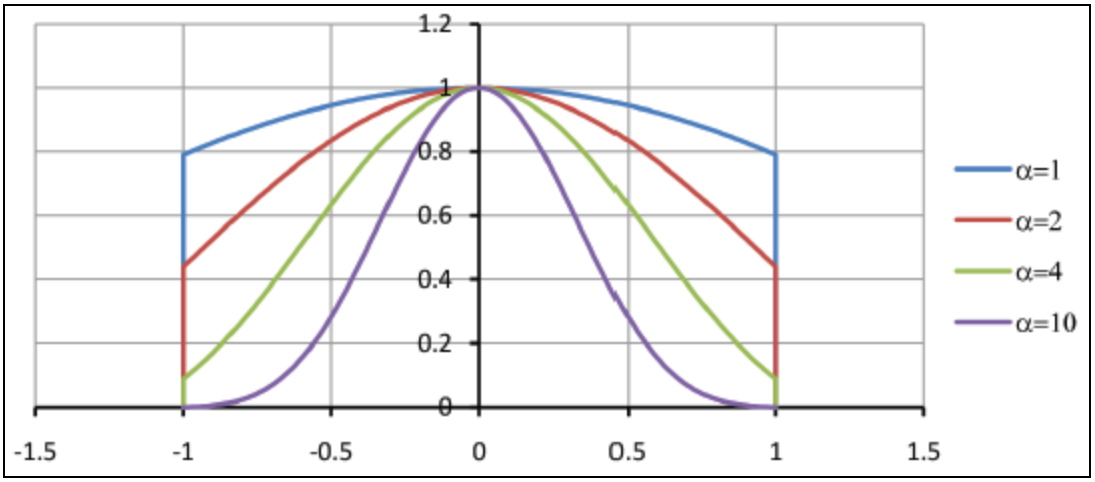


Figure 4. The Kaiser window for a width of *w*=2 and varying values

Because the spectral width *w* includes both positive and negative frequencies, it is twice the bandwidth of the sweep, *b*, which is equal to the (positive) upper frequency limit.

Ideal Step Response

It is immaterial whether we think of the window as multiplying the frequency sweep data, with the spectrum of the time-domain excitation having infinite bandwidth, or if we instead imagine we have infinite sweep data and a windowed excitation spectrum. With the latter interpretation, we can examine the effects of different windows on an ideal step without concern for what the sweep data looks like.

We will apply different windows to an ideal step function, which is approximated in HFSS and SIwave by choosing an edge and setting the rise time to 0. We continue to assume that we have a continuous spectrum, and will defer a discussion of the effects of discretization until later. The effect of the Welch window is shown in Fig. 5.

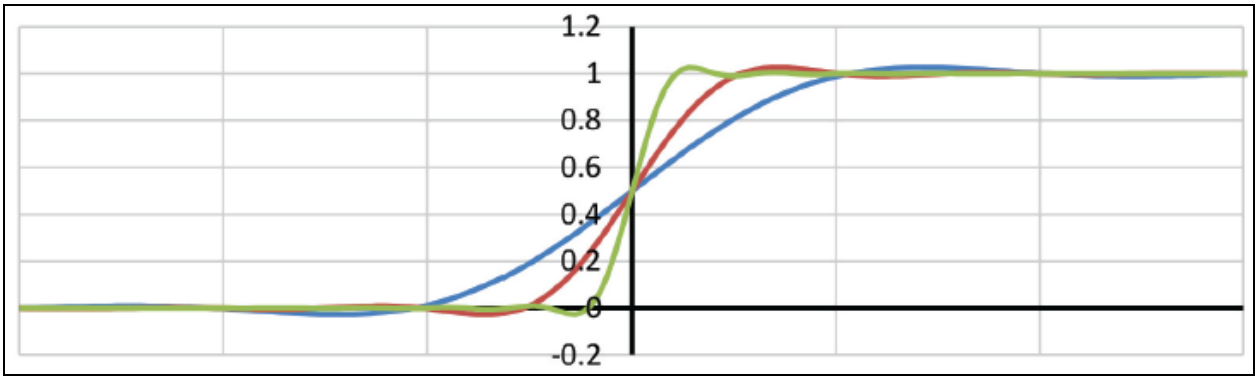


Figure 5. The effect of Welch windows of three different widths on an ideal step

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fig. 5 shows that the Welch window has substantially decreased the signal oscillation that was seen with the rectangular window. As Fig. 6 below demonstrates, the Blackman window results in almost no oscillation.

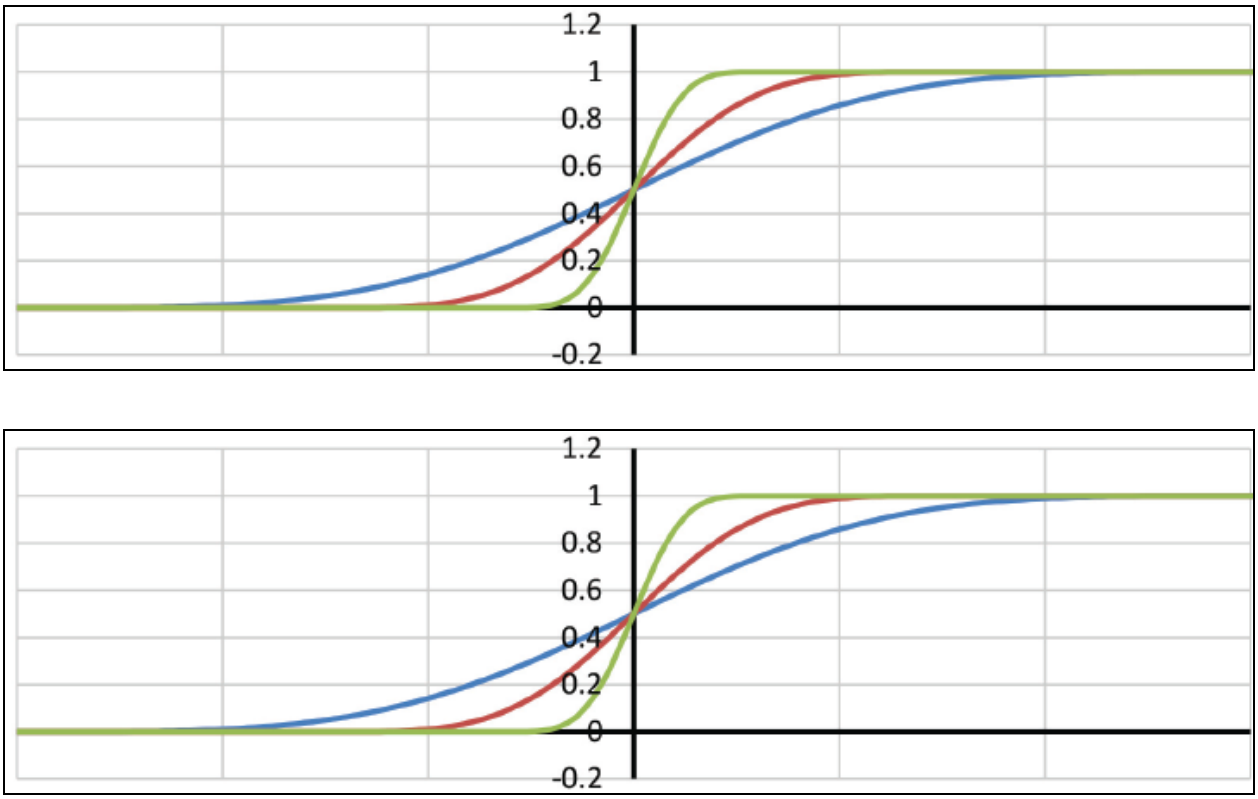
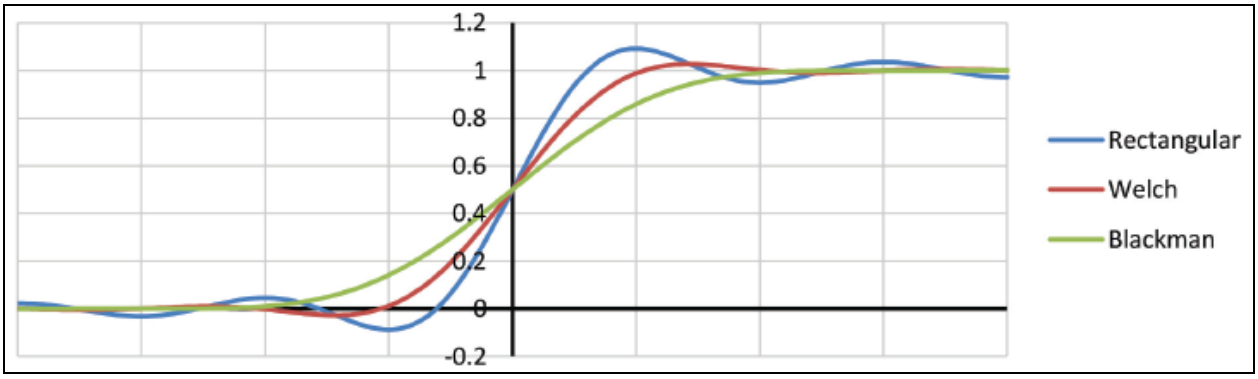


Figure 6. The effect of Blackman windows of three different widths on an ideal step.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

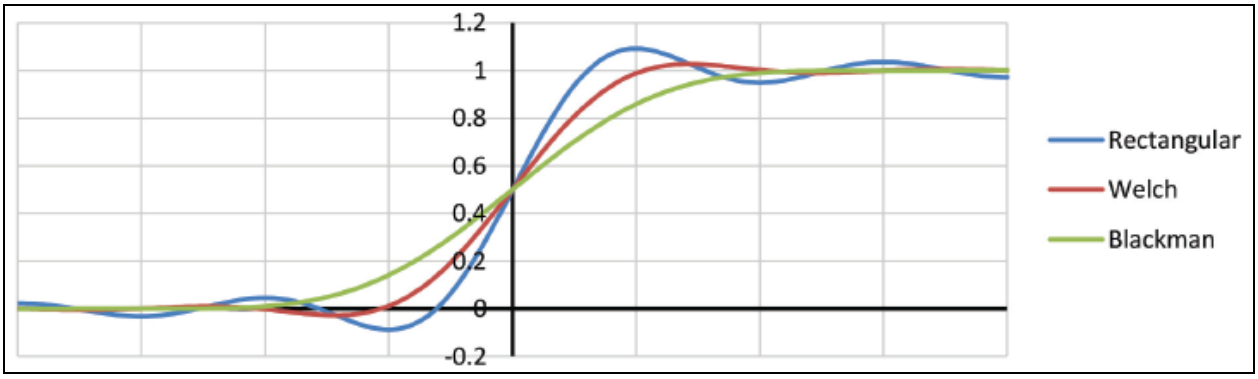


Figure 7. Step response for three different windows, each with the same bandwidth.

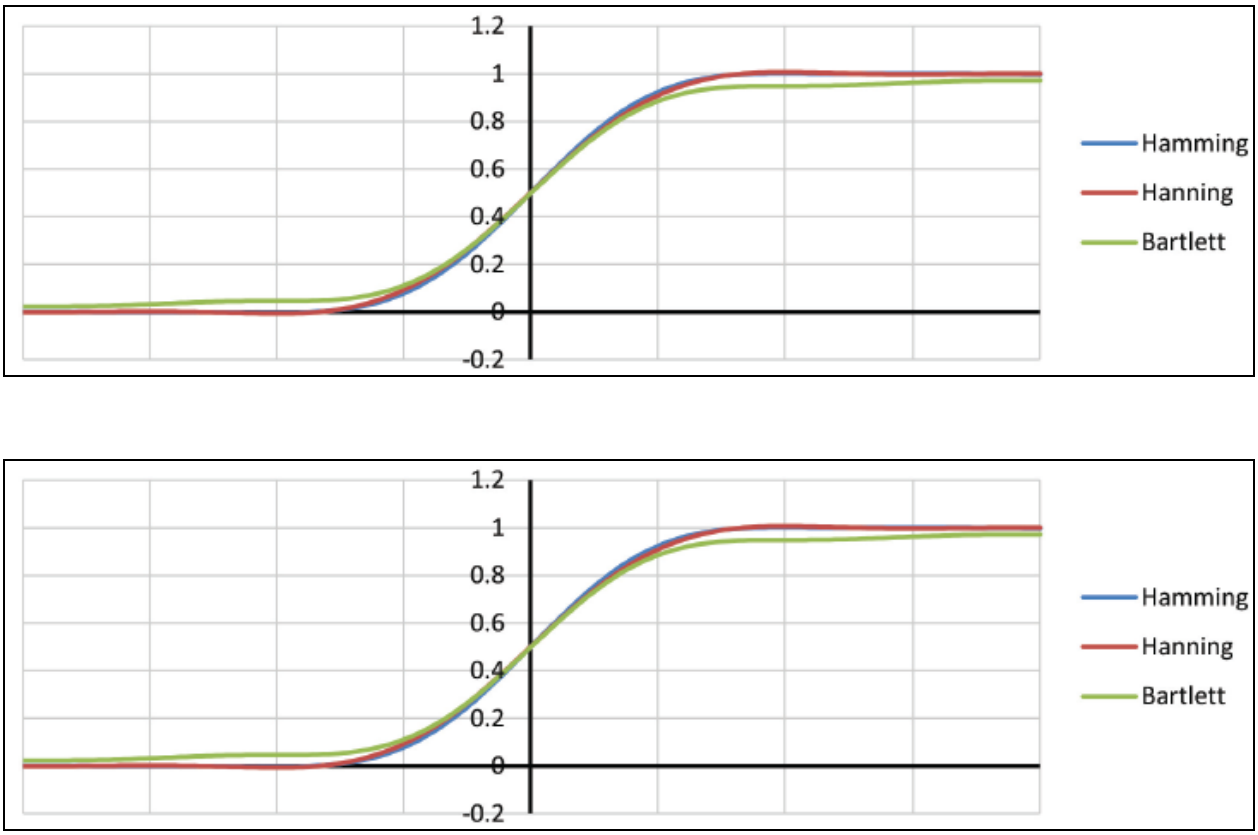


Figure 8. Hamming, Hanning, and Bartlett windows of equal bandwidth

As Fig. 8 suggests, the difference between Hamming and Hanning windows is usually quite small. The Bartlett window is generally not recommended, as it distorts the signal in the vicinity of the edge without providing any advantage over the Hamming and Hanning windows. The Kaiser window gives edges that are slower and less oscillatory with increasing α.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The rectangular, Welch, Hanning, and Blackman windows are sufficient to provide a good sampling of the edge-rate vs. oscillation tradeoff. Table 1 quantifies the characteristics of these windows on an ideal step. With the exception of the Blackman window, it is possible to derive reasonably simple expressions for the step response. In Table 1, *b* is the bandwidth or upper frequency limit of the sweep and Si(χ) refers to the sine integral function:

$$\mathrm{Si}(x) = \int_0^x \frac{\sin z}{z} dz$$

Table 1. Characteristics of selected window functions for continuous time.

Window	*Step Response* $f_s(t)$	*10-90 Edge Rate*	*Max Overshoot*
Rectangular	$\frac{1}{2} + \frac{\mathrm{Si}(2\pi bt)}{\pi}$	$\frac{0.45}{b}$	8.95%
Welch	$\frac{1}{2} + \frac{\cos(2\pi bt)}{2\pi^2 bt} - \frac{\sin(2\pi bt)}{4\pi^3 b^2 t^2} + \frac{\mathrm{Si}(2\pi bt)}{\pi}$	$\frac{0.70}{b}$	2.70%
Hanning	$\frac{2\pi + 2\mathrm{Si}(2\pi bt) - \mathrm{Si}(\pi - 2\pi bt) + \mathrm{Si}(\pi + 2\pi bt)}{4\pi}$	$\frac{0.97}{b}$	0.64%
Blackman	--	$\frac{1.19}{b}$	0.02%

Note that in the expressions for the step response, the time variable is always multiplied by the bandwidth. Changing the bandwidth scales the time response, but does not affect the shape of the edge.

Finite Edge Response

Finite edges can be simulated by providing a nonzero value for the rise time. For finite edges, the same edge rate vs. oscillation tradeoff applies. However, the spectrum of a finite edge declines with frequency at a faster rate than an ideal step. As a result, modest amounts of overshoot can be achieved even with a rectangular window. The continuous time finite edge response of a rectangular window is given by

$$f_e(t) = \frac{1}{2} - \frac{\mathrm{Si}\big(2\pi b(r-t)\big)}{\pi} + \frac{\cos(2\pi bt) - \cos\big(2\pi b(r-t)\big) + 2\pi bt\left(\mathrm{Si}\big(2\pi b(r-t)\big) + \mathrm{Si}(2\pi bt)\right)}{2\pi^2 br}$$

The value of the edge response at $t = 0$ is given by

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$f_e(0) = \frac{1}{2} + \frac{1 - \cos(2\pi br)}{2\pi^2 br} - \frac{\mathrm{Si}(2\pi br)}{\pi}$$

Along with the overshoot, f_e (O) is a useful metric for describing how closely the finite edge response approximates the ideal case, for which f_e (O) = O. The degree to which the windowed edge approximates an ideal finite edge depends only on *br*, the dimensionless product of the bandwidth and the rise time (Fig. 9).

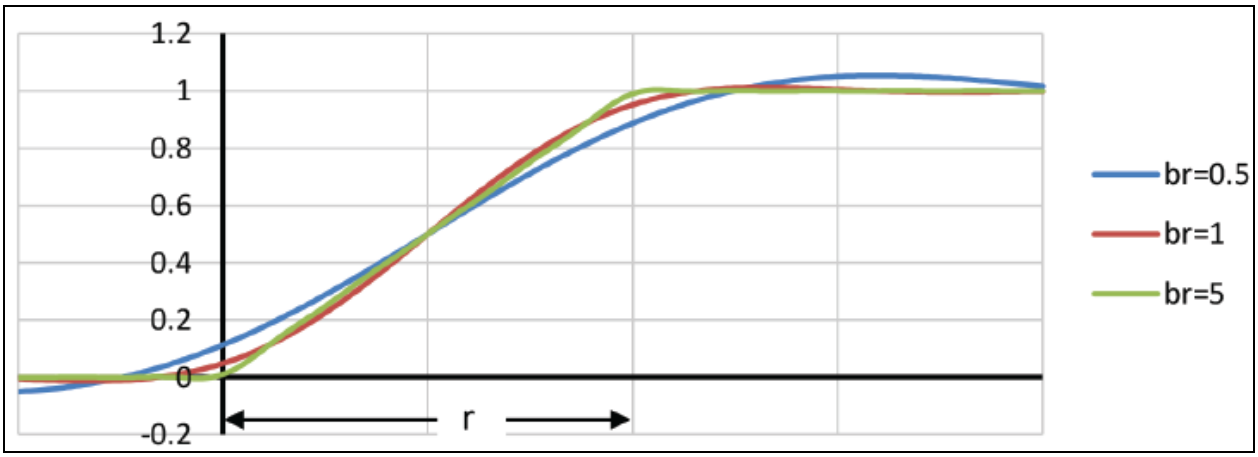


Figure 9. The effect of rectangular windows on edges with rise time *r*. The *y*-intercept and overshoot decline with increasing bandwidth *b*.

Table 2 below quantifies these relationships.

Table 2. Finite edge response for rectangular windows for continuous time

*Bandwidth*Risetime (br)*	*Edge Response at t = 0*	*Max Overshoot*
0.5	0.113	5.34%
0.75	0.056	2.41%
1	0.049	1.19%
2	0.025	0.72%
3	0.017	0.51%
5	0.010	0.32%
10	0.005	0.17%

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As Fig. 9 and Table 2 show, a fairly good finite edge can be achieved with a *br* of 1, but a *br* of around 5 is needed to give a very close approximation to the ideal finite edge.

Impulse Response

The principles behind the step and edge responses also apply to the calculation of impulse responses. Rectangular windows produce the sharpest impulses, but with the greatest amount of oscillation. Hanning and Blackman windows produce impulses that are more spread out, but with less oscillation (Fig. 10).

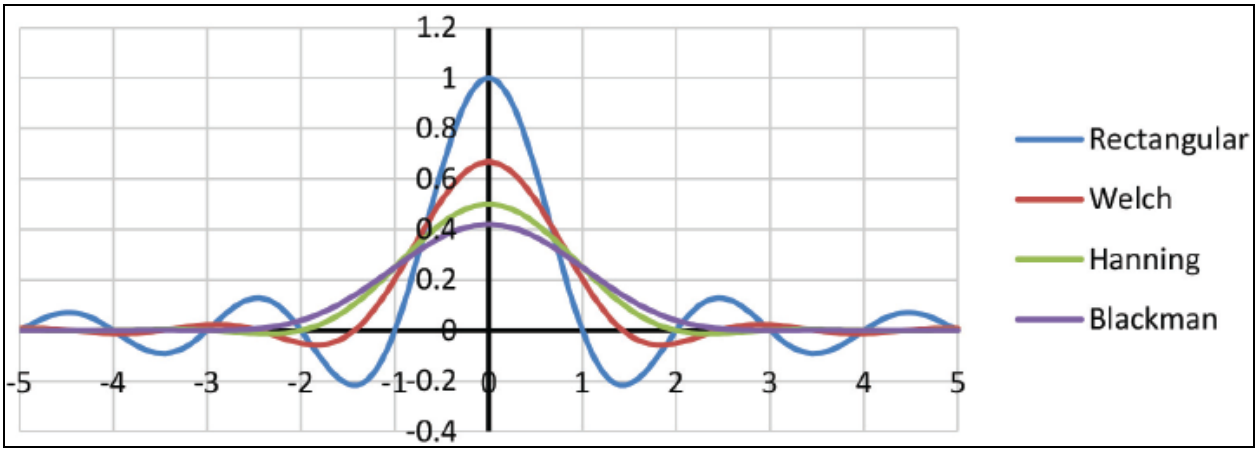


Figure 10. The impulse response for selected windows with a spectral width of 1

Discrete Time Domain Plotting

The preceding discussion treated frequency spectra as continuous functions, but in practice both the frequency and corresponding time data will be discrete. HFSS uses a discrete Fourier transform (DFT) to approximate a continuous time transform, with the frequency step size and upper limit determining the corresponding quantities in the time domain. The default time step and maximum time are given by

$$t_{step} = \frac{1}{2f_{max}} \qquad t_{max} = \frac{1}{2f_{step}}$$

Time resolution is controlled by the upper frequency in the sweep. The maximum time is controlled by the frequency resolution of the sweep. While t_{max} is fixed by the choice of frequency step and cannot be increased after the simulation, t_{step}, or the time delta, can be reduced from the default value within the **TDR Options** dialog box. Decreasing the time delta does not increase the bandwidth of the frequency data, but it does more closely approximate the band-limited continuous time spectra we have so far discussed. Although decreasing the time

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

delta will increase the time required to perform the DFT, the time required is rarely significant. Additionally, a smaller time delta has a significant benefit, as demonstrated in Fig. 11 below.

Fig. 11 shows the step response of a matched lossless transmission line for which the length is controlled by de-embedding the driving waveport, using rectangular window functions. The plots on the left are for a short transmission line length and those on the right correspond to a longer length. Fig. 11a shows the time response using the default values for t_{step}. There is some oscillation in the response, which is expected for a rectangular window, but the amplitude of the oscillation is different for the two length cases. This is problematic; since the line is matched and lossless, we expect that a length change will only affect the time delay of the response, not affect the shape or quality of the rising edge. The variation in the response is an undesirable artifact of the coarse time sampling. We can increase resolution by increasing the bandwidth of the sweep, but this requires additional simulation. Fig. 11b shows the same two cases, but with the time delta reduced using the TDR Options dialog box. The results in Fig. 11b agree with our intuition: the edge shape is the same for both line lengths and the only difference is the location of the edge. Setting the time delta to around 1/5 of the default value is generally sufficient, but finer time steps are needed for precise correlation to Tables 1 and 2.

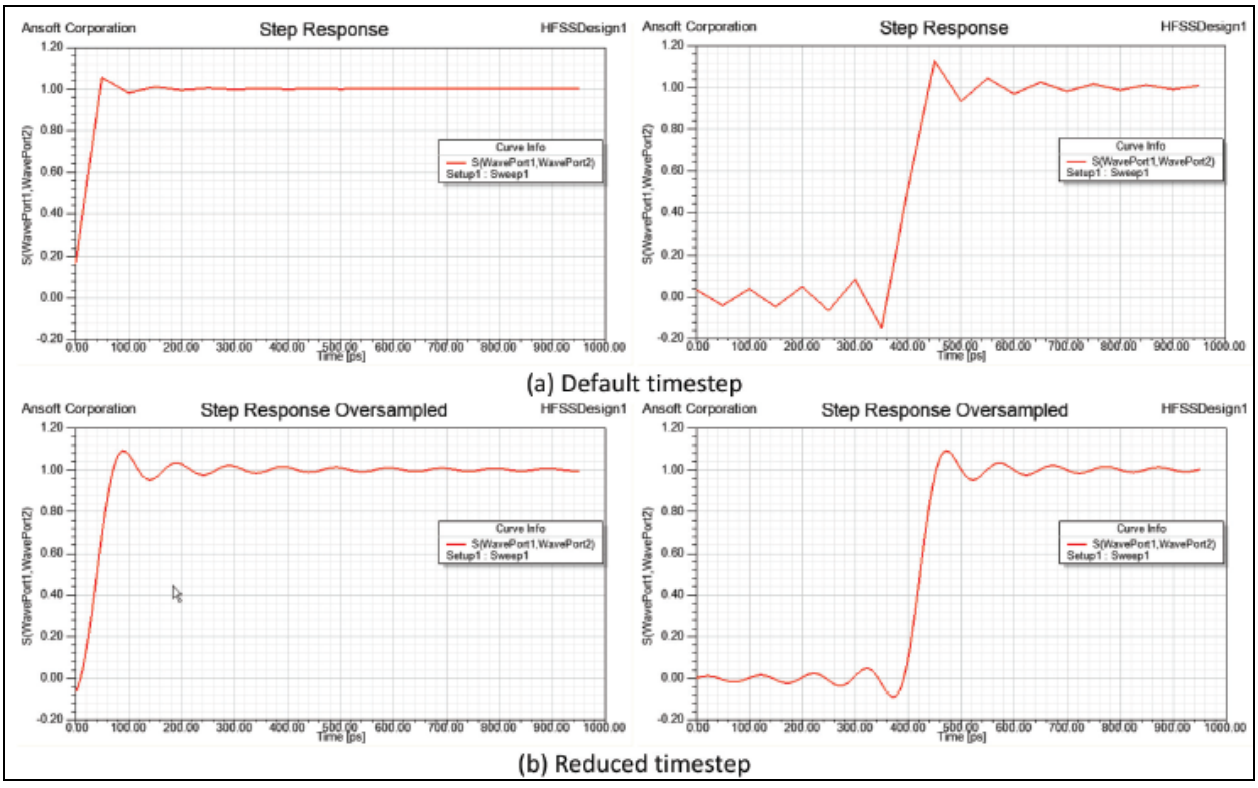


Figure 11. The time domain response of an ideal delay of two different lengths shows that a finer time sampling yields more intuitive results.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The frequency step size governs the length of the time range generated. Although a coarse frequency sampling is often sufficient to generate enough time data for a TDR plot, it is important not to set f_{step} too high in the frequency sweep. Discrete frequency spectra necessarily correspond to periodic time-domain functions, so the calculated step is actually more like a repeating series of long pulses. Fig. 12 shows the how the oscillation decays after the rising edge up to a point, but then begins increasing in anticipation of a falling edge.

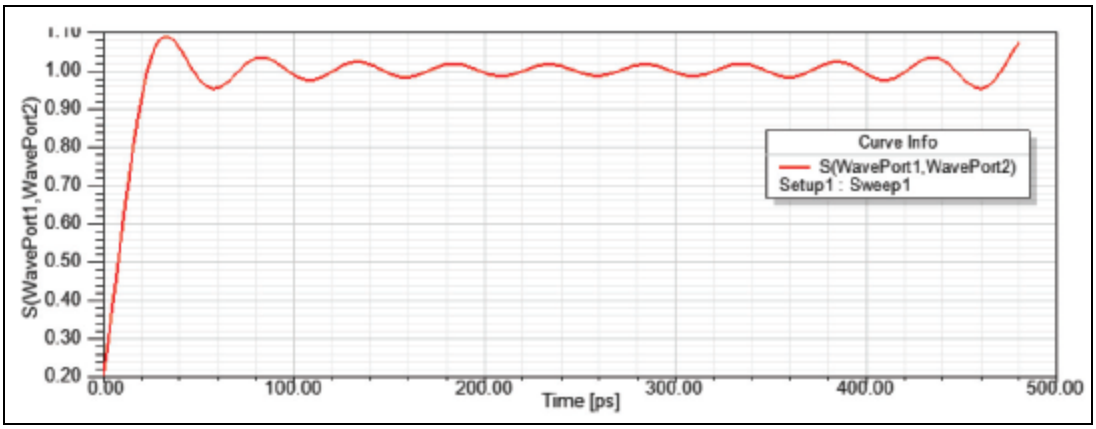


Figure 12. The oscillation caused by a rectangular window eventually starts increasing, due to the periodicity of the waveform

Setting f_{step} to a small value increases the length of the pulse, and minimizes the influence of the future falling edge. Additionally, a smaller f_{step} ensures that resonances and other sharp features in the frequency data are adequately captured. As t_{step} and f_{step} approach zero, the calculated results will converge on the continuous time descriptions given earlier.

Applications

When simulating a TDR plot, we want the fastest edge possible for the bandwidth of our simulation, subject to our preference for oscillation control. Therefore an edge with a rise time of zero is a good choice. Fig. 13 shows TDR plots of a transmission line with several impedance discontinuities. The results for a rectangular and Hanning window with a 20GHz bandwidth are compared with those for a Hanning window with a 50GHz bandwidth, which will necessarily be more accurate due to the higher bandwidth, and can be used as a reference. In all cases, the time step was set substantially lower than the default.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

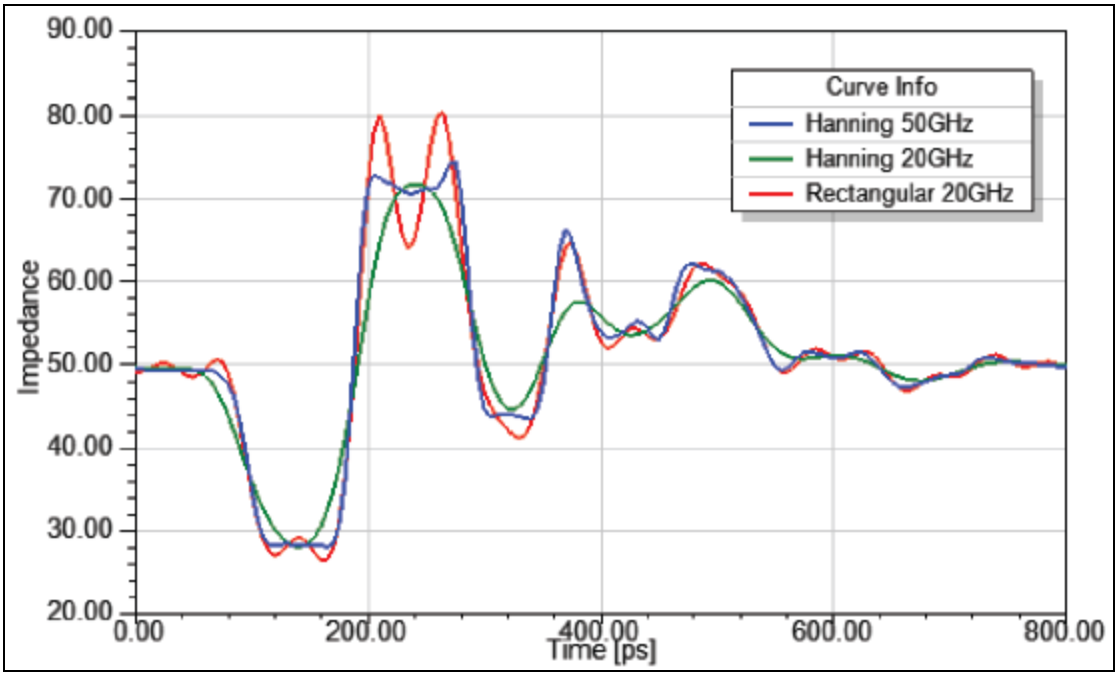
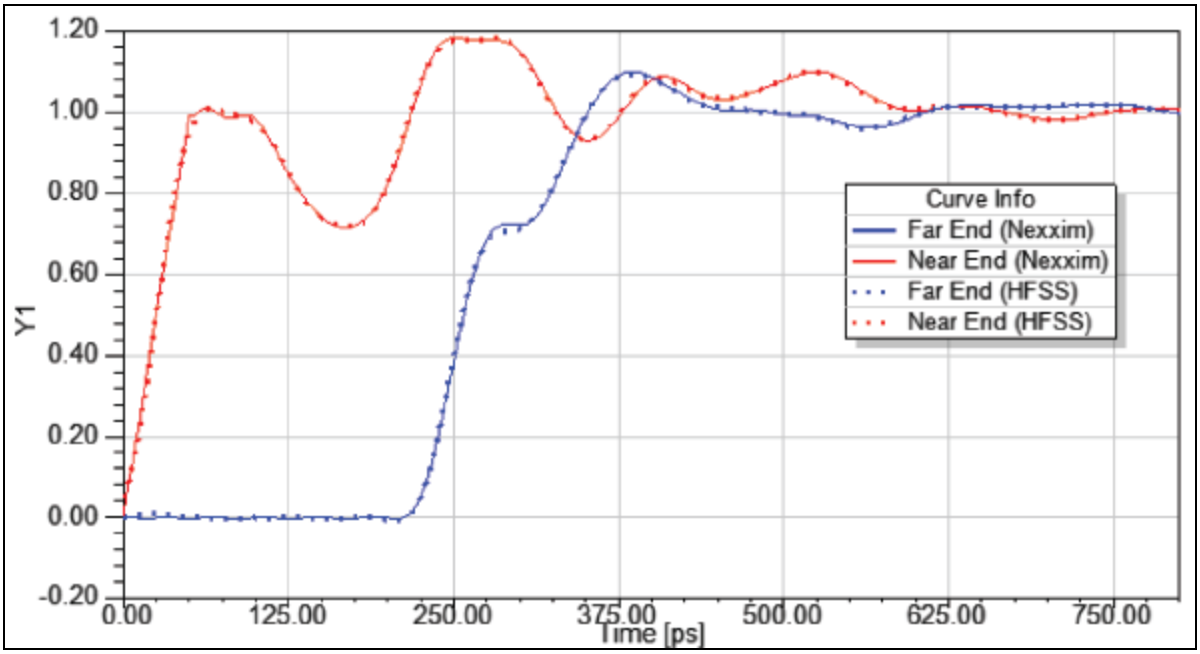


Figure 13. TDR plots for a transmission line with several impedance discontinuities

Fig. 13 shows that the rectangular window effectively captures the sharp impedance transitions, but also displays spurious oscillation. The 20GHz Hanning window does not suffer any oscillation, but gives less resolution on the sharp edges. These results are consistent with the step response characteristics of the different windows we have previously shown.

We can also use time-domain plotting to approximate how a structure would behave in a Nexxim transient simulation. When comparing the results to a transient simulation that uses a pulse or piecewise linear source, it makes sense to use a finite edge with a rectangular window. Fig. 14 compares HFSS and Nexxim results for the transmission line, using a rise time of 50ps and a rectangular window with a 20GHz bandwidth *(br* = 1).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



As Fig. 14 shows, very good agreement between Nexxim and HFSS is possible when appropriate settings are used for time domain plotting.

References

Haykin, S., and M. Moher. *Introduction to Analog and Digital Communications*, 2nd ed., Wiley, Hoboken,N.J., 2007.

Kammler, D.W. *A First Course in Fourier Analysis*. Prentice-Hall, Upper Saddle River, N.J., 2000.

Lathi, B.P. *Linear Systems and Signals*, 2nd ed. Oxford University Press, New York, 2005.

Appendix: Window Function Formulas

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rectangular	$\text{Rect}\left(\frac{f}{w}\right)$ where $\text{Rect}(x) = \begin{cases} 1 & -\frac{1}{2} \le x \le \frac{1}{2} \\ 0 & \text{otherwise} \end{cases}$
Bartlett	$1 - \left\lvert\frac{2f}{w}\right\rvert$
Blackman	$0.42 + 0.5\cos\left(\frac{2\pi f}{w}\right) + 0.08\cos\left(\frac{4\pi f}{w}\right)$
Hamming	$0.54 + 0.46\cos\left(\frac{2\pi f}{w}\right)$
Hanning	$0.5\left[1 + \cos\left(\frac{2\pi f}{w}\right)\right]$
Kaiser	$\frac{1}{I_0(\alpha)} I_0\left(\alpha\sqrt{1 - 4\left(\frac{f}{w}\right)^2}\right)$ where $I_0(x)$ is a modified Bessel function of the first kind.
Welch	$1 - \left(\frac{2f}{w}\right)^2$

Animated Reports

The following sections describe how you can postprocess field overlay displays to create and then view various animated reports. Any of the field overlay displays can be animated by cycling the overlays as a series of frames. Refer to the following topics for details on creating the field overlays:

- Overlaying Surface Currents on a 3D View
- Overlaying Far Fields on a 3D View
- Overlaying Near Fields on a 3D View

There are two modes of animation: frequency-based and phase-based. Frequency-based animation creates a series of frames by calculating the field overlay at each of the swept frequencies. Phase-based animation creates a series of frames by calculating the field overlay at a fixed set of phase deviations around a selected frequency. The animation controls can also change the simulation design point.

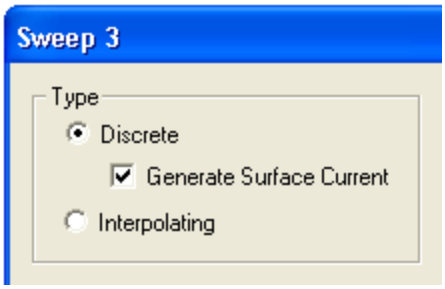
Overlaying Surface Currents on a 3D View

Surface currents calculated as the results of a Planar EM simulation can be displayed as overlays on the 3D viewer.

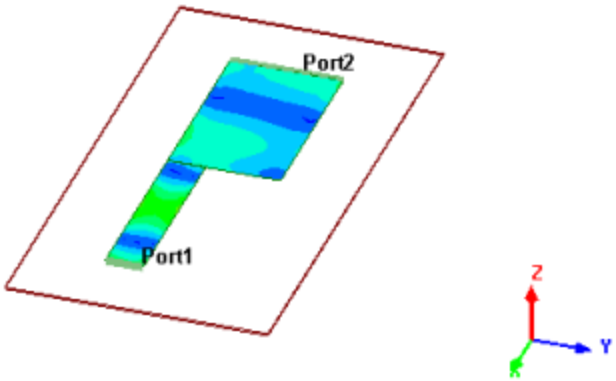
1. To ensure that the surface current information is generated, the sweep setup must specify a **Discrete** frequency sweep, and the **Generate Surface Current** option must be enabled

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(checked):

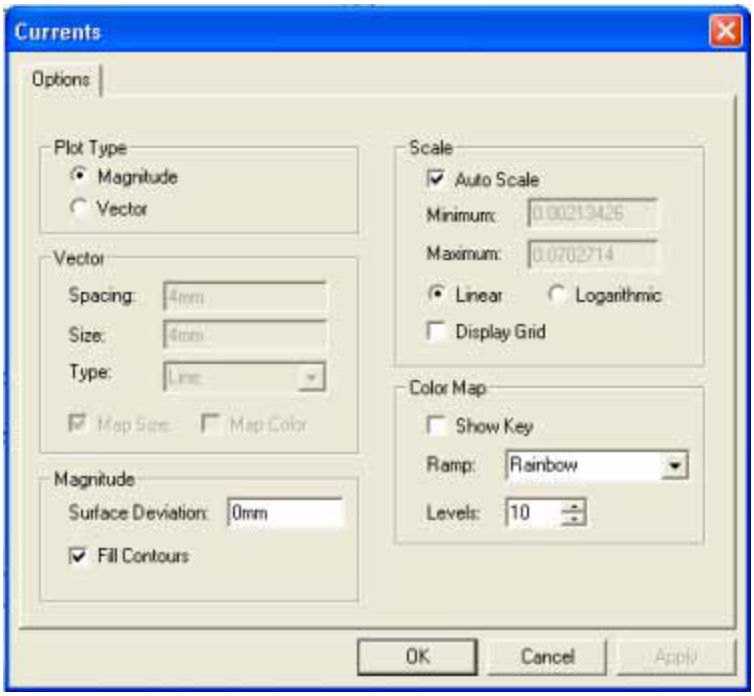


2. Run the Planar EM analysis with the sweep.
3. To display the surface current as an overlay, expand the **Analysis** icon in the Project window, and select **Setup***m* **> Sweep***n* **> Results > Display Currents** (*m* and *n* identify the particular solution setup and sweep setup, respectively). You can also select from a list of corresponding Setup/Sweep overlay choices which are displayed when you right-click **Field Overlays** in the **Project** tree. The 3D viewer window appears with the current values overlaid on the geometry:



4. To change the display properties of the surface current overlay, expand the **Results** icon in the Project window, and select **Setup***m***:Sweep***n***:Currents***k* **> Properties** (*m*, *n*, and *k* identify the particular solution setup, sweep setup, and surface current setup, respectively). The **Currents** dialog opens:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **Plot Type** panel, select **Magnitude** to enable the **Magnitude** panel options or select **Vector** to enable the **Vector** panel options.

In the **Scale** panel, select **Auto Scale** (the default), or deselect **Auto Scale** and enter custom **Minimum** and **Maximum** scaling values. Select **Linear** or **Logarithmic** scaling (the default is **Linear**), and toggle **Display Grid** on or off (the default is off).

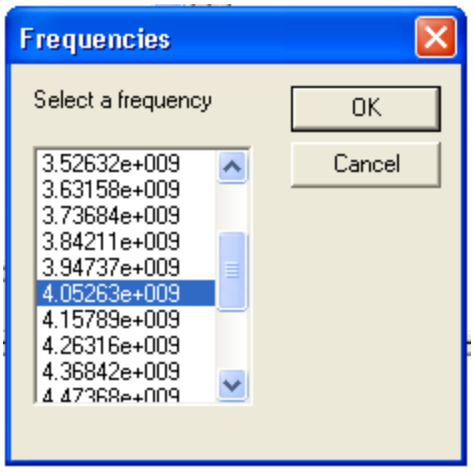
In the **Color Map** panel, select the **Ramp** type (**Rainbow** is the default; other options are **HueScale**, **Magenta**, and **Temperature**), set the number of **Levels** (the default is 10 levels), and toggle the color key (**Show Key**) on and off (the default is off).

Click **Apply** to apply any changes to the display without closing the dialog box. Click **OK** to apply any changes and close the dialog box. Click **Cancel** to close the dialog without changing any options.

5. To select the frequency for the current overlay, expand the **Results** icon in the Project window, and select **Setup*m*:Sweep*n*:Currents*k* > Frequency** (*m*, *n*, and *k* identify the particular solution setup, sweep setup, and surface current setup, respectively). The

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frequencies dialog opens:



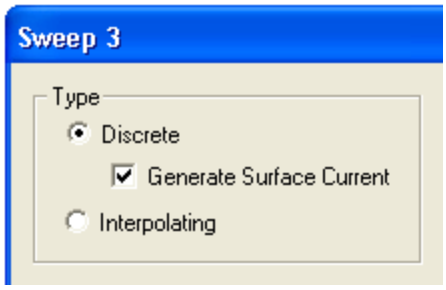
The list displays the frequencies that were swept in the analysis. When you select a frequency from the list, the overlay displays the surface current values calculated at that frequency. Click **OK** to leave the overlay at the selected frequency, or click **Cancel** to close the dialog without applying any frequency changes to the overlay.

6. To dismiss the overlay, expand the **Results** icon in the Project window, right-click **Setup***m*:**Sweep***n*:**Currents***k*, and select **Delete** from the drop-down menu **s** (*m*, *n*, and *k* identify the particular solution setup, sweep setup, and surface current setup, respectively).

Overlaying Far Fields on a 3D View

Far fields calculated as the results of a Planar EM simulation can be displayed as overlays on the 3D viewer.

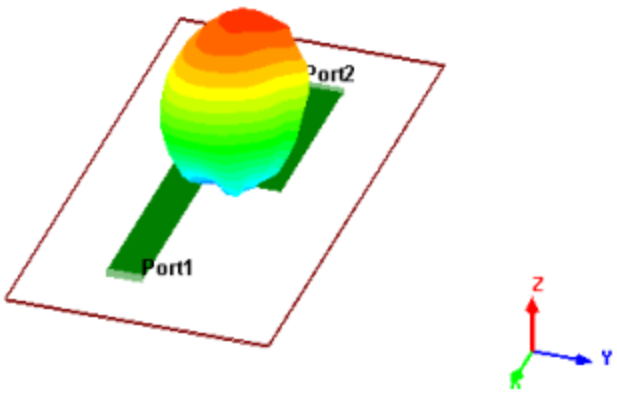
1. To ensure that the far field information can be generated, the sweep setup must specify a **Discrete** frequency sweep, and the **Generate Surface Current** option must be enabled (checked):



2. Run the Planar EM analysis with the sweep.

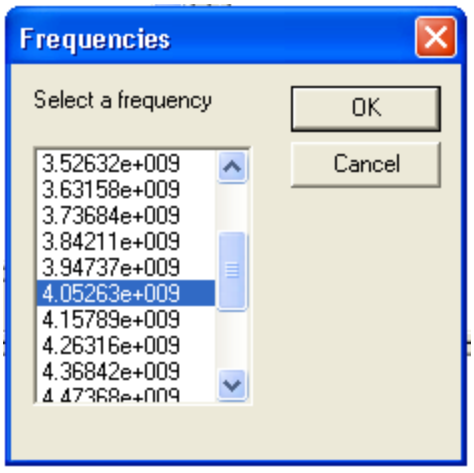
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. To display the far field as an overlay, expand the **Analysis** icon in the Project window, and select **Setup***m* **> Sweep***n* **> Results > Far Field** (*m* and *n* identify the particular solution setup and sweep setup, respectively). You can also select from a list of corresponding Setup/Sweep overlay choices which are displayed when you right-click **Field Overlays** in the **Project** tree. The 3D viewer window appears with the far field values overlaid on the geometry:



The display properties of the Far Field overlay cannot be changed. The Ramp type is Rainbow, and the number of levels is 20.

4. To select the frequency for the far field overlay, expand the **Results** icon in the Project window, and select **Setup***m***:Sweep***n***:Far Field***k* **> Frequency**. The **Frequencies** dialog opens:



The list displays the frequencies that were swept in the analysis. When you select a frequency from the list, the overlay displays the far field values calculated at that frequency.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

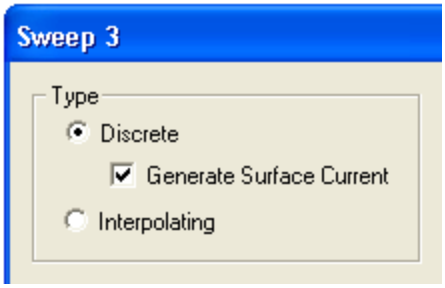
Click **OK** to leave the overlay at the selected frequency, or click **Cancel** to close the dialog without applying any frequency changes to the overlay.

5. To dismiss the overlay, expand the **Results** icon in the Project window, right-click **Setup***m*:**Sweep***n*:**Far Field***k*, and select **Delete** from the drop-down menu (*m*, *n*, and *k* identify the particular solution setup, sweep setup, and far field setup, respectively).

Overlaying Near Fields on a 3D View

Near fields calculated as the results of a Planar EM simulation can be displayed as overlays on the 3D viewer.

1. To ensure that the near field information can be generated, the sweep setup must specify a **Discrete** frequency sweep, and the **Generate Surface Current** option must be enabled (checked):

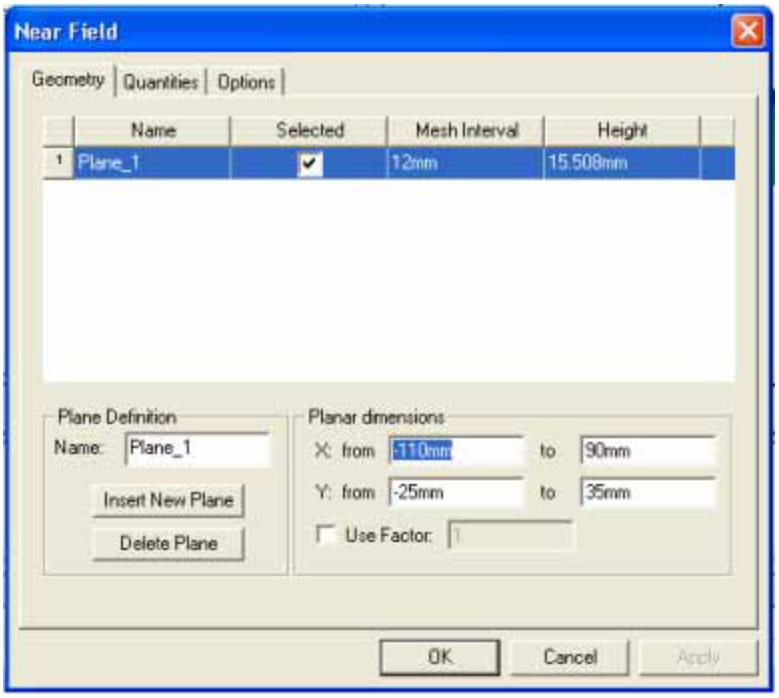


2. Run the Planar EM analysis with the sweep.
3. To display the near field overlay, expand the **Analysis** icon in the Project window, and select **Setup***m* **> Sweep***n* **> Results > Near Field** (*m* and *n* identify the particular solution setup and sweep setup, respectively). You can also select from a list of corresponding Setup/Sweep overlay choices which are displayed when you right-click **Field Overlays** in the **Project** tree.
4. The **Near Field** dialog opens. The dialog has three tabs, described below. At the bottom of each tab are three buttons:

- **Apply** is activated whenever you change a value. Click **Apply** to start the display, and then to see the effect of each change. The dialog stays open.
- When no values were changed on any tab, the **OK** button starts the display. When one or more values have been changed, **OK** applies the changes. In either case, **OK** closes the dialog and adds an icon for the overlay under the **Results** icon in the Project window.
- **Cancel** is active as long as no changes have been applied. **Cancel** closes the dialog without changing any values. If the overlay is already displayed, it does not change. If **Cancel** is pressed before any overlay is displayed, the overlay is canceled.

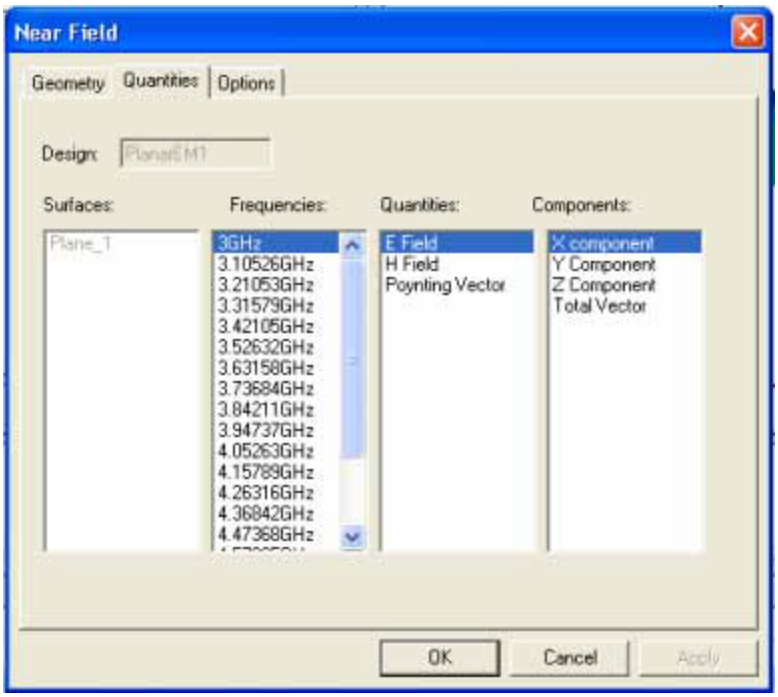
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. The **Near Field** dialog opens with the **Geometry** tab displayed:



Use the options in the **Geometry** tab to select (or define) one or more planes for the calculation, including the dimensions to be used, and a scale factor if desired.

Use the **Quantities** tab to select the near field quantity to be calculated:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

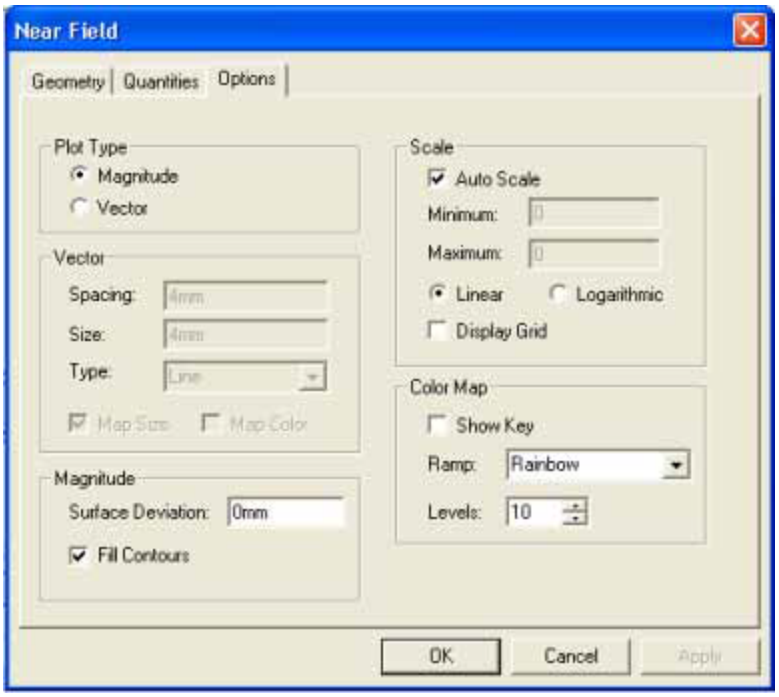
When more than one surface is involved, select a surface from the **Surfaces** list.

Select a frequency from the **Frequencies** list. The frequencies are the ones swept in the analysis.

Select a field type from the **Quantities** list. **E** is the electronic field, **H** is the magnetic field, and the **Poynting Vector** is the (E×H*) field, where H* is the complex conjugate of the H matrix.

Select a vector component from the **Components** field.

Use the **Options** tab to specify the display options for the near field overlay:



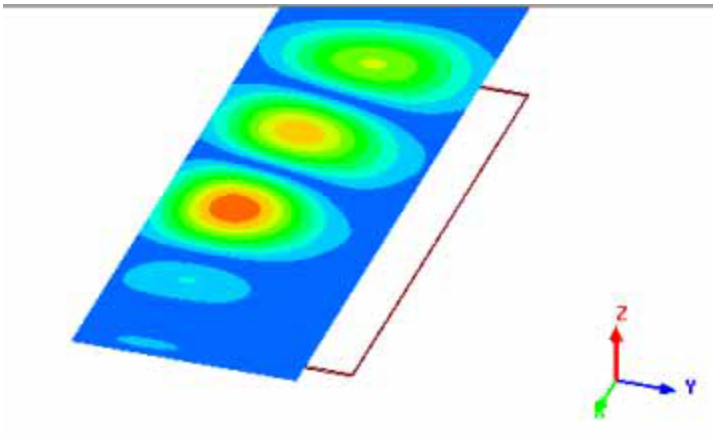
In the **Plot Type** panel, select **Magnitude** to enable the **Magnitude** panel options or select **Vector** to enable the **Vector** panel options.

In the **Scale** panel, select **Auto Scale** (the default), or deselect **Auto Scale** and enter custom **Minimum** and **Maximum** scaling values. Select **Linear** or **Logarithmic** scaling (the default is **Linear**), and toggle **Display Grid** on or off (the default is off).

In the **Color Map** panel, select the **Ramp** type (**Rainbow** is the default; other options are **HueScale**, **Magenta**, and **Temperature**), set the number of **Levels** (the default is 10 levels), and toggle the color key (**Show Key**) on and off (the default is off).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. When you click **Apply** or **OK** in any of the **Near Field** dialog tabs, the 3D viewer window appears with the near field values overlaid on the geometry:



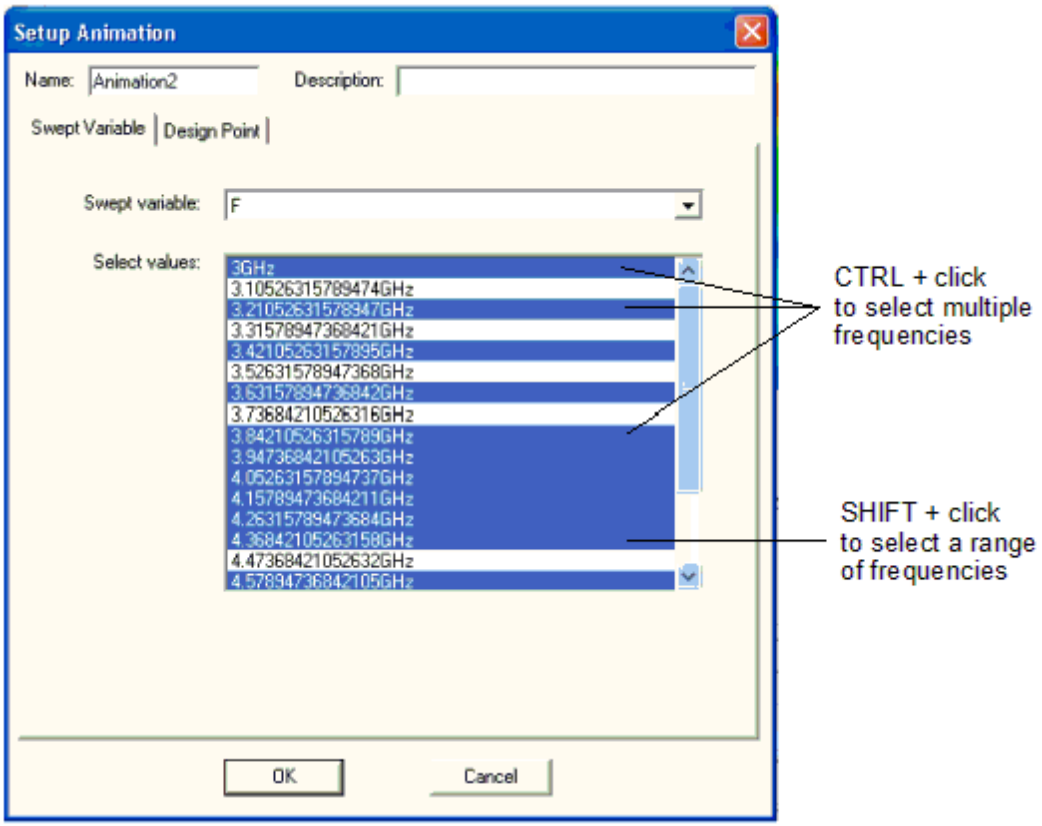
7. To dismiss the overlay, expand the **Results** icon in the Project window, right-click **Setup*m*:Sweep*n*:Near Field*k***, and select **Delete** from the drop-down menu.

Frequency Animation

1. To initiate the animation of the overlay that is currently displayed, do one of the following:
   - Select **Animate** from the **View** menu.
   - Expand the **Results** icon in the Project window, right-click the overlay entry, and select **Animate** from the menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. If no animations have been defined previously, the **Setup Animation** dialog opens:



- Specify a name in the **Name** field (or accept the default, **Animation***n*, where *n* is a numeral). Optionally, enter a description.
- For a frequency animation, select **F** as the **Swept Variable**.
- By default, all the frequencies are selected (highlighted). Hold down the Ctrl key to select multiple individual frequencies, or hold down the Shift key to select a contiguous range of frequencies. [These selection modes are illustrated in the dialog example above].
- Click **OK**.

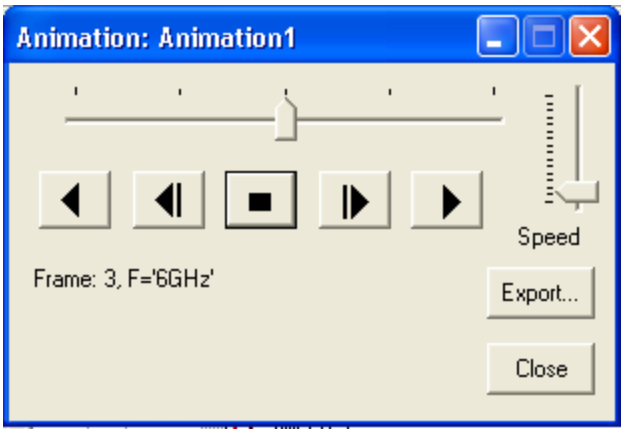
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If one or more animations have been defined, selecting **Animate** from one of the menus opens the **Select Animation** dialog:

The **Select Animation** dialog provides the following operations:

- Click to select one of the animations and click **OK** to start that animation.
- Click **New** to open the **Setup Animation** dialog described above, and close the **Select Animation** dialog.
- Select an animation and click **Delete** to delete the definition.

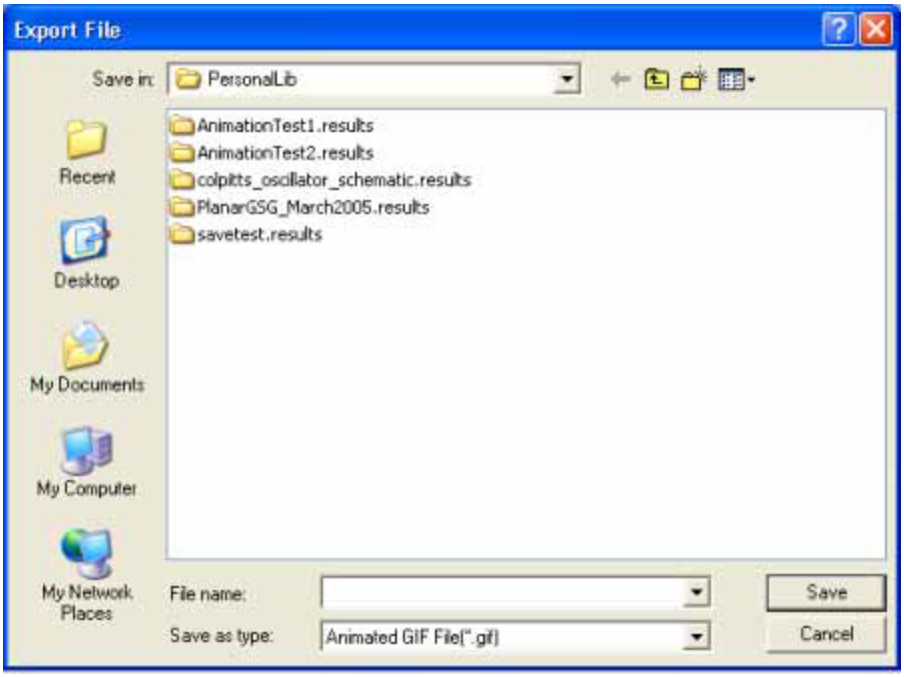
3. The frame data is automatically calculated. If the Progress window is displayed, you can monitor the progress of the calculation. When the frames have been calculated, the animation begins and the **Animation** control panel opens:



- Use the “VCR” buttons to play the animation. From left to right, the buttons are **Reverse**, **FastReverse**, **Stop**, **Fast Forward**, and **Forward**. The indicator at the top of the dialog shows the progress of the animation. Use the **Speed** slider to control the speed of the animation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To export the frame data to a file, click **Export**. The **Export File** dialog opens:



Specify the directory and file name. Use the **Save as type** menu to select the file format (Animated GIF or AVI). Click **Save** to save the data and close the dialog.

- Click **Close** on the **Animation** control panel to stop the animation and close the panel.

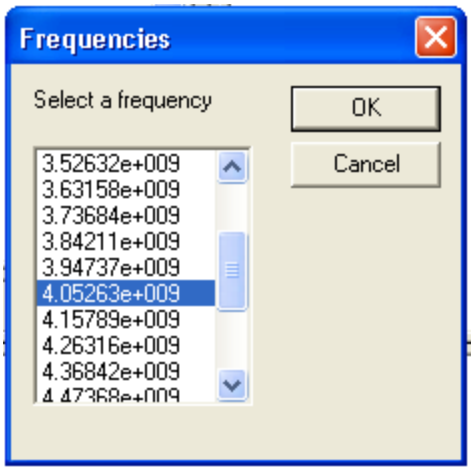
Phase Animation

1. To prepare for phase animation, you must select the base frequency.

- For surface current and far field overlays, expand the **Results** icon in the Project window, right-click the overlay, and select **Frequency** from the menu. The **Frequencies** dialog

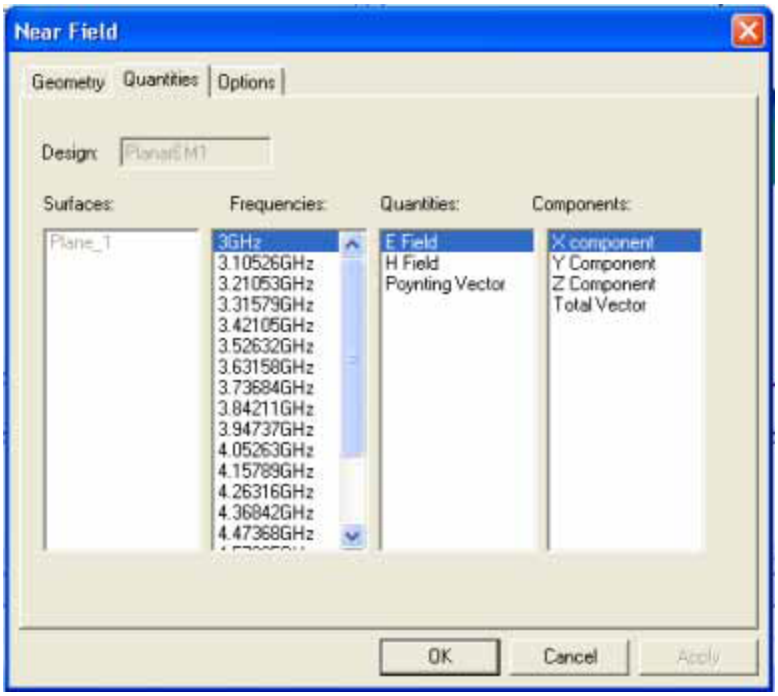
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

opens:



The list displays the frequencies that were swept in the analysis. Select a frequency from the list; the overlay displays the field values calculated at that frequency. Click **OK** to leave the overlay at the selected frequency.

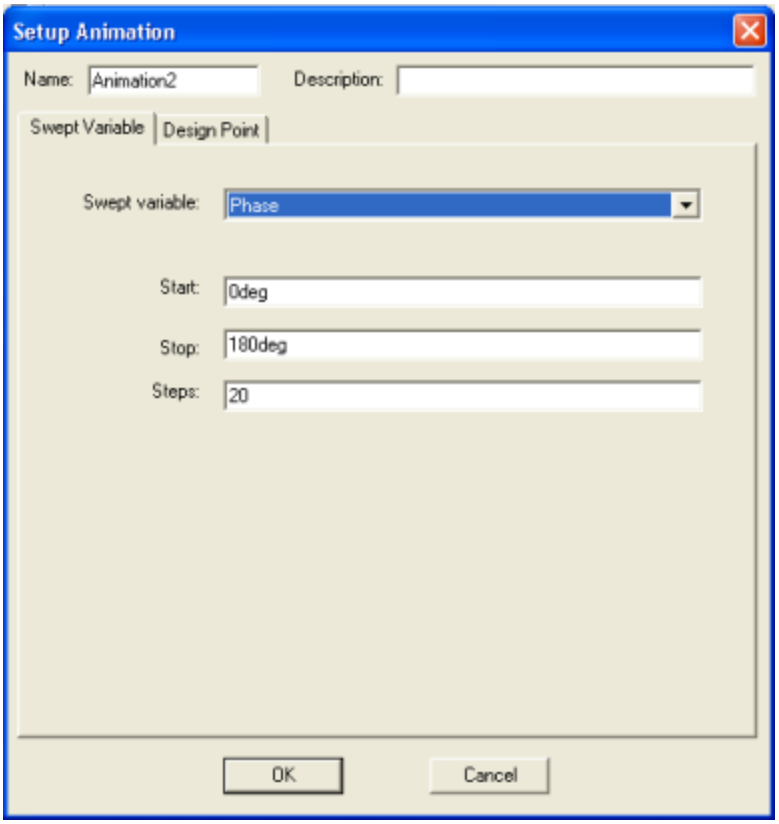
- For near field overlays, expand the **Results** icon in the Project window, right-click the overlay, and select **Properties** from the menu. Use the **Quantities** tab to select a reference frequency for the phase animation:



Click **OK** to apply the frequency and close the dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. To initiate the animation of the overlay that is currently displayed, do one of the following:
   - Select **Animate** from the **View** menu.
   - Expand the **Results** icon in the Project window, right-click the overlay entry, and select **Animate** from the menu.
3. If no animations have been defined previously, the **Setup Animation** dialog opens:



- Specify a name in the **Name** field (or accept the default, **Animation***n*, where *n* is a numeral). Optionally, enter a description.
- For a frequency animation, select **Phase** as the **Swept Variable**.
- Select the Start and Stop phases in degrees.
- Click **OK**.

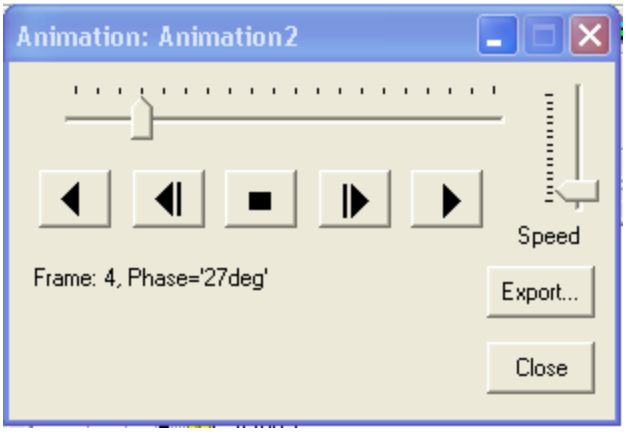
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If one or more animations have been defined, selecting **Animate** from one of the menus opens the **Select Animation** dialog:

The **Select Animation** dialog provides the following operations:

- Click to select one of the animations and click **OK** to start that animation.
- Click **New** to open the **Setup Animation** dialog described above, and close the **Select Animation** dialog.
- Select an animation and click **Delete** to delete the definition.

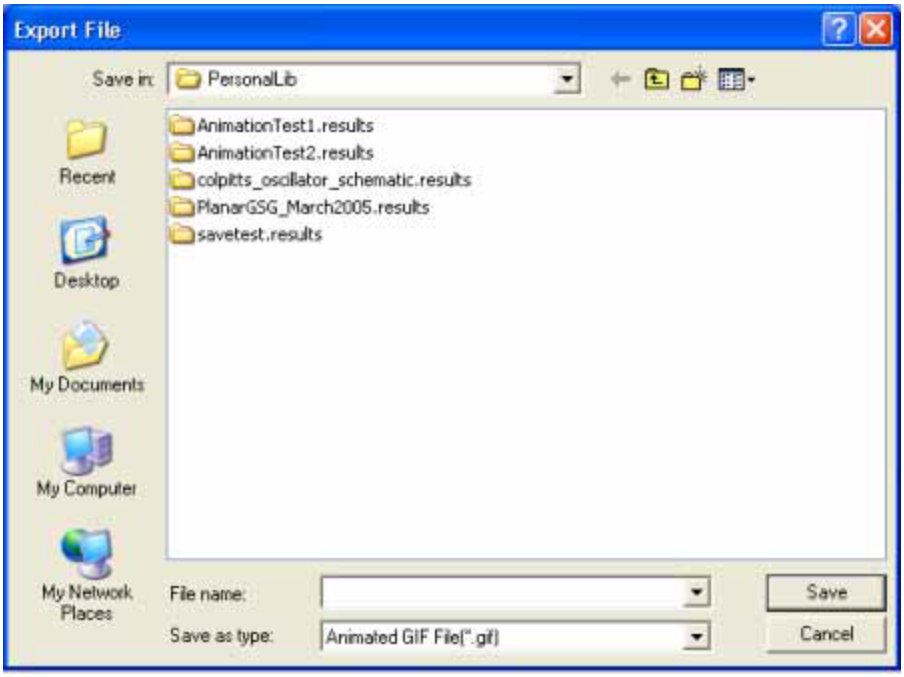
4. The frame data is calculated automatically. If the Progress window is displayed, you can monitor the progress of the calculation. When the frames have been calculated, the animation begins and the **Animation** control panel opens:



- Use the “VCR” buttons to play the animation. From left to right, the buttons are **Reverse**, **FastReverse**, **Stop**, **Fast Forward**, and **Forward**. The indicator at the top of the dialog shows the progress of the animation. Use the **Speed** slider to control the speed of the animation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To export the frame data to a file, click **Export**. The **Export File** dialog opens:



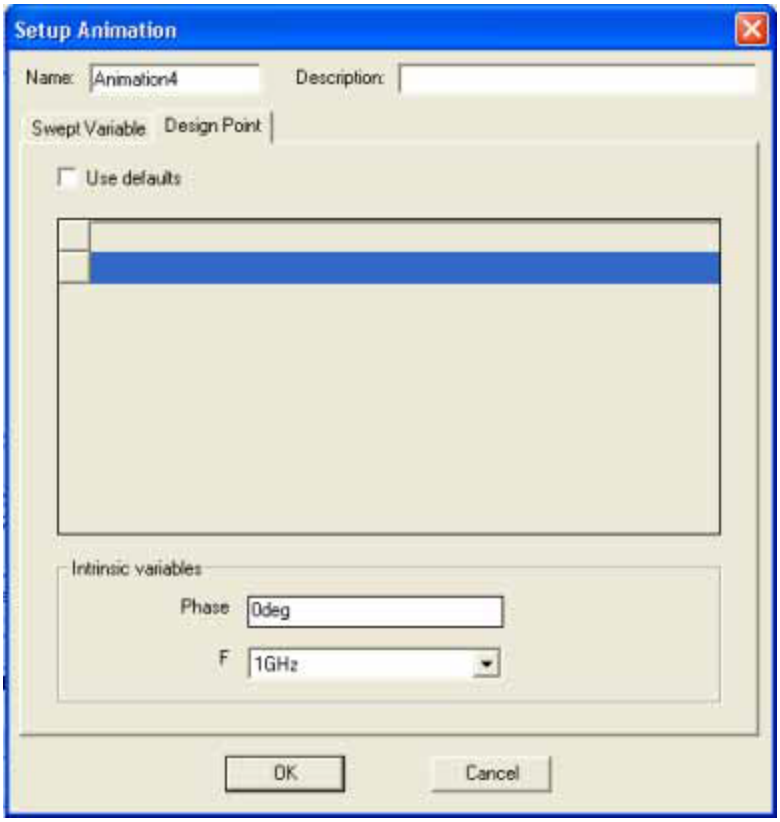
Specify the directory and file name. Use the **Save as type** menu to select the file format (Animated GIF or AVI). Click **Save** to save the data and close the dialog.

- Click **Close** on the **Animation** control panel to stop the animation and close the panel.

Changing the Design Point

Calculating frames for an animation is equivalent to re-simulating the planar design. You can specify design point parameters for the animation calculations that are different from the ones used in the original simulation. On the **Animation Setup** dialog box, select the **Design Point** tab and deselect the **Use defaults** option. The following fields are displayed:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Make any desired changes, and then click **OK** to apply the changes and close the dialog box. Clicking **Cancel** closes the dialog without making any changes.

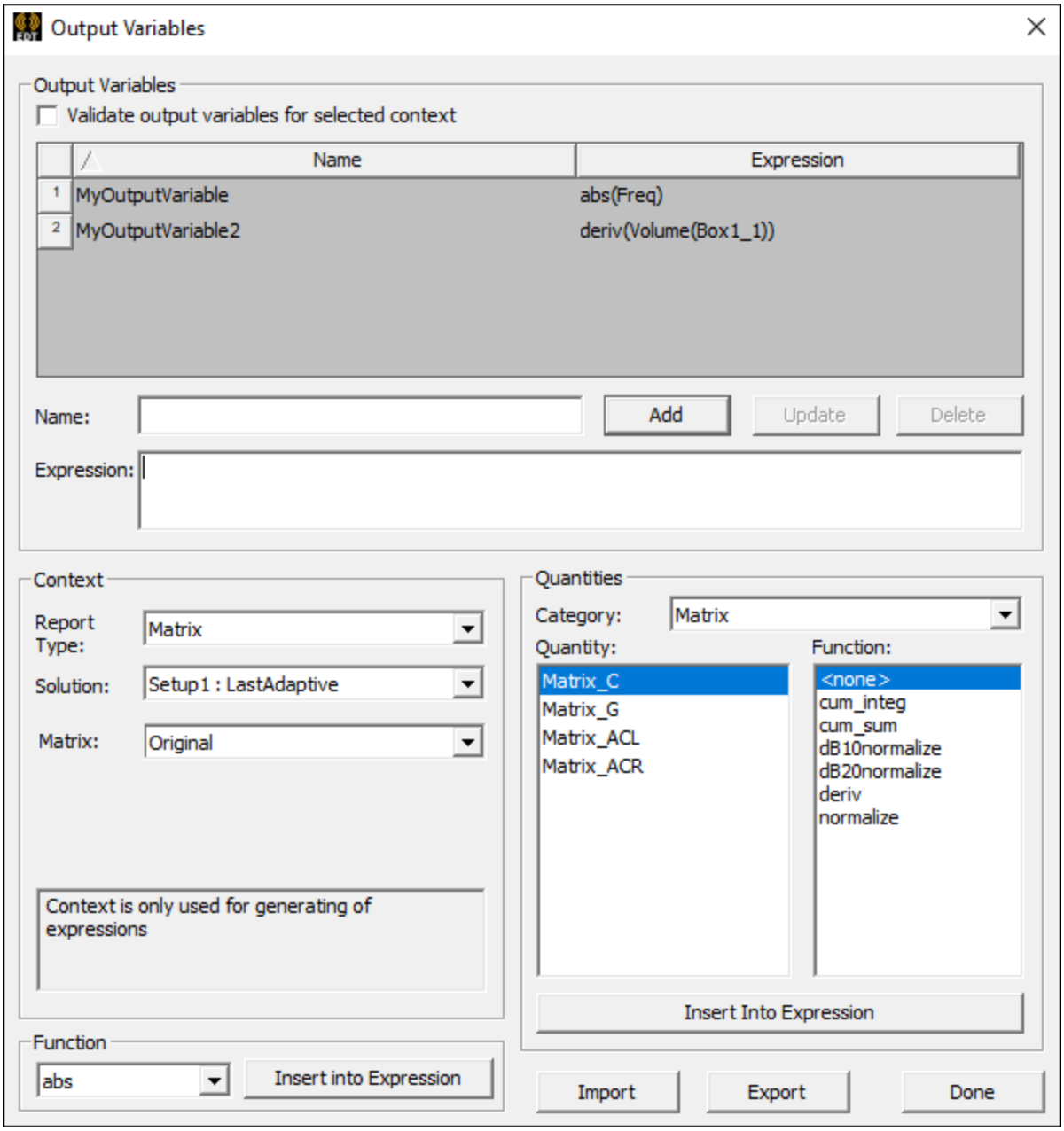
Specifying Output Variables

You can define output variables to use calculated expressions of results as adaptive convergence goals and for reports. You can use output variables in the Expression Cache tab of the **Solution Setup** dialog box and select them as Categories in the **Reports** dialog box, as well as the **Output Variables** window.

Access the **Output Variables** window one of several ways:

- Click **HFSS** > **Results** > **Output Variables**.
- In the Project Manager, right-click **Results** and select **Output Variables** from the shortcut menu.
- In the **Solution Setup** dialog box, select the **Expression Cache** tab, click **Add...** to display the **Add to expression cache** dialog box, and click the **Output Variables** button.
- Click the **Output Variables** button in the **Report** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Output Variables** window contains four sections:

- **Output Variables** – contains a list of existing output variables, and allows you to specify the name and expression for a new output variable. The **Validate output variable...** check box allows you to validate an expression before adding it. Valid expressions appear in blue and invalid expressions appear in red.
- **Context** – specifies the Report Type and Solution. Depending on the type of report selected, additional fields may appear. All selections affect the functions and quantities

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

listed.

> **Note:**
>
> The context is used only for generating expressions.

- **Quantities** – allows you to insert quantities into the Expression area of the **Output Variables** section.
- **Function** – allows you to insert completed expressions into the Expression area of the **Output Variables** section.

To add an output variable:

1. Enter a **Name** for the variable.
2. Specify the **Context**. This impacts the available quantities and functions.
3. Specify the **Quantities** (required) and **Functions** (optional) using the **Insert Into Expression** buttons.
4. Review the **Expression**. When it is ready, click **Add**.

   The new output variable appears in the **Output Variables** list.

To delete an output variable:

1. Remove all references to the output variable from the project.
2. Save the project to erase the command history.
3. Open the **Output Variables** window using one of the methods above.
4. Select the variable from the **Output Variables** list and click **Delete**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The evaluated value of an expression is always interpreted in SI units. However, when an angle quantity is plotted in a report, you have the option to plot values in units other than SI. For example, if you want to plot the polar angle of a complex simulation result S_{11}, you can choose between ang_deg(S_{11}) and ang_rad(S_{11}). Both of these return the exact same angle quantity, but in degree and radian units respectively.

When using non-SI units in expressions, surprising outcomes might result. For example, the expression 1+ang_deg(S_{11}) represents an angle and the number 1 is treated as 1 rad. The angle SI unit is attached to any unitless number that is added/subtracted from an angle value. If you want to treat 1 as degrees, make it explicit and use 1deg + ang_deg(S_{11}) instead.

If you are interested in unitless degree values, two additional functions exist: ang_deg_val(S_{11}) and cang_deg_val(S_{11}). These return simple numbers and are treated as such by any expression. For example, if the complex S_{11} lies on the positive Y axis, ang_deg_val(S_{11}) would be 90 and 1 + ang_deg_val(S_{11}) would be 91.

Function List for Output Variables

The **Output Variables** window includes a second function list containing functions to enter directly into the Expression field. These functions can also be applied to previously specified Quantities and Functions.

Some of these functions can operate along an entire curve. These are: min, max, integ, avg, rms, pk2pk, cang_deg and cang_rad.

You can select from the functions in the **Output Variables** dialog box's **Function** list or type them directly into the Expression field, if necessary.

Scaling a Source's Magnitude and Phase

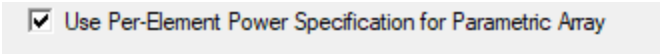
You can scale the magnitude and set the phase of ports, voltage and current sources, Eigenmodes, Characteristic Modes, and incident waves in the **Edit post process sources** dialog box. For HFSS modal projects you specify the total voltages. For HFSS terminal projects, you can also select either Incident Voltage or Total Voltage. The incident voltage selection includes magnitude, phase, and the impedance of a hypothetical external line. It is the user specified voltage at an input. The Total Voltage is incident voltage plus reflected voltage, that is, $V^T = V^i + V^r$. For HFSS Transient Network projects, the **Edit Sources** dialog includes tabs that let you specify scales for Spectral and Transient data separately. For HFSS Transient, **Edit Sources** is disabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For Driven Modal and Terminal Analysis solution types that involve near or far fields, a **Source Contexts** tab appears on the **Edit post process sources** dialog box. This provides a means to select the sources to use as context when creating a radiated field report. The tab lists the same sources as the Spectral Fields tab. Sources are scaled by exciting a single source (the selected context in the reporter dialog) while turning off all other sources.

For designs containing finite arrays, you have the option to create a Source group. The new Source Group will allow the user to activate a subset of the virtual excitations (e.g. array [5,5]port1:2 …) instead of the physical excitations associated with the unit cell(s).

You may select **Use Per-Element Power Specification for Parametric Array** in **Edit Sources** when only a Parametric Array exists in the scene.



When you select the option, **Edit Sources** defines the input power of an element before applying the Aperture Taper configured in Array Weightings. The Parametric Array with this option gives results similar to HFSS post-processing feature of Antenna Array, and couplings analysis and Enhanced Radiated Power Accuracy are not supported.

You can also **Save to File** and **Load from File**, where the file is comma delimited data (.csv) format or tab delimited data (.txt). This can help in defining source values for projects with a large number of sources. An example of the .csv format for HFSS follows:

```
Name,Magnitude,Phase
Port1:1,1W,0deg
Port2:1,0W,0deg
Port3:1,0W,0deg
```

An example of the tab delimited format follows:

Name Magnitude Phase

Port1:1 1W 0deg

Port2:1 0W 0deg

Port3:1 0W 0deg

Note: A Beta feature for HFSS, Hfss Interactive Dialogs, allows you run the **Edit Sources** dialog and the Antenna Parameters dialog in modeless fashion, that is, if you **Edit Sources** for magnitude or renormalization, affected Antenna Parameters update. Edits for sources that cause an Update of grids include:

- Source is created. A new row will be added to grids.
- Sources are deleted. Corresponding sources in grids will be deleted.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Source name is changed.Name updates in grids.

All changes will be saved in the dialog after the **Edit Sources** Sheet is closed, but they will not be applied until you click **Apply** or **OK**. The dialog supports undo and redo for creating, deleting and renaming elements.

If you edit excitations or ports, or create source groups in the Far Field Setup, the **Edit Source** dialog will close.

Scaling Sources for HFSS

Scaling Sources for HFSS Transient Network

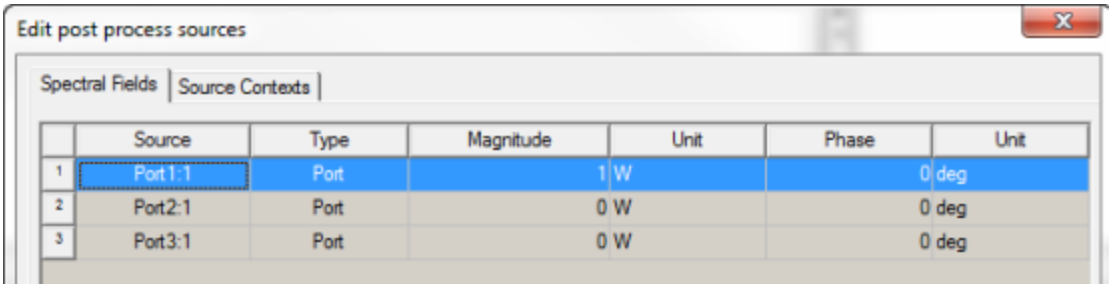
Specifying Source Contexts for Creating Radiated Field Reports

Defining Antenna Arrays

Scaling Sources for HFSS

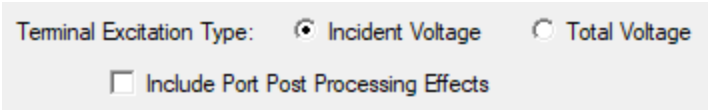
1. Click **HFSS** >**Fields**>**Edit Sources**.

   The **Edit post process sources** dialog box appears For modal projects, the Spectral fields tab shows the following column headings.



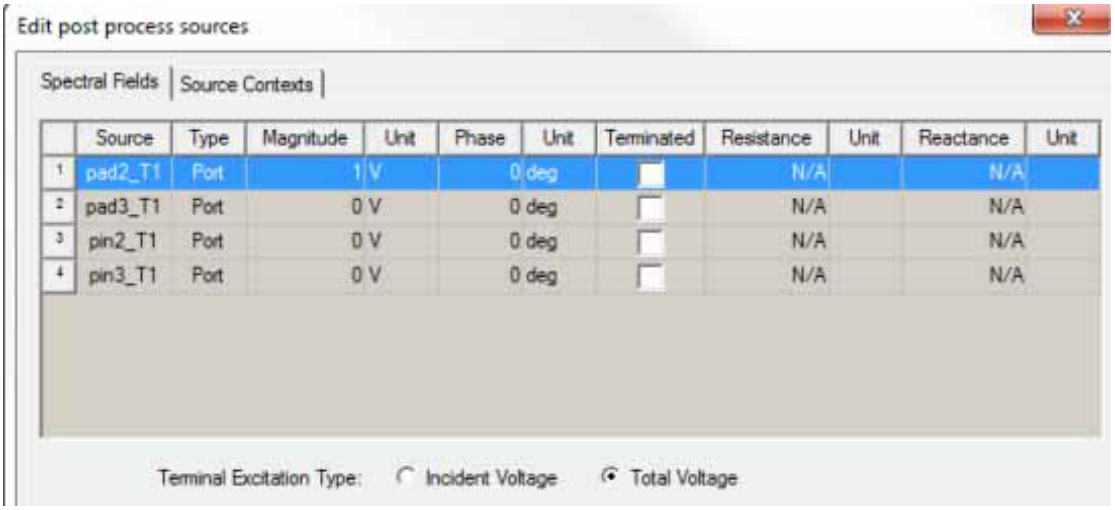
   If the project contains symmetry planes, the dialog includes a reminder that you may need to adjust the scaling factor accordingly.

   For HFSS Terminal solutions, you can select a Terminal Excitation Type as Incident Voltage or Total Voltage.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting Total Voltage adds more columns to the table.



Note that in the **modal** case a unit stimulation means 1 Watt of incident power at the port; in the **terminal** case a unit stimulation means 1 volt of total voltage at the terminal. After converting the voltage stimulation to the equivalent power stimulation the antenna results agree perfectly. In particular, the "ratioed" antenna parameters such as gain, directivity, and efficiency agree between the modal and terminal projects, while absolute antenna quantities such as incident power, accepted power may initially appear different. This is a direct result of the difference in edit-sources stimulations in the two types of projects.

You can scale the sources individually through the source table in the **Edit post processing sources** dialog box, or you can **Save to File** or **Load From File**, using the .csv format or tab delimited data (.txt) format. This feature can help for projects with many sources. The steps below describe how to directly edit the source table.

2. Select the source whose magnitude and phase you want to scale.
3. In the **Magnitude** text box, enter the magnitude you want. Design variables can be used as source scalings.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> You may not enter a negative voltage. To obtain the equivalent of a negative magnitude, add or subtract 180 degrees from the phase value.
>
> If you use a design variable as a scaling factor note that solutions are invalidated if the variable is changed.
>
> If the model contains symmetry planes, the **Edit Sources** dialog alerts you that you may need to adjust the scaling factor accordingly.
>
> At least one source should be excited (non-zero). If you set all sources to zero, you will receive a warning, but the values do go through.

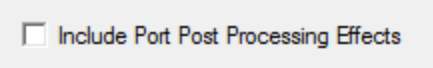
4. In the **Phase** text box, enter the new phase for the source.

The phase of the source is changed by the value that you enter.

5. Optionally, if your solution type is driven terminal, you may specify a complex reference impedance:
   a. For the selected terminal, select **Terminated**.

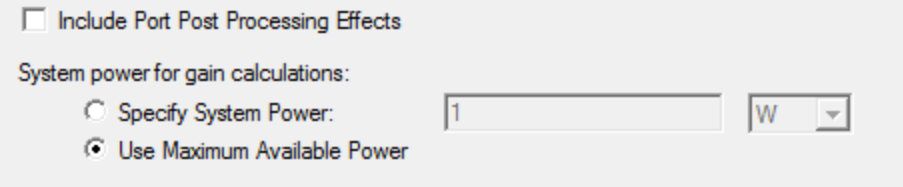
   This disables the values to the left of the check box, and enables the Resistance and Reactance text boxes. Use the scroll bar to view them.

   b. Enter the **Resistance** and the **Reactance** and select the units. Ohms is the default.
6. By option, you can click a check box to **Include Post Processing Effects**. Note that the post processing effect has no impact on the impedance of a terminated port.



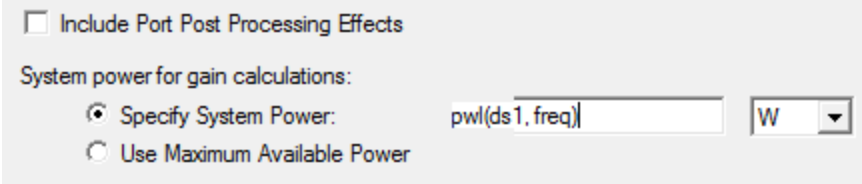
Checking this box also enables an **Apply** button.

7. For gain calculations, you can check whether to Specify System Power or Use Maximum Available Power. Selecting **Specify System Power** enables a field for value for power and select units. **Use Maximum Available Power** provides a default. The system power typically comes from an external circuit (e.g., a Circuit project), and is used in computing System Realized Gain in antenna parameters and System Gain in far field reports.

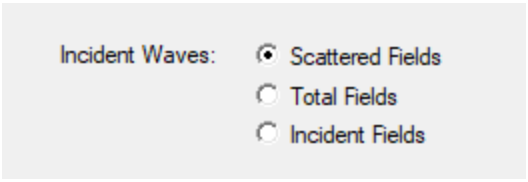


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can specify incident power for gain calculations as a function of frequency, for example, pwl(ds1,freq).



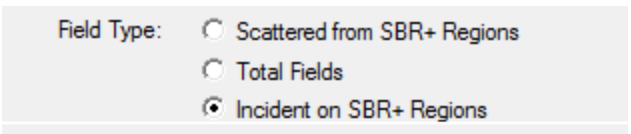
8. For HFSS, if an incident wave is present, use the radio buttons at the bottom of the panel to select one of the following field types to use:



Scattered Fields	The differential field formed by subtracting the incident field from the total field.
Total Fields	The physically measurable field that exists with the model present and a non-zero incident field.
Incident Fields	The plane-wave field that would exist in the absence of the model.

For the total field of a Floquet port *also chosen* as the source in **Edit post process sources**, you should always select **Scattering Fields** as the 2D surface fields, rather than the Total Fields. The Floquet ports are added in the far field calculation when the Solution setup **Save radiated fields only** option is checked. However, if you then create custom radiation surfaces, the 3D volume total fields are used in the calculations. In this specific situation, this causes inconsistent results between the default radiation surfaces the custom surfaces. Otherwise, there is no difference between scattering fields and total fields. since "total fields" = "incident fields (source fields)" + "Scattering fields".

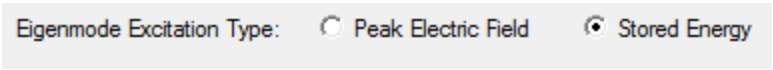
For HFSS with SBR+ designs that include ports, incident waves, or linked incident waves, the Field Type options take this form:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

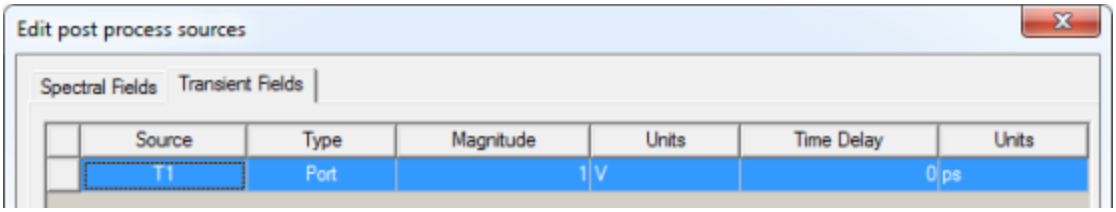
In the case of SBR+ with incident waves, the field type can only be Scattered from SBR+ Regions. For SBR+ designs with Port sources or linked incident waves, you can select from all three types.

For Eigenmode solutions, the **Edit post process sources dialog** displays a Spectral Fields tab and radio buttons for selecting as the Eigenmode Excitation Type, either Peak Electric Field or Stored Energy.



For Stored Energy, the table lists Magnitude and Unit as editable fields. For Peak Electric field, the editable fields are Magnitude, Unit, Phase, and Unit.

9. The Source Contexts tab lets you select the sources to use as context when you create radiated fields reports.



10. Click **OK to apply the changes and close the dialog box, or click Apply t**o view the changes without closing the dialog.

The magnitude and phase are assigned to the selected excitation.

> **Note:**
>
> When you scale an excitation, keep in mind that the original value of the excitation remains unchanged.

Guidelines for Scaling a Source's Magnitude and Phase

When specifying the magnitude of a source keep the following guidelines in mind:

For ports, driven modal case	• The excitation's magnitude specifies time-averaged incident power in watts. • If you are using a symmetry plane, remember to scale the input signal appropriately. For example, if you have one symmetry plane, use an input value of 0.5 watts to excite the full structure with 1 watt; if you have two symmetry planes, use an input value of 0.25 watts to excite the full

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	structure with 1 watt, and so forth. • Generally, use the default value of **1**. This specifies that the solution's E- and H-fields be scaled such that the excitation wave delivers 1 watt of power. To view the solution at some other power, enter a positive value. • Only port-mode combinations with non-zero magnitudes will be used.
For voltage and current sources	• The source magnitude for voltage and current sources specifies peak value volts and peak value amperes, respectively. • If you have defined multiple voltage and current sources, you can "remove" them by setting their magnitudes to **0**. This enables you to easily observe the effects that individual or specific groups of sources have on the problem.
For incident waves	• Source magnitude specifies peak value E-field in volts per meter. • When you scale the incident E-field, the scattered E-field and the total E-field are scaled as well. • This scaling factor affects all incident angles in the incident wave setup.
For ports, driven terminal case	• The excitation's magnitude specifies peak value volts. This is the sum of the incident and reflected waves at this terminal. See the equations here.
For Eigenmodes	• Source magnitude is unitless and represents a relative value. • When you enter a scaling factor for an eigenmode the relative source magnitude is amplified by this value. Exactly one eigenmode must be excited by setting its scaling factor to a non-zero positive number.
Ports of Transient Network solutions	• See Ports and Edit Sources Behavior for Transient Network

When specifying the new phase for ports, generally use zero. This zero-phase solution results from excitations phased in such a way that, at $\omega t = 0$, peak values occur at the port faces.

Scaling Sources and Setting Delays for HFSS Transient Network

For HFSS Transient Network solutions you can separately edit sources for Spectral Fields and Transient Fields. The **Transient** tab on **Edit Sources** controls transient far fields and fields saved on the mesh. The **Spectral Fields** tab on **Edit Sources** controls Frequency domain far fields and antenna parameters.

- Editing Sources for Spectral Fields for Transient Network
- Editing Sources for Transient Fields for Transient Network

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When non-port sources exist, the port post processing effects are off. For both Spectral and Transient fields, you can scale the sources individually through the source table in the **Edit Sources** dialog box, or you can **Save to File** or **Load From File**, using the .csv format or tab delimited data (.txt) format. This feature can help for projects with many sources. The linked steps describe how to directly edit the source table.

In Transient Network analysis each source is solved separately which allows arbitrary superposition of the results from all sources controlled by **Edit Sources** settings. This feature enables users great flexibility in studying a large set of excitation variations in an efficient and convenient manner.

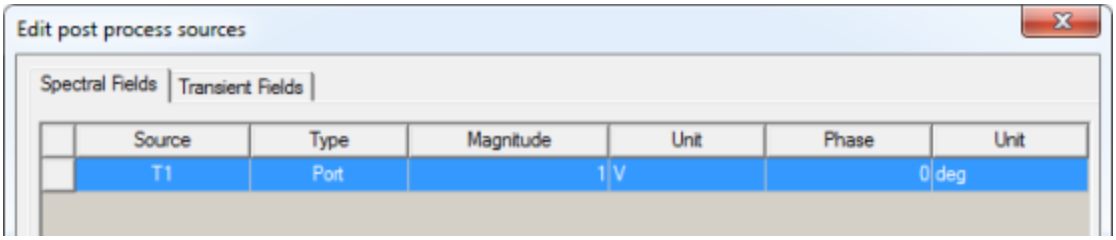
> **Note:**
>
> When superposing results it is assumed that the fields have decayed to zero at the end of the simulation for each source. In practice, the user settings for Target Residual and time duration can cause this assumption to be violated. In this case the displayed field quantities may contain significant error.
>
> For example, consider a design with two sources where the user has specified a large Target Residual setting, and one source converges at 1ns while the other converges at 2ns. If the user had not specified any time shifts in **Edit Sources**, the displayed field quantities will not be reliable in the range 1ns to 2ns because the fields from the first source cannot be assumed to be zero.

Editing Sources for Spectral Fields for Transient Network

For Spectral fields, you can optionally include port post-processing effects. The **Spectral Fields** tab on **Edit post process sources** controls Frequency domain far fields and antenna parameters. The voltage specifications are only for the incident voltage.

1. Click **HFSS**>**Fields**>**Edit Sources**.

   The **Edit post process sources** dialog box appears with the **Spectral Fields** tab selected.



   For HFSS Transient Network, the **Edit post process sources** dialog displays information for sources for Spectral Fields in table format. You can edit the Magnitude and Phase values including the use of variables, and set the units for each. You can scale the sources

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

individually through the source table in the **Edit Sources** dialog box, or you can **Save to File** or **Load From File**, using the .csv format. This feature can help for projects with many sources. The steps below describe how to directly edit the source table.

Note:

You may not enter a negative voltage. To obtain the equivalent of a negative magnitude, add or subtract 180 degrees from the phase value.

If you use a design variable as a magnitude note that solutions are invalidated if the variable is changed.

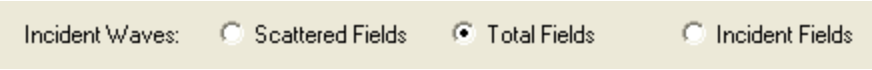
If the model contains symmetry planes, the **Edit Sources** dialog alerts you that you may need to adjust the scaling factor accordingly.

At least one source should be excited (non-zero). If you set all sources to zero, you will receive a warning, but the values do go through.

2. By option, you can click a check box to **Include Post Processing Effects**.

Checking this box also enables an **Apply** button.

3. For HFSS, if an incident wave is present, use the radio buttons at the bottom of the panel to select one of the following field types to use:



Scattered Fields	The differential field formed by subtracting the incident field from the total field.
Total Fields	The physically measurable field that exists with the model present and a non-zero incident field.
Incident Fields	The plane-wave field that would exist in the absence of the model.

Editing Sources for Transient Fields for Transient Network

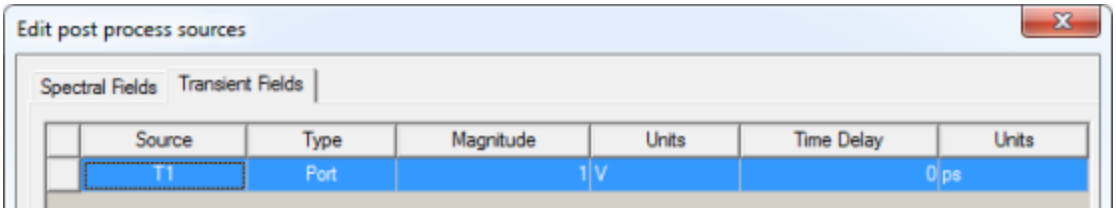
For Transient fields you can set magnitude and time delay for each source. The **Transient** tab on **Edit post process sources** controls transient far fields and fields saved on the mesh. The voltage specifications are only for the incident voltage.

1. Click **HFSS**>**Fields**>**Edit Sources**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Edit post process sources** dialog box appears with the **Spectral Fields** tab selected.

2. Select the **Transient Fields** tab.
3. Edit the Magnitude and Time Delay values as desired, and select the Units for each.



You can scale the sources individually through the source table in the **Edit Sources** dialog box, or you can **Save to File** or **Load From File**, using the .csv format. This feature can help for projects with many sources. An example of the format for HFSS Transient follows:

```
Name,Transient Magnitude,Delay
bw_tr_1_T1,1V,0ps
bw_tr_2_T1,0V,0ps
sb_tr_1_T1,0V,0ps
sb_tr_2_T1,0V,0ps
```

The magnitude and delay are assigned to the selected excitation.

> **Note:**
>
> When you change the Magnitude of an excitation, keep in mind that the original value of the excitation remains unchanged.

Ports and Edit Sources Behavior for Transient Network

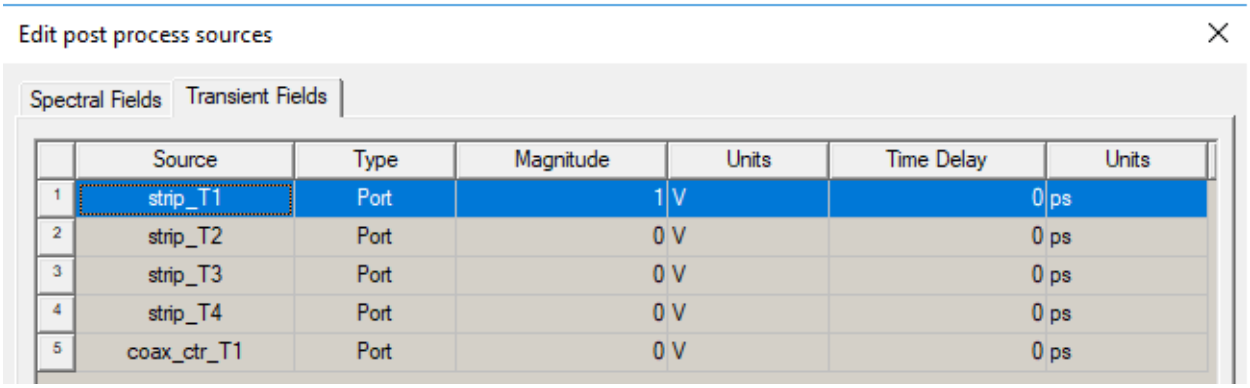
Transient Network solutions support the following port features:

- Just as in Driven Terminal we have two types of ports namely wave ports and lumped ports.
- Both implicit and hybrid transient solvers can solve dispersive and/or inhomogeneous wave ports, as well as single TEM terminal ports such as coax and gap ports. The S-parameter results from implicit transient designs will agree with mesh-linked frequency driven terminal designs. Dispersive means the wave-port modes are frequency dependent.
- Wave ports are matched loaded.
- Lumped ports are loaded with the user defined "Full Port Impedance".

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Both ports support renormalization of the spectral quantities S-parameters and far fields including antenna parameters.
- Neither port supports renormalization of transient quantities.

Edit Sources for Transient Network involves specifying the incident voltages at the terminals of the ports not the total voltages. This means that all ports are terminated with the port settings and if the edit source for a given port is set to zero it means that port is loaded with the port setting and it does not imply a short.



	Source	Type	Magnitude	Units	Time Delay	Units
1	strip_T1	Port	1	V	0	ps
2	strip_T2	Port	0	V	0	ps
3	strip_T3	Port	0	V	0	ps
4	strip_T4	Port	0	V	0	ps
5	coax_ctr_T1	Port	0	V	0	ps

As mentioned above, transient fields are not affected by any renormalization but the frequency domain far fields and antenna parameters are affected by renormalization.

In Transient Network analysis each source is solved separately which allows arbitrary superposition of the results from all sources controlled by Edit Sources settings. This feature enables users great flexibility in studying a large set of excitation variations in an efficient and convenient manner.

Note:

When superposing results it is assumed that the fields have decayed to zero at the end of the simulation for each source. In practice, the user settings for Target Residual and time duration can cause this assumption to be violated. In this case the displayed field quantities may contain significant error.

For example, consider a design with two sources where the user has specified a large Target Residual setting, and one source converges at 1ns while the other converges at 2ns. If the user had not specified any time shifts in Edit Sources, the displayed field quantities will not be reliable in the range 1ns to 2ns because the fields from the first source cannot be assumed to be zero.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specifying Source Contexts for Creating Radiated Field Reports

For Driven Modal and Terminal Analysis solution types that involve near or far fields, a **Source Contexts** tab appears on the **Edit post process sources** dialog box. This provides a means to select the sources to use as context when creating a radiated field report. The **Source Contexts** tab lists the same sources as the **Spectral Fields** tab. Sources are scaled by exciting a single source (the selected context in the reporter dialog) while turning off all other sources.

All types of sources can be enabled as Source context (ports, currents, incident waves). The default for all sources is "disabled". This applies to newly created sources and sources in legacy projects.

Each singly selected source is excited with factory default values. The phase will always be 0 deg and the magnitude will be 1 Watt for modal design and 1 Volt (Total voltage) for terminal design. These factory default values cannot be changed by users.

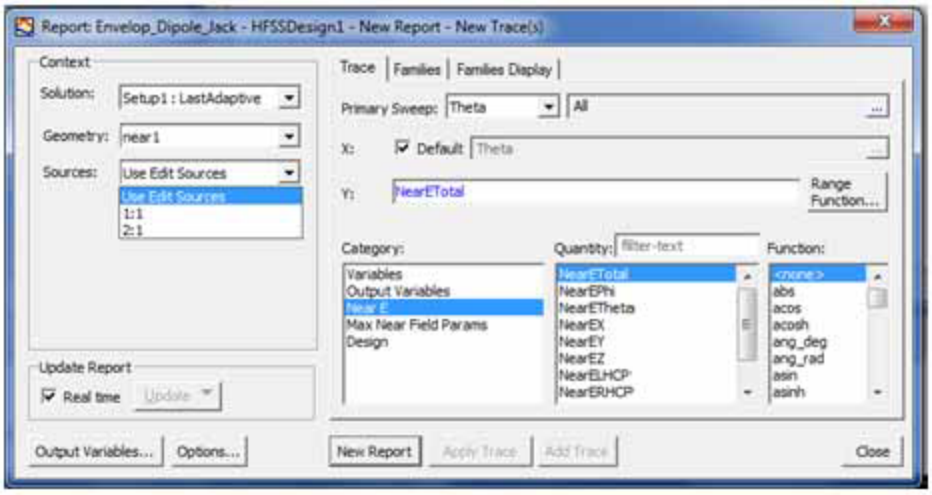
Design **Edit post process sources** settings such as "Include Port Post Processing Effects" are applicable for these singly excited sources. Thus changing any setups on the first tab of the **Edit post processing sources** dialog invalidates all traces that are defined with source context.

Enabling any source as context does not impact traces that are already defined

Disabling source as context invalidates an existing trace that uses that source as context.

This **Edit post process sources** selection applies to a Source selection in the Reporter that allows you to excite a single source.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



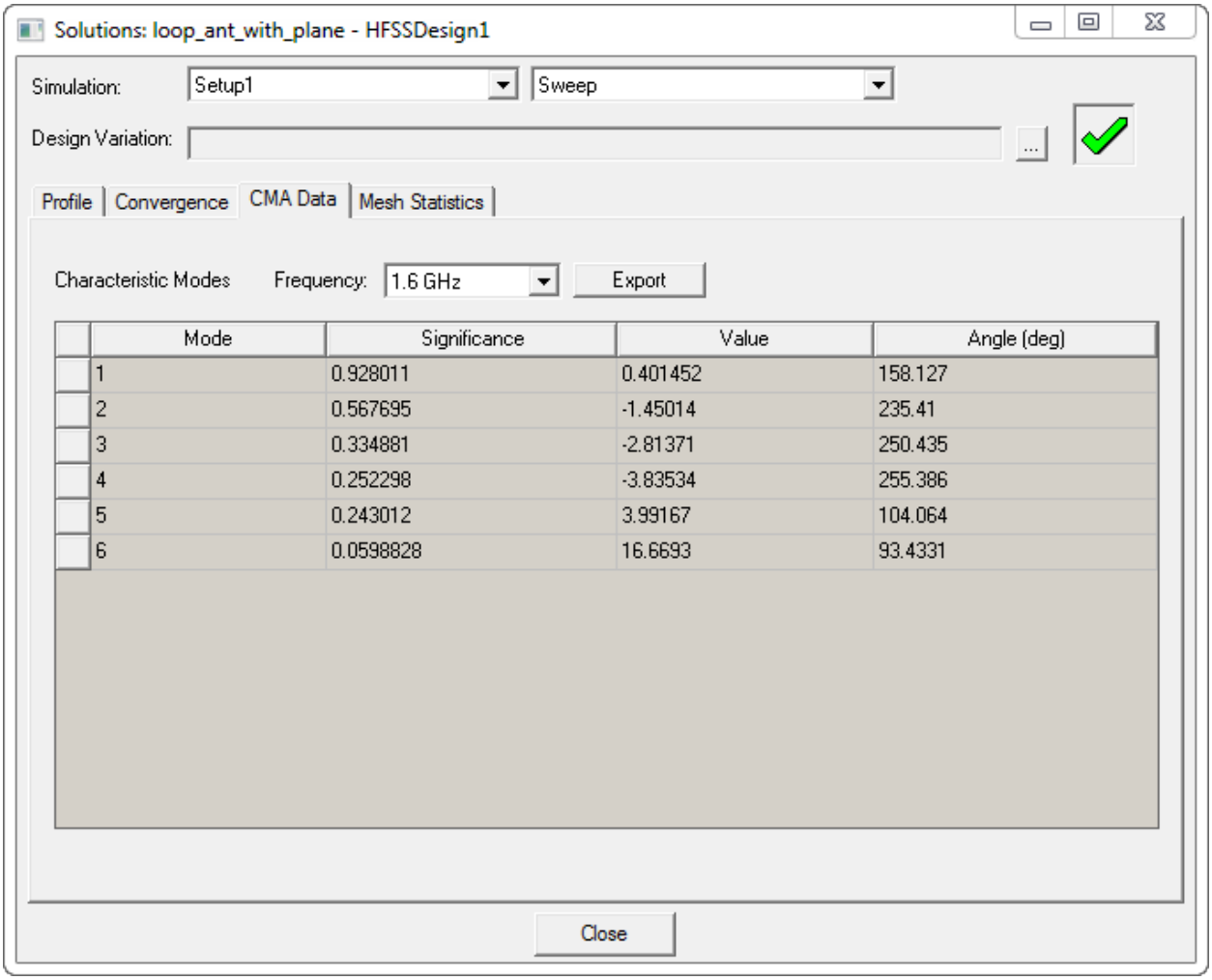
The Reporter Context selection for "Sources" appears when there is at least one source selected for "Source Contexts". The default selection of this combo box is "Use Edit Sources". When a source is selected, the **Edit Sources** setup in the design will be ignored during trace population. Instead, the trace will be calculated with the selected source excited while turning off all the other sources in the design. The above figure illustrates the extended **Create Near Field Report** dialog box. Note that "1" and "2" are port names and each has 1 mode.

For an example use of these features, see the User Defined Solution for MIMO Calculations example, that describes python scripts for the Toolkits and User Defined Solution features to generate calculations and reports.

Edit Sources Panel for CMA

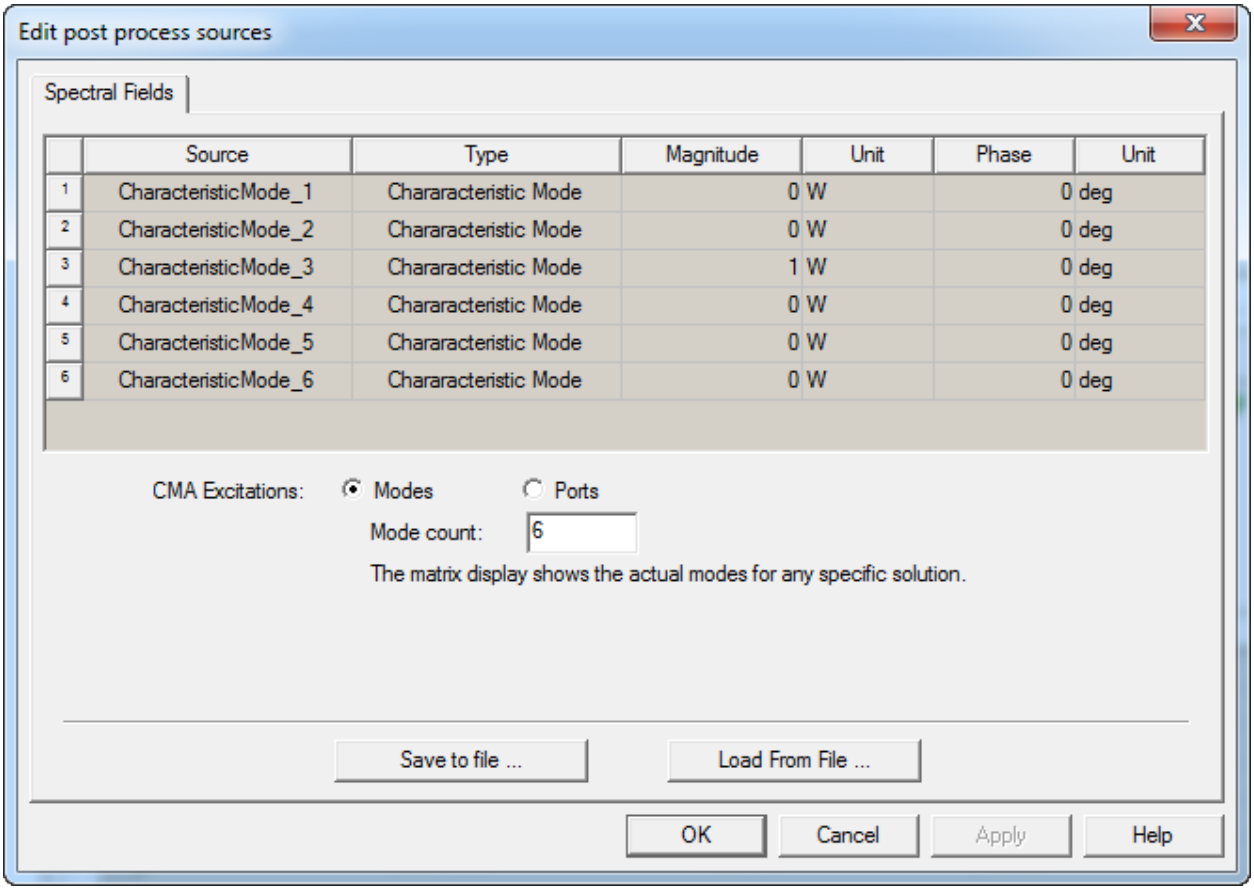
For characteristic mode analysis, the Edit Sources panel allows you to specify the modal weight for each characteristic mode used to form the total field displayed. In addition, for designs where ports are defined, you can specify the excitation values of the ports. In the case of port excitations, a linear combination of the solved characteristic modes will be determined, which matches the specified port values. The default value of Mode count in the Edit Sources panel is 10. The CMA data panel in the Solution Data dialog displays the actual modes above the minimum modal significance for any specific solution.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



	Mode	Significance	Value	Angle (deg)
	1	0.928011	0.401452	158.127
	2	0.567695	-1.45014	235.41
	3	0.334881	-2.81371	250.435
	4	0.252298	-3.83534	255.386
	5	0.243012	3.99167	104.064
	6	0.0598828	16.6693	93.4331

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Enter this value in the Mode count field. For example, if the matrix display shows 6 modes above the specified value of minimum modal significance, enter 6 in the Mode count field. Excite the magnitude and/or phase of the mode whose current pattern you wish to see in the design. For example, if you want to see surface currents due for Mode 3, define the magnitude or phase for CharacteristicMode_3 and set the other characteristic modes to 0. You can also define the magnitude and/or phase of a combination of characteristic modes.

Derivative Tuning for Reports

The Derivative Tuning feature available in the results menu is based on the derivatives that you can request for selected variables in the solution setup. It is limited to quantities like S-parameters and Far Fields, where far-field tuning supports the following:

- Incident Wave
- Linked Field
- Voltage Source
- Current Source

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

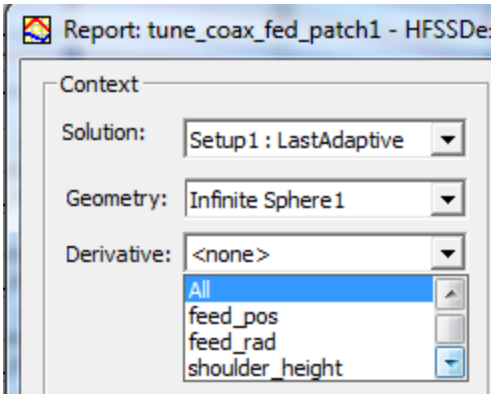
- Magnetic Bias : Note: Sensitivity analysis of a design variable associated with the magnetic bias region assumes that the magnetic bias field does not change. In some cases this assumption may not be appropriate and requires user discretion to ensure its applicability especially for non-uniform ferrite models.

Tuning far field quantities determines the derivatives of the response of electromagnetic devices with respect to variations in geometric or material properties. Designs can include port excitations or non-port excitations (that is, incident waves, linked field, voltage source and current source.

Note: Sensitivity analysis of a design variable associated with the magnetic bias region assumes that the magnetic bias field does not change. In some cases this assumption may not be appropriate and requires user discretion to ensure its applicability especially for non-uniform ferrite models.

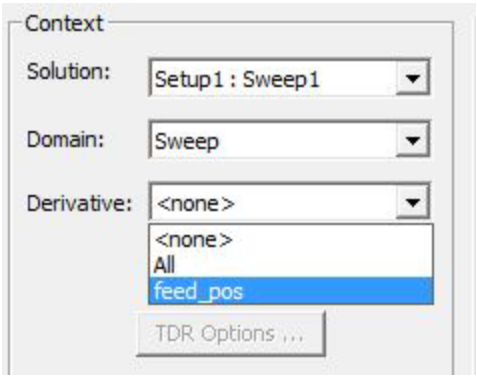
Results obtained by the “Tune Reports” functionality is only an estimate, it is recommended that designs be re-analyzed with the variables values changed to include any tuning offsets to get an accurate result.

Frequencies, and local quantities cannot be tuned. If you have defined variables in the solution setup, the Context area of the Reporter displays the Derivative field, from which you can select the variable(s) for a tunable report.

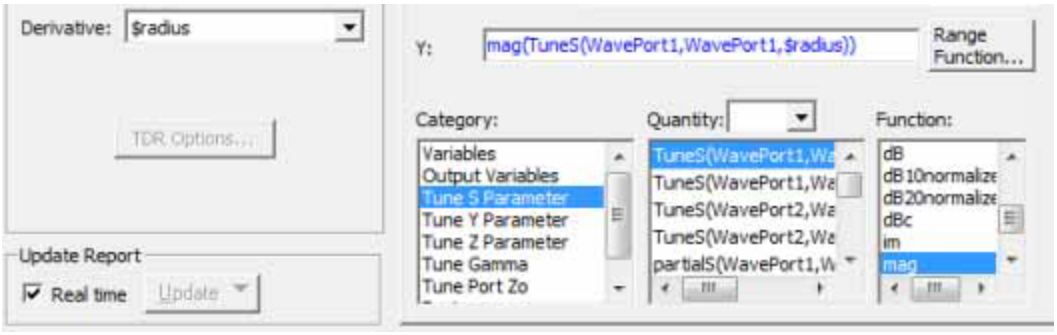


A common way to use this feature is to produce, for an output quantity of interest, two curves in one plot. One curve is produced with the **Report** dialog selection for the Context Derivative as <none>. The other has the All or specific variable selection.

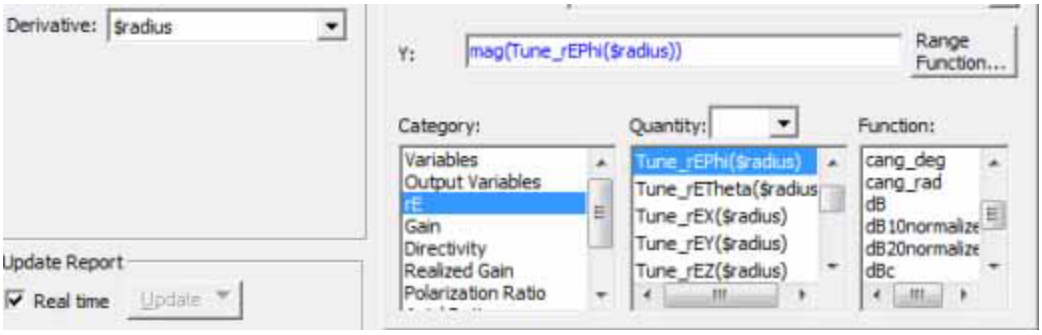
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When you select All or a variable, notice that the Context field of the Report dialog shows the names of various parameters prefixed with Tune. Some of the Quantities associated with the category also show a Tune prefix. This labeling makes it easier to distinguish the Tuned traces from the reference traces.

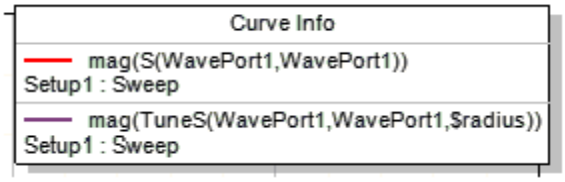


In the case of a Far Fields plot, the Tune prefix appears in only in the Quantity field.

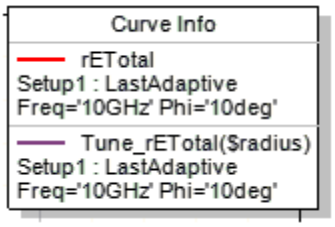


This gives you a plot with initially two identical curves, one on top of the other. The curve info table shows the trace names. This example shows the curve info for a plot with a normal S-parameter trace with a TuneS trace that can be tuned using the value of the $radius variable.

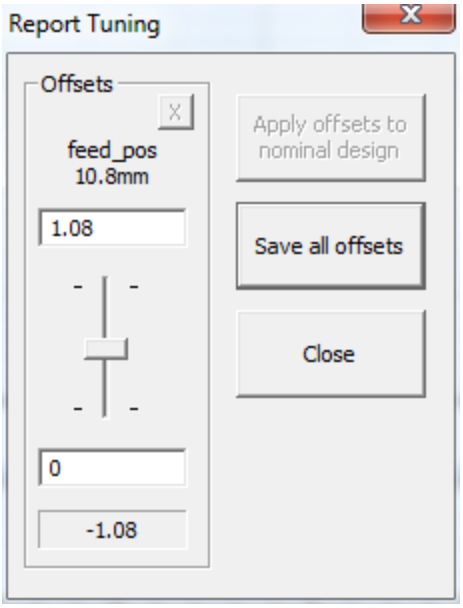
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This example shows the curve info for a plot with a normal rETotal trace with a Tune_rETotal trace that can be tuned using the value of the $radius variable.



You can then right-click Results in the Project tree and select **Tune Reports**. The **Report Tuning** window appears. You can use the slider to tune the Tune curve interactively while the reference curve stays to provide a reference. This way you can see interactively how small changes in variables affect the result. You can then apply those offsets to the original variable values and re-solve the design.



The overall procedure for Report Tuning follows.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

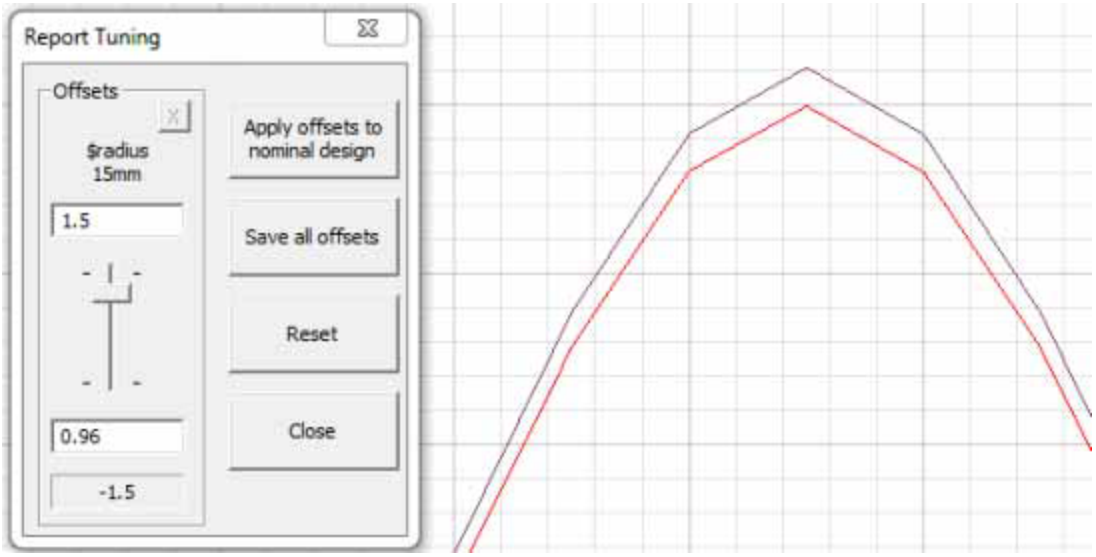
1. Generate a solution with one or more variables for which you select Use on the **Derivatives** tab of the solution setup.
2. Use the Reporter to create one curve with Derivative selection in the Context panel of the report dialog set to None.
3. Then create another curve, but in the Context pane. select for Derivatives, All or the variable of interest. Select to build your new trace from the Categories and Tune Quantities listed.

This gives you two identical curves, one top of the other. Notice that you are not limited in the number of traces that you define. You may choose to limit the number of traces for more ease in observing the results of tuning.

4. Click the **HFSS>Results>Tune Reports** or right-click **Results** in the Project tree and click **Tune Reports** from the short cut menu. The menu item is disabled if no variables have been selected in the Derivatives tab of the solution setup.

This displays a **Report Tuning** dialog which lists the variables available for tuning. The example above shows a **Report Tuning** dialog for a design with only one variable but it can show more.

5. You can use the slider to adjust the value of each available Tune variable. When you move a slider, the **Apply offsets to nominal design** button and the **Save all offsets** buttons are enabled. The Report shows the change to all Tune traces, relative to any reference traces you define.



The Tune dialog displays the change to the variable selected. For example, if the variable is $length with a value of 1mm, and the slider shows 0.1mm, then the effective value of $length for the purpose of derivatives is 1.1mm. If you exit the dialog by applying the offset or offsets

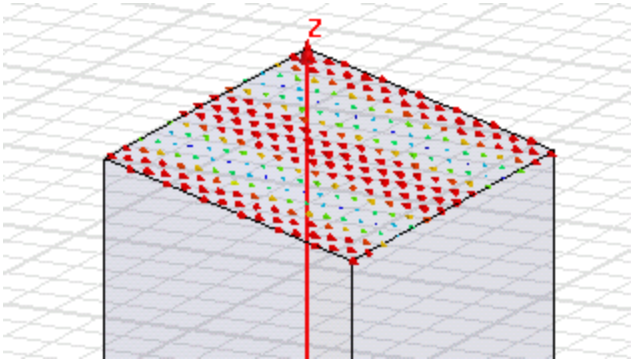
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(click **Apply offsets to nominal design**), $length is assigned a value of 1.1mm. You can then re-solve, and get results based on the derivative's prediction.

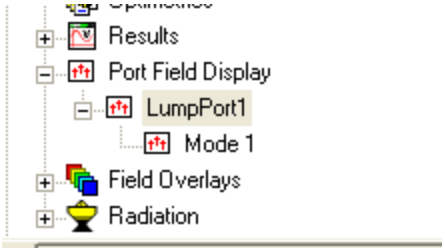
6. **Close** lets you close the dialog.

Port Field Display

The Port Field Display feature lets you see a visualization of the electric field patterns belonging to the modes of the 2D port solution. To have an available solution," Save Fields" must be checked for the Solution setup Advanced tab, or for a fast or discrete sweep. If you select a port mode listed in the **Project** tree, the **Modeler window** displays the a visualization of the selected mode for the port field.

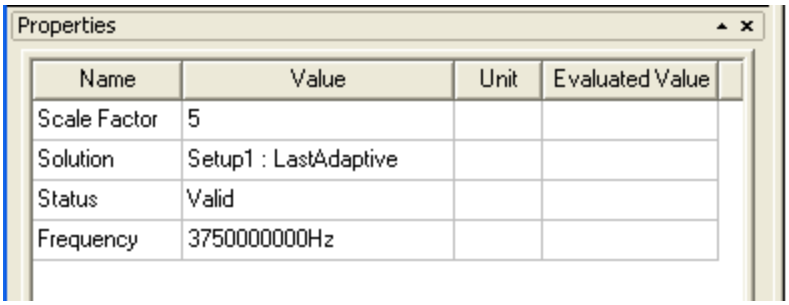


If an HFSS design has ports, they are listed Port Field Display icon in the Project tree. The modes for each port appear under the port name.



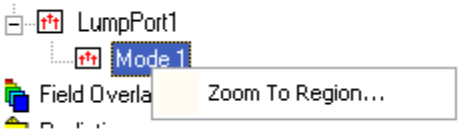
Select a port name to display a **Properties** dialog with the properties for that port, or for individual Modes listed under that port. These are used as subsequent plot defaults for the port field display. Your control of port field displays is purely through the docked **Property** window. You can edit the Scale Factor field, which governs the sizes of the arrows. If multiple solutions or frequencies are available, you can select from a drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If no 2D solutions exist for the selected port or mode, the Status line of the **Properties** dialog will say "No solutions available."

Depending on the view you begin with, and the location and size of the mode in the design, you may want to right-click the selected mode to display a **Zoom to Region** command.

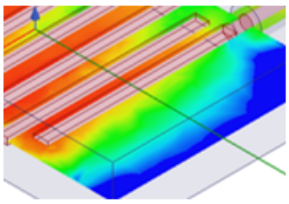


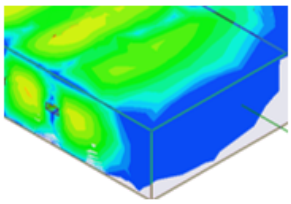
The difference between the Port Field Display and the Field Overlay is that the Port Field Display gives the pure 2D port solution. A Field Overlay provides a the 3D field solution on the port, which is the sum of the excitation in the port and the reflections by the 3D structure. The Port Field display lets you examine the field patterns with which the 3D structure is excited.

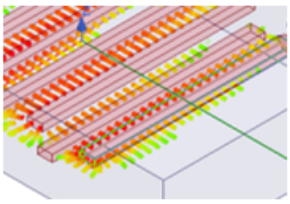
Plotting Field Overlays

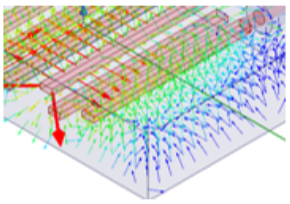
Field overlays are representations of basic or derived field quantities on surfaces or objects for the current design variation. You must have selected **Save Fields** in the Solution setup in order to enable the **Plot Fields** selections.

You must select a geometry to create a plot. Selection of line, surface, plane, arbitrary point list, or solid object, and of scalar or vector quantities, or of the streamline option affects the display. You have more control over the display via Modify Plot Attributes.


Mag E on Surface

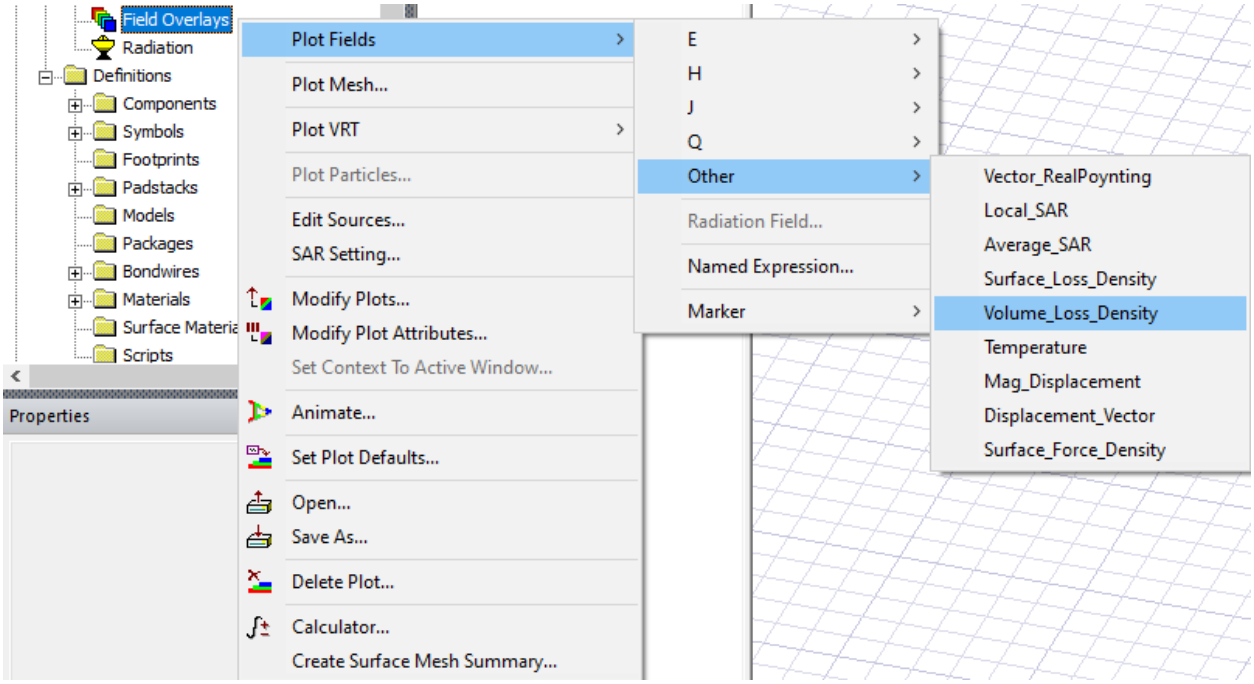

Mag E on Solid


Vector E Streamline

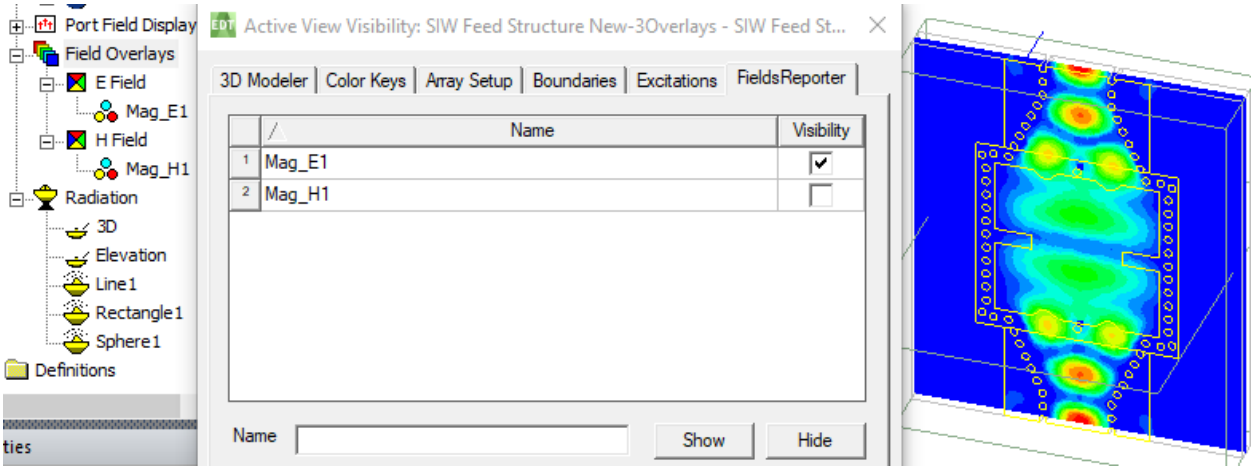

Vector E

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Plot Fields** menu selections depend on the solver selected.

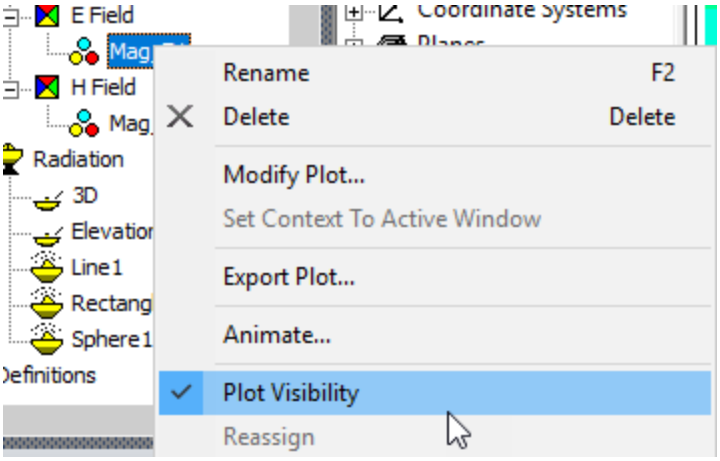


Electronics Desktop allows for the display of existing field overlays in the **3D Model** window and **Layout Editor** views. You can also create plots in Layout view for specified layers and/or nets.

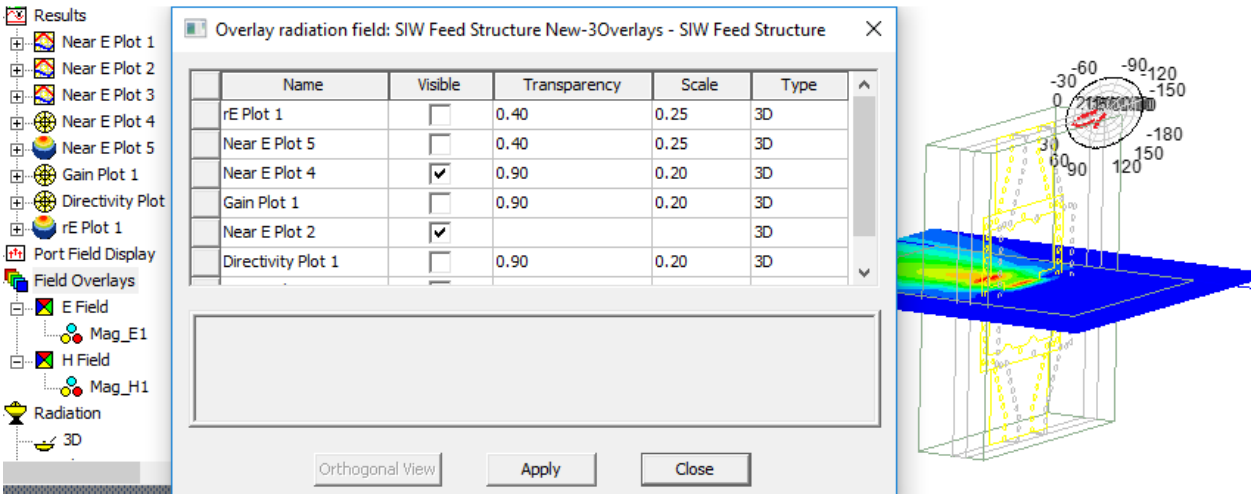


You can control the visibility of field overlays by means of the **View** > **Visibility** > **Active View Visibility** window, **Fields Reporter** tab (shown above) or by right-clicking on the Field plot in the Project Manager and checking or unchecking **Plot Visibility**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

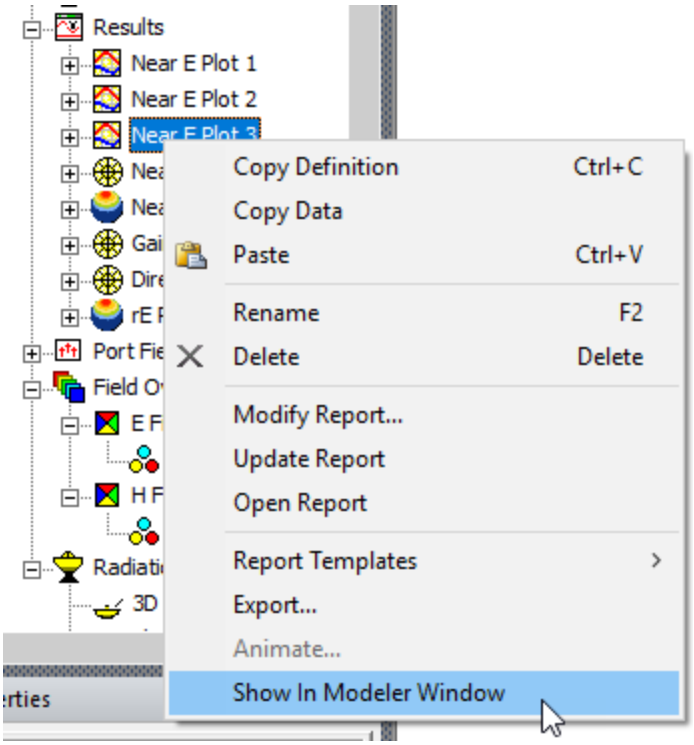


Radiation plots and antenna parameter tables available for overlay can be viewed by right-clicking **Field Overlays** in the **History Tree**, and selecting **Plot Fields** > **Radiation Field...**. This displays an *Overlay radiation field* window that lists each results plot available for overlay.



You can also control the visibility of **Results** plots available for the Modeler by right-clicking on the report and selecting **Show in Modeler Window**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can overlay existing 3D Polar Plots of near or far fields, near field contour plots on a near field rectangle, 2D Radiation Pattern Plots, and Monostatic or Bistatic RCS 2D plots in the model window using the **HFSS 3D Layout** or **HFSS** > **Fields** > **Plot Fields** > **[*Field*]** command, or by right-clicking on **Field Overlays** in the **Project Manager** and selecting **Plot Fields** > **[*Field*]**. You can also create animations of field plots.

You can overlay a 2D contour plot including Range Doppler color mapped plot to model window as a 2D overlay (which means it cannot be rotated like a geometry model). The overlay of near field on Rectangle contour plot or 2D radiation pattern plot is different. It is a 3D overlay as it can be rotated along with geometry model in 3D space.

You can also overlay tables of selected antenna parameters.

If the design includes Layout Components, the **Create Field Plot** dialog contains additional functionality for nets and layers.

To Plot Fields

To plot a basic field quantity after you have run a simulation with **Save Fields** checked on the Solution setup:

1. Select a point, line, surface, plane, cutplane, or object to create the plot on or within. Your choice affects the way the fields are plotted. If you selected an object, you must check **Plot on surface only** to enable the **Surface Smoothing** button

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

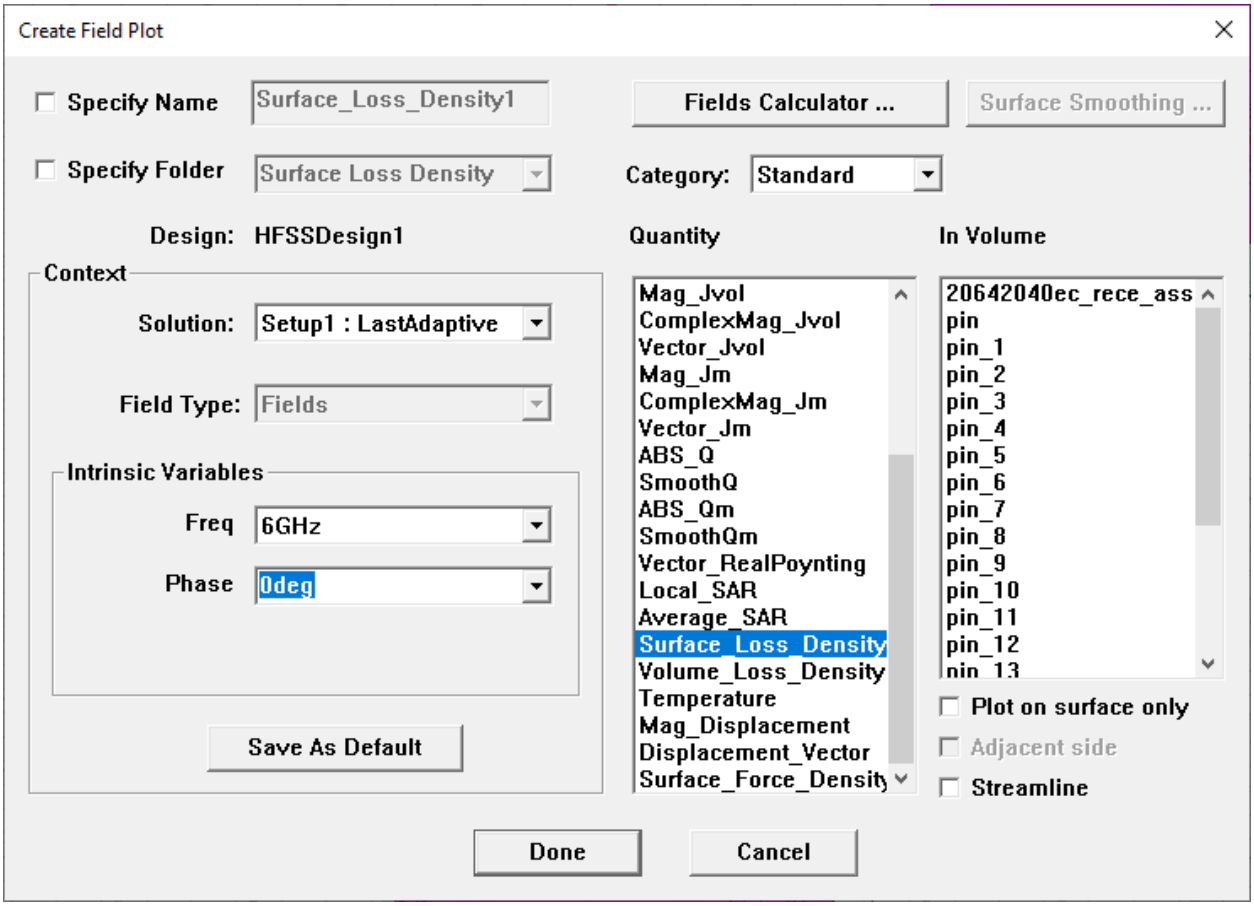
2. Click **HFSS** > **Fields** > **Plot Fields**, or right-click the Field Overlay icon in the Project Manager and select **Plot Fields** , or right-click in the modeler window, and select **Plot Fields** from the context menu.
3. In the **Plot Fields** menu, click the field quantity you want to plot.

   Available selections depend on the solved solution. For definitions of the usual quantities, see the list under Quantity command.

   If you select a scalar field quantity, a scalar surface or volume plot will be created. If you select a vector field quantity, a vector surface or volume plot will be created. If you select a vector quantity, you will be able to specify a Streamline plot. If the quantity you want to plot is not listed, see Named Expression Library.

   For projects with Temperature dependent materials, the **HFSS 3D Layout, HFSS** > **Fields** > **Plot Fields** > **Other...** menu selections include **Temperature**. For Transient projects, the menu selections show "_t" to show that they represent time dependent quantities, such as E_t, H_t, J_t, and so forth.

   After you select the field quantity to plot, the *Create Field Plot* window appears.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Specify Name** field shows a name based on the field quantity you selected, and the **Quantity** list shows the field quantity selected. If the design includes Layout Components, the **Create Field Plot** dialog contains additional functionality for nets and layers.

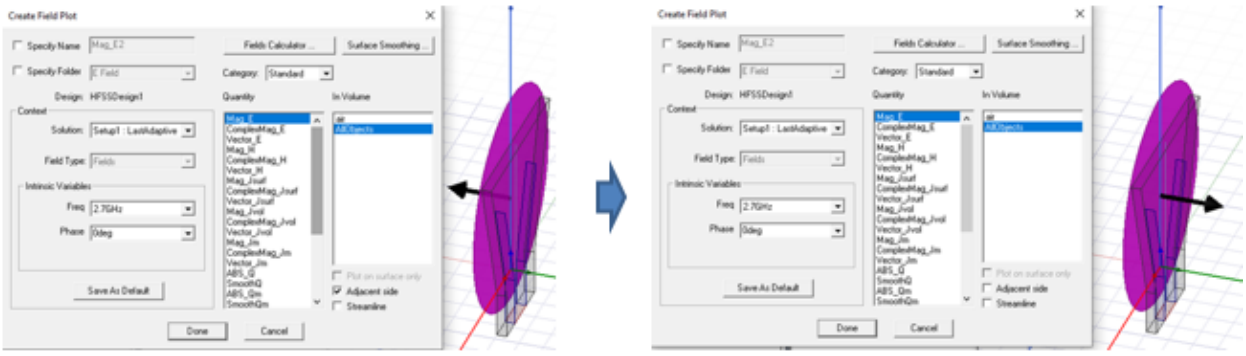
4. To specify a name for the plot other than the default, select **Specify Name**, and then type a new name in the **Name** field.
5. Select the solution to plot from the **Solution** drop-down menu. Set the design variation via the *Set Design Variation* window. Access this window from the *Solution Data* window by clicking the ellipsis button on the right of the Design Variation field, or via the **HFSS** > **Results** > **Apply Solved Variation** command.
6. To specify a folder other than the default in which to store the plot, select **Specify Folder**, and then click a folder in the **Plot Folder** drop-down menu, or type the name you wish to use. Plot folders are listed under **Field Overlays** in the Project Manager. Plot folders let you group plots with the same quantity together. All field plots under the same folder share the same color key.
7. Under **Intrinsic Variables**, select the frequency and phase angle at which the field quantity is evaluated.
8. If desired, you can select a different field quantity to plot from the **Quantity** list. For scalar quantities plotted on the surface of a geometry, the **Surface Smoothing** button is enabled. For vector quantities, the **Surface Smoothing** button is disabled. For details on its use, see Modifying Field Plots.
9. Select the volume (region) in which the field will be plotted from the **In Volume** list.

   This selection enables you to limit plots to the intersection of a volume with the selected object or objects. You can select and deselect any items in the **In Volume** list. You can mix model objects with non-model boxes. For example, you may want to see a plot from part of two model objects by restricting the region to a non-model box overlapping those parts.

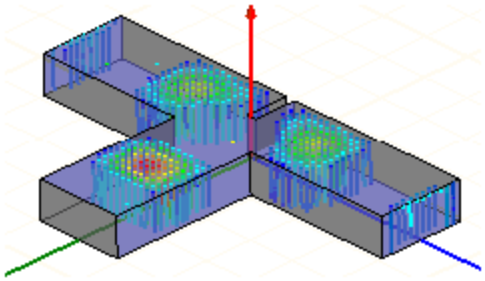
   > **Note:** Multiple selection should be used when there is a discontinuous field on a surface. If not, the field on both sides of the surface are plotted and each interferes with the other.

10. If you are creating a plot on a sheet object, the **Adjacent** checkbox allows you to create field overlays on a sheet object using data from either side of the sheet. An arrow indicates where the data is coming from. While the data is usually the same on the two sides of a sheet object, in a situation where the sheet object is assigned to be a shell element, the field may be different on its two sides. (Shell elements are an option for Layered Impedance Boundaries and Finite Conductivity Boundaries with DC conductivity and two-sided selected.) In such cases, you can visualize the field data on the side you want.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



11. If you selected a **vector** quantity, you can use the check box to select **Streamline** plot. Streamlines are often used to indicate magnetic flux lines, etc. in plots.



Notes for creating Streamline plots:

a. Before creating the plot, select the starting edge (in 2D), starting surface (in 3D), or the starting points for both 2D and 3D.

b. In the Creating Field Plot menu, select "In Volume: Region" which is the volume in which the streamlines will appear and is outside of the sources.

c. After the plot is created, on the Attributes/Plot tab you can, you can reduce "Seeds density" to show more streamlines. If no streamlines appear, reduce this by a factor of 10 (or 100) because the default seeding was too large.

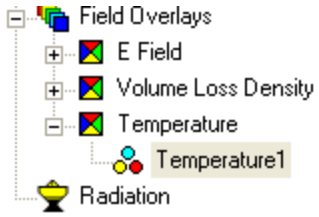
See Setting Field Plot Attributes for adjusting the streamline display and Setting Fields Reporter Options for setting Streamline defaults. For exporting Streamline plots, see Exporting Field Plots.

12. Click **Done**.

The field quantity is plotted on the surfaces or within the objects you selected. The plot uses the attributes specified in the **Plot Attributes** window.

The new plot appears in the view window. It is listed in the specified plot folder in the project tree. Each category of plot is listed separately in the Project Manager.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you have created a field plot on a simulation in progress, the field plot is updated after the last adaptive solution. If you want to update the field overlay before then, to view progress in the solution, select the Field icon in the Project tree that contains the field plot of interest, right-click to display the shortcut menu, and select **Update Plots**.

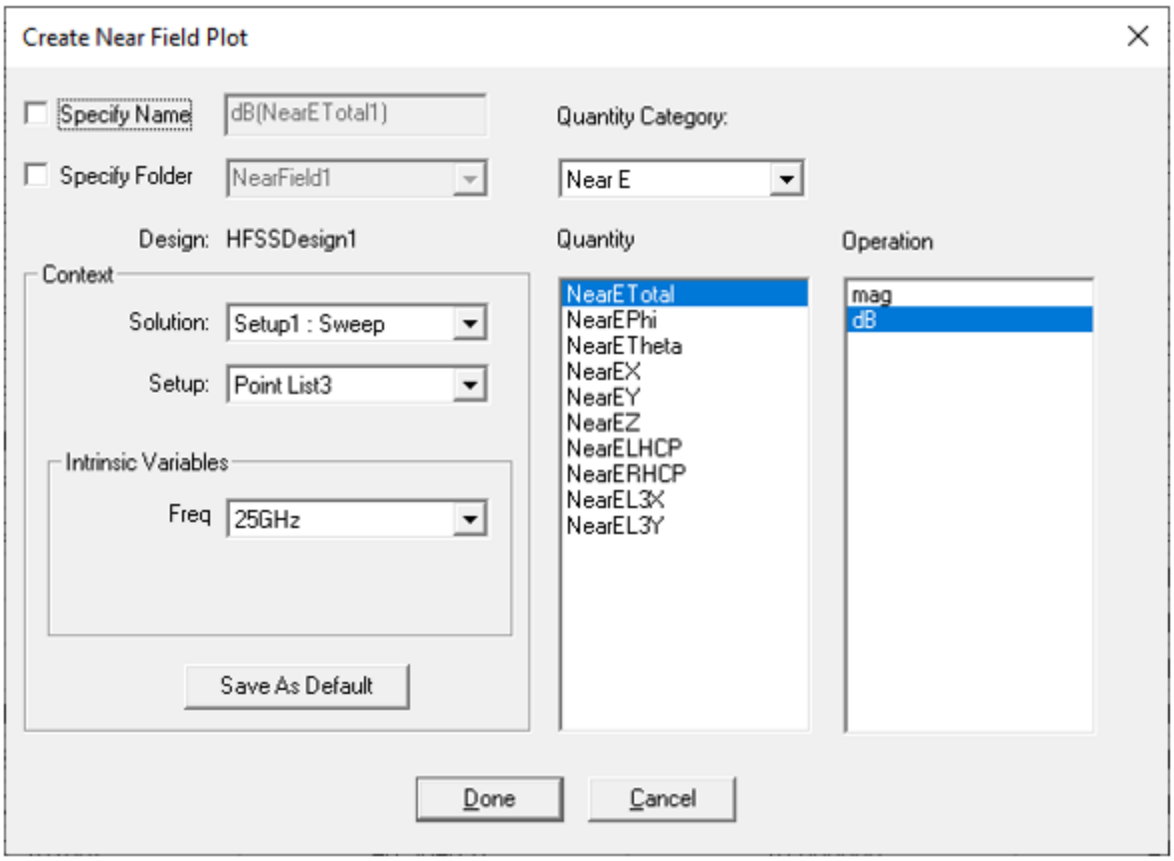
To turn off the display of the plot, right-click on the plot and select **Plot Visibility** from the shortcut menu. Deselect **Plot Visibility** to turn off the plot display.

Generating a Sampled Near Field Plot

To create a Sampled Near Field Plot, based on an arbitrary Point List:

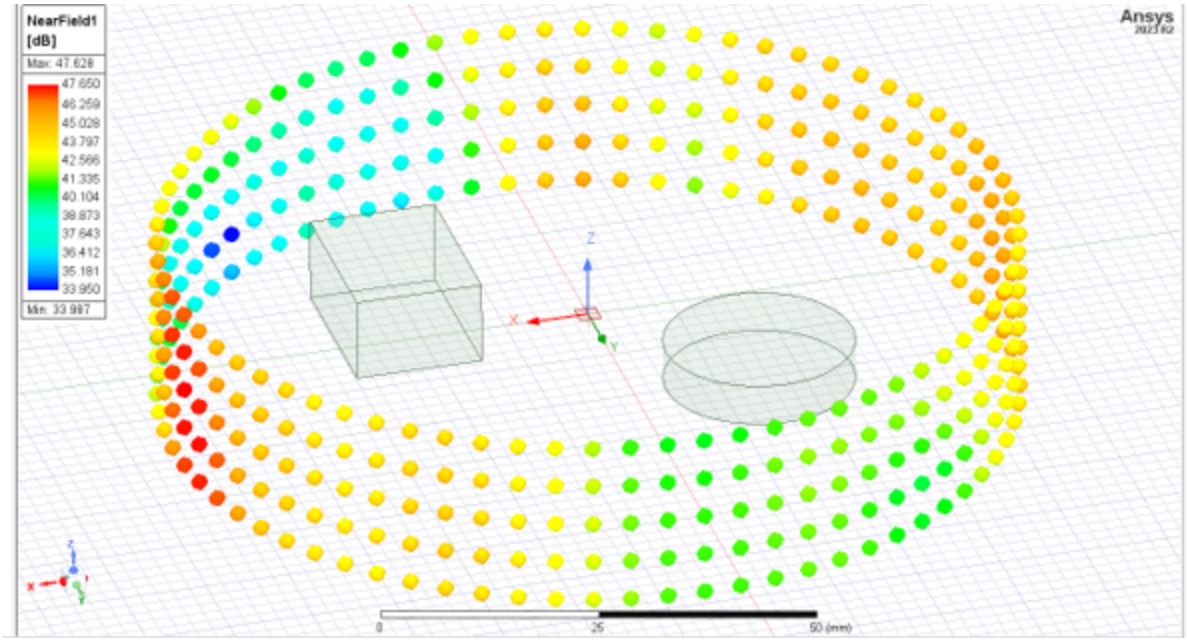
a. Right-click the Field Overlays node in the Project tree, and from the shortcut menu, select **Plot Sampled Near Field ...**. This opens the Create Near Field Plot dialog.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can use the **Create Near Field Plot** dialog to configure which solution, quantity, and so forth.

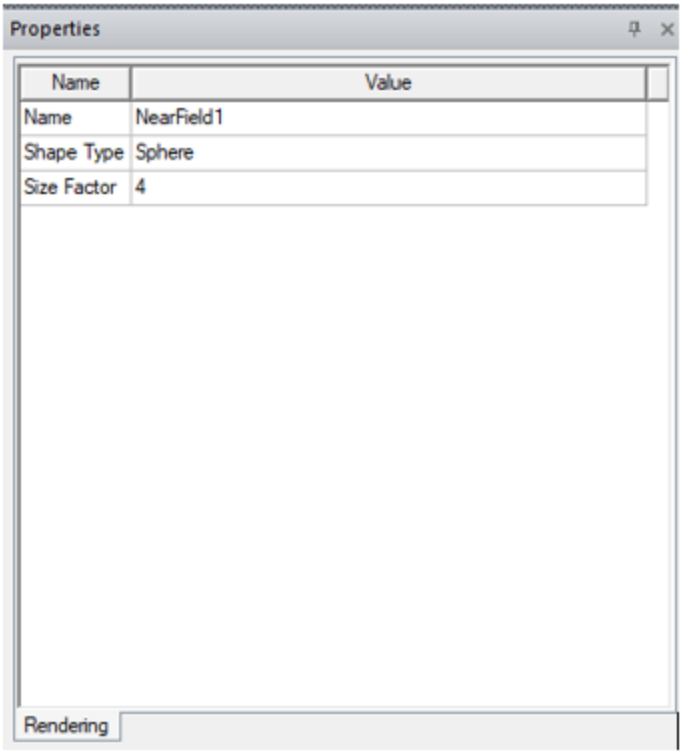
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Properties for the Plot lets you specify:

- Shape Type [Point, [Point, Sphere, Box, Dot, Plus]
- Size Factor (sets size of shape marker)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



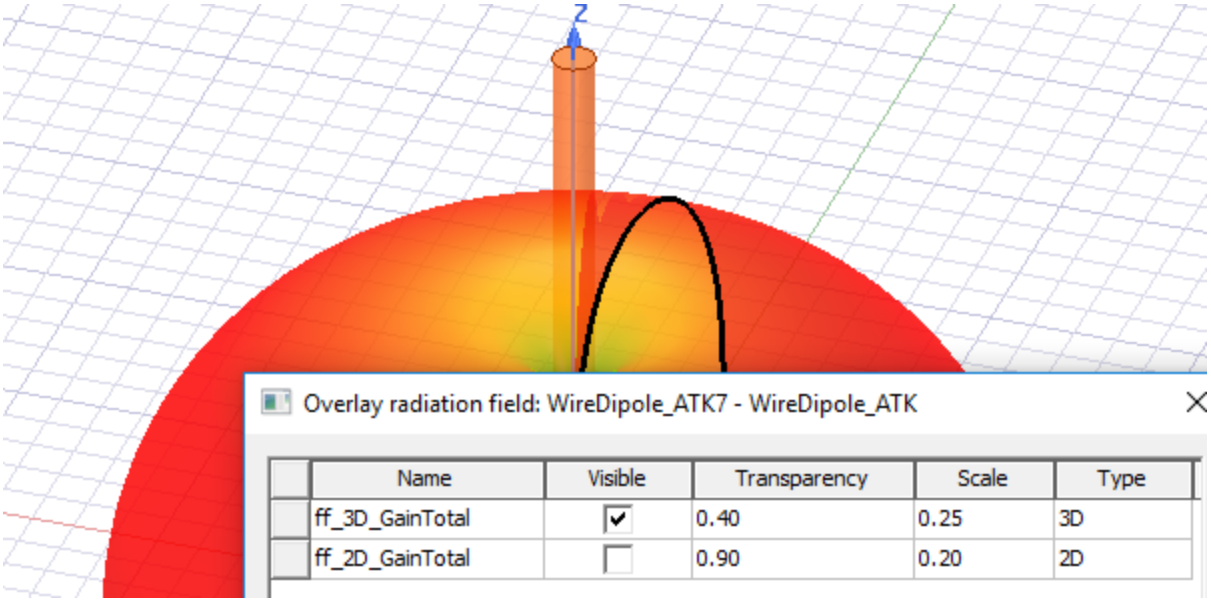
Other attributes can be modified via the Field Plot Attributes dialog box. These attributes include:

- Colormap Spectrum
- Auto Scale [On/Off] - automatically set Minimum/Maximum color scale range
- Minimum/Maximum - color scale range (linear)
- Number of Levels - sets the number of discrete color levels to use in linearly interpolating colors of shape/markers

Overlaying 3D Polar Plots of Near or Far Fields on Models

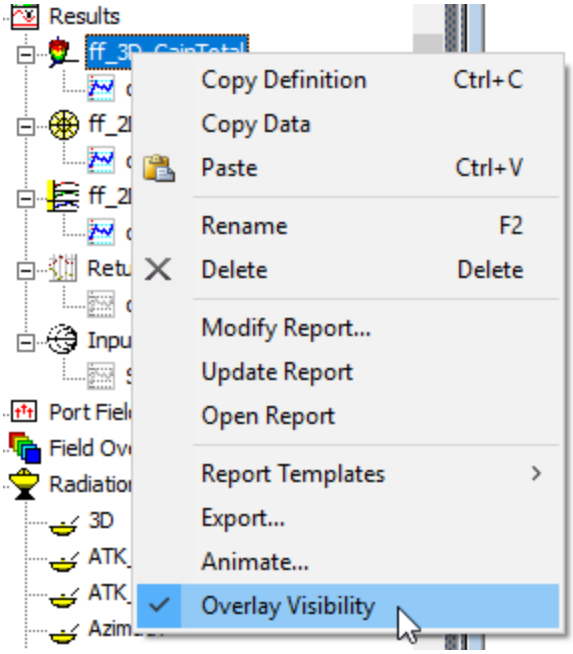
You can overlay existing 3D Polar Plots of near or far fields on the 3D model window by using the **Overlay Radiation** field dialog invoked by using the [*solver*]**>Fields>Plot Fields>Radiation Field...** command, or by right-clicking on Field Overlays in the **Project** tree and selecting **Field Overlays>Radiation Field...**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can edit the visibility, transparency and scale of the polar plot by using the dialog.

You also toggle the overlay of an existing plot by right-clicking on the plot in the **Project** tree and clicking **Overlay Visibility** on the shortcut menu.

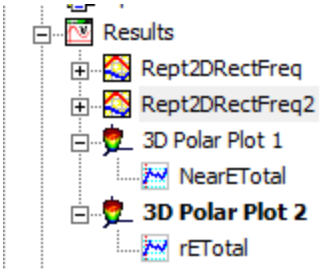


Using the Overlay Radiation Field Dialog

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For a 3D polar plot to be eligible for overlay, it must have its primary and secondary sweep from variables Phi and Theta or IWavePhi and IWave Theta in that order. If the plot is unsuitable, the **Overlay** commands are disabled.

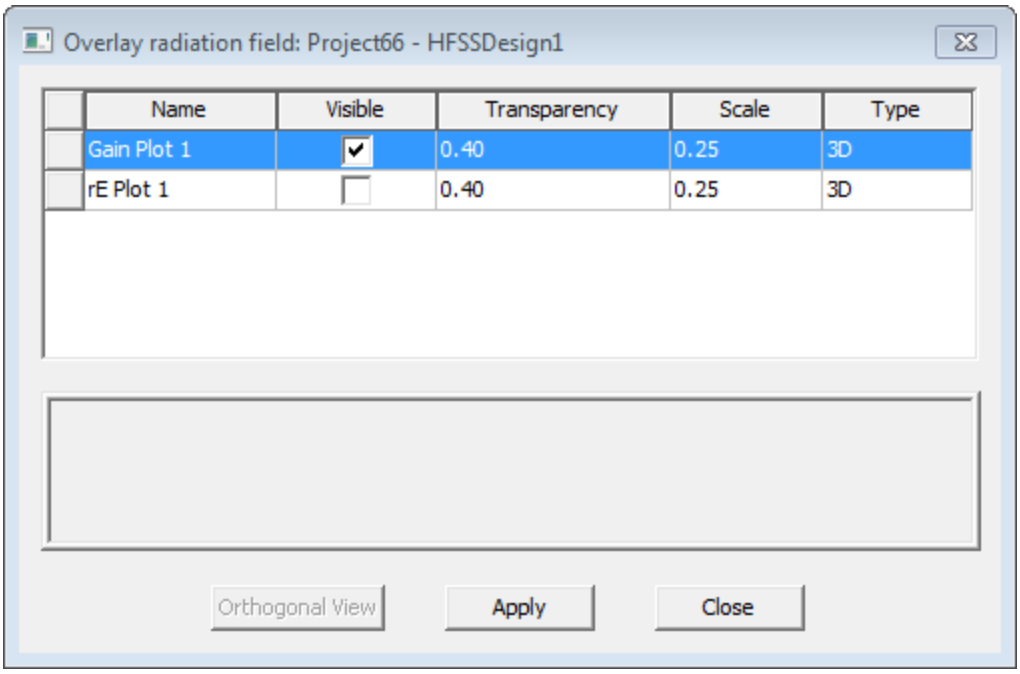
1. Create one or more suitable 3D Polar Plots of near or far fields for your model.



This enables the **Radiation Field...** command on the **Plot Fields** submenu.

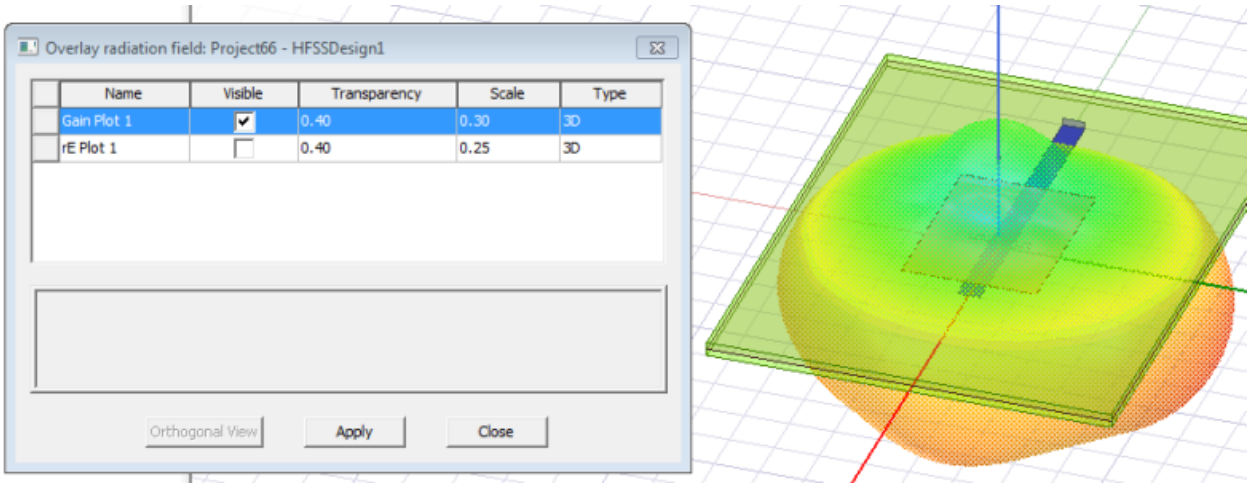
2. Click the [*solver*]**>Fields>Plot Fields>**[*field*] command, or by right-clicking on Field Overlays in the Project tree and selecting **Plot Fields>Radiation Field...**

This displays a dialog listing any existing 3D polar plots of near or far fields.



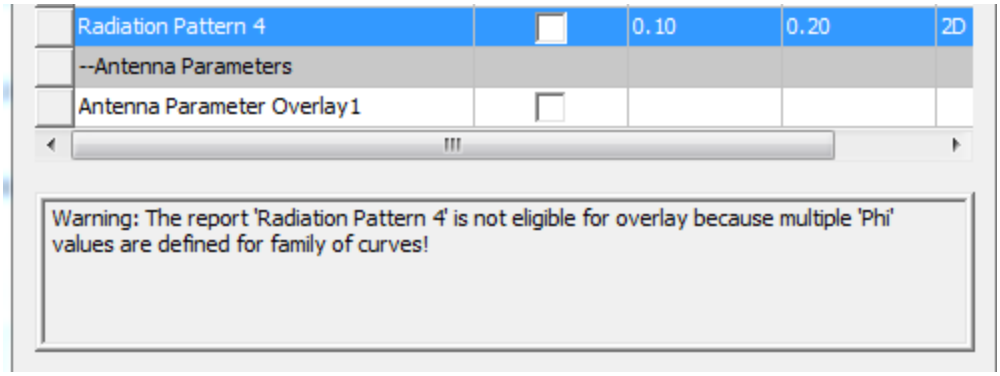
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Check the Visible box and click **Apply** to cause that plot to appear in the model window.



You can also edit the Transparency and Scale in the dialog box. Other properties of the 3D plot are controlled in its properties window.

If a 3D polar plot is unsuitable for overlay, the message field provides a description of the issue.



If you have Saved Antenna Parameters for Overlay, you also use this dialog to control whether to display a table of antenna parameters. See Antenna Parameter: Save for Overlay.

Overlaying 2D Radiation Field Plots on Models

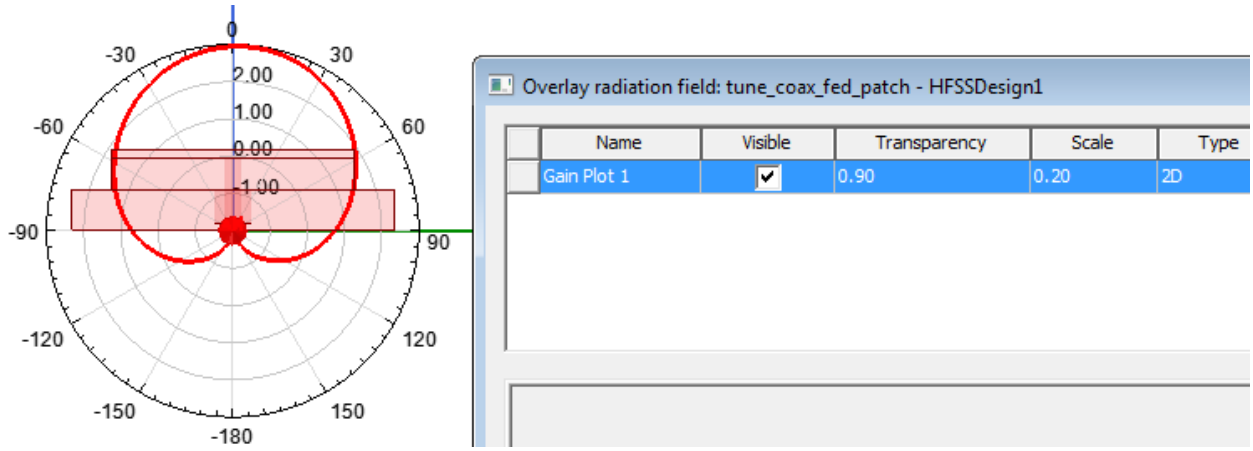
You can overlay Radiation Pattern plots (with restrictions) on the geometry in the Modeler window. The eligibility depends on whether the plot is defined on a single plane. For a Radiation Field plot, you right-click **Results** in the Project tree, and click **Results>Create Far Fields Report>Radiation Pattern**.

There are two cases of eligible radiation pattern plots for overlay, defining a single plane in a 3D Polar Plot.

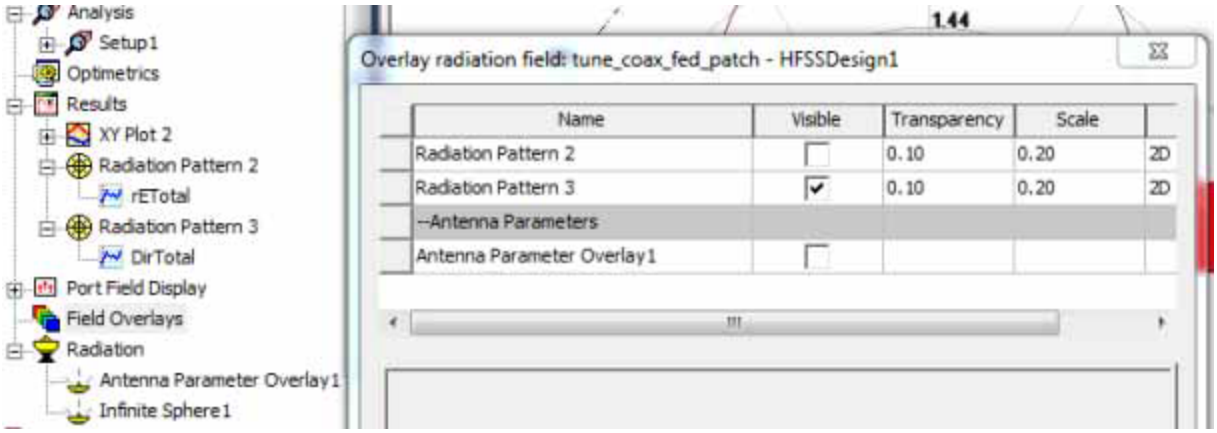
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Primary sweep on "Theta" and with one value for "Phi" in "Families" of curves, typically 0 or 90.
- Primary sweep on "Phi" and with one value for "Theta" in "Families" of curves and "Theta" is either 90deg or -90deg.

Note that the plot can contain multiple traces within this plane - for example traces for multiple frequencies of a discrete sweep. The Radiation Pattern plot will overlay on the 3D view, in the appropriate rotated plane of the far field CS, and include all traces and the axes/grid. There is no need to show the title or legend.



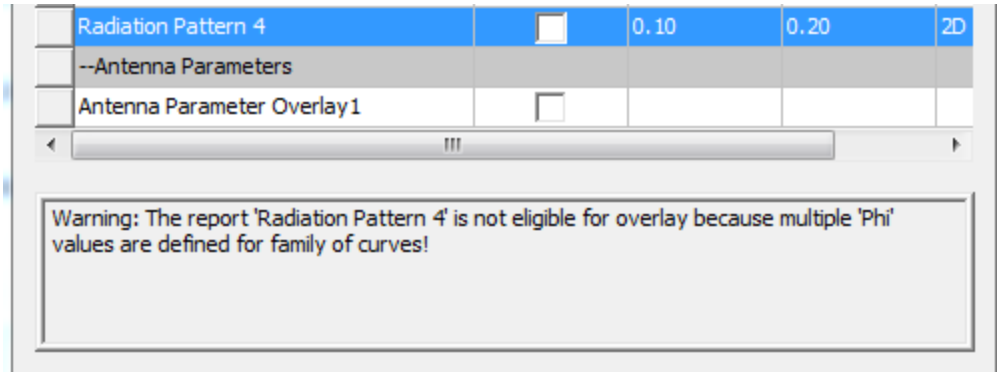
Radiation Pattern plots are added to the **Overlay radiation field** dialog that displays when you right-click **Field Overlays** in the Project Tree and select **Plot Fields>Radiation Field...** from the shortcut menu. The dialog lets you toggle visibility for the Model window and set plot parameters for Transparency, Scale, and Type (2D or 3D).



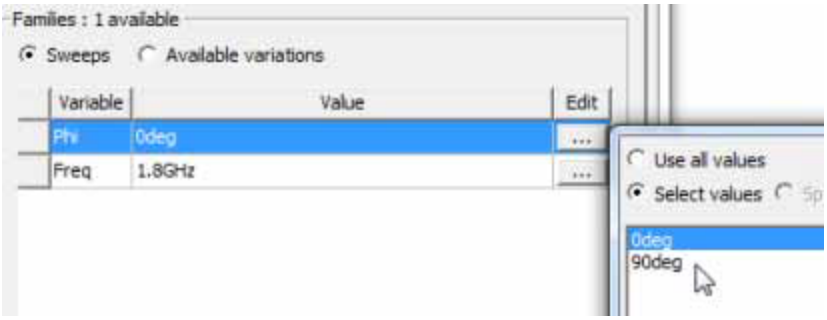
There are several cases mentioned above where an overlay will not be valid for display. This includes any radiation plot that has multiple planes. The **Overlay radiation field** dialog always lists all possibly overlays for selection, whether they are valid or not. However, if you check the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

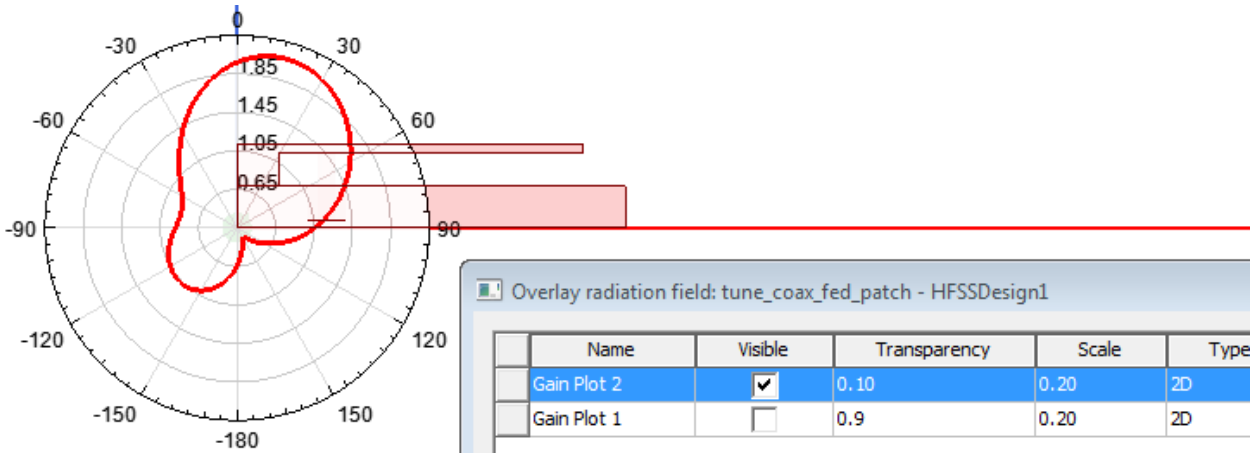
Visibility column an overlay that is not valid, and then click **Apply**, the text field in the dialog gives a warning describing why the overlay is not valid, and then unchecks that overlay. Similarly, if a valid overlay is already being shown, and then you edit the plot settings or antenna parameters such that the overlay becomes invalid, the visibility column becomes unchecked and the overlay stops displaying.



You can then use **Modify Report** to change the plot based on the warning to make it valid for display. Typically, you can select the **Families** Tab, and select the desired Phi value.



When the plot is valid, **Apply** then shows the plot in the modeler window.



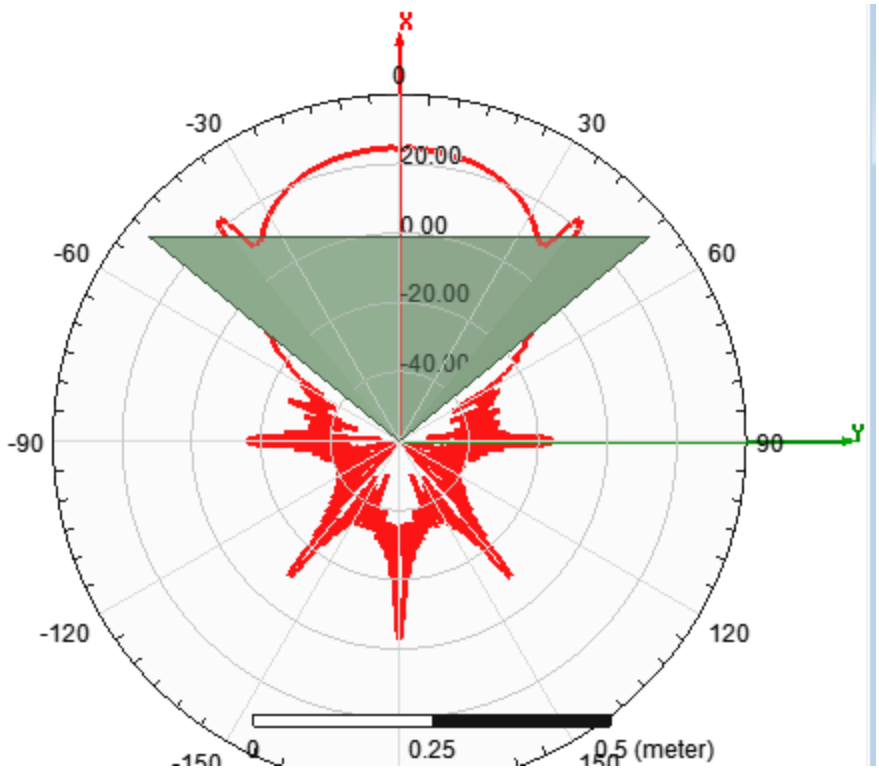
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you have Saved Antenna Parameters for Overlay, you also use this dialog to control whether to display a table of antenna parameters. See Antenna Parameter: Save for Overlay.

Also see Creating 2D Radiation Field Plot Overlay Animations

Overlaying Monostatic and Bistatic RCS 2D Plots

You can Monostatic and Bistatic 2D radiation pattern plots, in a similar way that you can overlay Monostatic RCS 2D Polar plots and Bistatic RCS 2D Polar plots, as well 3D Polar Plots and 2D Radiation patterns. As with regular radiation pattern plots, the eligibility depends on whether the RCS plot is defined on a single plane. The following is an example of a Monostatic RCSTotal Plot for a TriHedral SBR+ model:

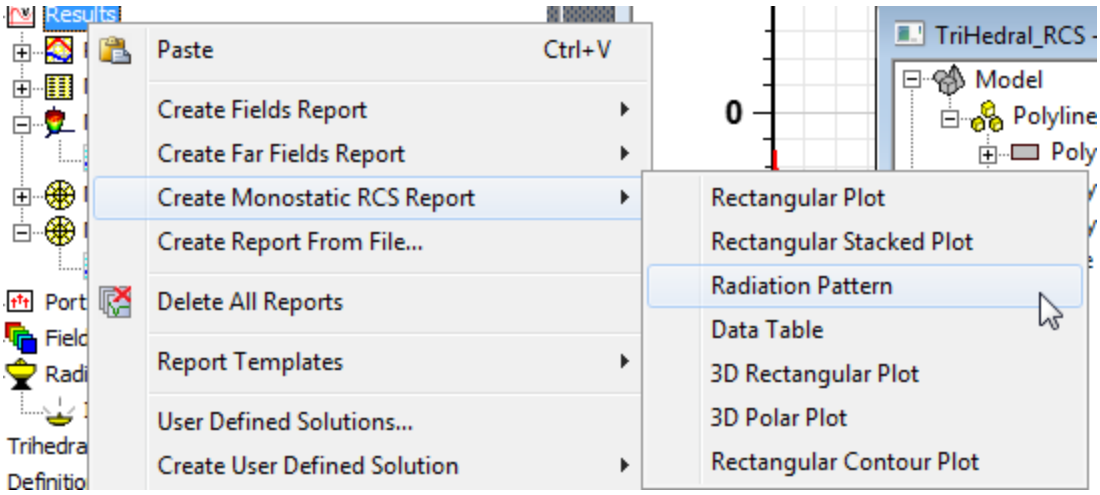


There are two cases of eligible Monostatic and Bistatic plots for overlay, defining a single plane in a 2D Polar Plot.

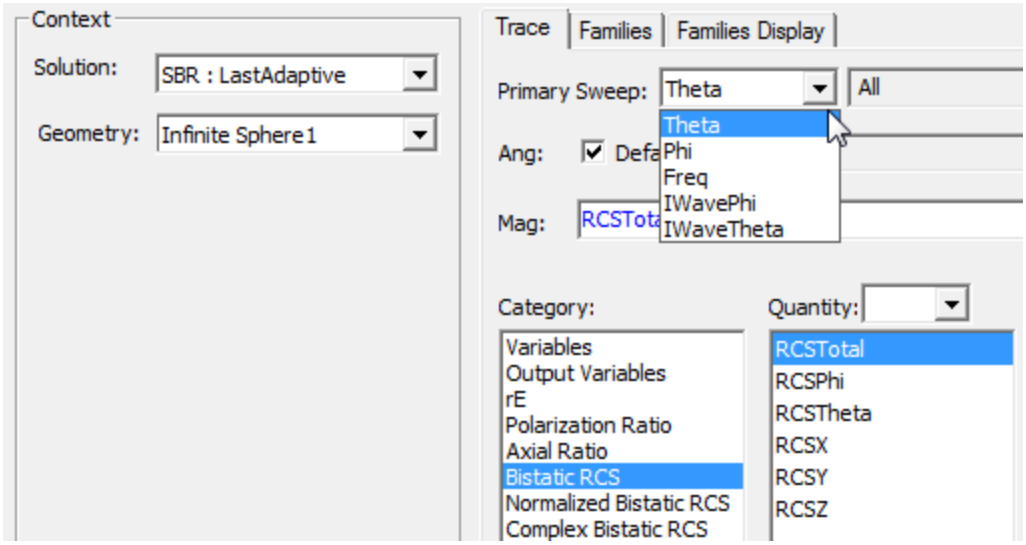
- For Primary sweep on Theta or IWave Theta, with the Phi or or IWavePhi to be any single value of the sweep, typically 0 or 90, but any single phi value is allowed.
- For Primary sweep on Phi or IWavePhi, Theta or IWaveTheta is either +/-90 deg.

Monostatic RCS Reports appear as options for models with SBR+ regions.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

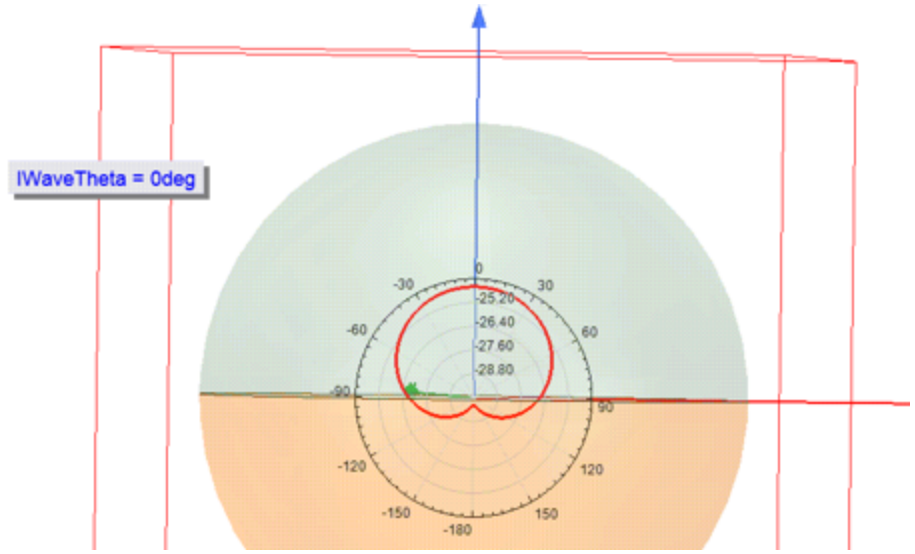


Bistatic RCS appears under Category for Far Field Reports in projects that contain SBR+ Regions.

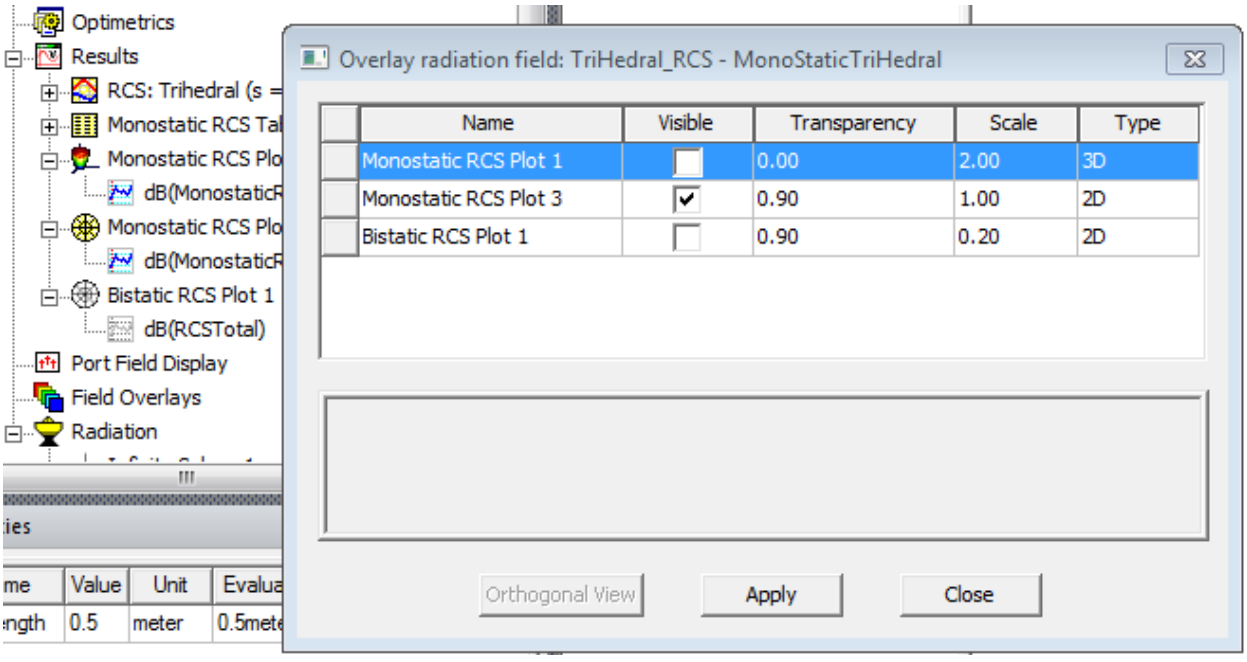


The following figure shows a Bistatic RCS Plot during animation with respect to Theta angle.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



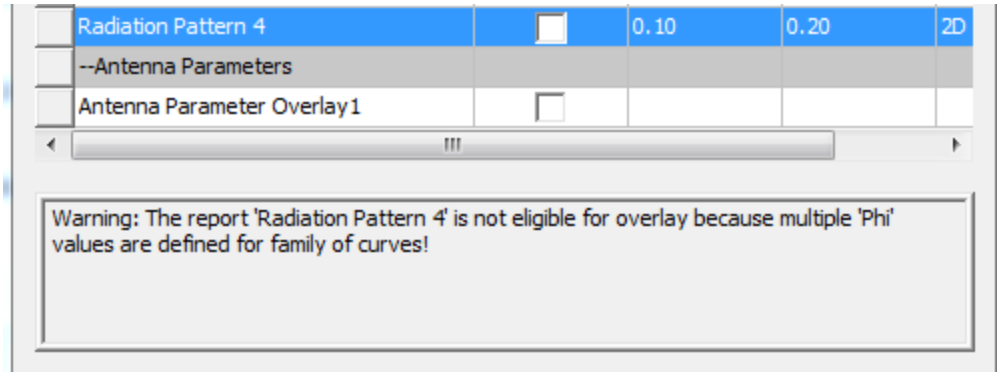
Radiation Pattern plots are added to the **Overlay radiation field** dialog that displays when you right-click **Field Overlays** in the Project Tree and select **Plot Fields>Radiation Field...** from the shortcut menu. The dialog lets you toggle visibility for the Model window and set plot parameters for Transparency, Scale, and Type (2D or 3D).



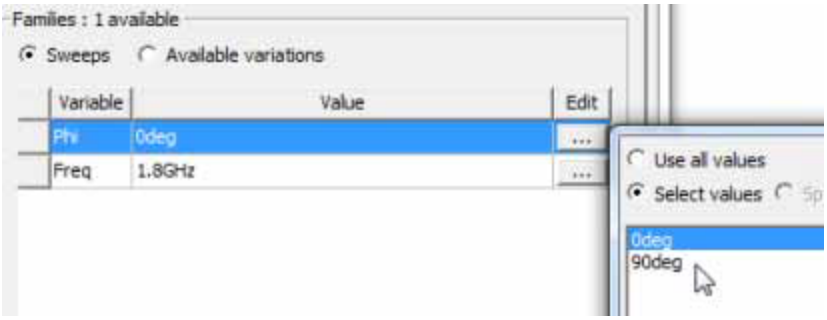
There are several cases mentioned above where an overlay will not be valid for display. This includes any radiation plot that has multiple planes. The **Overlay radiation field** dialog always lists all possibly overlays for selection, whether they are valid or not. However, if you check the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Visibility column an overlay that is not valid, and then click Apply, the text field in the dialog gives a warning describing why the overlay is not valid, and then unchecks that overlay. Similarly, if a valid overlay is already being shown, and then you edit the plot settings or antenna parameters such that the overlay becomes invalid, the visibility column becomes unchecked and the overlay stops displaying.



You can then use **Modify Report** to change the plot based on the warning to make it valid for display.



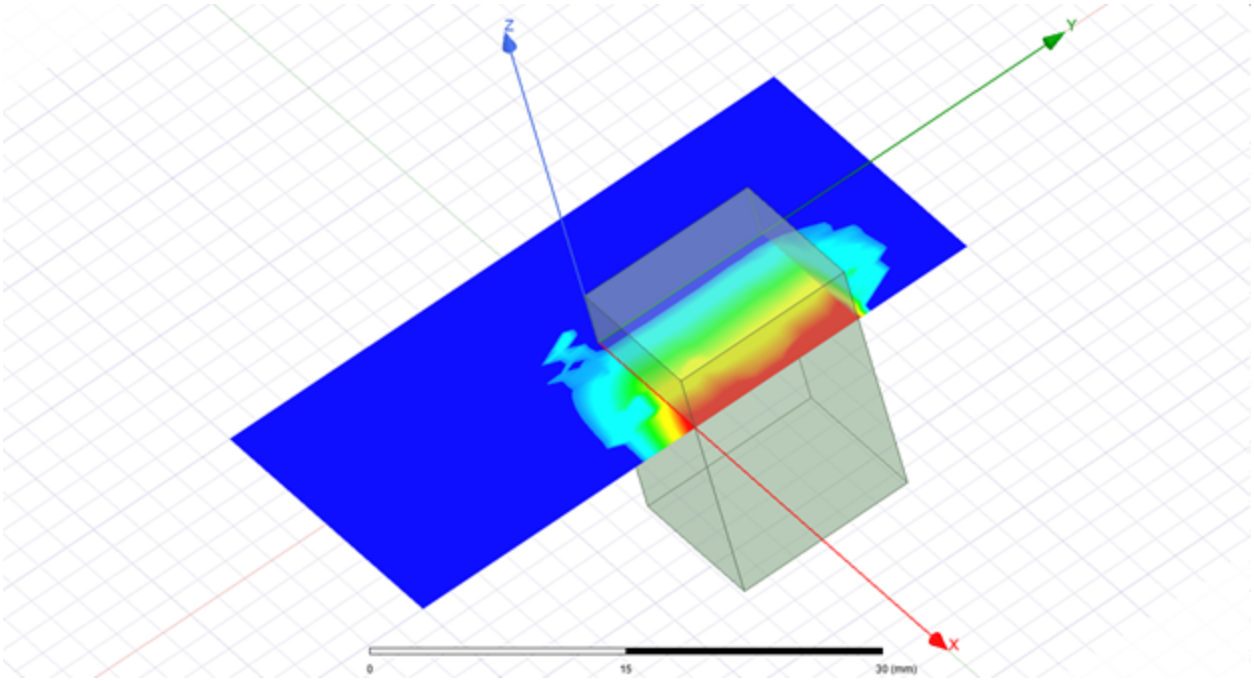
When the plot is valid, **Apply** then shows the plot in the modeler window.

If you have saved Antenna Parameters for Overlay, you also use this dialog to control whether to display a table of antenna parameters. See Antenna Parameter: Save for Overlay.

Overlay of Contour Plot of Near Field on Rectangle

For some contour plots like that for Near Field on a Rectangle radiation setup defined in a local coordinate system with physical dimensions, it makes sense to show the overlay of 2D contour plot in its actual defined rectangle in 3D space.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

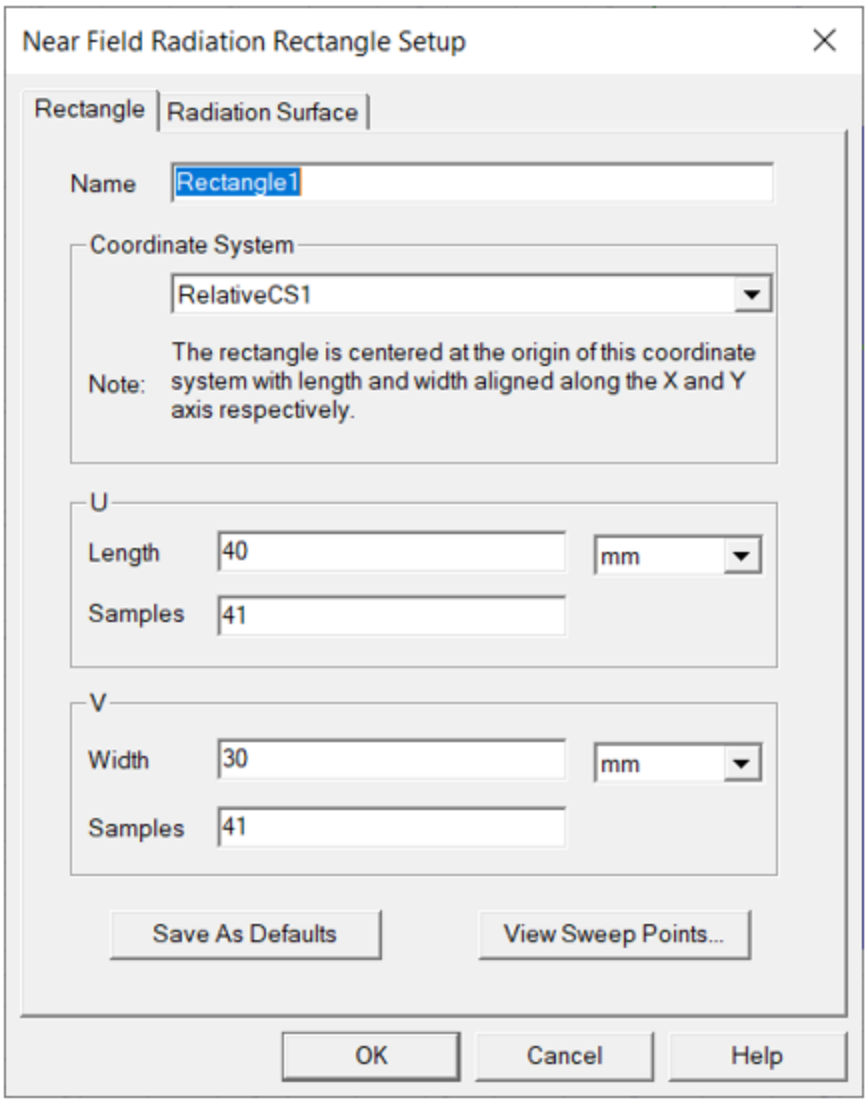


This feature is to allow user to have 3D overlay and its animation for this type of contour plot. The complete criteria of eligibility for overlay of this type of contour plot is as follows:

- Only a single trace plot
- Primary/Secondary sweep is _u and _v or _v and _u
- X/Y component of the trace is _u and _v or _v and _u
- No range function is used for Z component

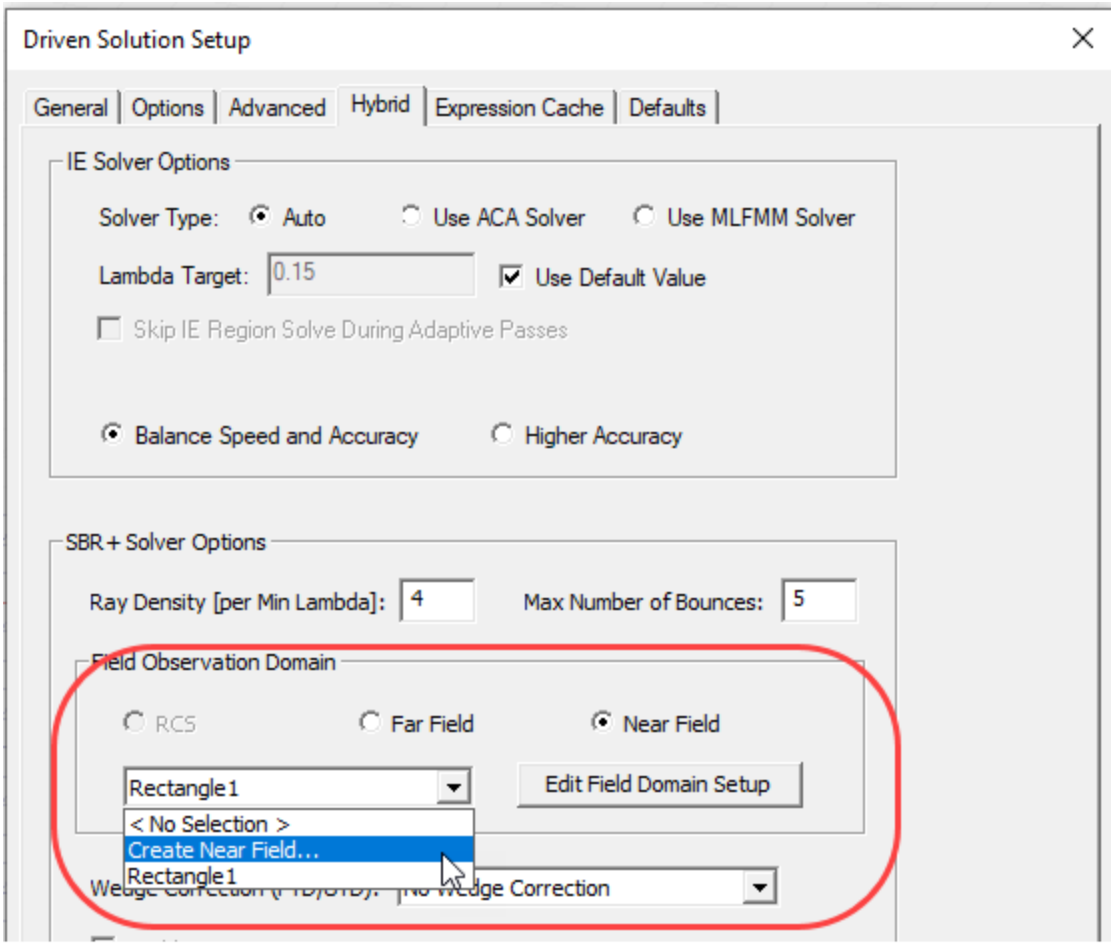
The Near Field rectangle setup looks something like this.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



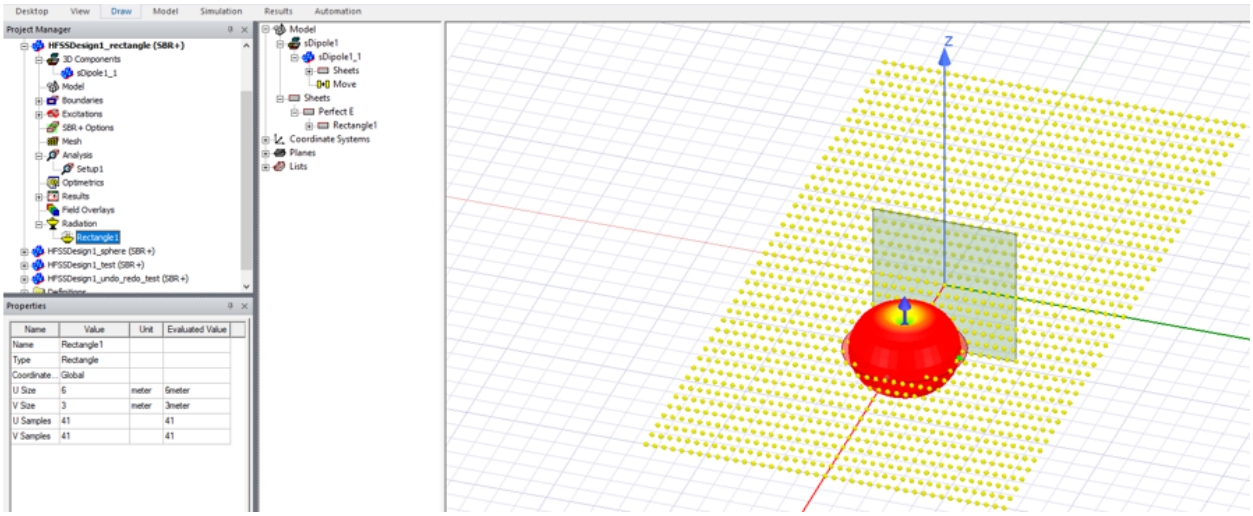
For HFSS Modal or Terminal solution types using SBR+ Regions, or HFSS SBR+ solution types, you must create the Near Field Rectangle before simulation. You can do this through the Field Observation Domain setting on the Hybrid tab of the Advanced Driven Solution Setup for and HFSS Driven Modal or Terminal Solution, or the Options Tab of the Driven Solution setup for an HFSS SBR+ Solution type setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



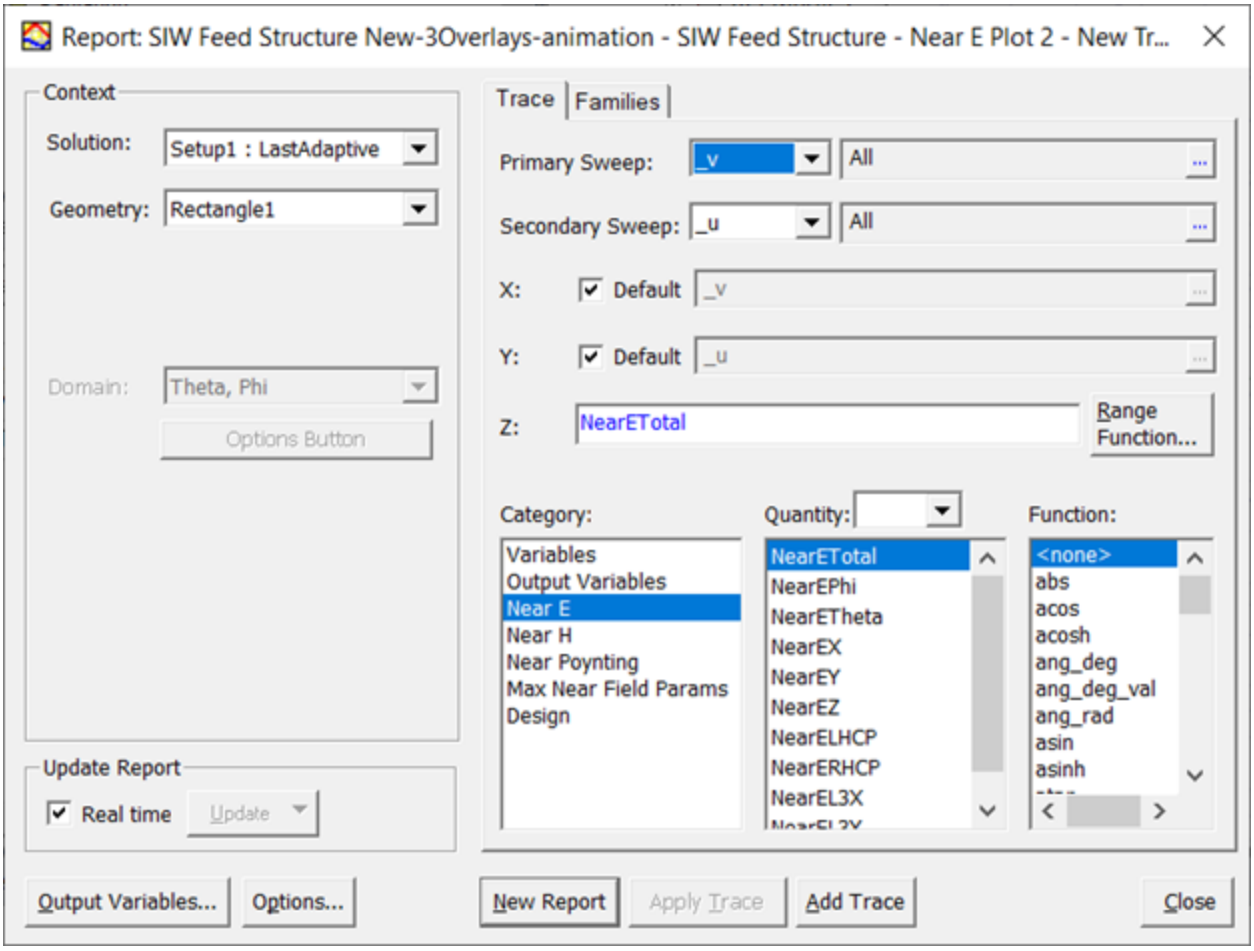
To preview the sampling points for a Near Field Rectangle, select the setup under the Radiation icon in the Project tree.

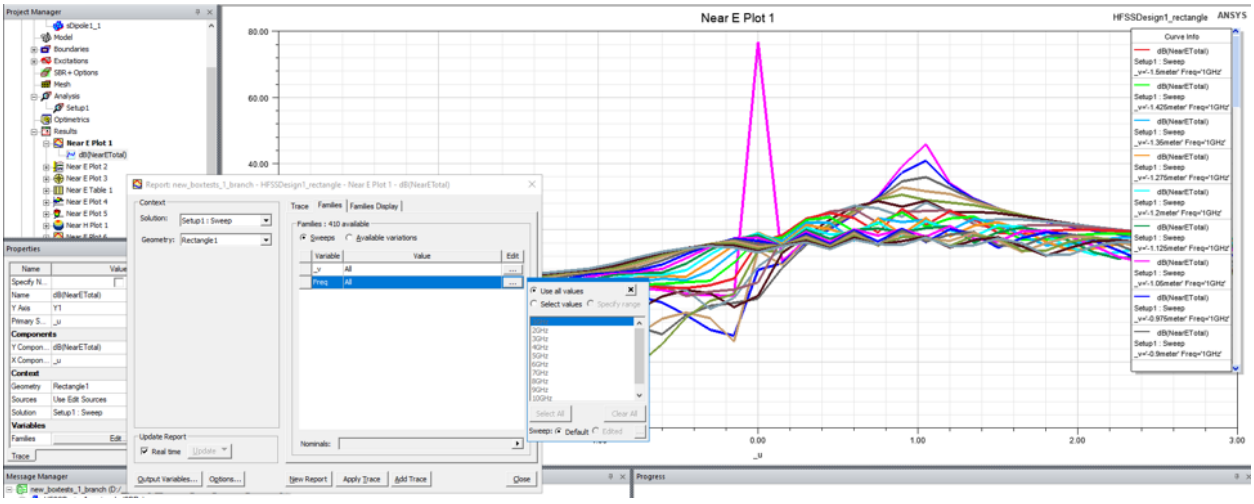
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Report** dialog for a Rectangular Contour plot of near field, using a Near Field Rectangle as the geometry, a combination of _u and _v as the Primary and Secondary sweeps is as follows.

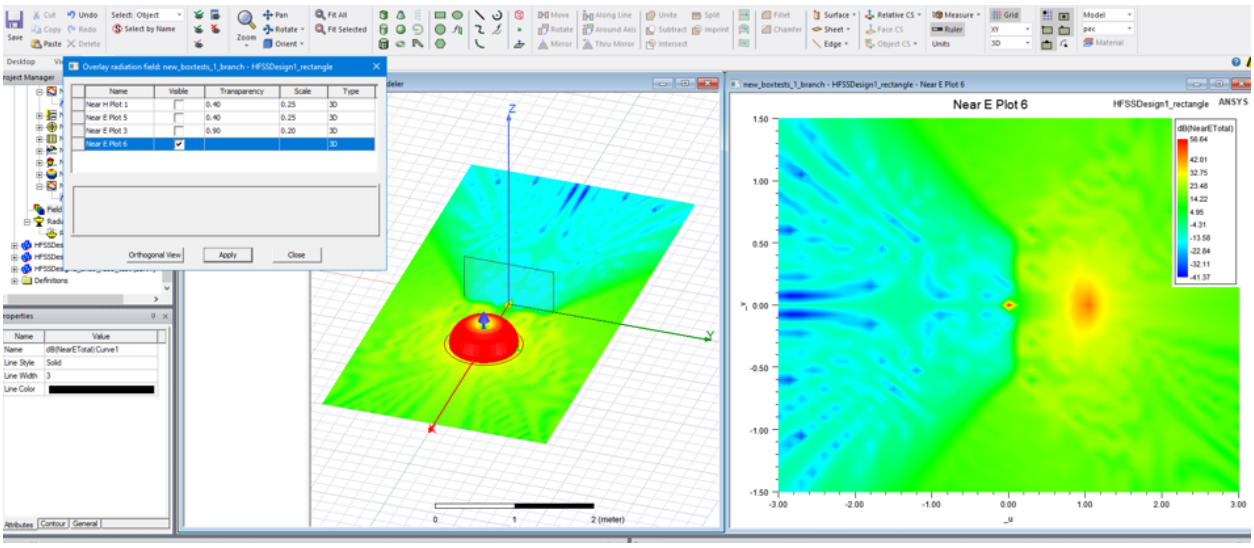
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A rectangular Near Field contour plot can be viewed as a report plot, or overlaid in the model window, just like other overlays, through the context menu. Rectangle size is not reflected in its 2D plot, it is only available in its overlay. The same applies to the coordinate system.



This Near Field Rectangular contour plot overlay can also be animated if there is a sweeping variable which could change the plot as the variable changes (e.g. it could change the rectangle location, etc.)

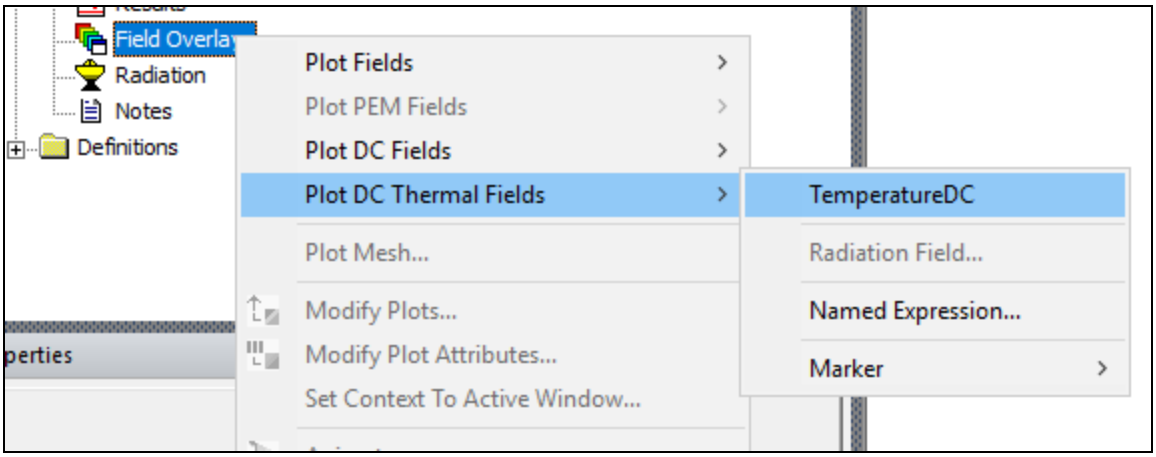
Also see Creating 2D Radiation Field Plot Overlay Animations

Overlaying DC Temperature Fields (General Mode Only)

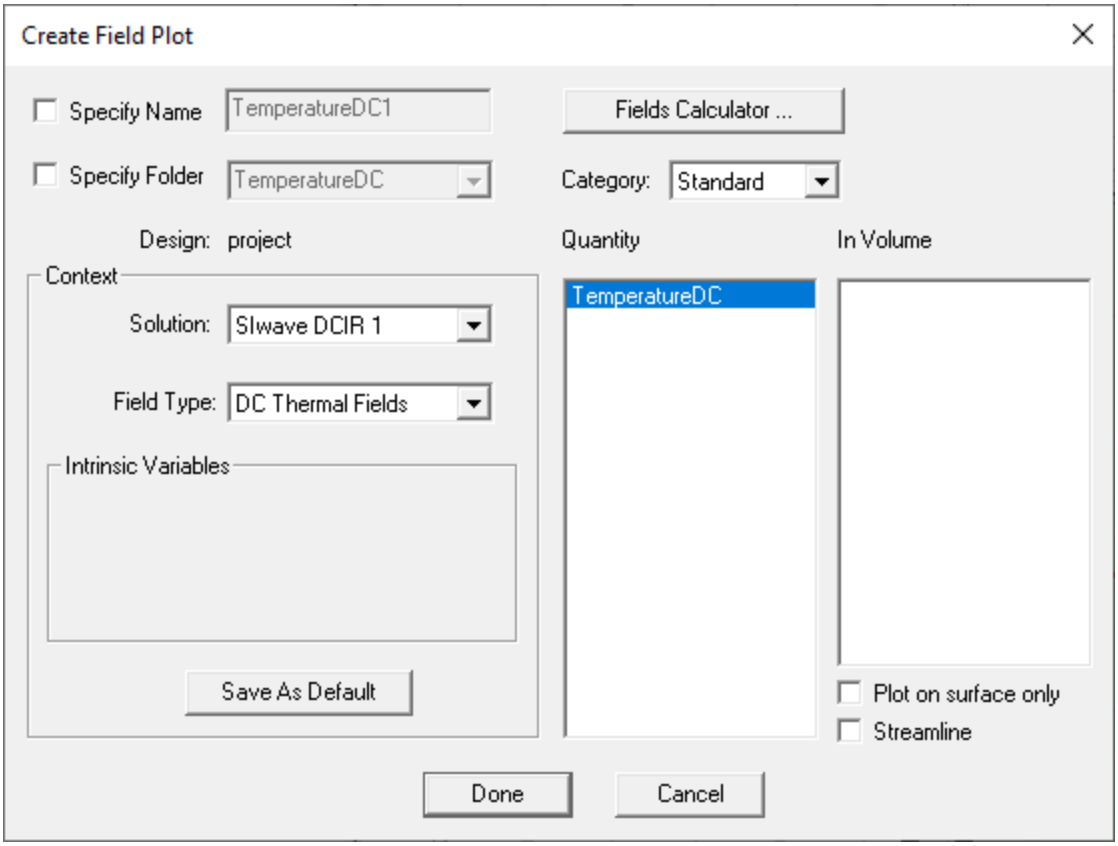
Projects with two-way coupling between Icepak and HFSS 3D Layout can have a field overlay that displays the DC temperatures. The 3D Layout project must have an Icepak DC IR analysis. To create such an overlay:

1. In the Project Manager, open the 3D Layout design and right click **Field Overlays** > **Plot DC Thermal Fields** > **TemperatureDC**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



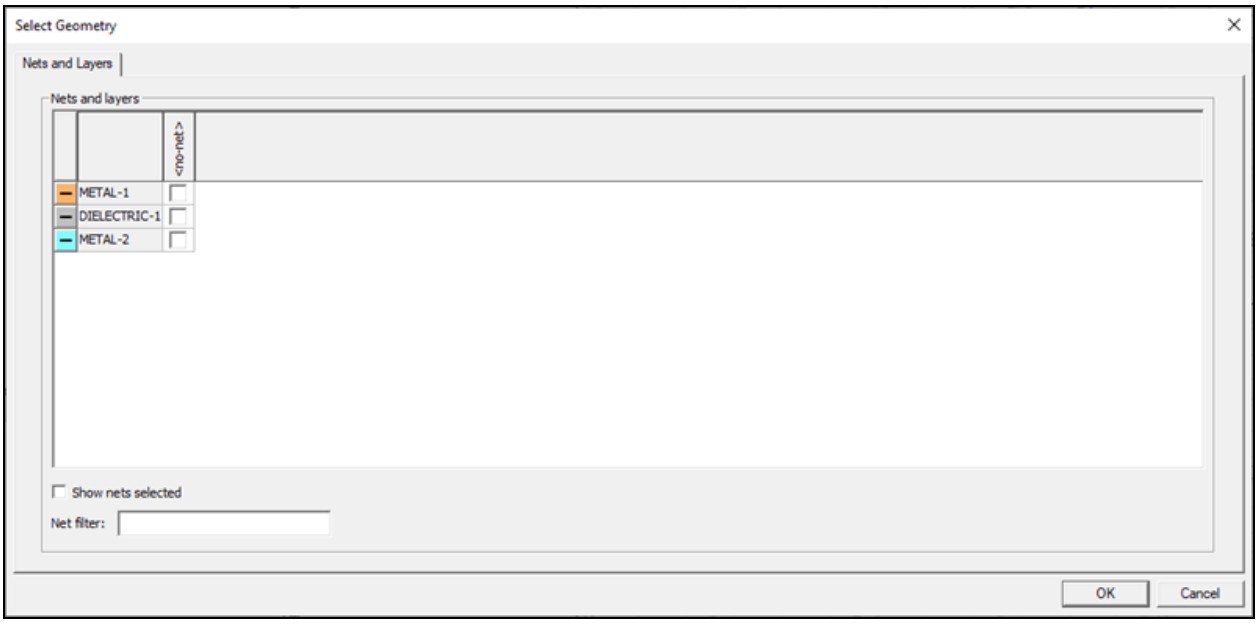
The **Create Field Plot** window opens.



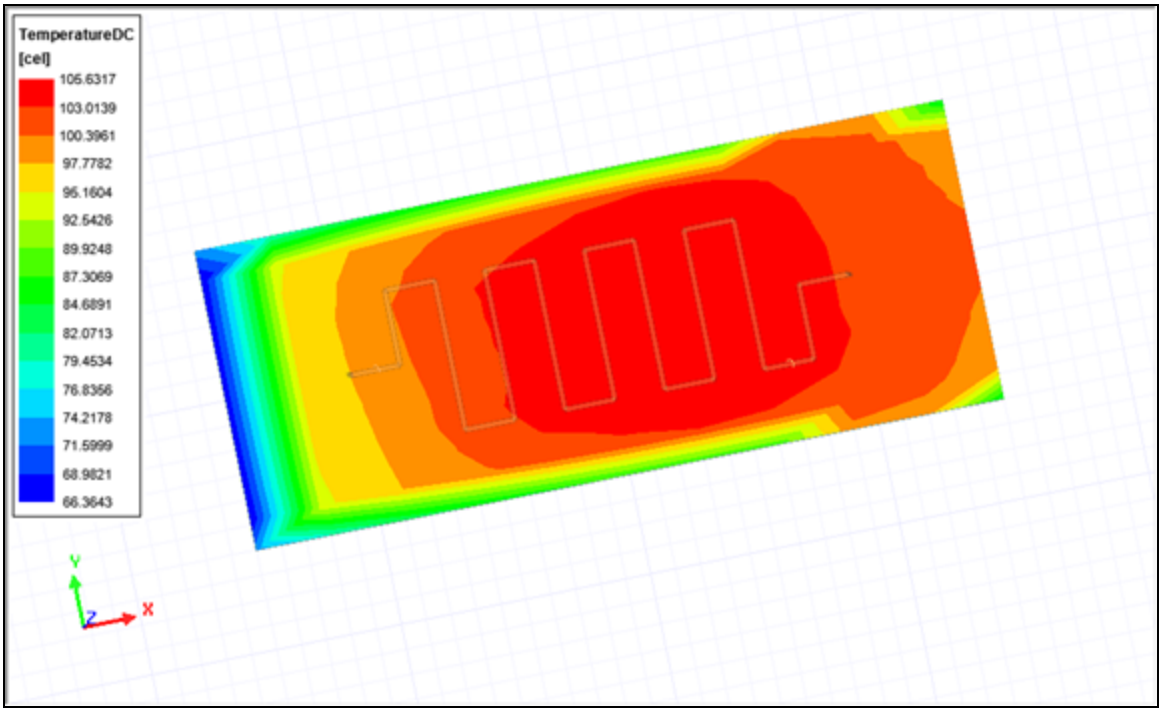
2. Select the **TemperatureDC** quantity. It is selected automatically.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Done**. The **Create Field Plot** window closes, while the **Select Geometry** window opens to its **Nets and Layers** tab.



4. Select the layer whose temperature field should display.
5. Click **OK**.
   The overlay appears.

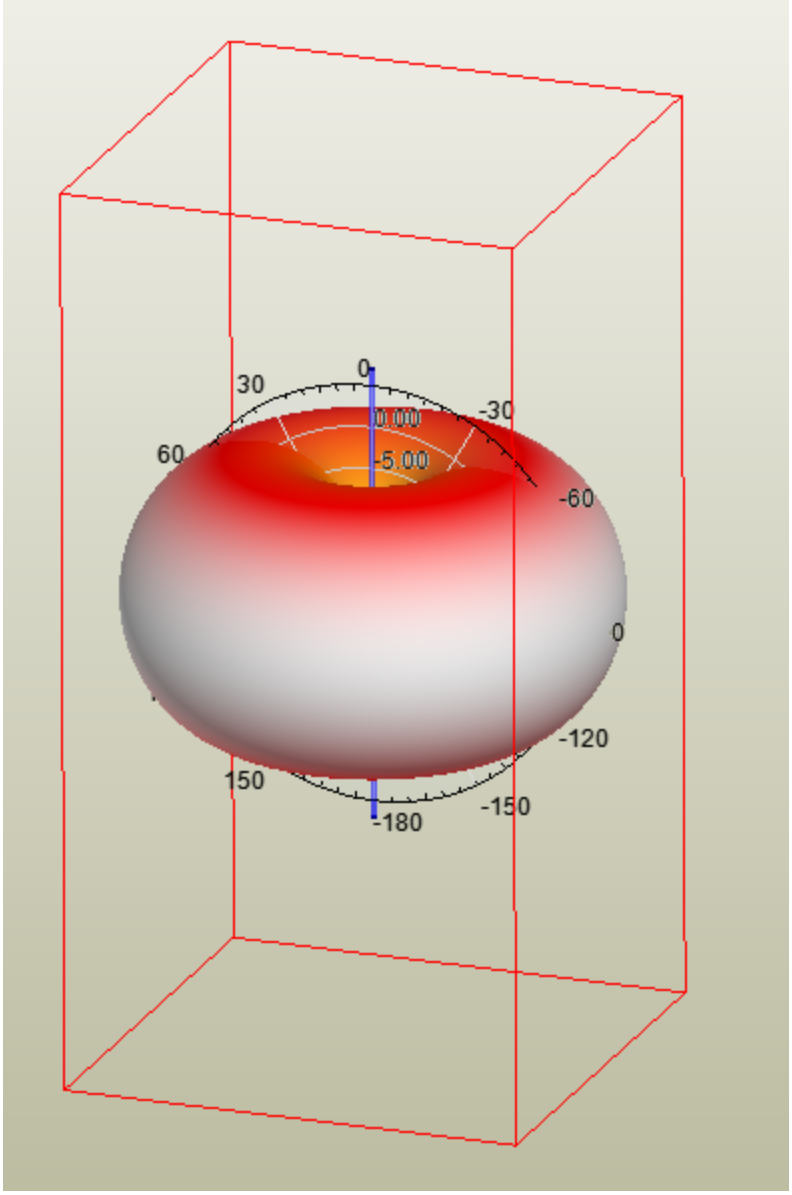


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23-1 An example of the DC temperature fields overlay

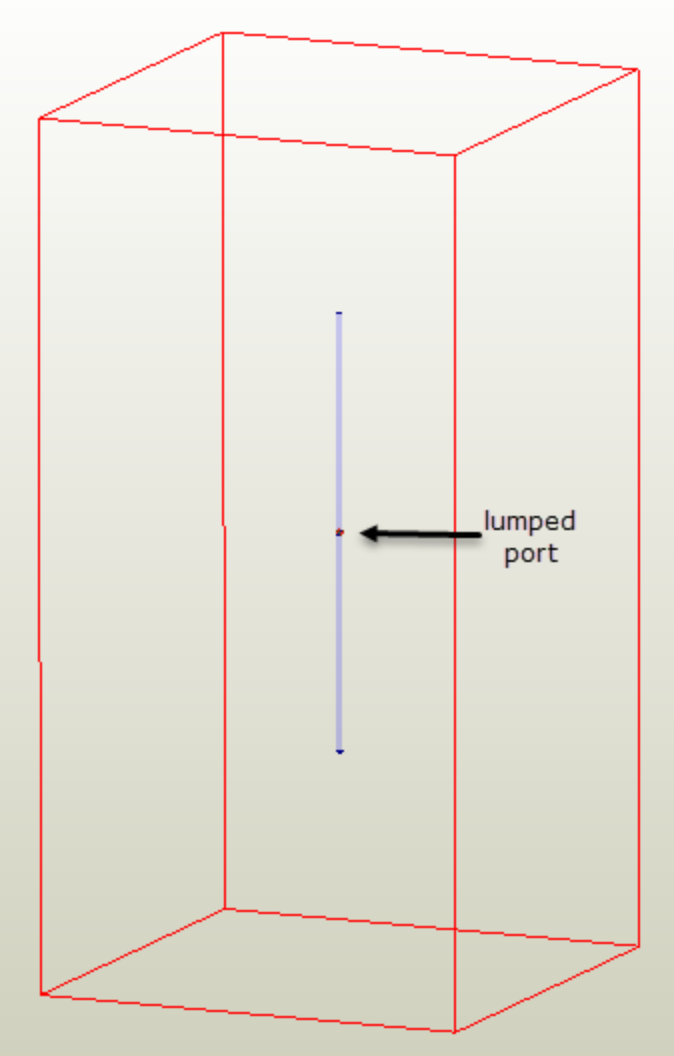
Overlay Visibility

This section describes how to plot radiation fields like Gain or Directivity right there on the geometry in the HFSS 3D Modeler window. Just right-click the plots under **Results** in the project tree and select the **Overlay Visibility** option from the short-cut menu to display the plots on the geometry.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

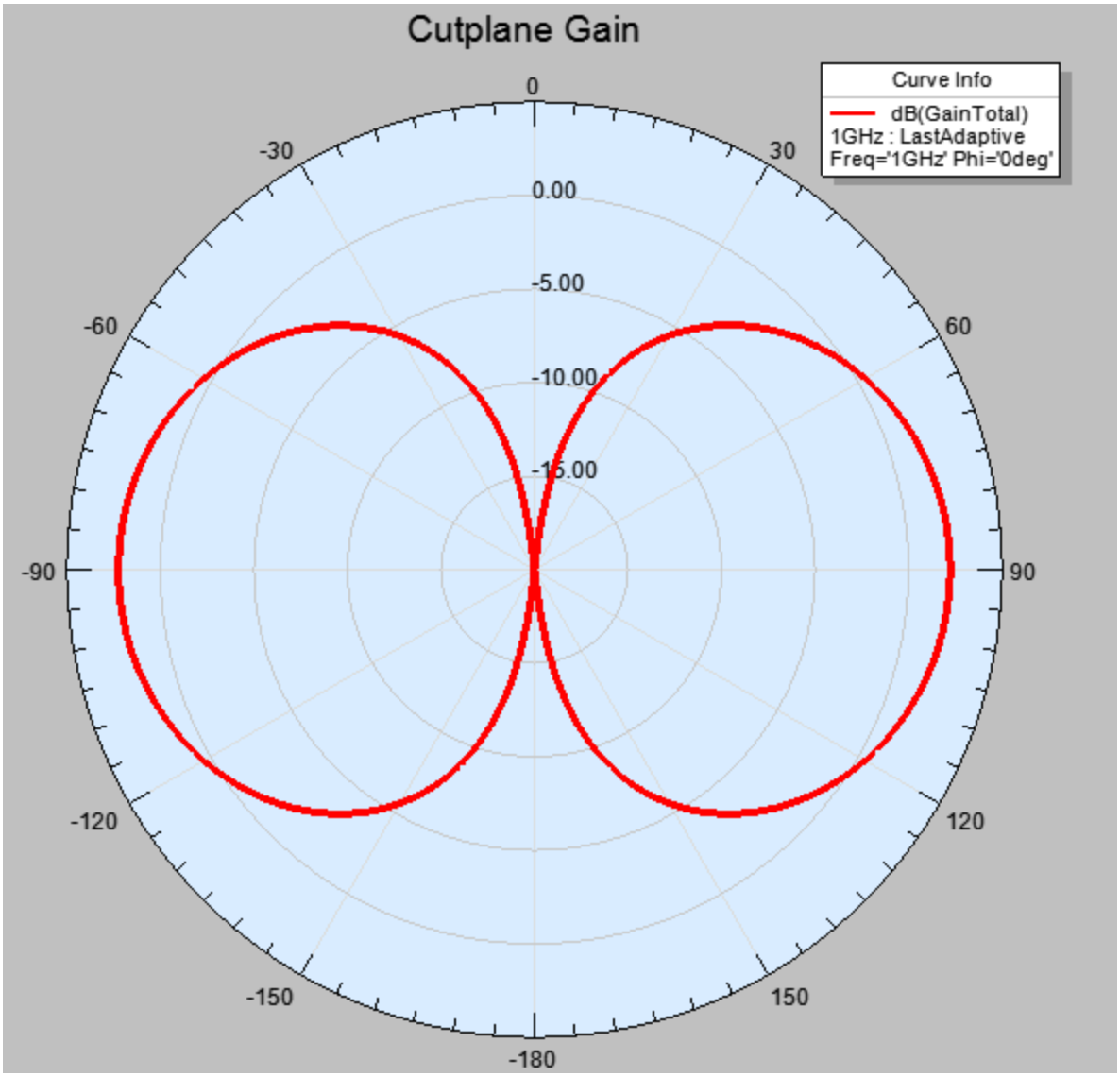
To illustrate this feature, consider a basic dipole antenna design shown in the following figure. The antenna is excited with a lumped port between the two nodes of the dipole.



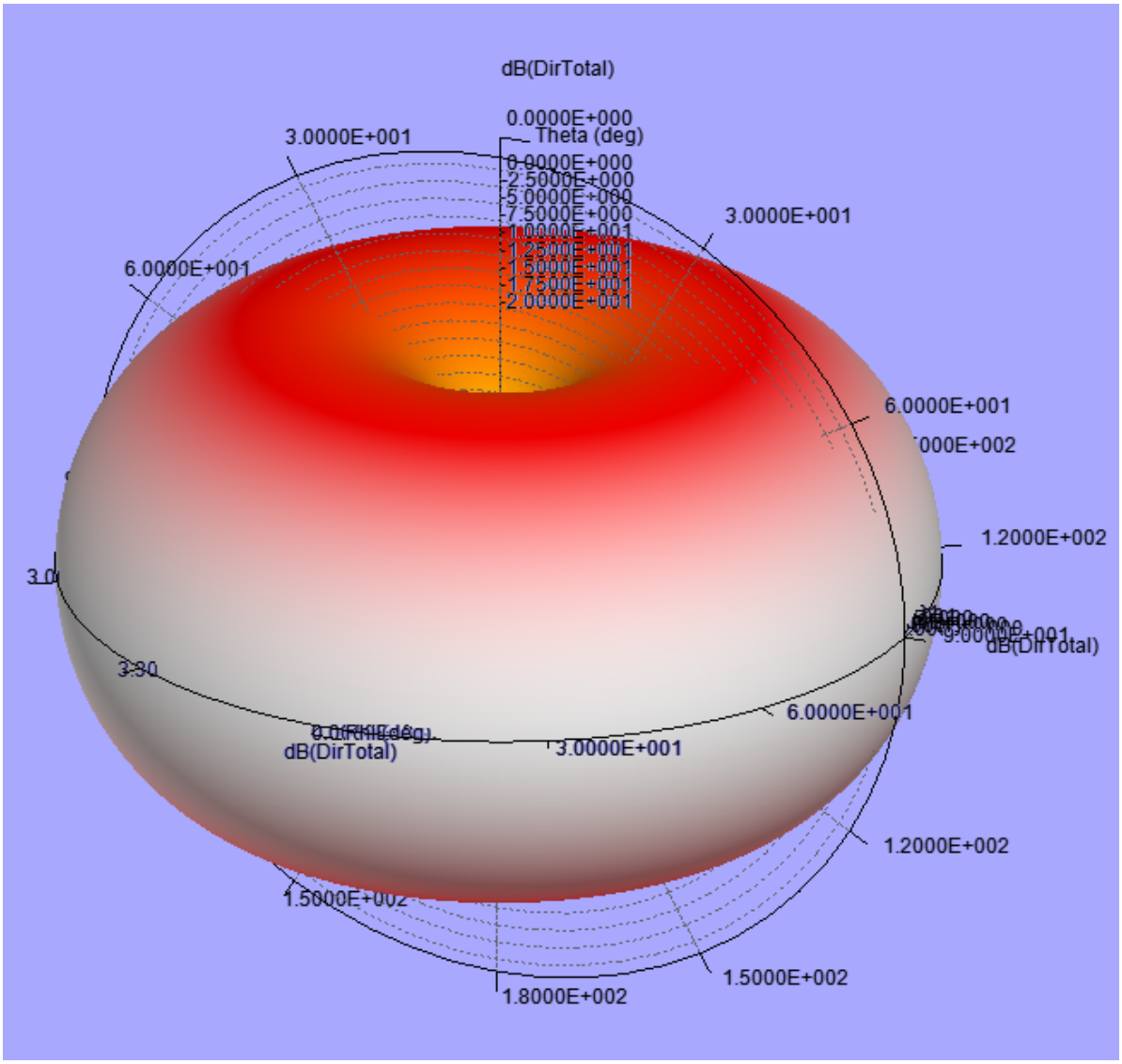
The antenna is simulated at 1 GHz. The generated Cut plane Gain and 3D far field plots can be displayed in their individual report windows. These are shown in the figures below.

> **Note:**
>
> You can also change the background color of these plot windows.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

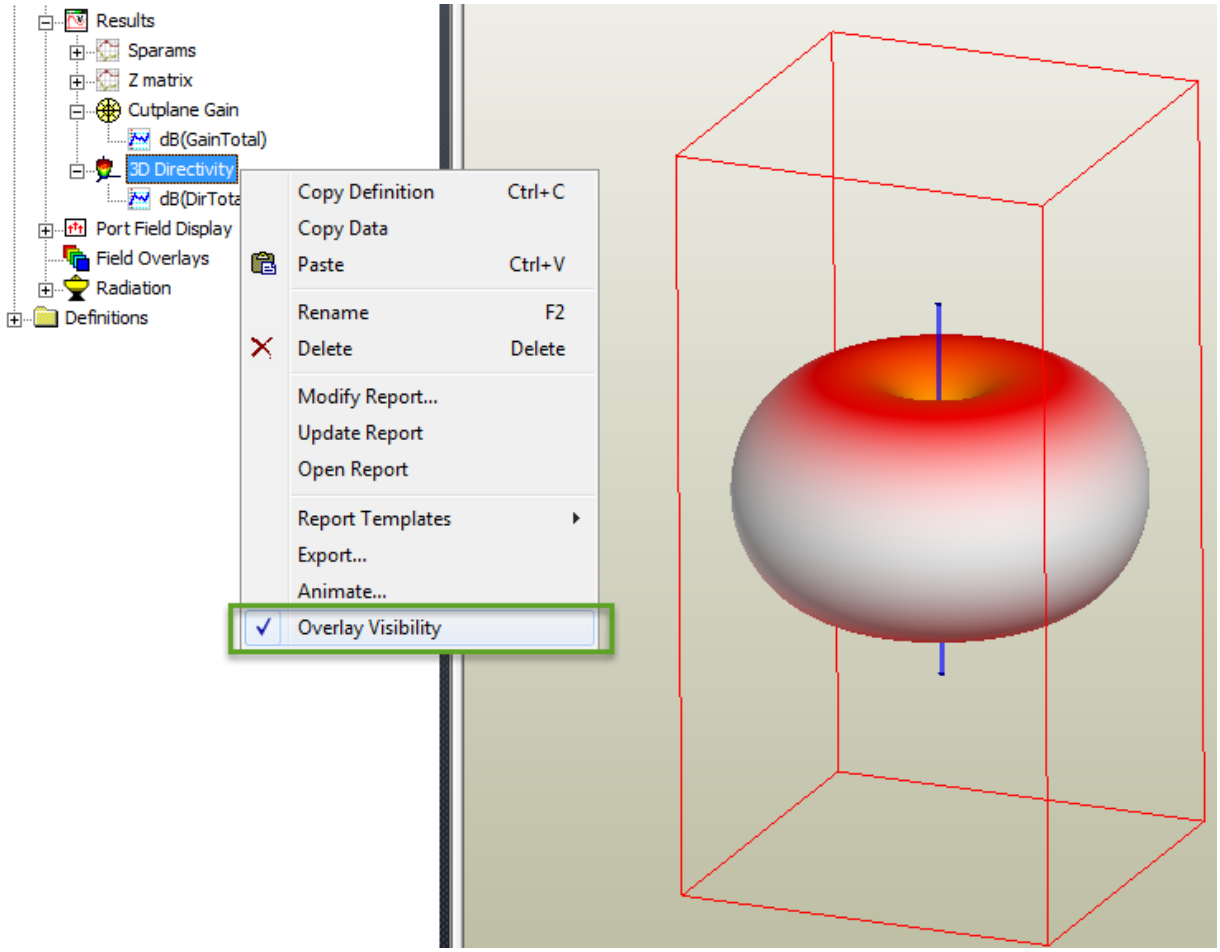


You can overlay these far field radiation patterns on the dipole antenna in the 3D modeler window by using a simple short-cut command. This overlay visibility feature can be very useful if you want to understand the orientation of the far field pattern relative to the structure of the antenna. You can directly apply the cut plane plots or the 3d polar plots right there on the antenna design as follows:

- Right-click the plot under **Results** in the project tree and select the option **Overlay Visibility** as shown below.

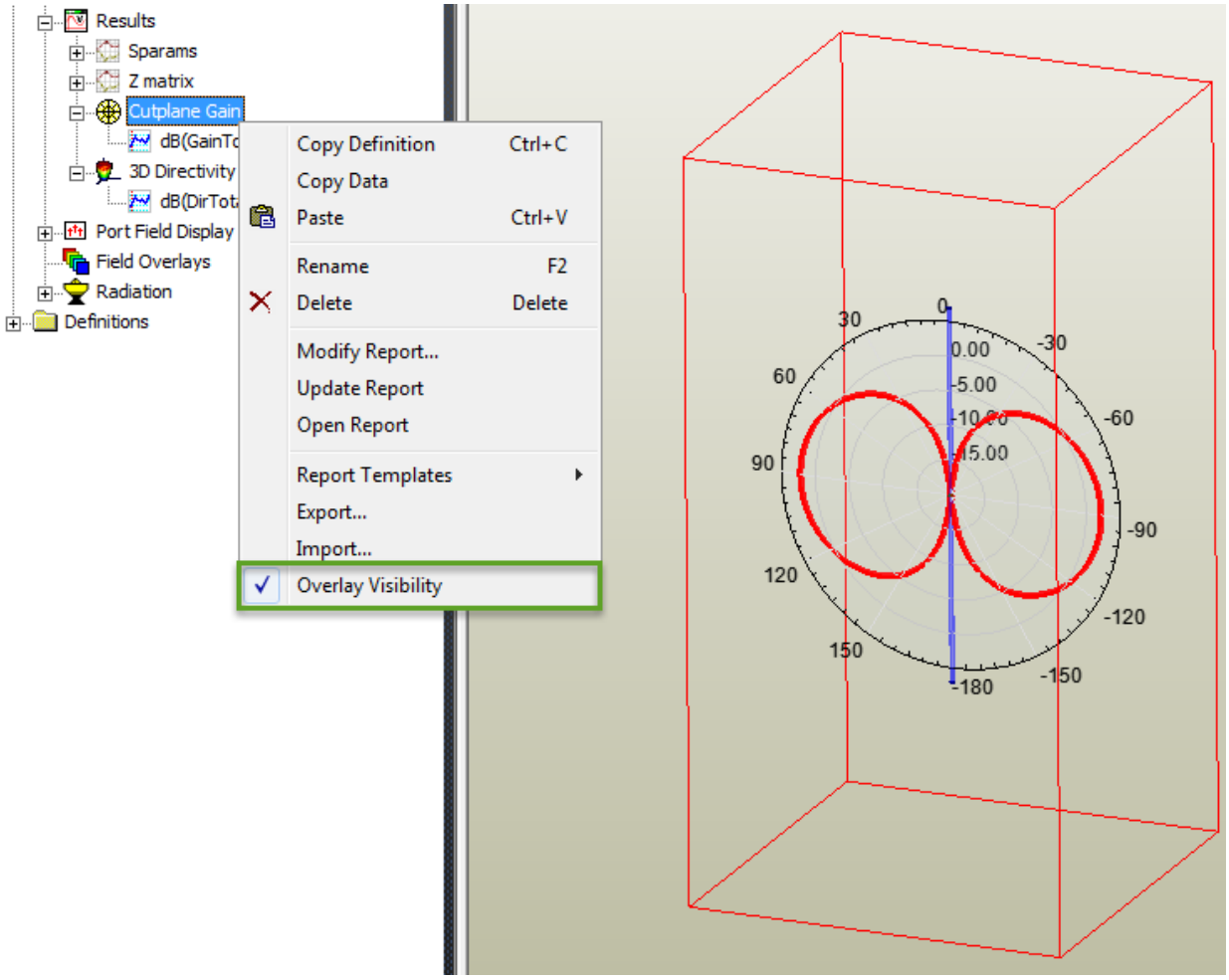
  This command overlays the plot on the antenna design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The same one-step process also overlays a 2D radiation pattern as shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Plotting Derived Field Quantities

Derived field quantities are field quantity representations that have been deduced from the original field solution using the Fields Calculator.

1. Select a point, line, surface, or object to create the plot on or within.
2. On the **HFSS** menu, or right-click on the **Field Overlays** icon in the Project tree, and point to **Fields>NamedExpression**.
3. Select the derived quantity you want to plot, and then click **OK**.

The **Create Field Plot** dialog box appears.

4. To specify a name for the plot other than the default, select **SpecifyName**, and then type a new name in the **Name** text box.
5. Select the solution to plot from the **Solution** drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. To specify a folder other than the default in which to store the plot, select **Specify Folder**, and then click a folder in the **Plot Folder** drop-down menu, or type the name you wish to use. Plot folders are listed under **Field Overlays** in the project tree.
7. Under **Intrinsic Variables**, select the frequency and phase angle at which the field quantity is evaluated.
8. Select the derived field quantity to plot from the **Quantity** list.
9. Select the volume, or region, in which the field will be plotted from the **In Volume** list.

   This selection enables you to limit plots to the intersection of a volume and the selected object.

10. Click **Done**.

    The derived field quantity you created in the Fields Calculator is plotted on the surfaces or objects you selected. The new plot is listed in the project tree under **Field Overlays**.

Creating 2D Reports from Named Expressions

You can create a 2D report from named expressions that evaluate as a single scalar value or that evaluate along polylines.

To create a report from a named expression evaluating as a single scalar value:

1. Click **HFSS**, and then click **Results** > **Create Report**.

   The *Create Report* dialog box appears.

2. In the **Target Design** drop-down menu, click the design containing the solution data you want to plot.
3. Select **Fields** from the **Report Type** drop-down menu.
4. In the **Display Type** drop-down menu, select the type of report you want to create.
5. Click **OK**.

   The *Traces* dialog box appears.

6. In the **Solution** drop-down menu, click the solution containing the data you want to plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Add one or more traces to include in the report.

> **Note:**
>
> If you select a point for the **Geometry** and a **Category** that is a named expression based on the point (or other scalar, non-3D value), then the **Quantity** list only lists expressions that returns a single scalar value. If you have added named expressions that take the magnitude of the ScalarX(), ScalarY(), or ScalarZ() of a vector point value, then you can create output variables for those expressions only in this case.

8. Click **Done**.

   The report appears in the view window and is listed in the project tree. Once you have created a report, additional options become available on the **Results** submenu.

To create a report from a named expression evaluating along a polyline:

1. Click **HFSS** and then click **Results> Create Report**.

   The *Create Report* dialog box appears.

2. In the **Target Design** drop-down menu, click the design containing the solution data you want to plot.
3. Select **Fields** from the **Report Type** drop-down menu.
4. In the **Display Type** drop-down menu, select the type of report you want to create.
5. Click **OK**.

   The *Traces* dialog box appears.

6. In the **Solution** drop-down menu, click the solution containing the data you want to plot.
7. Select the geometry you want to plot from the **Geometry** drop-down menu.
8. Add one or more traces to include in the report.

> **Note:**
>
> If you select a polyline for the **Geometry** and a **Category** that is a named expression based on the line, then the **Quantity** list only lists corresponding expressions. The list will not contain scalar values when a line is selected as the geometry.

9. Click **Done**.

   The report appears in the view window and is listed in the project tree. The default name is based on the Report Category you selected, (for example, S Parameter Plot *n* or rE Plot

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

n). You can edit the plot names in the project tree and the plot header text in the report synchronizes. Once you have created a report, additional options become available on the **Results** submenu.

Creating Scalar Field Plots

A scalar plot uses shaded colors or contoured lines to illustrate the magnitude of field quantities on surfaces or volumes.

1. Do one of the following:
    a. To create a scalar surface plot, select the faces on which you want to plot the fields.
    b. To create a scalar volume plot, select the objects within which you want to plot the fields.
2. Click **HFSS** > **Fields** > **Plot Fields**.
3. On the **Plot Fields** menu, click the scalar field quantity you want to plot.

    The **Create Field Plot** dialog box appears.
4. Follow the procedure for plotting field overlays.

The plot uses the attributes specified in the **Plot Attributes** dialog box.

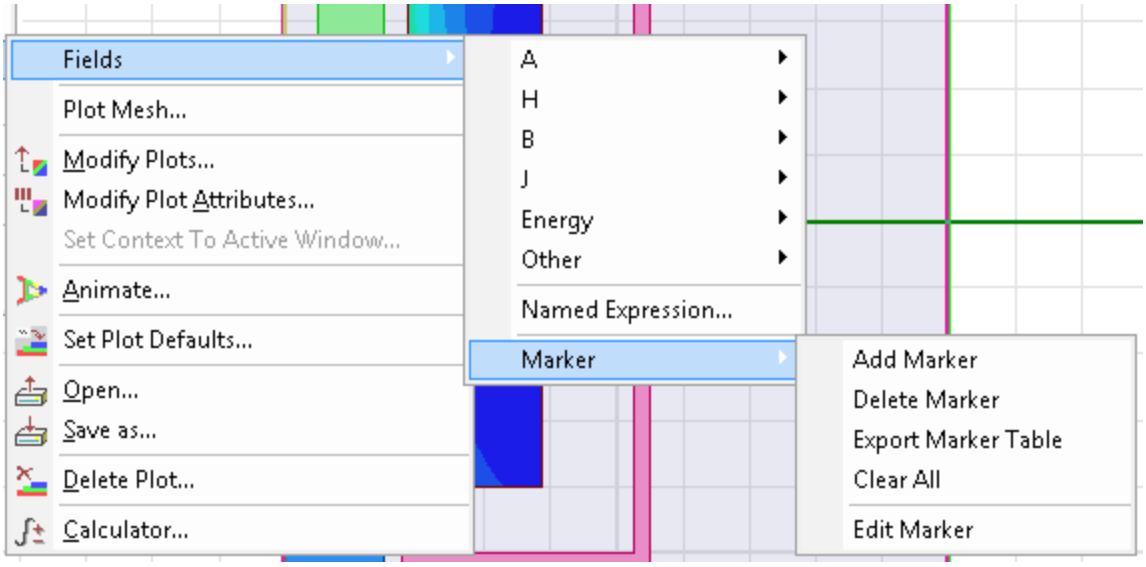
The new plot will be listed in the specified plot folder in the project tree.

Working with Scalar Field Plot Markers

The field overlay plot marker feature enables you to create a marker at selected points in the scalar field overlay plot geometry, and to obtain the field value at that point. The fields Marker sub-menu enables you to:

- Add Marker
- Delete Marker
- Export Marker Table
- Clear All markers
- Edit Marker

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Adding a Field Plot Marker

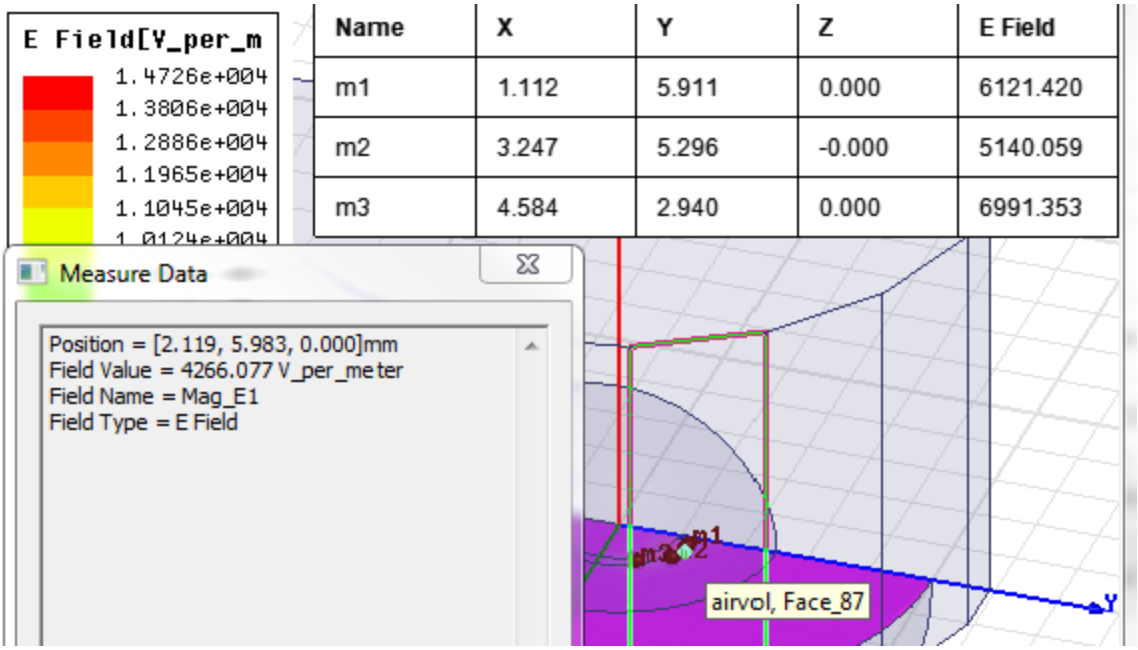
To add one or more field plot markers to a scalar field:

1. On the main menu click **HFSS** and then select **Fields** > **Plot Fields** > **Marker** > **Add Marker**. Alternatively, you can right-click anywhere in the modeler window, or on the Field Overlays folder icon in the Project Manager, and select **Fields** > **Marker** > **Add Marker**.

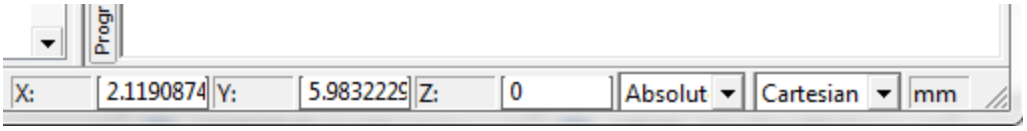
   The *Measure Data* dialog box opens, and a round dot appears at the tip of the cursor.

2. Drag the dot over the spot on the field overlay plot where you want to add a marker. The Measure Data dialog box shows detailed information for the spot currently under the dot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Click the desired point in the field overlay plot to add the marker at that location. Alternatively you can select the position of the marker by entering the values manually in the edit fields at the bottom of the window as shown below.



A table showing the marker coordinates and associated field value is also created and added to the modeler window.

4. Repeat as desired to add additional markers.
5. Press the **Esc** key when finished adding markers.

Deleting a Field Plot Marker

To delete a field plot marker:

1. Click on the marker you want to delete to select it. The row corresponding to the selected marker will be highlighted in the marker table.

   Press and hold the **Ctrl** key and click to select multiple markers.

2. On the main menu click **HFSS** and then select **Fields> Plot Fields> Marker> Delete Marker** to delete the selected marker(s).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Alternatively, you can right-click anywhere in the Modeler window, or on the Field Overlays folder icon in the Project Manager, and select **Plot Fields> Marker> Delete Marker**.
- You can also simply press the **Delete** key to delete the selected marker(s).

Exporting a Field Plot Marker Table

You can export a field plot marker table to either a comma- or tab-delimited file as follows:

1. On the main menu click **HFSS** and then select **Fields> Plot Fields> Marker> Export Marker Table**.

   Alternatively you can right-click anywhere in the modeler window, or on the Field Overlays folder icon in the Project Manager, and select **Plot Fields> Marker> Export Marker Table**.

2. In the *Export As* dialog box, choose the export format, either **.csv** or **.tab**, and save the file in the desired location.

   The exported file can then be imported into another application such as a spreadsheet.

	A	B	C	D	E
1	Name	X	Y	Z	H[A_per_m]
2	m1	2	22.935	1.517	1234
3	m2	1.517	17.373	1.517	123
4	m3	2.488	23.516	1.517	234
5	m4	2.751	23.828	1.517	1324

Clearing All Field Plot Markers

To clear all field plot markers in the active modeler window do one of the following:

- On the main menu, click **HFSS** and then select **Fields> Plot Fields> Marker> Clear All**.
- Alternatively, right-click anywhere in the Modeler window, or on the Field Overlays folder icon in the Project Manager, and select **Plot Fields> Marker> Clear All**.

Deleting all the markers also removes the marker table.

Editing Field Plot Markers

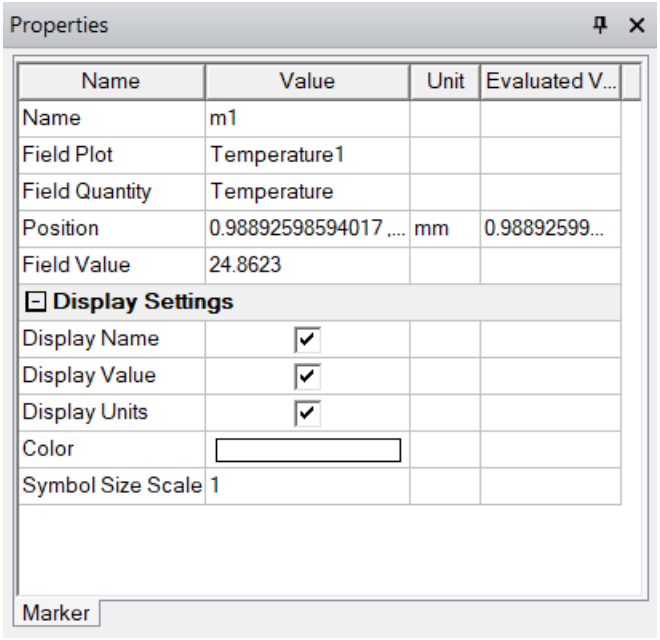
To edit field plot markers:

1. In the marker table, click on the marker you wish to edit. The marker in the 3D Modeler window and its row in the table are highlighted. The properties of the marker are displayed

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in the **Properties** window.

> **Note:**
>
> Press **Ctrl** and select multiple marker rows in the table to edit marker properties simultaneously.



2. You can edit the following marker properties:
   - **Name**: Specify the name that appears next to the marker in the 3D Modeler window.
   - **Display Name**: Toggle to show or hide the marker's name.
   - **Display Value**: Toggle to show or hide the marker's value.
   - **Display Units**: Toggle to show or hide the marker's units.
   - **Color**: Click the color to change the marker's color in the **Color** dialog box.
   - **Symbol Size Scale**: Enter a different number to adjust the size of the marker in the 3D Modeler window.
3. Optionally, when a marker is selected, press the **Delete** key to delete the marker and its corresponding entry in the marker table.

Showing Field Plot Markers

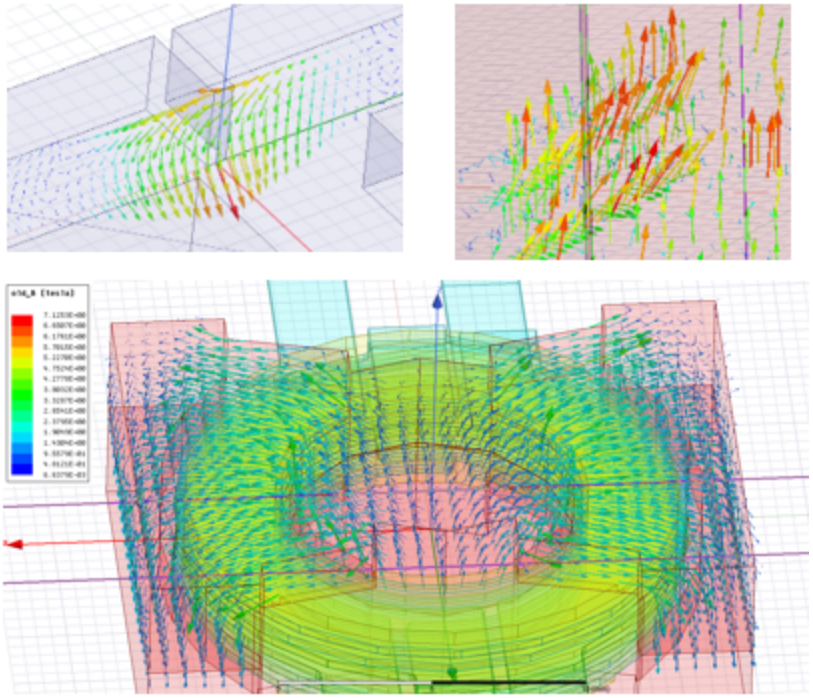
To display field plot markers in the 3D Modeler window, on the main menu click **HFSS** and then select **Fields** > **Plot Fields** > **Marker** > **Show Marker**. The table showing the marker coordinates and associated field value is also displayed in the modeler window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternatively, you can right-click anywhere in the modeler window, or on the Field Overlays folder icon in the Project Manager, and select **Plot Fields** > **Marker** > **Show Marker**.

Creating Vector Field Plots

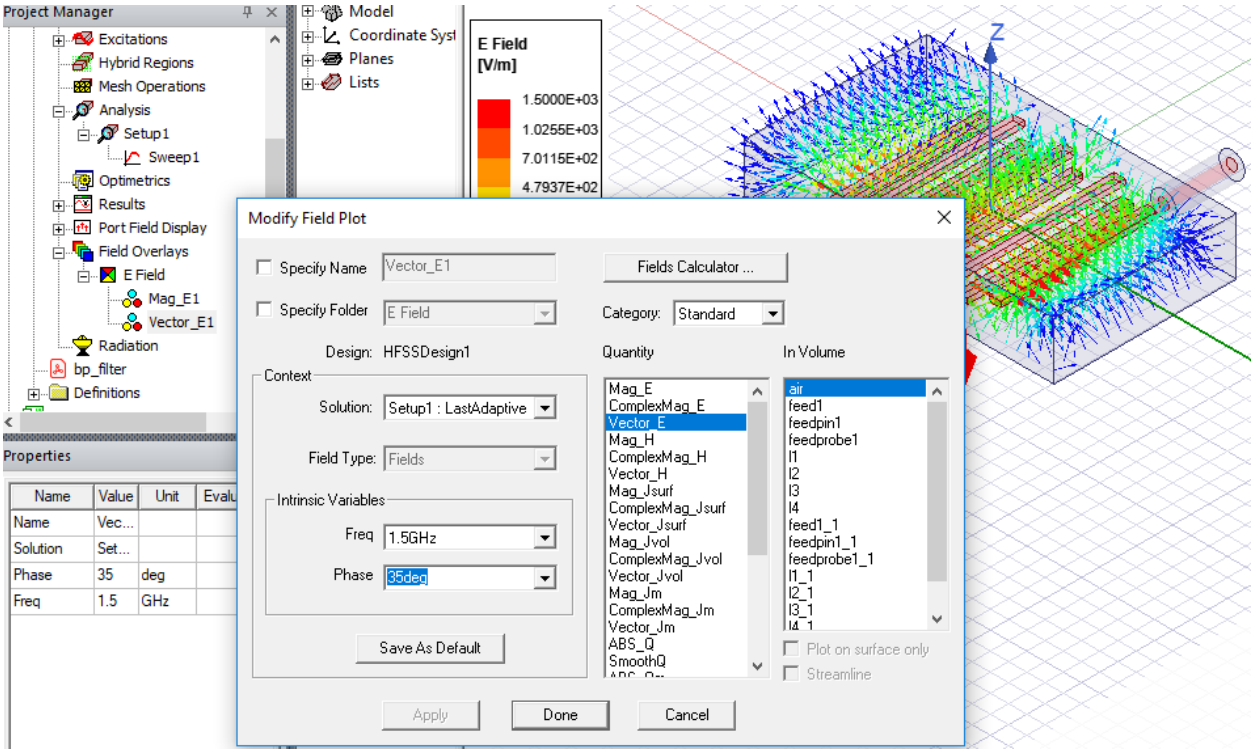
A vector plot uses arrows to illustrate the magnitudes of the x-, y-, and z-components of field quantities. Vector plots can be created on surfaces or volumes. The vector plots calculates a good initial spacing (min, max, and spacing range). The initial min, max spacing is based on a percentage of the plot geometry size and they are in model units. You can edit the vector min and max threshold spacing and the arrow representation.



1. Do one of the following:
    a. To create a vector surface plot, select the faces on which you want to plot the fields.
    b. To create a vector volume plot, select the objects within which you want to plot the fields.
2. Click **HFSS** > **Fields** > **Fields**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

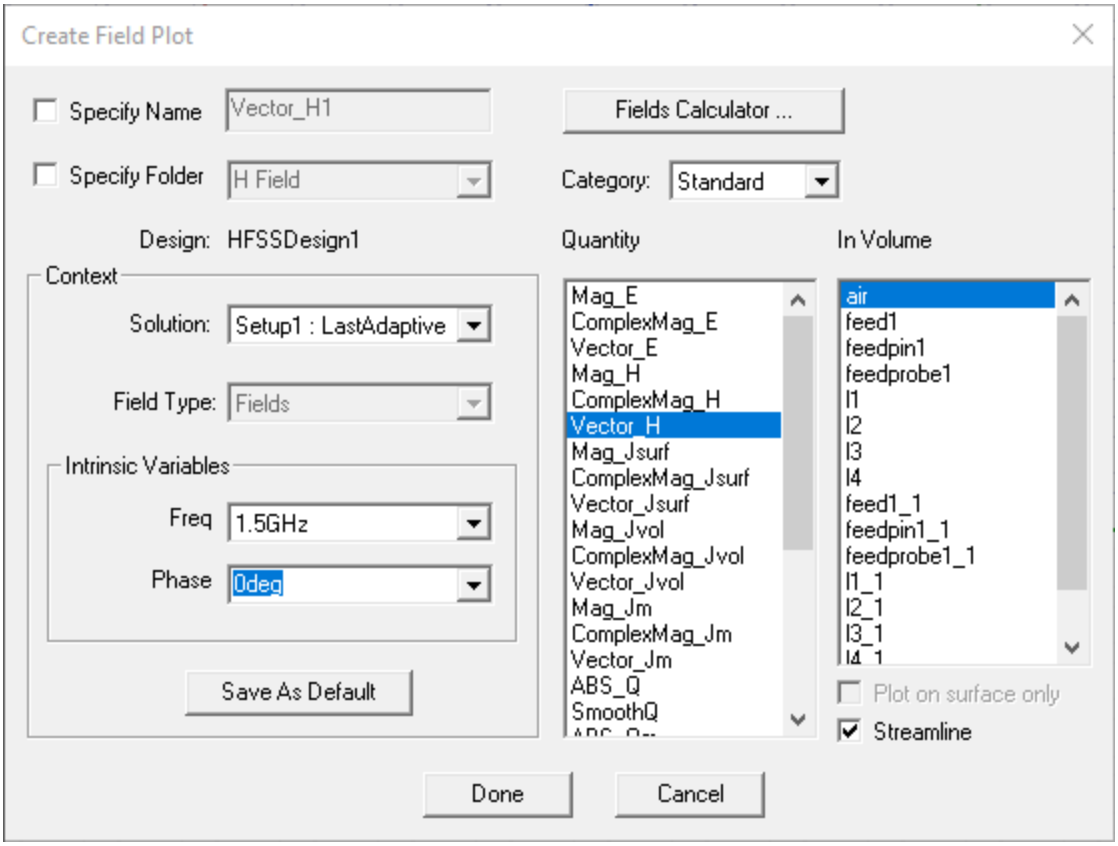
3. From the **Fields** submenu, click the vector field quantity you want to plot.



4. Follow the procedure for plotting field overlays.

If you select a vector quantity when creating a plot, you can also check Streamline for the plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

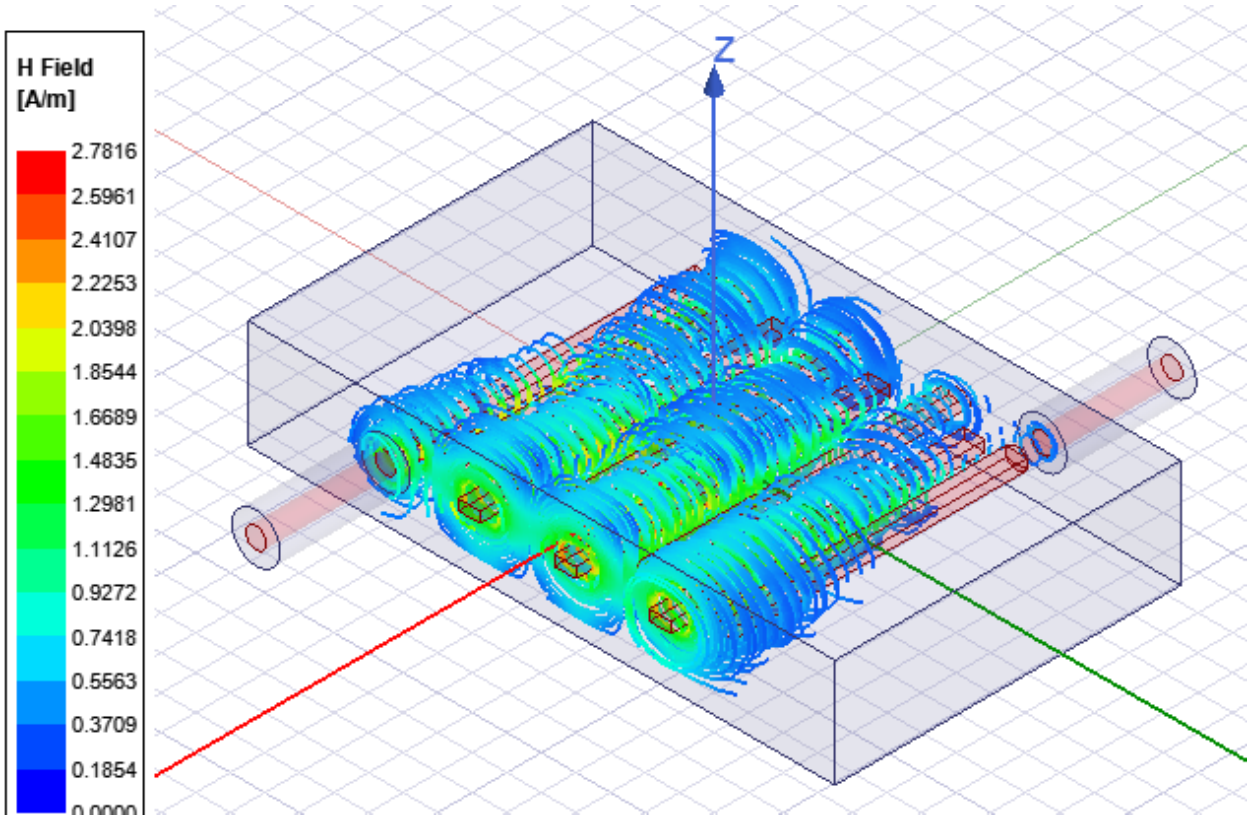


Notes for creating Streamline plots:

1. Before creating the plot, select both starting and ending edges (in 2D) or surfaces (in 3D).
2. In the Creating Field Plot menu, select "In Volume: Region" which is the volume in which the streamlines will appear and is outside of the sources.
3. After the plot is created, on the Attributes/Plot tab reduce "Seeds density" to show more streamlines. If no streamlines appear, reduce this by a factor of 10 (or 100) because the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

default seeding was too large.



Modifying Field Plots

1. Click **HFSS**> **Fields**> **Modify Plots** , or in the Project Manager, select the **Field Overlays** icon, right-click, and select **Modify Plots**, or press the **M** hotkey.

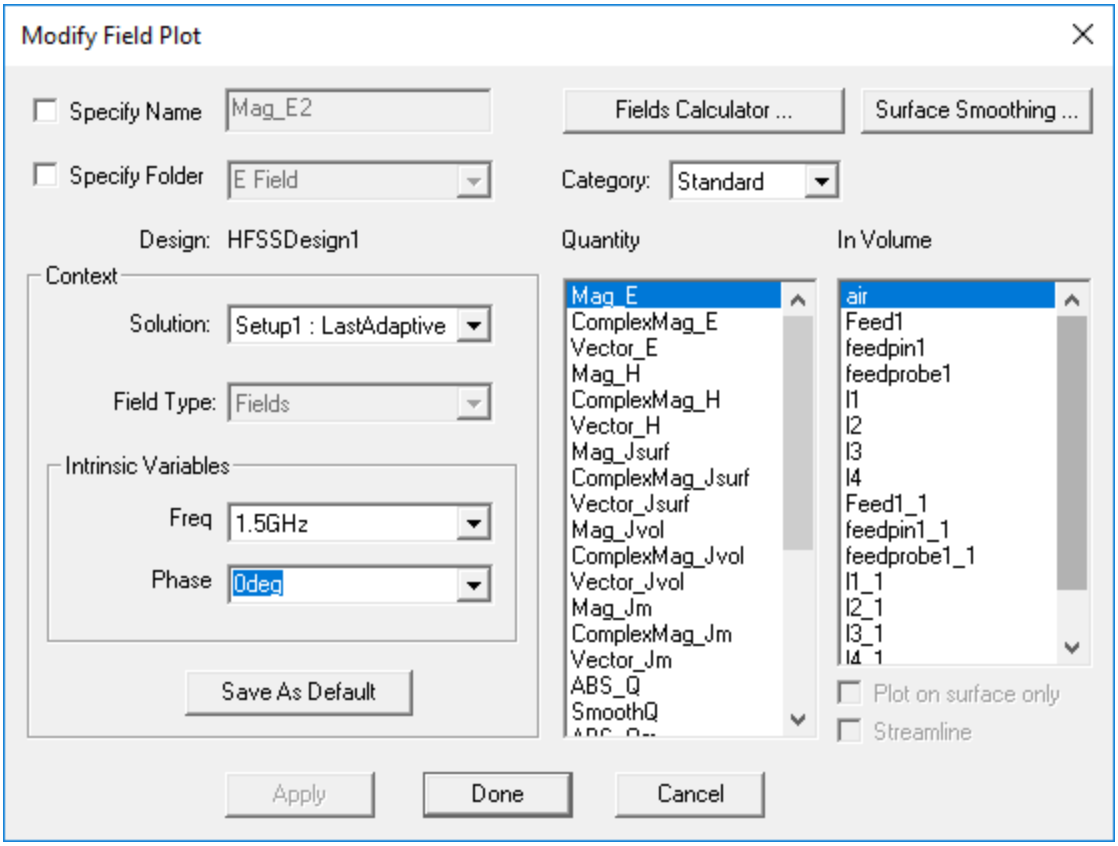
   The **Select Field Plot(s)** dialog box appears.

   Optionally, you can right-click an existing plot listed under the Field Overlays in the Project tree, bypass the **Select Field Plot** dialog box and go to step 3.

2. Select the plot you want to modify in the **Select** column, and then click **OK**.

   The **Modify Field Plot** dialog box opens.

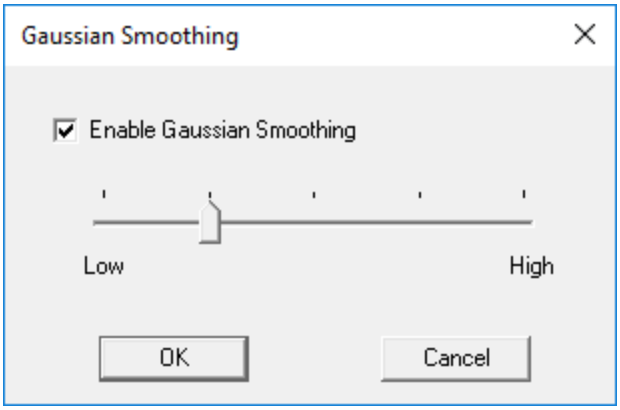
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Optionally, select the **Specify Name** check box to enable the name field.
- Optionally, select the **Specify Folder** check box to enable the drop-down menu.
- Optionally, select a different **Solution** from the drop-down menu.
- Optionally, select a different **Field Type**, if available.

3. Under **Intrinsic Variables**, specify the frequency and phase at which the field quantity will be evaluated.
4. Optionally, select a different field quantity to plot from the **Quantity** list.
   - To choose a calculated expression, select **Calculator** from the **Category** drop-down menu. If you choose Calculator, click the Fields Calculator button to display the calculator.
   - To choose a predefined field quantity, select **Standard** from the **Category** drop-down menu. Select from the Quantity list.
   - For scalar quantities plotted on the surface of a geometry, the **Surface Smoothing** button is enabled. For vector quantities, the **Surface Smoothing** button is disabled. Surface Smoothing applies a selected level of smoothing based on the Gaussian Blur algorithm. Clicking Surface Smoothing opens the Gaussian Smoothing dialog box.

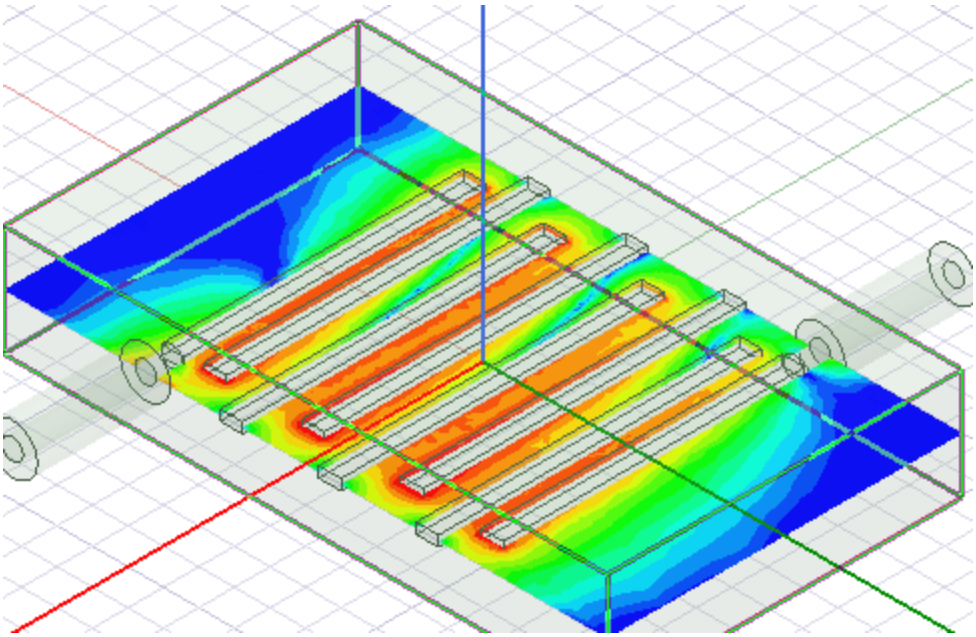
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Select the check box to enable the feature and use the slider to set the degree of smoothing.

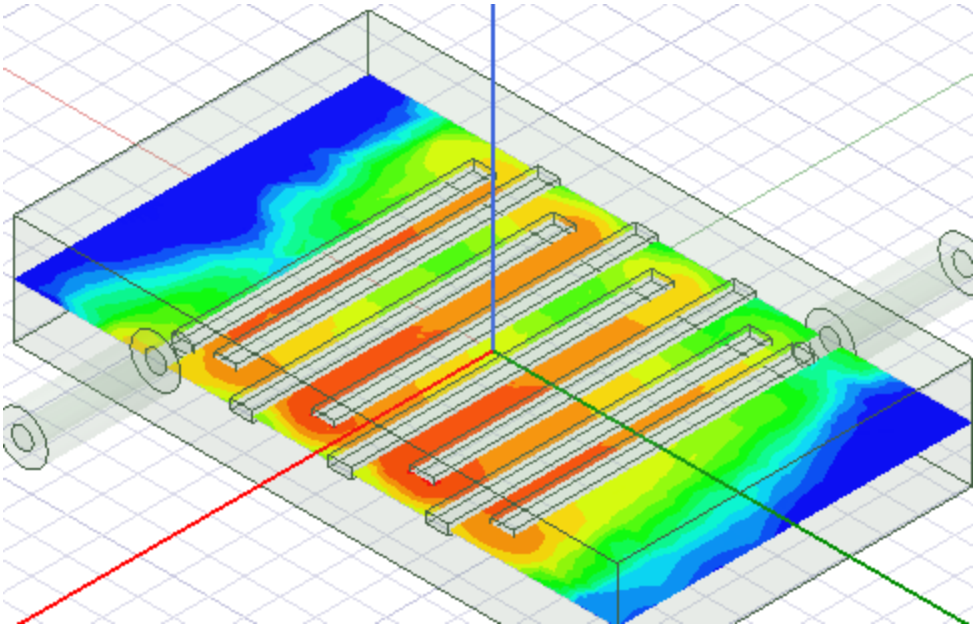
Important: You *must* click **Apply** in the *Modify Field Plots* dialog box to see the changes.

For example, this is a Mag_E fringe plot before applying the smoothing:



This is a Mag_E fringe plot after applying High smoothing:

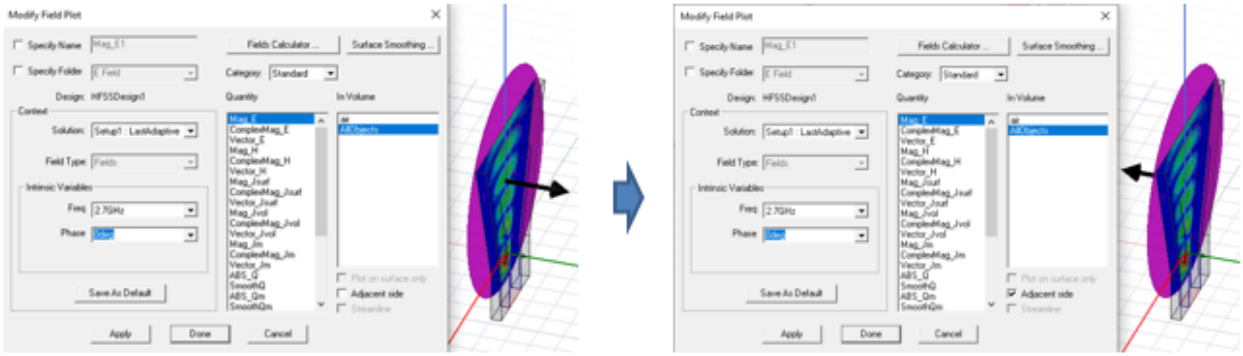
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Select the volume, or region, in which the field will be plotted from the **In Volume** list.

   This selection enables you to limit plots to the intersection of a volume and the selected object.

6. If you are modifying a plot on a sheet object, the **Adjacent** check box allows you to generate field overlays on a sheet object using data from either side of the sheet. An arrow indicates where the data is coming from. While the data is usually the same on the two sides of a sheet object, in a situation where the sheet object is assigned to be a shell element, the field may be different on its two sides. (Shell elements are an option for Layered Impedance Boundaries and Finite Conductivity Boundaries with DC conductivity and two-sided selected.) In such cases, you can visualize the field data on the side you want.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Click **Apply** to make the changes and leave the dialog box open, or click **Done** to apply the changes and close the dialog box.

   The field quantity is plotted on the surfaces or within the objects you selected. The modified plot is listed in the specified plot folder in the project tree.

   The plot uses the attributes specified in the **Plot Attributes** dialog box. For details see Setting Field Plot Attributes.

Setting Field Overlay Attributes

After creating a mesh or field overlay on a surface or volume, you can modify its appearance by changing the settings in the **Plot Attributes** dialog box. You will modify the settings for a plot folder and all plots in that folder will use the same attributes.

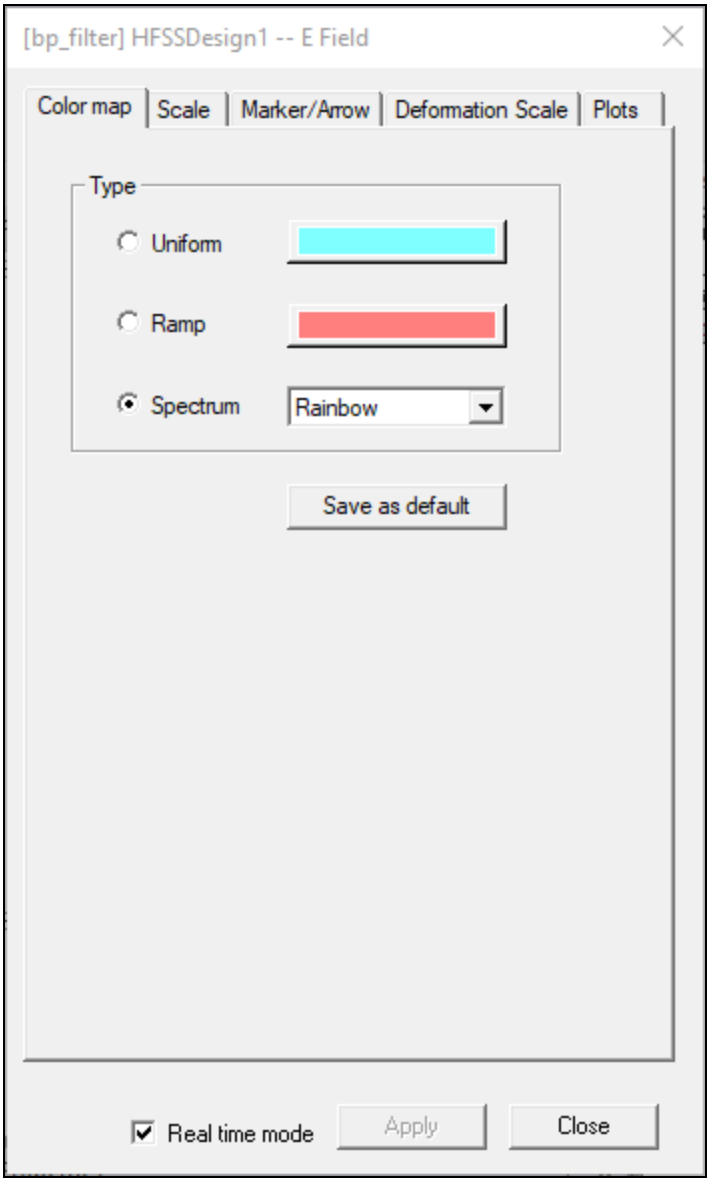
1. Click **HFSS** > **Fields** > **Modify Plot Attributes**. Alternatively, right-click **Field Overlays** in the Project Manager and select **Modify Attributes** from the shortcut menu.

   The **Select Plot Folder** dialog box appears.

2. In the **Select Plot Folder** dialog box, select the plot type that you want to modify and click **OK**. You can also right-click the specific plot in the Project Manager and choose **Modify Attributes** from the shortcut menu.

   A dialog box with attribute settings for the selected plot type (whether a field result or a mesh plot) appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. For field result overlays (not mesh plots), the available plot attributes under each tab in the dialog box are listed in the following table:
    - **Color map:**

      The number of colors used and how they are displayed. The field data must be available for the color key to appear.

    - **Scale tab:**

      The scale of field quantities, including the number of divisions in the scale, whether to use dB as the units, whether to use a linear or log scale, auto schall options, and plot number format.

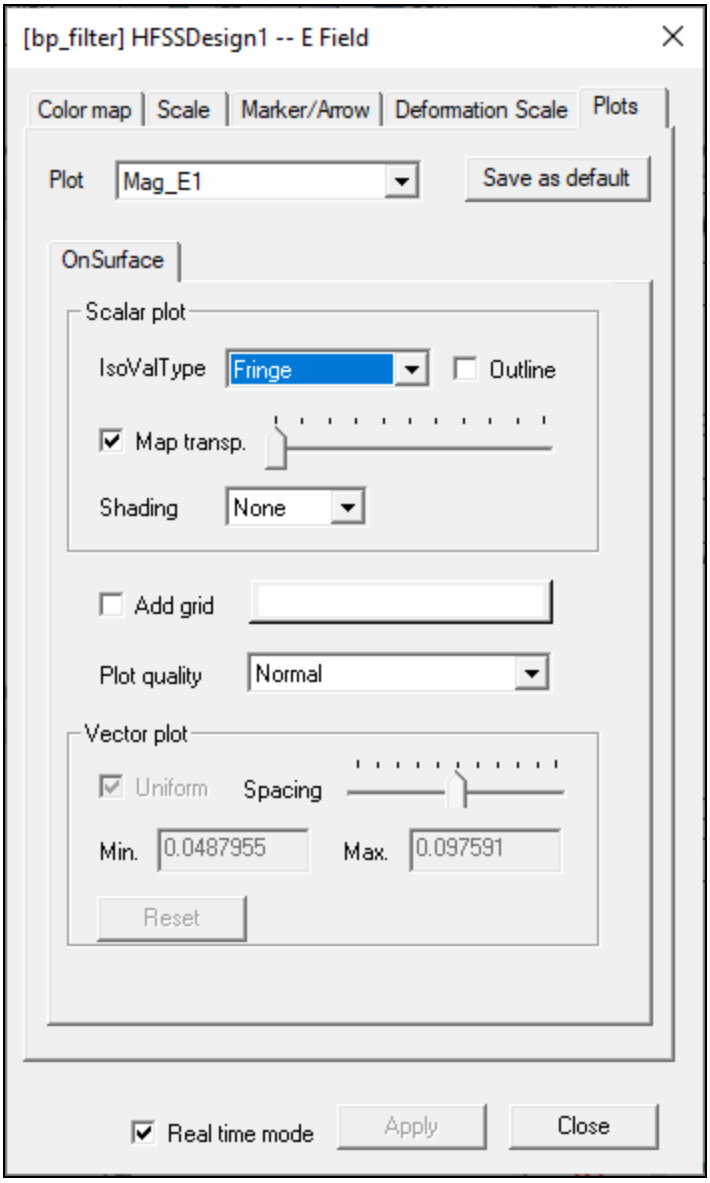
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Marker/Arrow**
  - The appearance of points (for scalar point plots).
  - The appearance of arrows (for vector plots).
  - Magnitude filtering (for vector plots). That is, you uncheck Map size, and specify a Min and Max Magnitude, or use a slider to set the Min threshold.
- **Deformation Scale:**

  This is for use with plots that include Stress feedback from Ansys Workbench Integration.

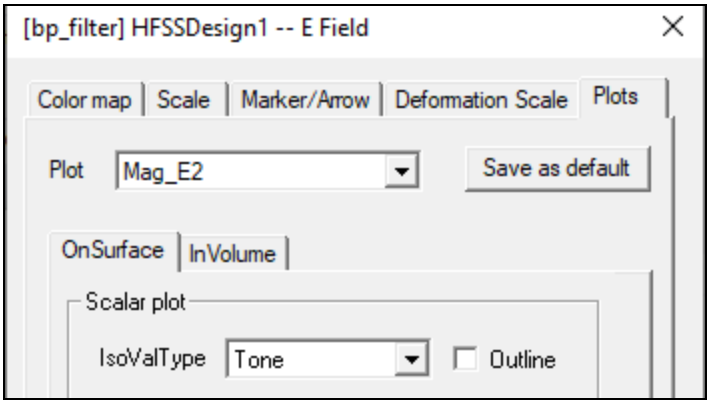
  See Modify Plot Attributes Dialog for Stress Feedback Projects.
- **Plots (if not vector or streamline):**
  - The **Plot** selected. If multiple plots are available, you can select from a drop down menu. Plots can be OnSurface or InVolume. If both types are available, the dialog includes tabs for each.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



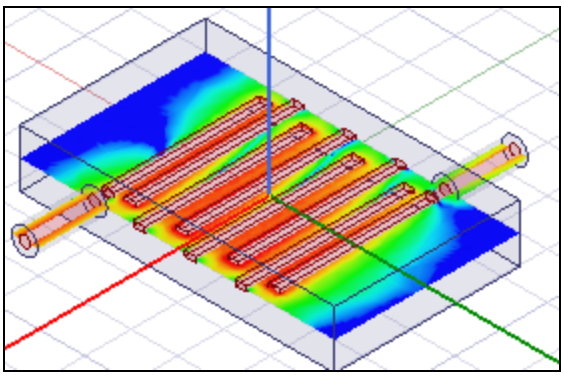
- Plots can be **OnSurface** or **InVolume**. If both types are available, the dialog includes tabs for each.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



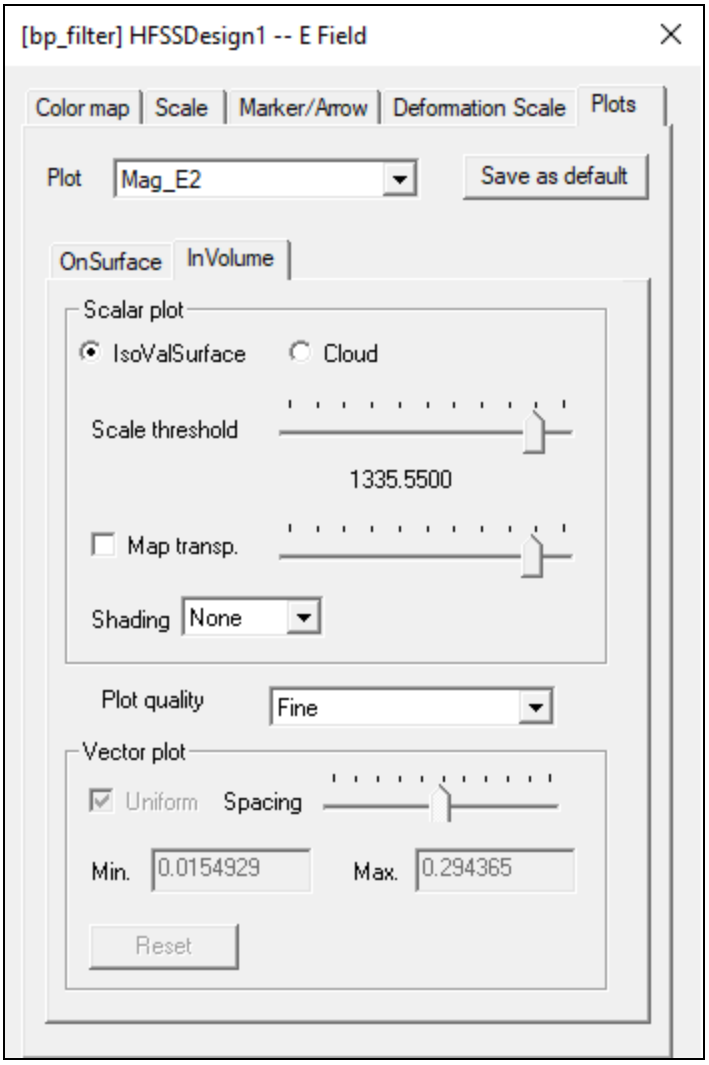
- The type of isovalue display (for scalar plots.) For *Line, Fringe* and *Tone* **IsoValType**, *Outine* is enabled. For *Gourad*, it is disabled. The *IsoValue* display selection also affects *InVolume*, which also has other options discussed below.
- Whether to use Shading (for scalar plots), if lighting is turned on. By default, the **Shading** is set to *None* which is equivalent to "Do not use lighting."
- The transparency based on solution value.
- Whether to add a grid (that is, a mesh overlay), and to set the grid color.
- Specify the plot resolution as **Coarse**, **Normal**, **Fine**, or **Very Fine**.

  This affects the use of memory for animating plots. For large plots with more frames to animate, use **Coarse** or **Normal** to reduce memory requirements and improve performance. For smaller plots with few frames, if higher resolution is required, use **Fine** or **Very Fine**.



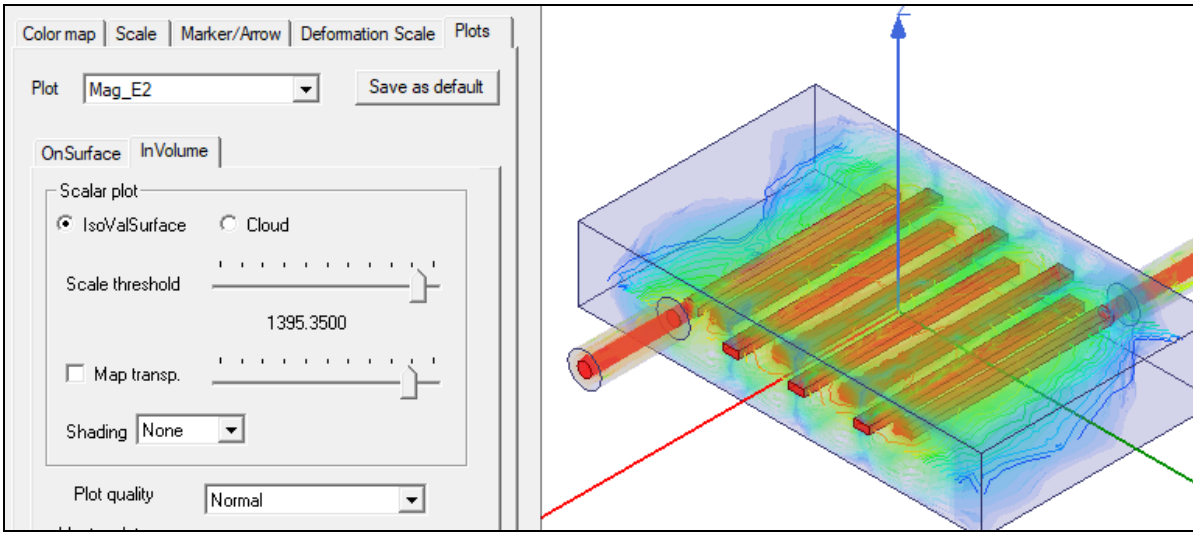
If an InVolume plot is available, and you select the tab, additional display options are available to allow visibility of internal layers and objects, particularly those with high field values. For *IsoValSurface* for an *InVolume* Scalar plot, the choices are for **Scale threshold** and **Map transp**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

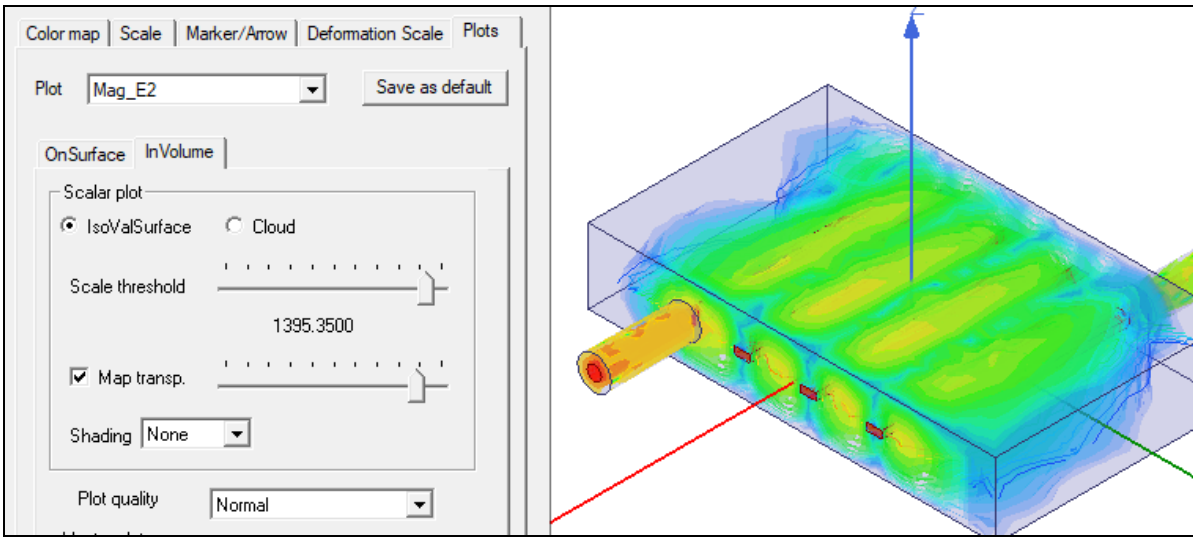


- The "**Scale threshold**" slider bar allows you to set a field value threshold so that iso surfaces with field value lower than the given threshold will show as translucent with given transparency specified by the slider bar below it.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



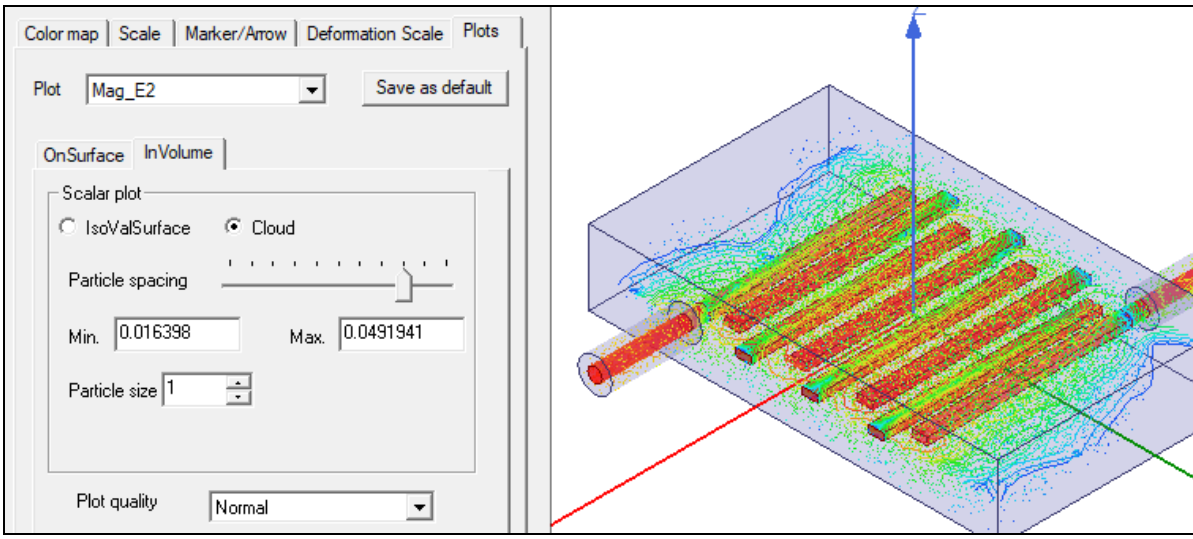
- Iso surfaces with field value higher than the threshold will show as opaque. The range of the "**Scale threshold**" is the same as the color key range from *Min* to *Max*.
- When "**Map transp.**" is checked, translucent iso surfaces field values map to [0.0, transparency] so lower field value map to higher transparency while higher field value to lower transparency. If it is unchecked, all translucent iso surfaces will have the same given transparency from the slider bar.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> When the **Map transp.** slider bar is at the left most position, transparency is 0. This feature is essentially disabled, and the "**Scale threshold**" slider bar will be disabled. Also, high fidelity transparency should be enabled for best results.

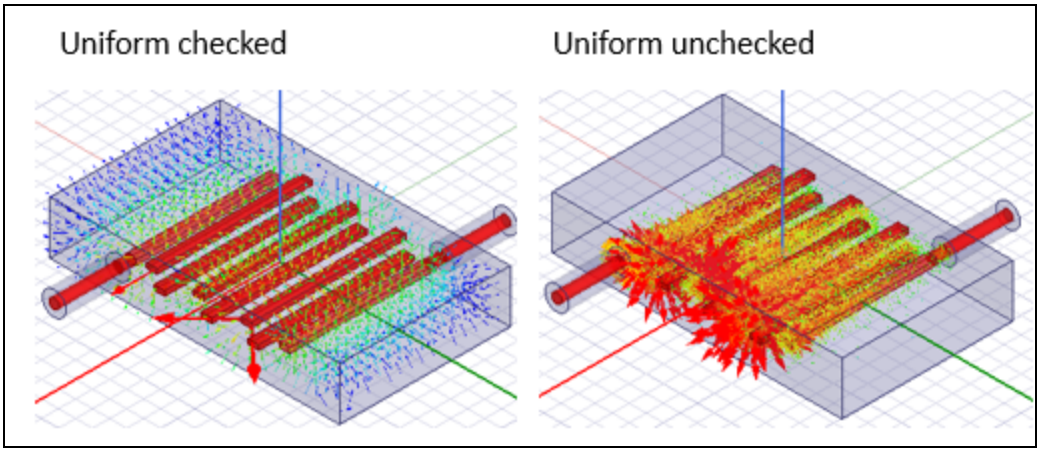
- For *Cloud* plots, field values are represented by points that illustrate the spatial distribution of the solution. You can use the **Particle spacing** slider bar to adjust the plot. The higher the solution value, the greater the cloud density. You can also specify **Min** and **Max** values (numeric values based on the active model length unit) and adjust **Particle size**.



- **Plots (for vector plots):**
  - The **Plot** selected.
  - The **Min** and **Max** spacing of arrows (for vector plots). These values are based on the model's active length unit and are only relevant when the **Uniform** option is selected.
  - **Plot quality**, as *Normal, Coarse, Fine,* or *Very Fine*.
  - **Uniform** vector spacing, based on the *Spacing* slider position and the *Min* and *Max* values.

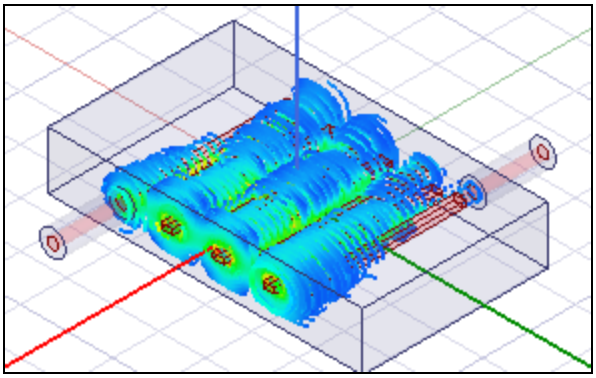
    When Uniform is *not* selected, a vector is rendered at every node (both corner and mid-side nodes of the elements).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Reset** button resets a good initial spacing (and thus Min, Max spacing range) without the need to recreate the field plot.

- **Plots (if streamline is checked):**
  - The **Plot** selected
  - The **Line style** as *solid* or *cylinder* from drop-down menu.
  - **Line width**, specified using a slider.
  - Whether to show *Marker* on *Streamline*.
  - **Seeds density** spacing. This affects the number of stream lines used to represent the quantity in the plot. Moving the slider to the left decreases the spacing and increases the number of stream lines. Moving the slider to the right increases the spacing and decreases the number of lines used to represent the quantity.
  - **Min**. and **Max**. values represented (in *Magnitude filtering* settings).



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Under each tab, click **Save as default** if you want the tab's settings to apply to field overlay plots created after this point.
b. Select **Real time mode** if you want the changes to take effect immediately in the view window.
c. If this option is cleared, click **Apply** when you want to see the changes.

Modifying Field Plot Colors

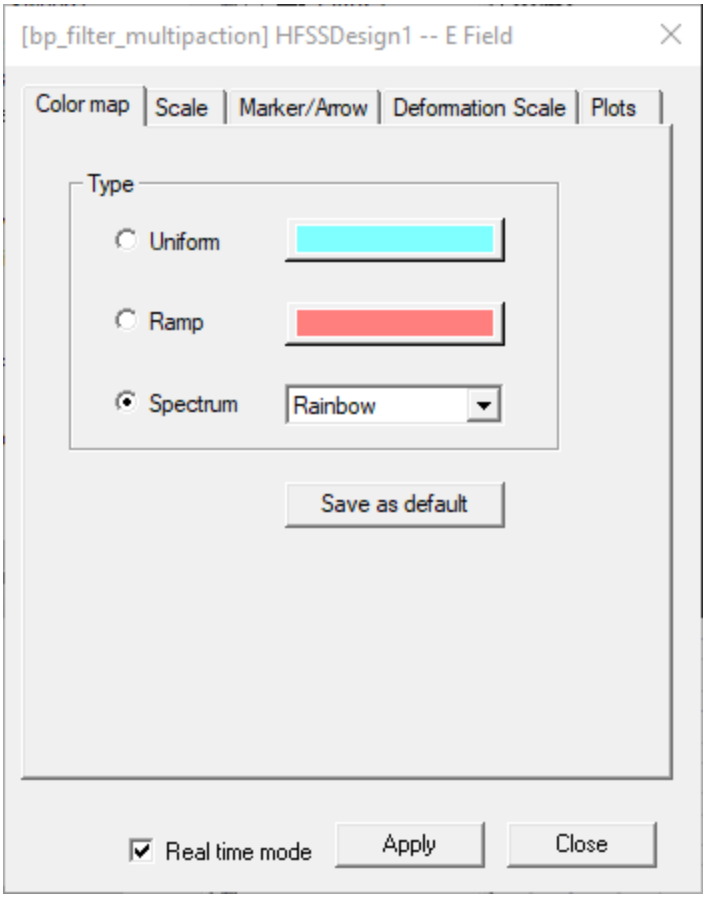
1. Click **HFSS** > **Fields**> **Modify Plot Attributes** or, in the Project Manager, right-click **Field Overlays** and select **Modify Plots** from the short-cut menu (or use the "m" hotkey).

   The **Select Plot Folder** dialog box appears.

2. Select the plot folder you want to modify and click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

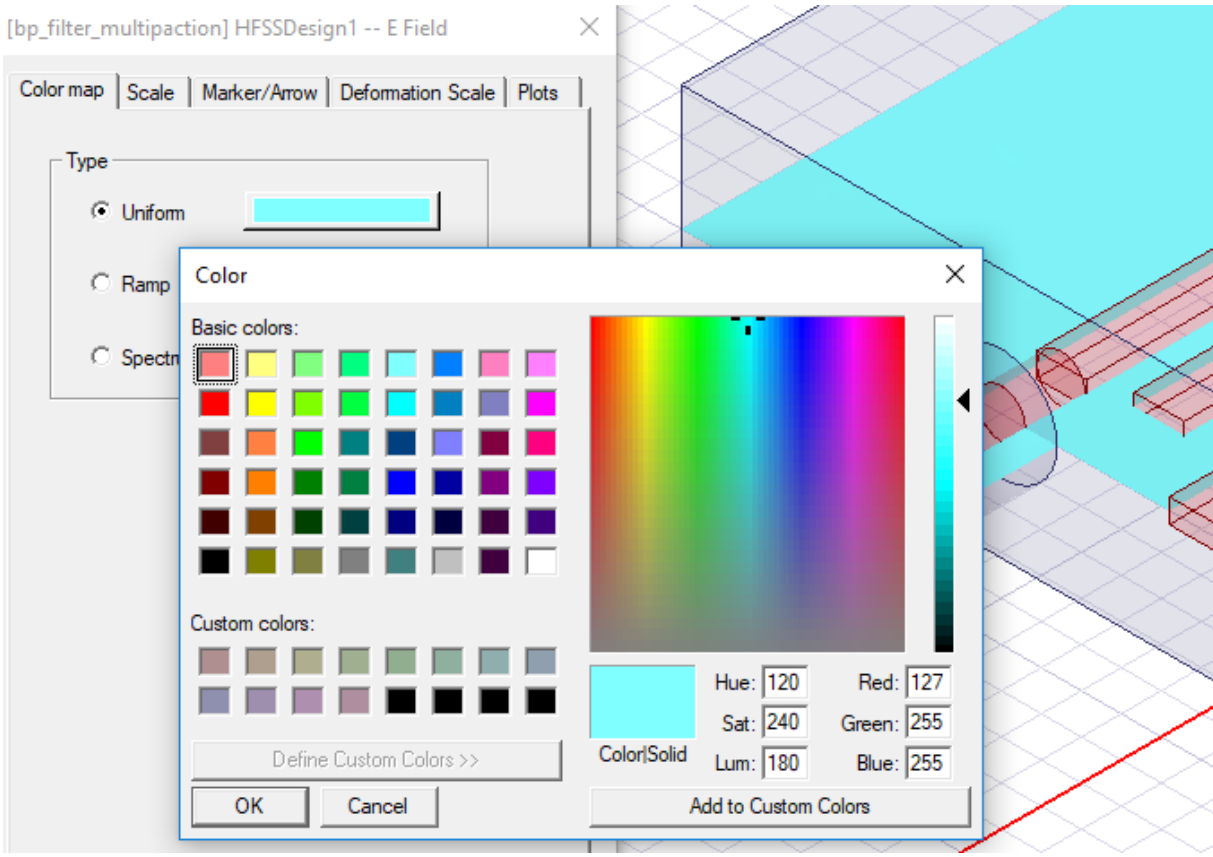


3. Click the **Color Map** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

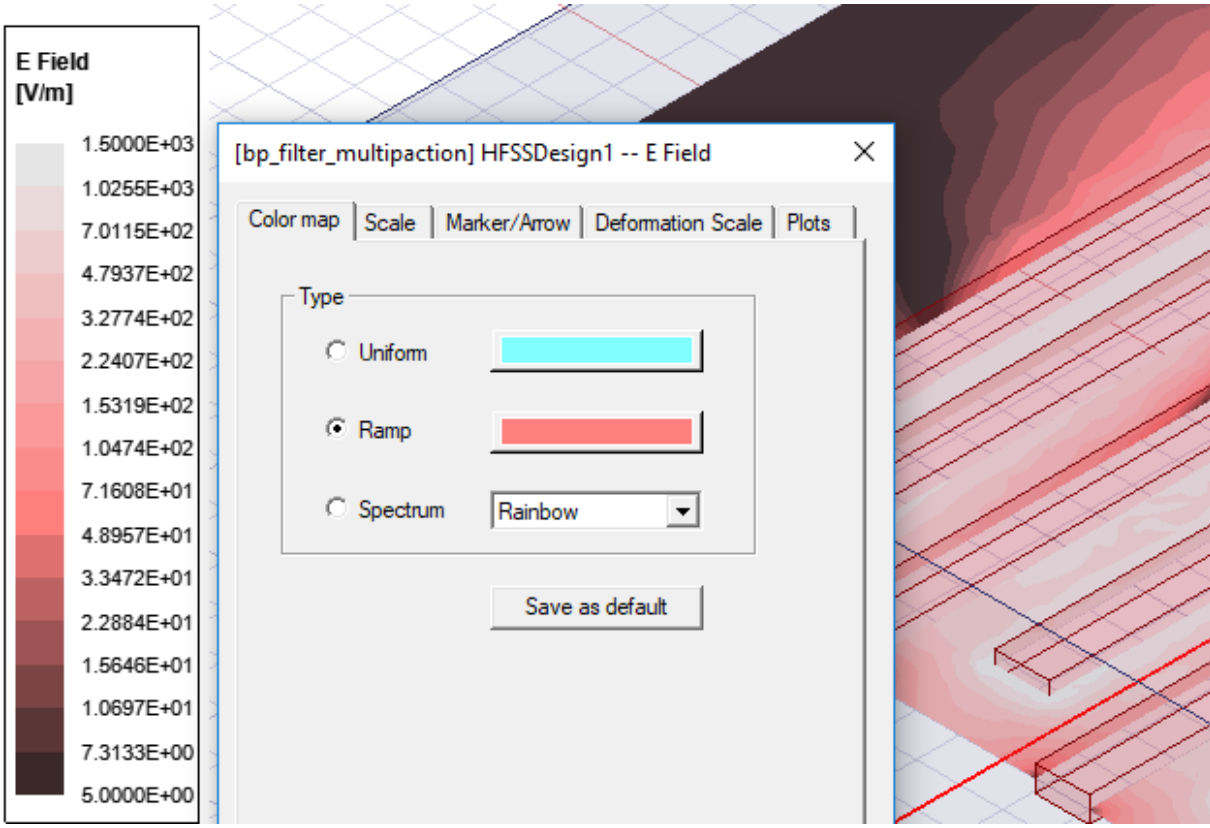
4. Select one of the following color types:

Uniform Field quantities are plotted in a single color. Click the button to choose the plot color from the **Color** button palette.



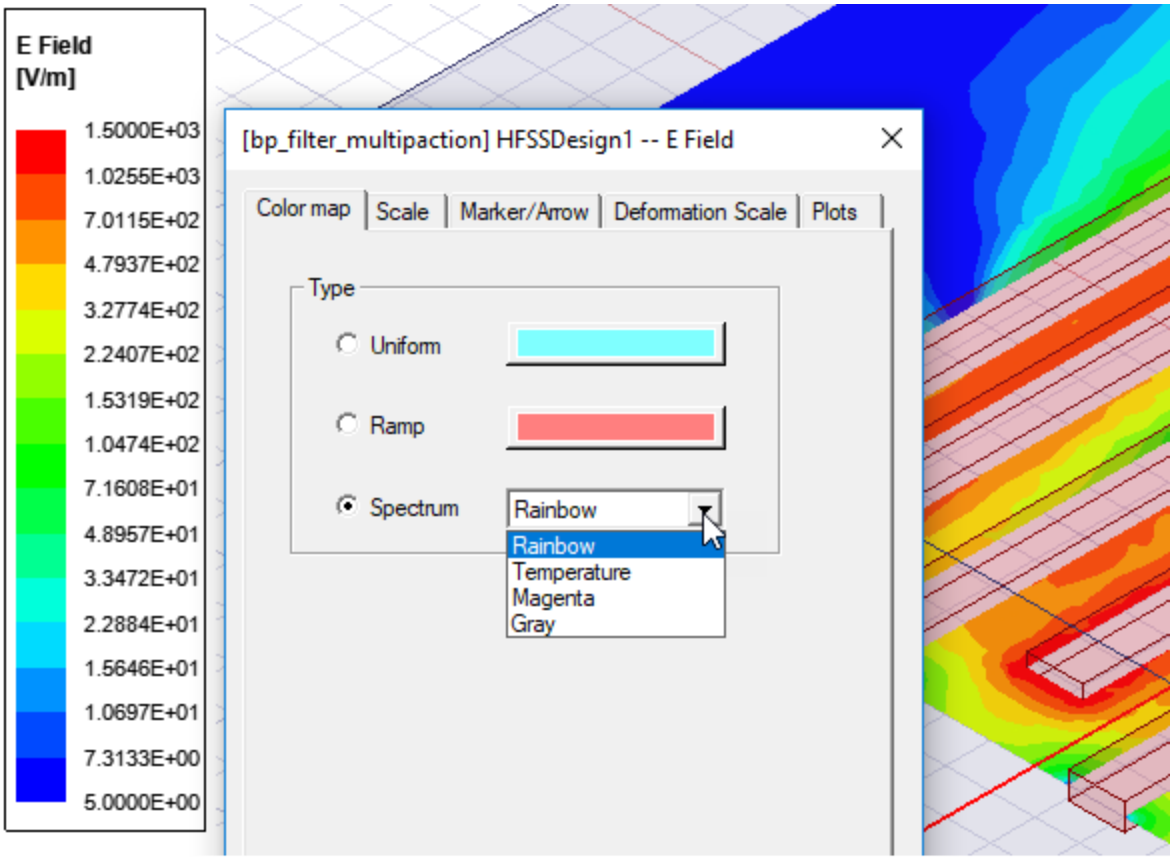
Ramp Field quantities are plotted in shades of a single color. Choose the plot color from the **Color** palette. The shade of the color corresponds to its field value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Spectrum Field quantities are plotted in multiple colors. Choose a color spectrum from the drop-down menu. The values are Rainbow, Temperature, Magenta, and Grey. Each field value is assigned a color from the selected spectrum.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can choose **Save as Default**, if you want to use the current settings.

Select **Real time mode** if you want these, or subsequent changes to take effect immediately in the view window.

If this option is cleared, click **Apply** when you want to see the changes.

5. Click the **Scale Tab**.
6. In the **Num. Divisions** field, enter the number of colors to use in the plot.

   You can choose **Save as Default**, if you want to use the current settings.

   Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.
7. Click **Close** to dismiss the dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color Key Settings

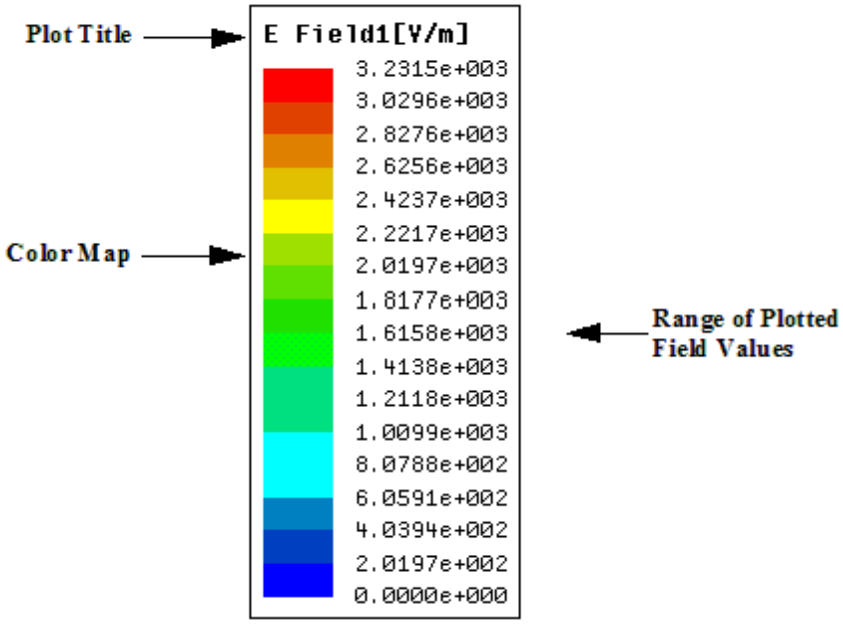
The color key (shown below) displays the range of plotted field values for a field overlay plot. It displays the colors that correspond to the range of field values on the plot. You can click and drag the color key to move it. In the case of no field data, the color key does not appear.

1. Click **View**> **Visibility**> **Active View Visibility**.

   The **Active View Visibility** dialog box appears.

2. Click the **Color Keys** tab.
3. In the **Visibility** column, select the field overlay or mesh plots in which you want to display the color key. Clear the plots in which you want to hide the color key from view.
4. Click **Done** to dismiss the dialog box.

Alternatively, to hide the color key, right-click on the color key in the active view window and choose **Hide** from the shortcut menu.



Color keys will only be visible for the plots that are selected in the **Active View Visibility** dialog box. If a hidden color key is shared by two or more overlay plots (such as a Heat Flux key, which is shared by heat flux magnitude and heat flux vector plots), then the color key will be hidden in all plots that share it. The **Active View Visibility** dialog box only lists a single incidence of a particular key, regardless of how many overlay plots share it. Use this dialog box to restore visibility of hidden color keys.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modifying the Field Plot Scale

To change how field quantities are scaled on the field overlay plot:

1. Click **HFSS** > **Fields** > **Modify Plot Attributes** or, in the Project Manager, right-click Field Overlays and select **Modify Plots** from the shortcut menu (or use the "**m**" hotkey).

   The **Select Plot Folder** dialog box appears.

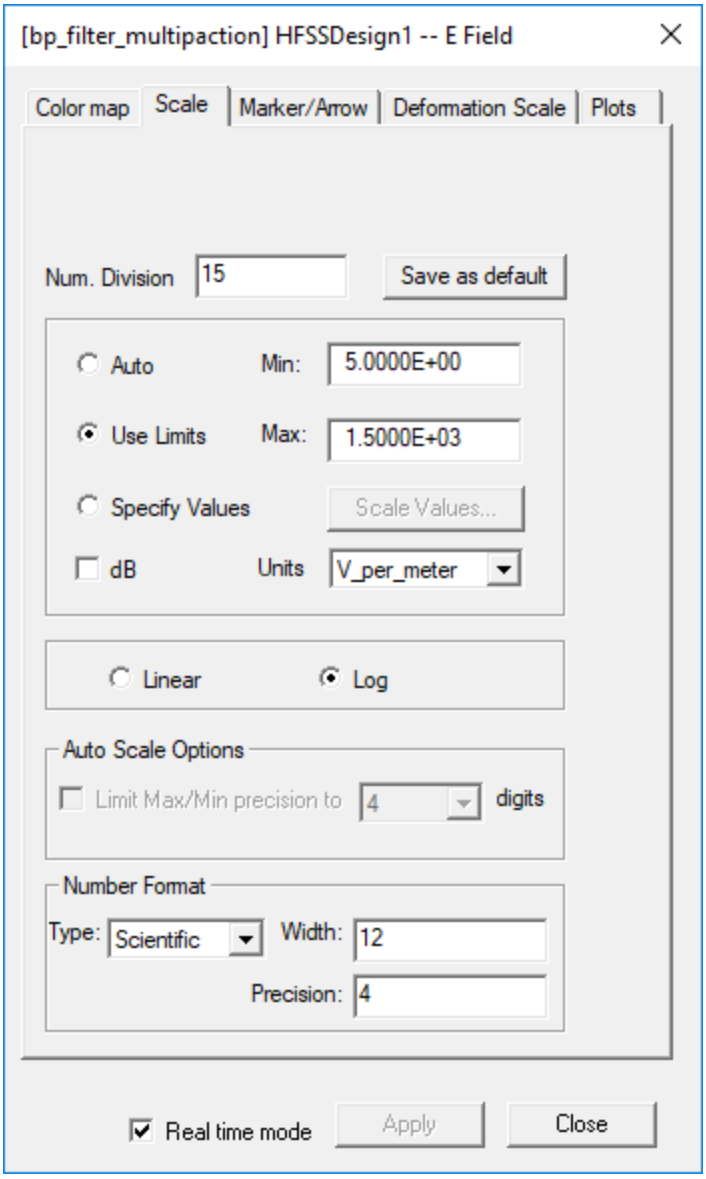
2. Select the plot folder you want to modify and click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

3. Click the **Scale** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Optionally, to change the number of divisions in the field plot scale, set the **Num. Division** field to a new value. You can click **Save as Default**, if desired.
5. Select one of the following scale options:

Auto The full range of field values will be plotted on the selected surface or volume. Selecting **Auto** enables the **Auto Scale Options** and disables the **Min** and **Max** fields. By default, precision is not limited and auto-min is the actual computed min on the plotted geometry.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use Limits	Only the field values between the minimum and maximum values will be plotted. Field values below or above these values will be plotted in the colors assigned to the minimum or maximum limits, respectively. Selecting **Use Limits** enables the **Min** and **Max** fields and disables the **Auto Scale Options**. Field values have a precision of at most 6 decimal places (field solution files are saved in floating precision), so Min/Max numbers are displayed to this precision.
Specify Values	This enables a **Scale Values** button.

Optionally, when dB is checked, dB scale is used for the plot. It disables the Units field, Linear and Log and have "Linear" selected.

6. Optionally, use the **Units** drop-down menu to select the default unit of measure for the plot.

   The units specified here appear on the Color map for the fields plot, and for the properties dialog for the field quantities.

7. If you selected **Use Limits**, enter the lowest field value to be plotted in the **Min.** text box and the highest field value to be plotted in the **Max.** text box.

   If you selected **Auto** or **Use Limits** without having dB checked, the **Auto Scale Options** are enabled. You should only changed for cases where auto-min is a small number. Use the 'Limits Max/Min precision to' check box to enable setting the drop-down menu for the precision limit.

   The auto-min is the greater of the following:

   - Actual computed Min
   - Max/pow(10, num digits of field precision)

   If you selected Specify Values, you can click the **Scale Values** button. This opens a dialog with an editable, scrollable list of the current scale values. To apply the changes you make, click **OK**. To close the dialog box without making changes, click **Cancel**.

8. If you selected **Auto** or use **Limits** or **dB**, you can select one of the following options:

Linear	Field values are plotted on a linear scale.
Log	Field values are plotted on a logarithmic scale. If field plots have negative and positive values and when auto-scale is selected, the log-scale choice automatically sets the Min value as the Max/Min Ratio. (If field plots have all negative values, **Log** is not allowed.)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

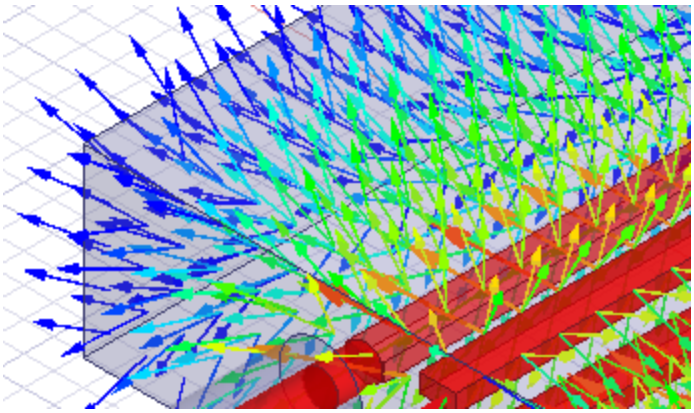
9. Specify the Number Format for the plot as Auto, Scientific Notation, or Decimal. You can also specify Width and Precision for the plot.
10. Select **Real time mode** if you want the changes to take effect immediately in the view window.

    If this option is cleared, click **Apply** when you want to see the changes.
11. Optionally, you can use the **Save As Default** button to save the following to the registry:
    - Whether to limit field precision
    - The number of digits of field precision
    - Whether to use log/linear scale
    - Whether to use dB

    Auto scale is the default for new plots. For scalar-in-volume plots, iso-surface (rather than cloud) is the default display
12. Click **Close** to dismiss the dialog box.

Modifying Vector Field Plot Arrows

To change the appearance of a vector field plot's arrows:



1. Click **HFSS**> **Fields**> **Modify Plot Attributes** or, in the Project Manager, right-click on **Field Overlays** and select **Modify Plot Attributes**. Alternatively, under **Field Overlays** in the Project Manager, right-click a subfolder (overlay type) heading and select **Modify Attributes**.

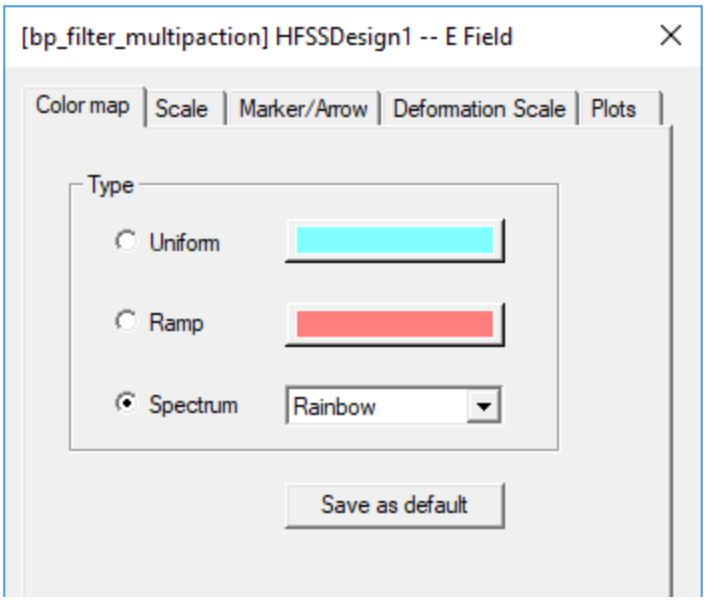
   If you select Modify Plot Attributes, the **Select Plot Folder** dialog box appears. If you select Modify Attributes, the appropriate attributes dialog box appears, bypassing the **Select Plot Folder** dialog box. In this case, skip to step 3.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

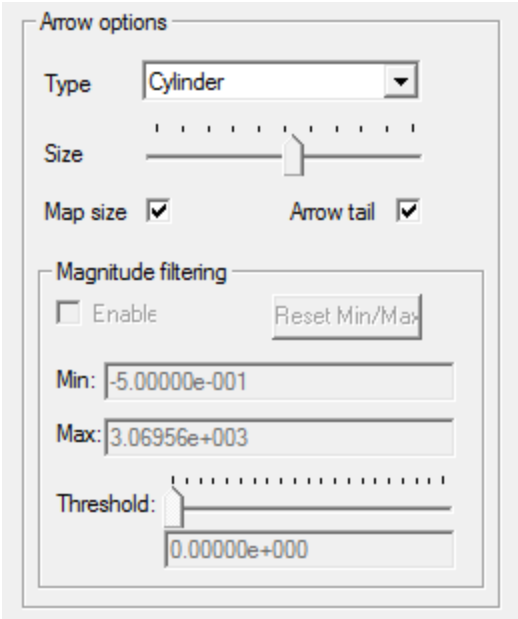
2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.



3. Click the **Marker/Arrow** tab.



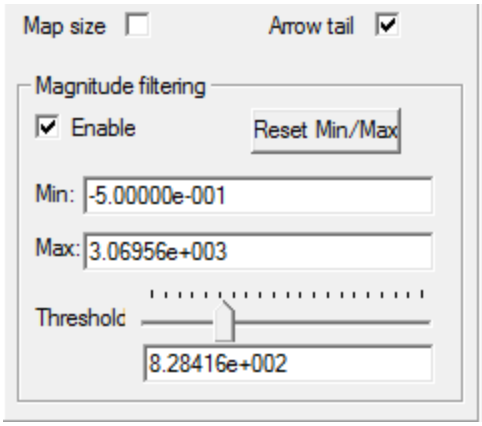
4. Under **Arrow Options**, select one of the following arrow types:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Line	The arrows are displayed as 2D/flat.
Cylinder	The arrow tails are displayed as cylinders. The arrowheads are displayed as 3D/round.
Umbrella	The arrow tails are displayed as 1D lines. The arrowheads are displayed as 3D/round.

5. Use the **Size** slider to increase (move to the right) or decrease (move to the left) the length and dimensions of the arrows. The arrows are resized relative to the size of the model geometry.
6. Select **Arrow tail** to include tails on all arrows.
7. Check **Map size** to scale the size of the arrows to the magnitude of the field quantity being plotted.

   If you uncheck **Map size**, the Magnitude filtering is enabled. You can specify the Min and Max magnitude filtering, or use a Threshold slider to set a threshold for plotting vectors. Vectors under the threshold or below the Min are not plotted.



   The **Reset Min/Max** button lets you reset min/max to the current intrinsic values.

8. Click the **Plots** tab.
9. HFSS plots arrows on a grid that is superimposed on the surface or object you selected for the plot. Under **Vector plot**, use the **Spacing** slider to increase (move to the right) or decrease (move to the left) the distance between arrows (grid points.)
   - Select **Uniform** if you want the arrows to be spaced equally.
10. Select **Real time mode** if you want the changes to take effect immediately in the view window.

    If this option is cleared, click **Apply** when you want to see the changes.

11. Click **Close** to dismiss the dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Mesh Visibility on Field Plots

To display or hide the mesh on field plots, or change the mesh's color:

1. Click **HFSS**> **Fields**> **Modify Plot Attributes** .

   The **Select Plot Folder** dialog box appears.

2. Select the plot folder you want to modify and click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

3. Click the **Plots** tab.
4. Select **Add Grid** to display the mesh.
5. Optionally, select a color for the mesh from the **Color** palette.
6. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.

7. Click **Close** to dismiss the window.

Modifying Scalar Field Plot Isovalues

1. Click **HFSS**> **Fields**> **Modify Plot Attributes**.

   The **Select Plot Folder** window appears.

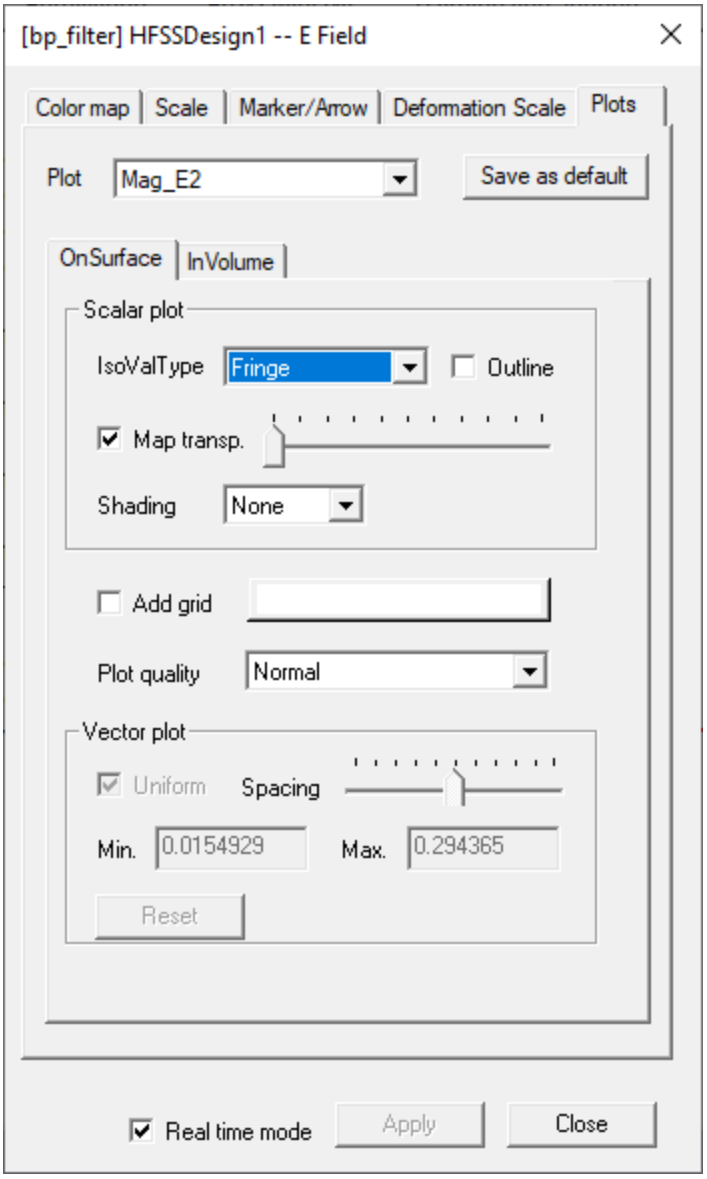
2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

3. Click the **Plots** tab. If you have more than one plot, a drop down lets you select. The OnSurface tab appears for surface plots and InVolume for volume plots.

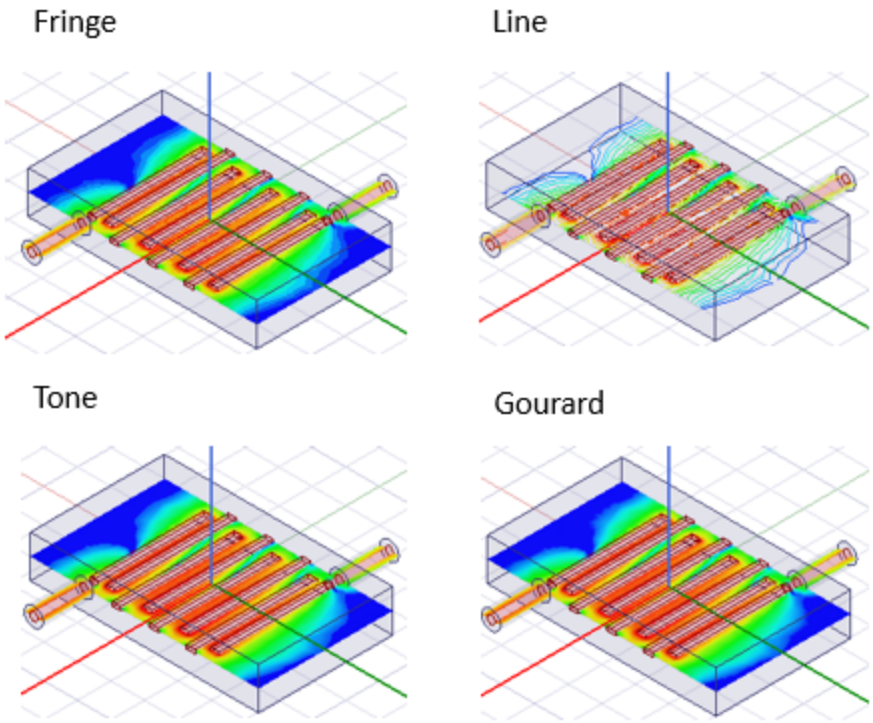
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. If the plot is a scalar surface plot, do the following:
   a. Select one of the following isosurface display types in the **IsoValType** drop-down menu:

Line	Lines are drawn along the isovalues.
Fringe	Color is constant between isovalues.
Tone	Color varies continuously between isovalues.
Gourard	Color varies continuously across the plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. Optionally, if you selected **Fringe** or **Tone**, select **Outline** to add a border line between isovalues.

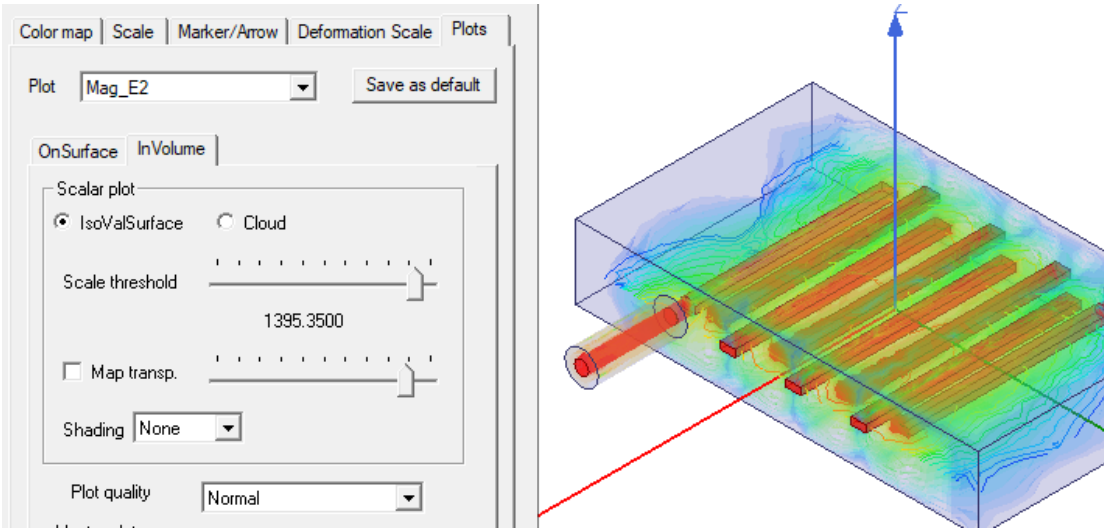
c. For shading on OnSurface plots, if you are using Lighting, you can adjust the Shading as None, Flat, or Smooth.

5. If the plot is a scalar volume plot, do the following:

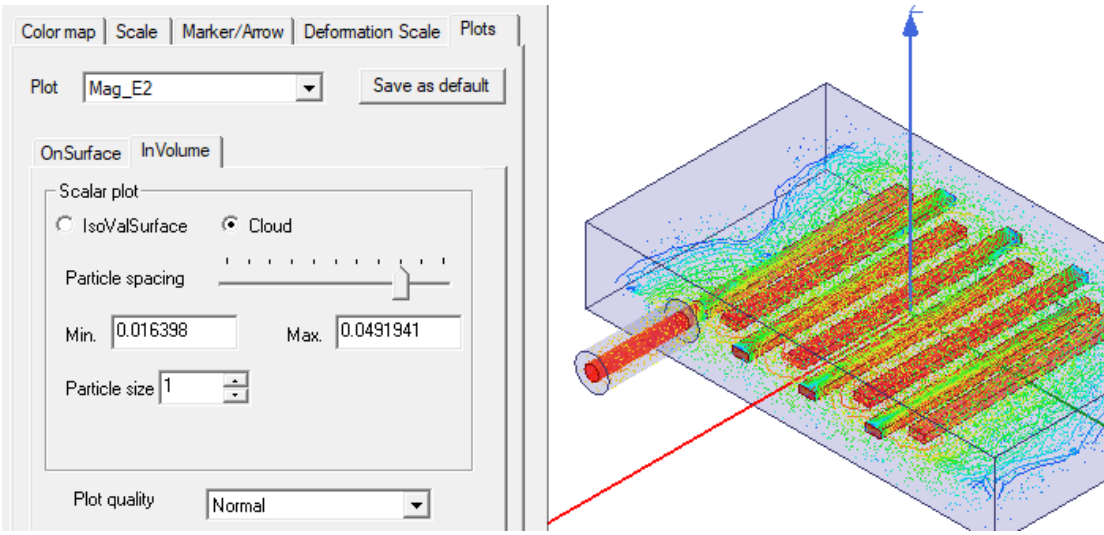
a. Select one of the following display types:

IsoValSurface	Color is drawn on the isovalues. This choice includes “Scale threshold” slider barthat allows you to set a field value threshold so that iso surfaces with field value lower than the given threshold will show as translucent with given transparency specified by the slider bar below it. When "Map transp." is checked, translucent iso surfaces field values map to [0.0, transparency] so lower field value map to higher transparency while higher field value to lower transparency. If it is unchecked, all translucent iso surfaces will have the same given transparency from the slider bar.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Cloud Field values are represented by points that illustrate the spatial distribution of the solution. The higher the solution value, the greater the cloud density. You can adjust particle spacing, Min. and Max, values, and Particle size.



6. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.

7. Click **Close** to dismiss the window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modifying Scalar Field Plot Shading

1. Click **HFSS** > **Fields** > **Modify Plot Attributes**.

   The **Select Plot Folder** window appears.

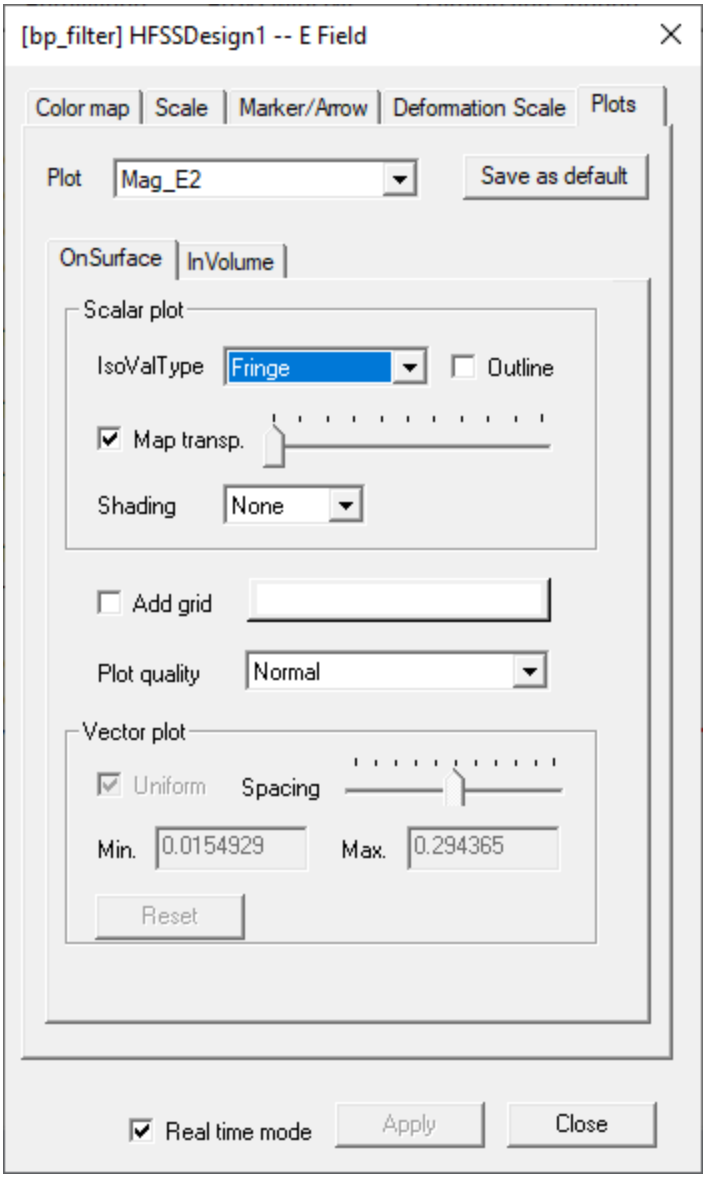
2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

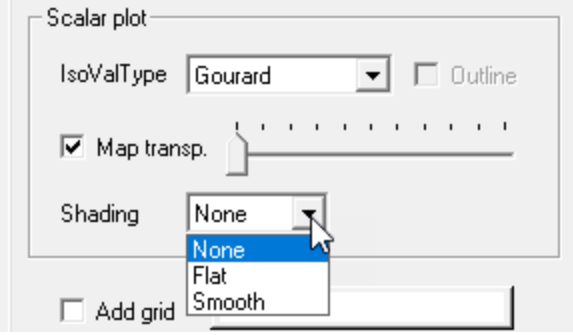
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click the **Plots** tab.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If the plot is a scalar surface plot, do the following:
    a. Select one of the following from the Shading drop-down menu:



None	Equivalent to selecting **Do not use lighting** on the Lighting Properties dialog box.
Flat	Provides a flat rendering of the field plot.
Smooth	Provides a smooth rendering of the field plot.

5. Select **Real time mode** if you want the changes to take effect immediately in the view window.

    If this option is cleared, click **Apply** when you want to see the changes.

6. Click **Close** to dismiss the window.

Mapping Scalar Field Plot Transparency to Field Values

1. Click **HFSS** > **Fields**> **Modify Plot Attributes**.

    The *Select Plot Folder* dialog box appears.

2. Select the plot folder you want to modify, and then click **OK**.

    All plots in the selected folder will be modified.

    A dialog box with attribute settings for the selected folder appears.

3. Click the **Plots** tab.
4. Use the **Map transp.** slider to increase (move to the right) or decrease (move to the left) the transparency of the plot.
    - If you select **Map transp.**, the transparency of field values increases as the solution values decrease.
5. Select **Real time mode** if you want the changes to take effect immediately in the view window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If this option is cleared, click **Apply** when you want to see the changes.

6. Click **Close** to dismiss the dialog box.

Modifying Markers on Point Plots

For scalar point plots, a marker is used to represent a field quantity at a selected point (for vector point plots, arrows are used).

Modify the shape and size of markers in the plot attributes window:

1. Click **HFSS**> **Fields**> **Modify Plot Attributes**.

   The **Select Plot Folder** window appears.

2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

3. Click the **Marker/Arrow** tab in the plot attributes window.
4. Under **Marker options**, select one of the marker types to represent the field quantity at the point:
   - **Sphere**
   - **Box**
   - **Tetrahedron**
   - **Octahedron**
5. Use the **Size** slider to increase or decrease the size of the marker.
6. Select **Map size** to scale the size of the marker to the magnitude of the quantity being plotted.
7. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.

8. Click **Close**.

Modifying Line Plots

Field quantities can be plotted directly on a line object. Scalar quantities are plotted as 3D color-shaded lines. Vector quantities are plotted as arrows that are based on the line.

To modify the appearance of line plots:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **HFSS>Fields>Modify Plot Attributes** .

   The **Select Plot Folder** window appears.

2. Select the plot folder you want to modify, and then click **OK**.

   All plots in the selected folder will be modified.

   A dialog box with attribute settings for the selected folder appears.

3. Click the **Plots** tab.
4. Select one of the following isosurface display types in the **IsoValType** drop-down menu:

Fringe	Color is constant between isovalues.
Tone	Color varies continuously between isovalues.
Gourard	Color varies continuously across the plot.

5. Select one of the following styles for the line object in the **Line style** drop-down menu:

Cylinder	The line object is shaped like a cylinder.
Solid	The line object is a 3D solid.
Dash-Dash	The line object is represented by dashed black line segments.
Dot-Dot	The line object is represented by a series of dots.
Dash-Dot	The line object is represented by a series of alternating dashed black line segments and dots.

6. Use the **Line width** slider to increase (move to the right) or decrease (move to the left) the thickness of the line.
7. By default, a polyline object is divided into 100 equally spaced points for post processing. To modify the number of points on the line, type a new value in the **Number of points** text box.
8. Select **Real time mode** if you want the changes to take effect immediately in the view window.

   If this option is cleared, click **Apply** when you want to see the changes.

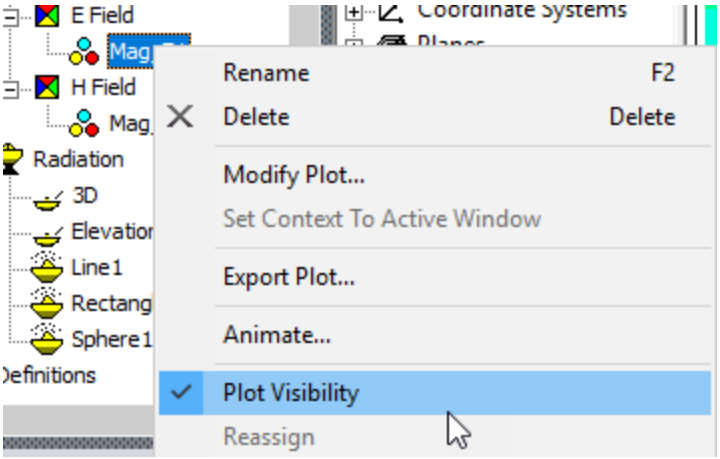
9. Click **Close** to dismiss the window.

Setting a Plot's Visibility

You can control the visibility of Field Overlays by means of the **View> Visibility> Active View Visibility** dialog box, **Fields Reporter** tab (shown below) or by right-clicking on the plot in the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

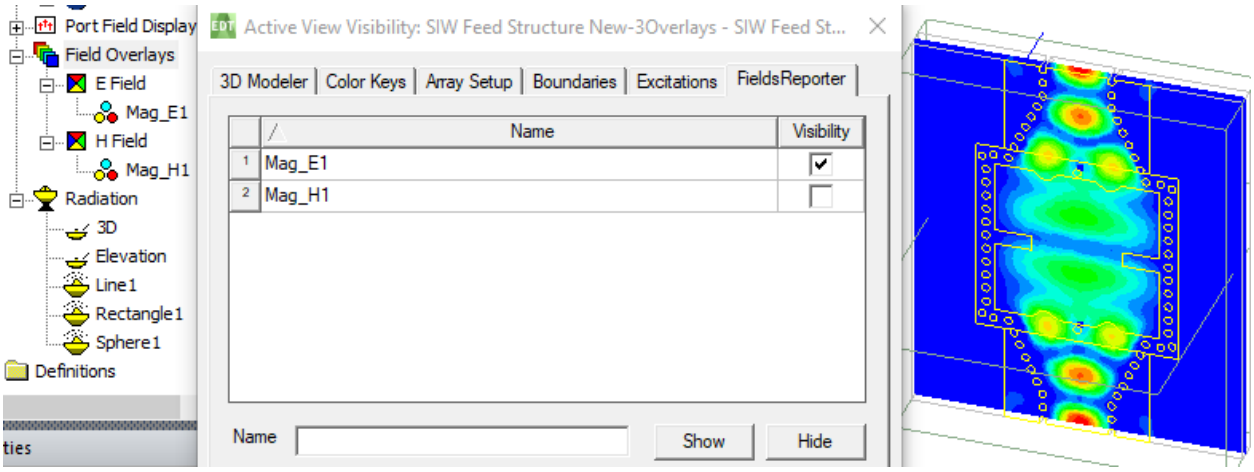
Project Manager and checking or unchecking **Plot Visibility**.



To display or hide a field overlay or mesh plot from view in the *3D Modeler* window:

1. Click **View> Visibility>** **Active View Visibility**. Alternatively, you can select the **Active View Visibility** icon from the **View, Draw,** or **Model** ribbon tabs.

   The *Active View Visibility* dialog box appears.

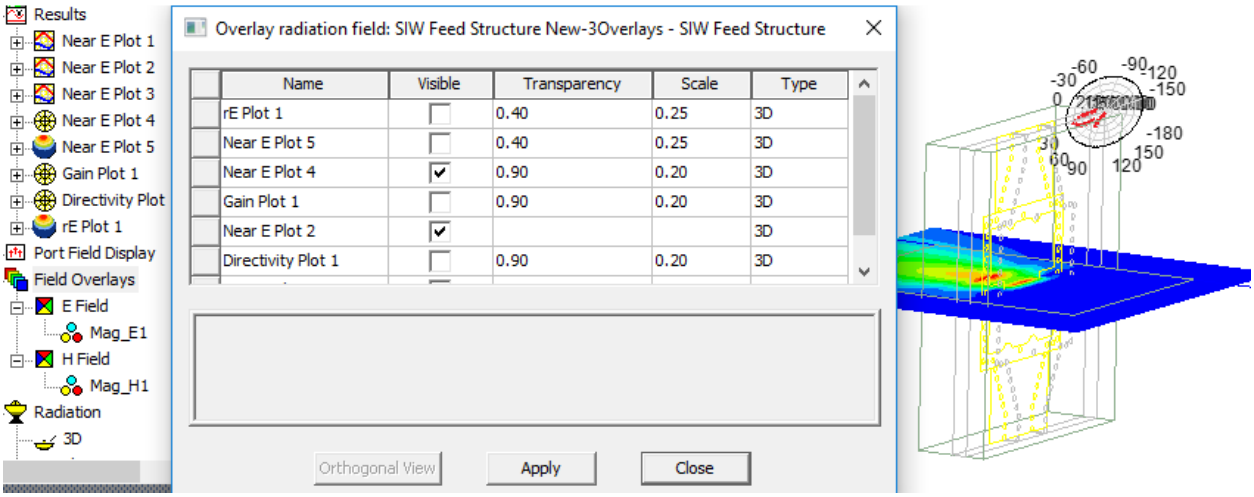


2. Click the **FieldsReporter** tab.
3. In the **Visibility** column, select the field overlay or mesh plots you want to display. Clear the plots you want to hide from view.

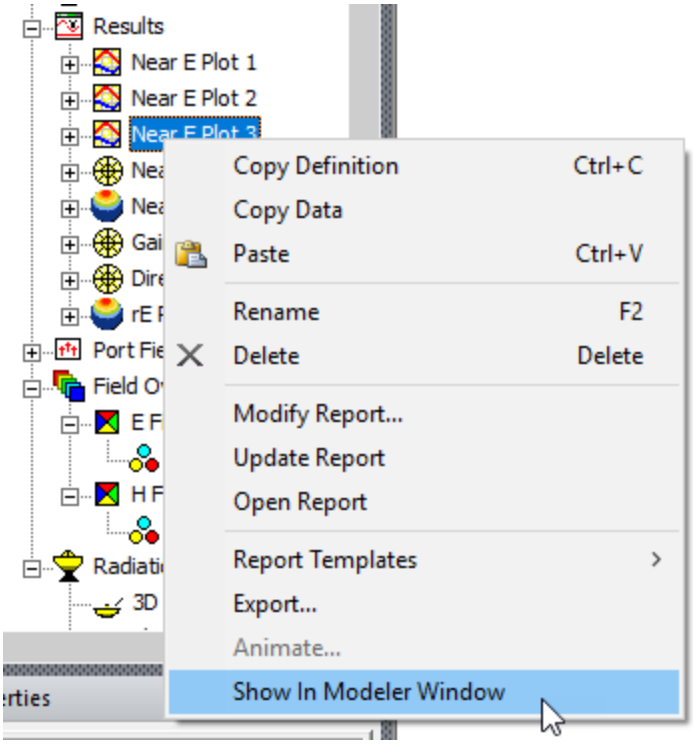
   Only the selected plots will be visible. Multiple plots can be shown at once.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Radiation plots and antenna parameter tables available for overlay can be viewed by right-clicking **Field Overlays** in the *History Tree*, and selecting **Plot Fields> Radiation Field...**. This displays an **Overlay radiation field** dialog that lists each Results plot available for overlay.



You can also control the visibility of **Results** plots available for **Overlay radiation field** by right-clicking on the report and selecting **Show in Modeler Window**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Saving a Field Overlay Plot

Field overlay and mesh plots are saved in the project file; however, you can save a plot to Field Plot File format (.dsp) and then open it in HFSS.

To save field overlay or mesh plot data to a .dsp file:

1. In the project tree, click the plot you want to export.
2. Click **HFSS** > **Fields** > **Save as** .

   The **Select Field Plot(s)** dialog box appears.
3. Select the plots you want to export by checking the **Select** box, and then click **OK**.

   The file browser appears. **Field Plot Files (.dsp)** is the selected file type.
4. Specify the name of the .dsp file and the location in which to save it.
5. Click **Save**.

The plot is exported to the specified .dsp file.

To open the file you created, click **HFSS** > **Fields** > **Open**.

Opening a Field Overlay Plot

To open a field overlay or mesh plot that you have saved to Field Plot File format (.dsp):

1. Click HFSS > **Fields** > **Open** .
2. The file browser appears. **Field Plot Files (.dsp)** is the selected file type.
3. Browse to the location of the .dsp file you want to open, and then click the file name.
4. Click **Open**.

The plot appears in the view window. It is listed under **Field Overlays** in the project tree.

Deleting a Field Overlay Plot

1. Click **HFSS** > **Fields** > **Delete Plot** .

   The **Delete Plots** dialog box appears.
2. Select the plots you want to delete by checking the **Delete** check box.
3. Click **OK**. The selected plot are deleted.

Alternatively, click the plot in the project tree that you want to delete, and then press **Delete** .

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting Field Overlay Plot Defaults

Each new field plot uses the default plot settings specified in the **Set Plot Defaults** dialog box.

To modify the default plot settings:

1. If a plot folder has not been created, click **Field Overlays** in the Project Manager.
2. Click **HFSS** > **Fields** > **Set Plot Defaults**.

   The **Set Plot Defaults** dialog box appears.
3. Select the solution to plot from the **Solution** drop-down menu.
4. Select the plot folder in which new plots will be stored from the **Quantity type** drop-down menu. Choose one of the following options:

New Folder	Each new plot will be stored in a separate folder in the project tree.
Automatic	Each new plot will be stored in a folder determined by HFSS as the most appropriate based on the plotted field quantity. For example, all surface magnitude E plots will be stored in the same folder.
An existing folder	Select the existing folder in which you want to store new plots.

> **Note:**
>
> Plots stored in the same folder will use the same color key. The **Auto** scale setting will be based on the maximum field solution value present in a plot.

5. Under **Intrinsic Variables**, specify the frequency and phase angle at which the field quantity is evaluated.
6. Click **OK**.

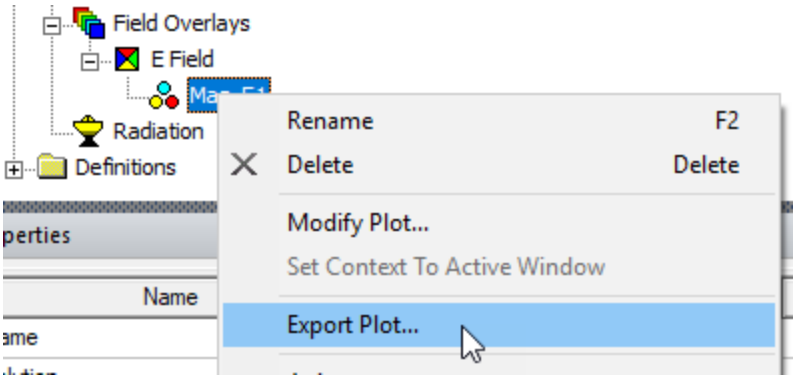
Exporting Field Plots

The **Export Plot...** command lets you export a field plot to a format that supports full data precision without the need to go through the Fields Calculator.

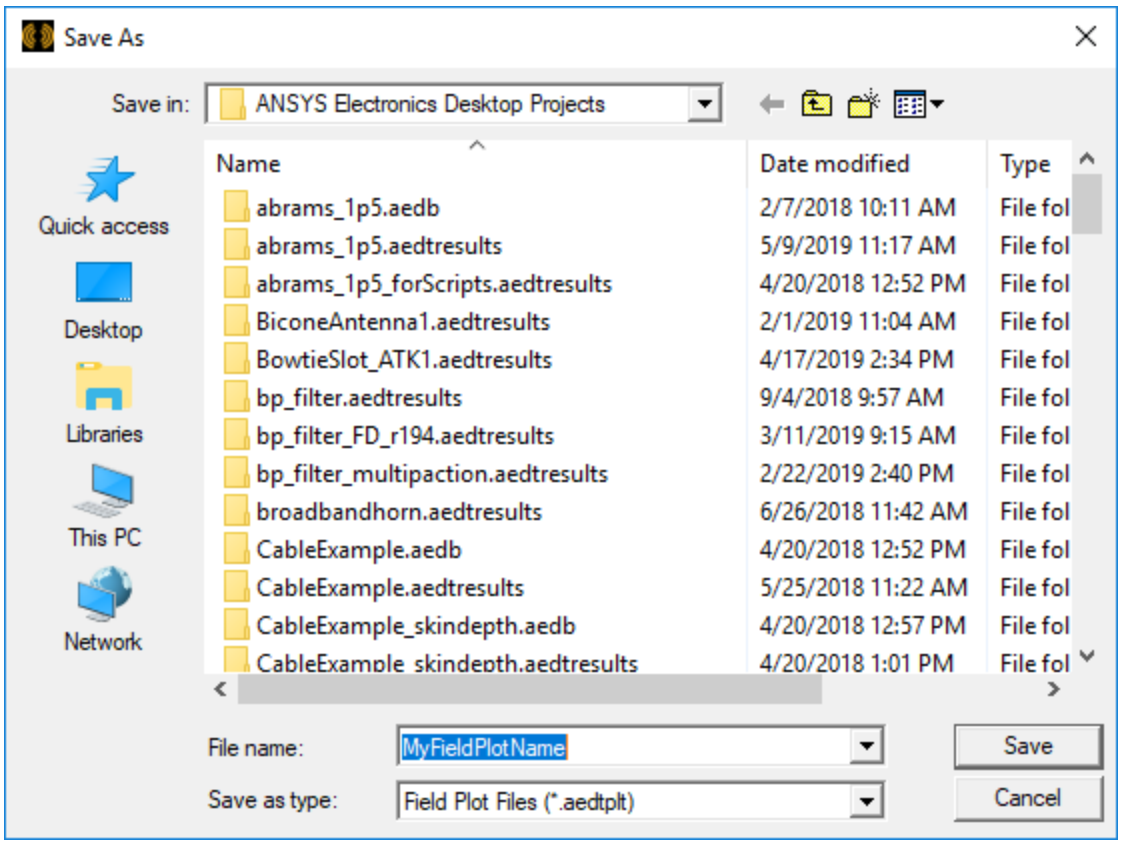
1. Create the field plot you want to export.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the **Project Manager tree**, right click on the field plot icon and in the shortcut menu, click **Export Plot...**.



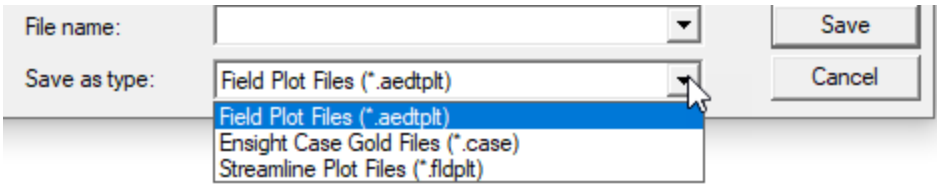
The **Save As** dialog box appears.



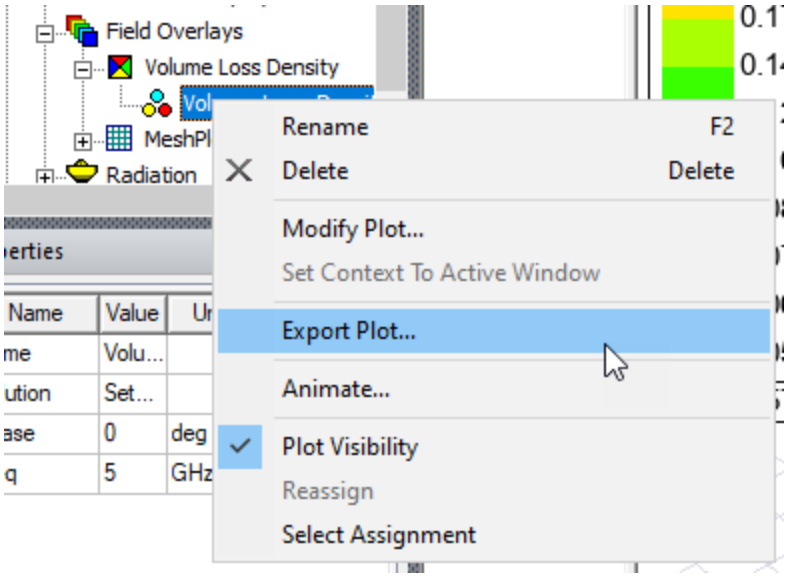
3. Specify the directory to **Save in**, the **File name** and use the **Save as type** as Field Plot Files (*.aedtplot) or Ensight Case Gold Files (*.case). If the Plot is a Streamline plot you can also choose Steamline files (*.fldplt). (See **Notes for creating Streamline plots**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

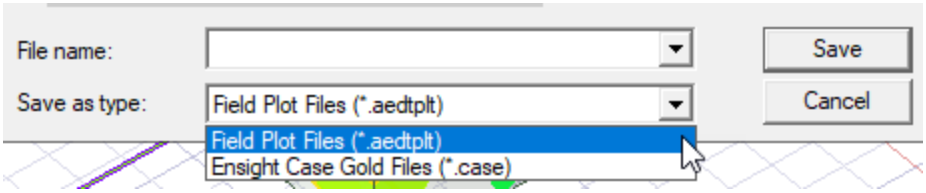
under Plotting Field Overlays).



You can export a Streamline plot in .fldplt format by right-clicking on the plot in the Project tree, and selecting **Export Plot...**



This opens a dialog for you to specify a plot name and location.



The Field Plot is exported to the file format you specified.

Field Plot Data Export Format (*.aedtplt)

A sample first order file with 1 drawing:

```
# Ansys ElectronicsDesktop 2025.1.0
# Field plot export file (*.aedtplt), version 1.0
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Number of drawing: 1
$begin Drawing_1
IsUniformMesh=true
HasCurvElem=false
BoundingBox(-1.2999999523162842e+00, 1.0000000000000000e+00, -
8.0000001192092896e-01, 1.3999999761581421e+00, -6.9999998807907104e-
01, 0.0000000000000000e+00)
Elements(27, 6, 4, 3, 3, 3, 10, 5, 23, 6, 24, 26, 7, 18, 19, 20, 3,
4, 3, 3, 3, 10, 5, 24, 7, 25, 27, 8, 18, 20, 21, 3, 4, 3, 3, 3, 10,
2, 14, 4, 16, 22, 7, 13, 17, 20, 3, 4, 3, 3, 3, 10, 7, 26, 6, 16, 15,
2, 20, 19, 13, 3, 4, 3, 3, 3, 10, 1, 9, 2, 12, 15, 6, 10, 13, 19, 3,
4, 3, 3, 3, 10, 6, 23, 5, 12, 11, 1, 19, 18, 10, 3)
Nodes(1.0000000000000000e+00, 1.4000000000000001e+00, -
7.0000000000000007e-01, 1.0000000000000000e+00, -8.0000000000000004e-
01, -7.0000000000000007e-01, -1.3000000000000003e+00,
1.4000000000000001e+00, -7.0000000000000007e-01, -
1.3000000000000003e+00, -8.0000000000000004e-01, -
7.0000000000000007e-01, 1.0000000000000000e+00,
1.4000000000000001e+00, 0.0000000000000000e+00,
1.0000000000000000e+00, -8.0000000000000004e-01,
0.0000000000000000e+00, -1.3000000000000003e+00, -
8.0000000000000004e-01, 0.0000000000000000e+00, -
1.3000000000000003e+00, 1.4000000000000001e+00,
0.0000000000000000e+00, 1.0000000000000000e+00, 3.0000000000000004e-
01, -7.0000000000000007e-01, -1.5000000000000013e-01,
1.4000000000000001e+00, -7.0000000000000007e-01,
1.0000000000000000e+00, 1.4000000000000001e+00, -3.5000000000000003e-
01, 1.0000000000000000e+00, 3.0000000000000004e-01, -
3.5000000000000003e-01, -1.5000000000000013e-01, 3.0000000000000004e-
01, -7.0000000000000007e-01, -1.5000000000000013e-01, -
8.0000000000000004e-01, -7.0000000000000007e-01,
1.0000000000000000e+00, -8.0000000000000004e-01, -
3.5000000000000003e-01, -1.5000000000000013e-01, -
8.0000000000000004e-01, -3.5000000000000003e-01, -
1.3000000000000003e+00, 3.0000000000000004e-01, -7.0000000000000007e-
01, -1.5000000000000013e-01, 1.4000000000000001e+00, -
3.5000000000000003e-01, -1.5000000000000013e-01, 3.0000000000000004e-
01, -3.5000000000000003e-01, -1.3000000000000003e+00,
3.0000000000000004e-01, -3.5000000000000003e-01, -
1.3000000000000003e+00, 1.4000000000000001e+00, -3.5000000000000003e-
01, -1.3000000000000003e+00, -8.0000000000000004e-01, -
3.5000000000000003e-01, 1.0000000000000000e+00, 3.0000000000000004e-
01, 0.0000000000000000e+00, -1.5000000000000013e-01,
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
3.0000000000000004e-01, 0.0000000000000000e+00, -1.5000000000000013e-
01, 1.4000000000000001e+00, 0.0000000000000000e+00, -
1.5000000000000013e-01, -8.0000000000000004e-01,
0.0000000000000000e+00, -1.3000000000000003e+00, 3.0000000000000004e-
01, 0.0000000000000000e+00)
ElemSolution(5.7354731844611963e-02, 2.9232772984855881e-01, 10,
2.1352990786467962e-01, 1.2797657961936443e-01, 1.9660806325882133e-
01, 1.1726155545957916e-01, 2.1885451415722262e-01,
2.4247258543532729e-01, 1.8703186257772084e-01, 1.1712555190711266e-
01, 1.0306551944577172e-01, 1.9222144469761518e-01,
1.6615464797768376e-01, 1.0804043743316410e-01, 6.6510438035030331e-
02, 1.7865861636680735e-01, 1.1040223877077059e-01,
2.5391393137507334e-01, 1.4691691383403402e-01, 9.2573922668716196e-
02, 1.7035856643167813e-01, 1.4641167161072380e-01,
1.4770701527152347e-01, 1.6261065519093762e-01, 2.3063204305627905e-
01, 1.2530038704664462e-01, 1.4614642445195816e-01,
1.1920618653295692e-01, 1.1233254336705870e-01, 1.2156273418014953e-
01, 9.0145063831456124e-02, 8.3219042948130054e-02,
1.2535082887941645e-01, 1.2753078005131815e-01, 1.3872888237187334e-
01, 1.2424658874600869e-01, 1.3005844785230156e-01,
1.4711693546682481e-01, 1.0390046361183673e-01, 1.2075181459543853e-
01, 1.1420762153323573e-01, 1.0510569422539225e-01,
2.5244183990075453e-01, 1.4468010704771497e-01, 1.1075980053186024e-
01, 1.0756449329788385e-01, 7.1852801171335884e-02,
5.7354731844611963e-02, 1.9153549687571236e-01, 1.2155254289403770e-
01, 1.0467070945529336e-01, 1.8204372604099889e-01,
1.6179743545861430e-01, 1.4548511943318404e-01, 1.7334644308737701e-
01, 1.1138082367857982e-01, 1.9054730731722422e-01,
2.9232772984855881e-01, 1.1136332643725760e-01, 1.5631026982928142e-
01, 2.1419476339746449e-01, 1.7884651132070065e-01)
ElemSolutionMinMaxLocation(1.0000000000000000e-03, -
8.0000000000000004e-04, 0.0000000000000000e+00, 1.0000000000000000e-
03, 1.4000000000000002e-03, -7.0000000000000010e-04)
ElemSolutionMinMaxIndices(4, 5, 5, 5)
$end Drawing_1
```

File Format Description

The first line starts with the symbol of “#” and continues with the product name and its version that is used to produce this file. The 2nd line also starts with the symbol of “#” and continue with the file format version. Currently it is version 1.0.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The next line, it shows the number of drawings in this file. The above example shows 1 drawing. The drawing has 2 big data blocks (the mesh data block and field data block). Each drawing starts with **$begin Drawing_***** and ends with **$end Drawing_***** (where ***** should be replaced with the drawing index, starting from 1).

Mesh data block

It contains all the mesh information about current drawing include element type, number of nodes of each element, node coordinates and element node connectivity. At the beginning of the block, it shows if it has curve element and then it lists the bounding box of this mesh. The bounding box format is **BoundingBox(lowerX, lowerY, lowerZ, upperX, upperY, upperZ)**. They are 6 double values.

Next comes with elements part starting with **Elements(**. First integer is the total number of nodes for this mesh and the next is the total number of elements. After that, it repeats the following information for each element: element type, reserved integer 1, reserved integer 2 and reserved integer 3, number of nodes for this element, node index 1, node index 2 till to the number of nodes for this element. All these numbers are integers. Please refer to the end of this document to see the list of element types. This part ends with the close paren, ")".

The last part of the mesh data block is the nodes part. It contains the coordinates(x,y,z) of all the nodes for this mesh. It starts with Nodes(and continue with a list of tuples of 3 double values which are the x, y, z coordinates of the first, 2nd, last nodes and it ends with a close paren,")".

Field data block

It contains two parts. The first part includes node field data of each element. Node field data could either be scalar or vector data which means either it has 1 component or it has 3 component (x, y, z).

This part starts with **ElemSolution(**, the next two double values are the minimal and maximal field value of this drawing. The next integer shows the total number field data for each element. It should be a multiple of the number of nodes of each element (e.g. if field data is vector data, then it should be 3 times of the number of nodes of each element). For vector data, each element should have data like (**node1_x, node1_y, node1_z, node2_x, node2_y, node2_z, …**). This part ends with close paren, ")".

The 2nd part shows two location points. It starts with **ElemSolutionMinMaxLocation(**. The first 3 double values are the (x,y,z) coordinates of the location where the minimal field value is located while the next 3 numbers for the maximal field value. This part ends with close paren, ")".

Element types:

Element type description	Element type value
point	0
line	1
triangle	2

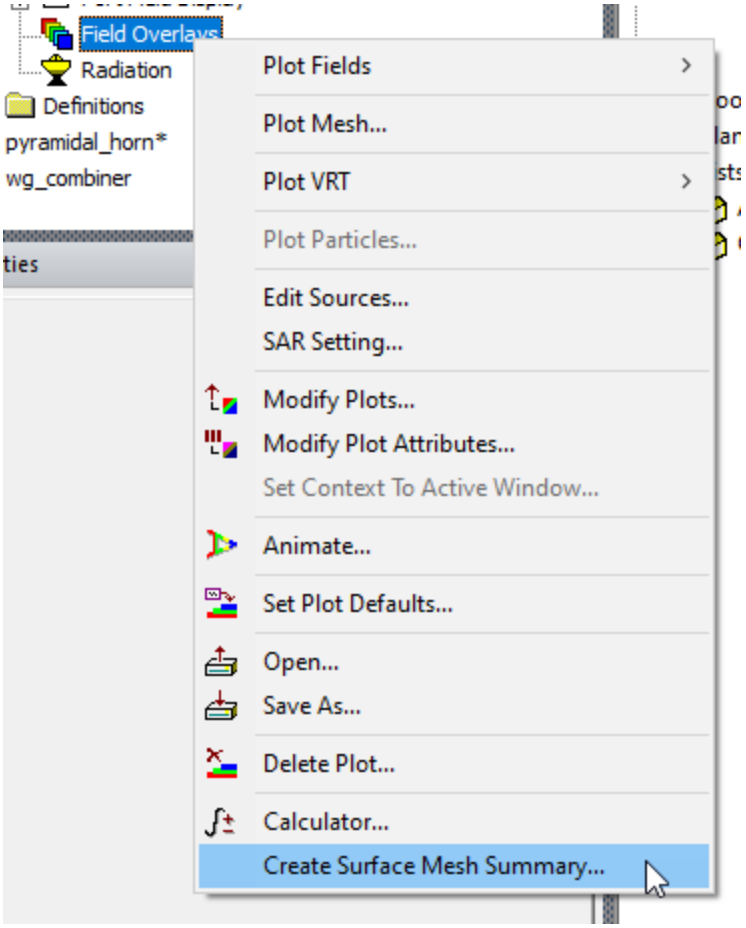
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element type description	Element type value
quad	3
tetradedron	4
pyramid	5
wedge	6
hexahedron	7

For second order element node index convention, Lagrange Parabolic is used.

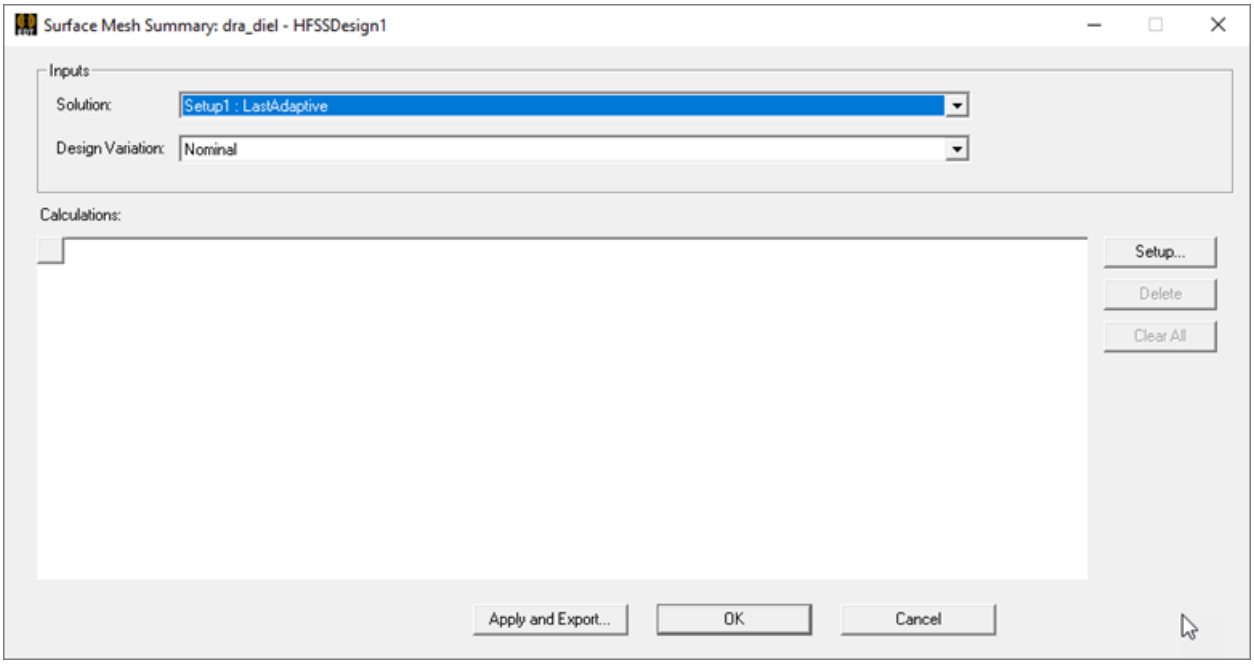
Create Surface Mesh Summary

You can use the **HFSS** > **Fields** > **Create Surface Mesh Summary...** command or the equivalent command on the short-cut menu on the **Field Overlays** icon in the **Project** tree to create a report of mesh statistics on selected surfaces or lists of surfaces.

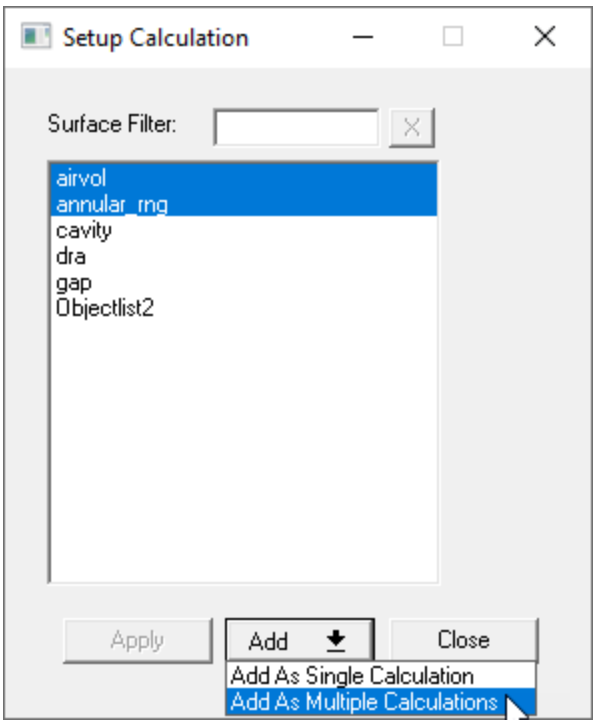


This command opens a dialog box in which you specify the Solution and Design Variation to use.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

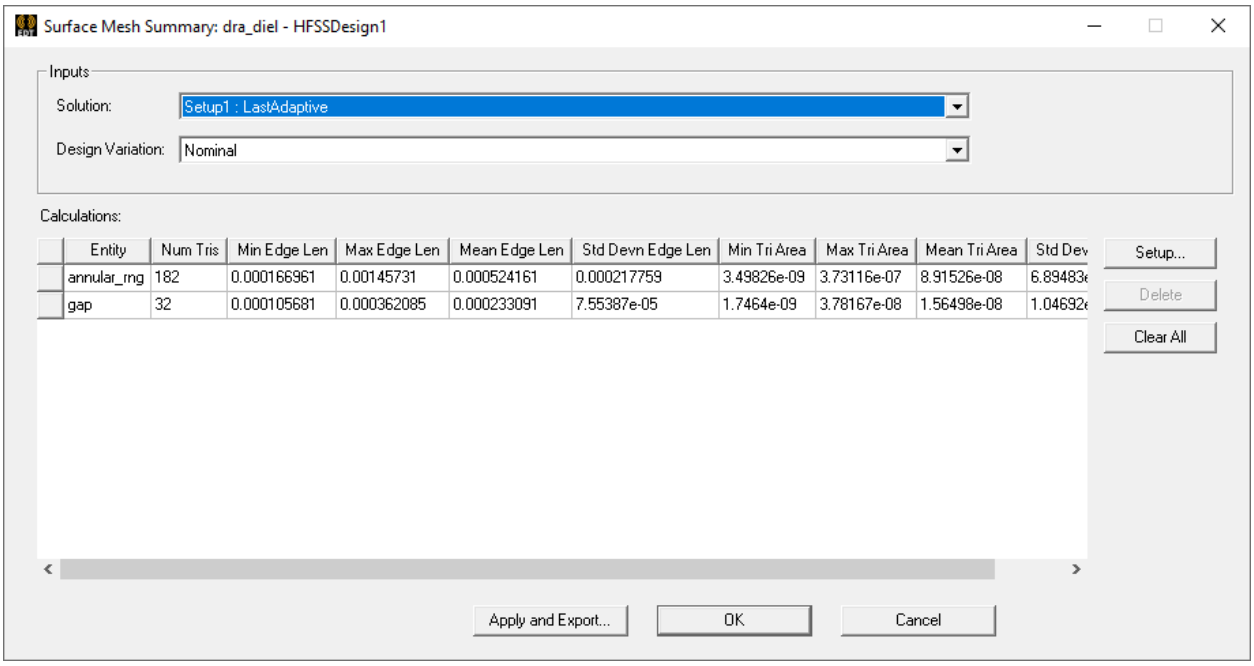


Select the **Setup...** button to open the **Setup Calculation** window, in which you specify the objects or lists for which to calculate mesh statistics. If the number of objects is large, you can use the Surface Filter text box to narrow the search.



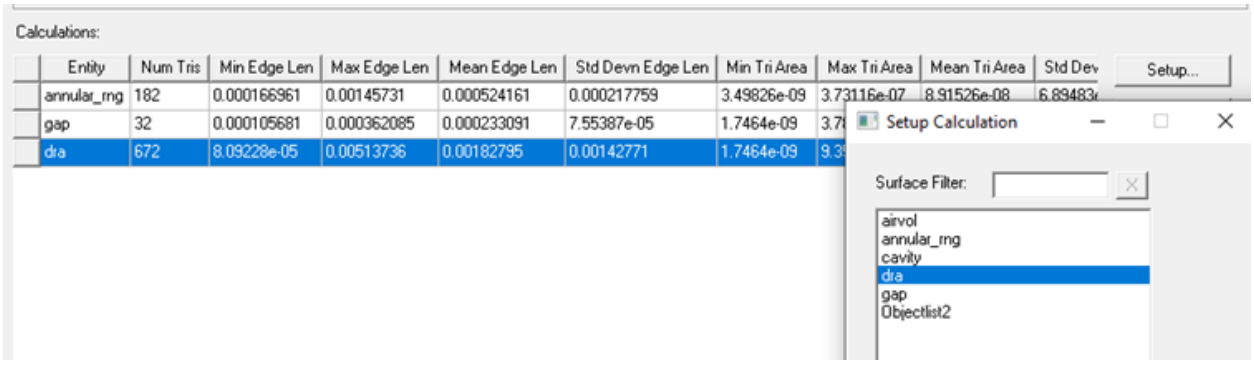
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you have made your selection, you can **Add as a Single Calculation** or **Add as Multiple Calculations**. The Calculations are displayed in table format, according to the **Setup** specified. If there is no mesh solution, the panel show only the selected entity name and a warning message reports “Failed to load surface mesh results”.



Statistics include Num Tris, Min Edge Length, Max Edge Length, Mean Edge Ledge, Standard Deviation Edge Length, Min Triangle Area, Max Triangle Area, Mean Triangle Area, and Standard Deviation Triangle Area.

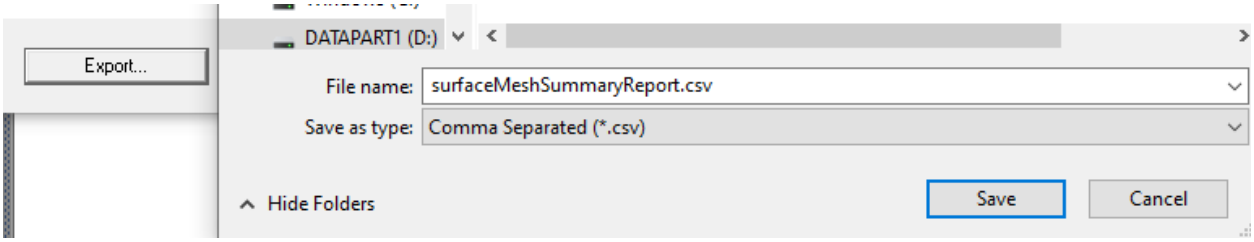
You can add additional rows, select selected rows, or clear all rows. Selecting only one row in the **Surface Mesh Summary** dialog enables the **Apply** button in the **Setup Calculation** dialog box. You can then make selections, then click on the **Apply** button, to use the selection to overwrite the content in the selected row by the entities chosen from the **Setup Calculation** dialog box.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Apply and Export** and **Export** buttons allow you to write out a comma separated (.csv) file of the statistics. Clicking either **OK** or **Apply and Export** saves the current data rows in the panel. The difference between the two buttons is:

- Clicking **OK** saves the data and then closes the dialog box.
- Clicking **Apply and Export** saves the data, then opens the **Save As** dialog box for .csv file export. After you click **Apply and Export**, the dialog box remains open and the button context is converted to **Export**, which means there is no new data to save.



The .csv file can be viewed in a spreadsheet as follows.

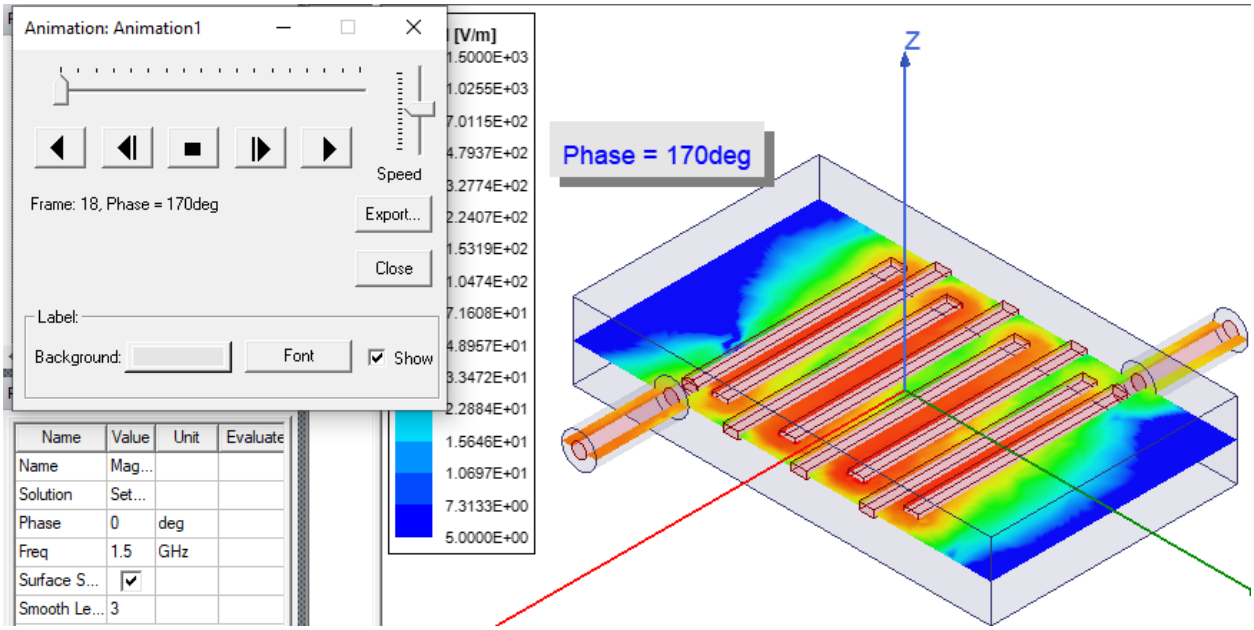
SolutionName

	A	B	C	D	E	F	G	H	I	J	K
1	SolutionN	Setup1 : LastAdaptive									
2	DesignVar	Nominal									
3											
4	Entity	Num Tris	Min Edge	Max Edge	Mean Edg	Std Devn I	Min Tri Ar	Max Tri Ar	Mean Tri A	Std Devn Tri Area	
5	annular_ri	182	0.000167	0.001457	0.000524	0.000218	3.50E-09	3.73E-07	8.92E-08	6.89E-08	
6	gap	32	0.000106	0.000362	0.000233	7.55E-05	1.75E-09	3.78E-08	1.56E-08	1.05E-08	
7	dra	672	8.09E-05	0.005137	0.001828	0.001428	1.75E-09	9.35E-06	1.43E-06	1.93E-06	
8											
9											

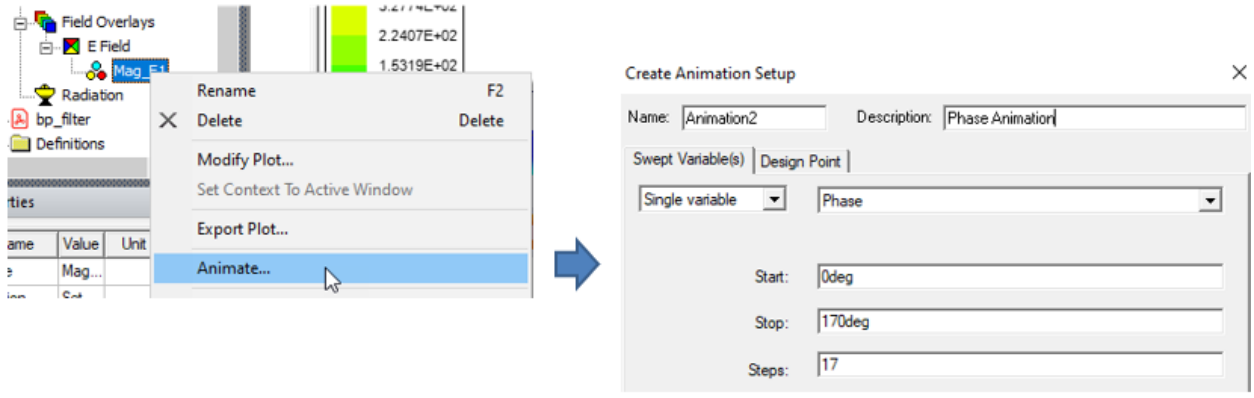
Creating Animations

An animated plot is a series of frames that displays a field, frequency, parametric value, mesh, virtual ray trace or geometry at varying values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To create an animated plot, you specify the values of the plot that you want to include, just as an animator takes snapshots of individual drawings that make up a cartoon. Each value is a frame in the animation. You specify how many frames to include in the animation.



The following sections describe how to create different kinds of animations.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Each animation frame requires memory for storage which depends upon the mesh size and type of plot. Memory usage may become very large during plot animations. To reduce memory usage, specify the minimum number of frames possible. See General Options for more information.

Note:

Graphics System requirements for Optimal Performance

In order to obtain optimal performance improvements on fields overlay plot, a workstation-class 3D-capable graphics card with at least 512 MB of memory that supports OpenGL version 2.0 or higher is needed.

On Windows, the default OpenGL version support is v1.1, so you might need to update graphics driver to the latest version;

If you access the application through Windows Remote Desktop which only supports Generic GDI (functionally equivalent to OpenGL v1.0), the performance improvement will also not be available;

To view OpenGL version/extensions supported by your card, the OpenGL Extension Viewer tool is accessible via softpedia.com.

If animation is slow, especially for complex models, for some older graphics cards, you can improve performance by accessing **NVIDIA Control Panel**> **3D Settings**> **Manage 3D Settings – Global Settings** tab and choosing the **Workstation App – Dynamic Streaming** option from the **Global Presets** drop-down menu.

You can export the animation to animated Graphics Interchange Format (GIF), to Audio Video Interleave (AVI) format, or to WebM (.webm) format.

Creating Phase Animations

You can create phase animations of field plots of frequency sweeps.

Prerequisites

Before creating a phase animation, you must solve a frequency sweep for the project.

Create a field plot.

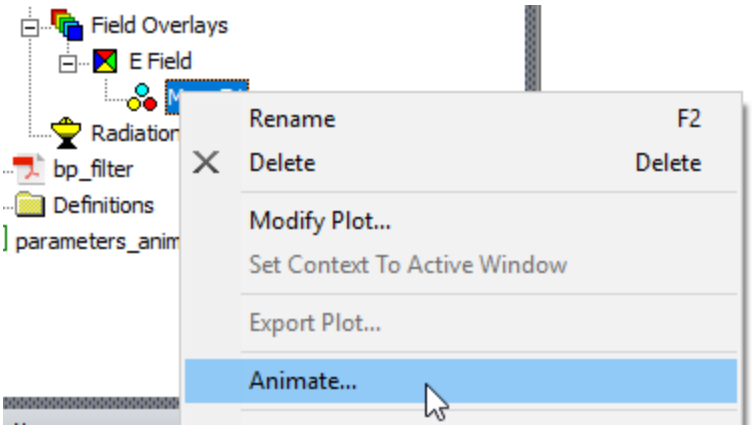
Procedure:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

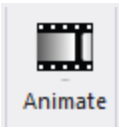
To animate a plot with respect to the phase of the plotted field:

1. Create a field overlay plot to animate.
2. Select the plot in the **Project** tree, and click **HFSS>Fields>Animate...**, or right-click in the **Modeler** window, and click **View>Animate...**.

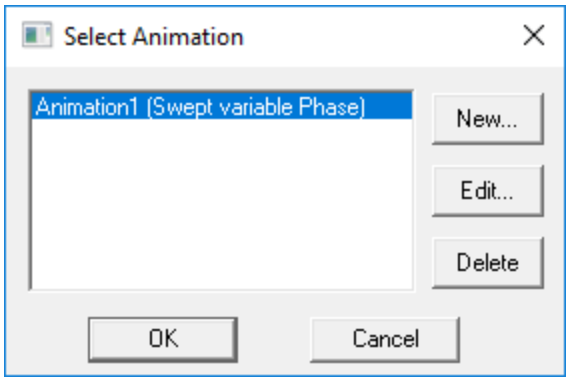
Or right-click the field overlay plot of interest and click **Animate...** from the shortcut menu,



Or select the field overlay plot of interest, select the **View** tab of the ribbon, and click the **Animate** icon.



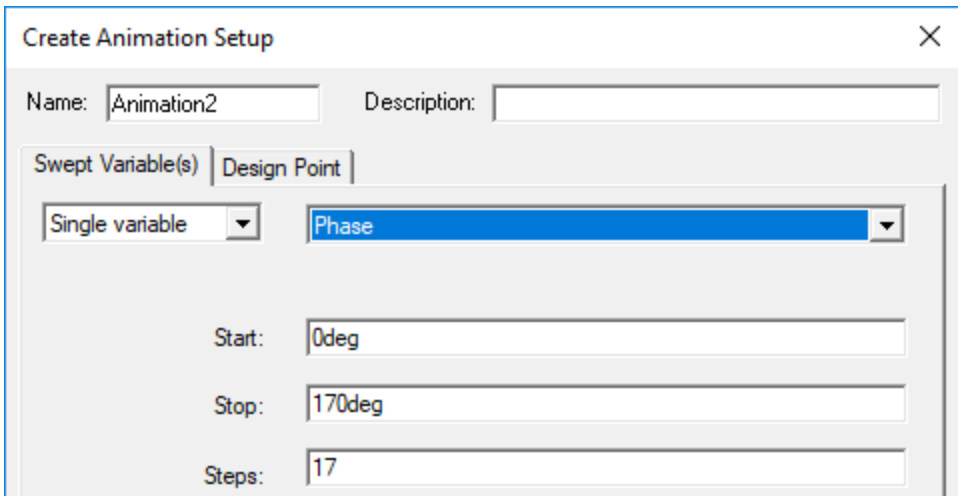
If you already created an animation, the **Select Animation** dialog box appears. Selecting an existing animation from that list starts it.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

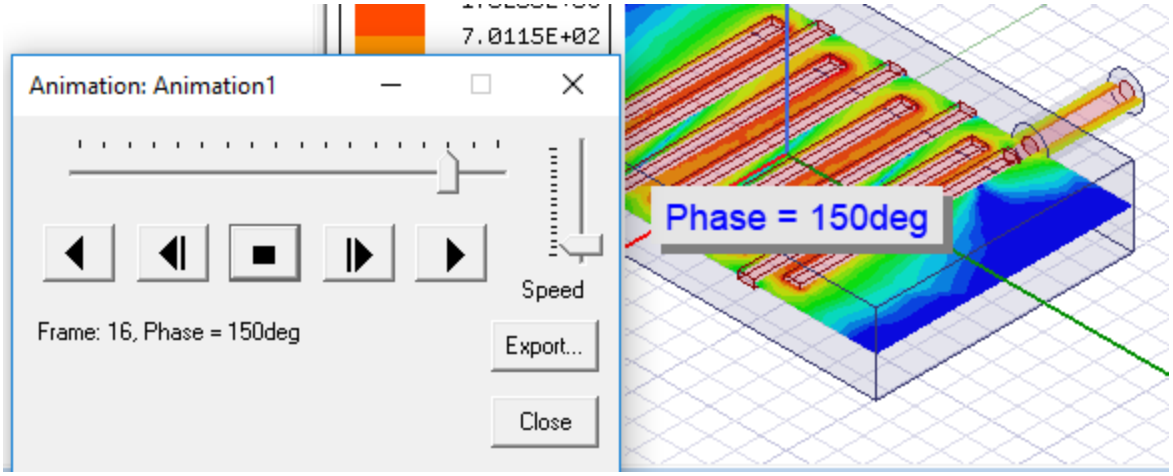
Selecting an existing animation from that list starts it when you click **OK**. Click **Edit...** to open the **Modify Animation Setup** dialog.To create a new animation, click **New**.

The **Create Animation Setup** dialog box appears.



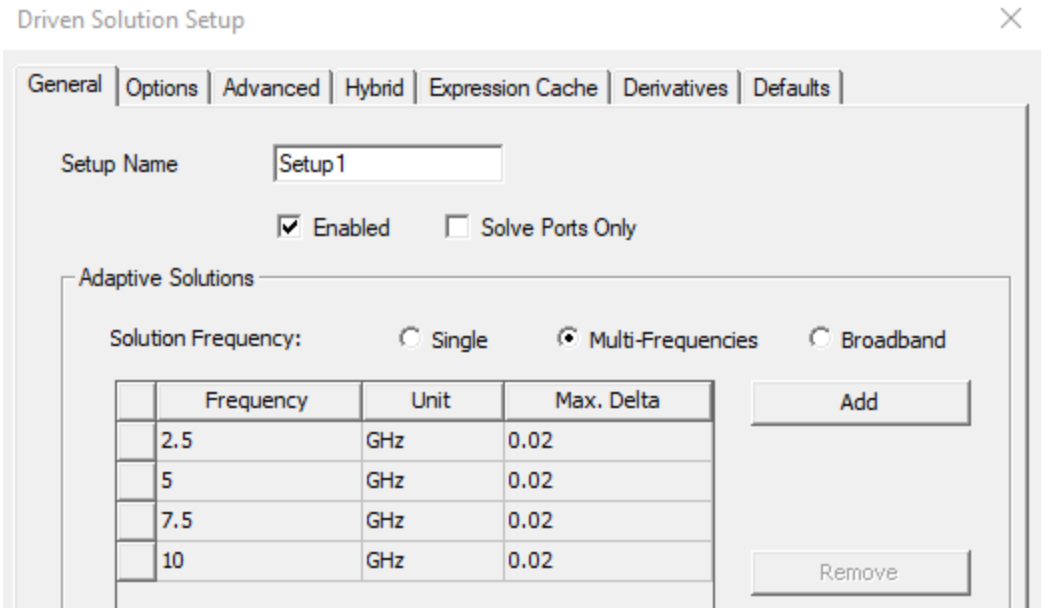
3. Under the **Swept Variable** tab, select **Single Variable** and accept Phase.
4. Specify the phase values you want to include in the animation, as Start, Stop, and Steps to include (e.g., if the **Start** value is **10**, the **Stop** value is **160**, and the number of steps is **10**, the animation will display the plot at 10 phase values between 10 and 160. The start value will be the first frame displayed, resulting in a total of 11 frames in the animation.). If the design has multiple project or intrinsic variables, click the **Design Point** tab to set the values of the non-animated variables.
5. Click the **Design Point** tab.
6. Deselect the **Use defaults** check box.
7. In the table, select the row corresponding to the variable setting of interest.
8. Click **OK** and the animation begins in the view window. The play panel appears in the upper-left corner of the desktop, enabling you to stop, restart, and control the speed and sequence of the frames.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Creating Frequency Animations

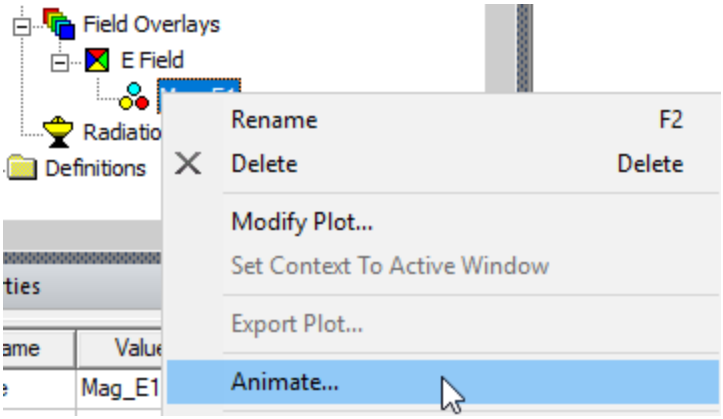
1. In order to create Frequency animations, you must define the frequencies of interest in the Solution setup, using the Multi-Frequencies option.



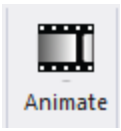
2. After running the analysis, Create a field overlay plot to animate.
3. Do one of the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

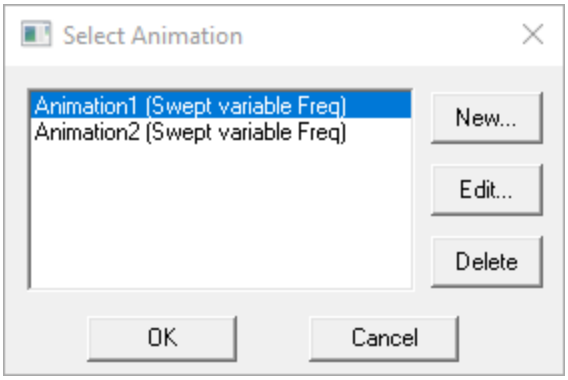
- Click **HFSS>Fields>Animate** .
- Right-click the field overlay plot of interest and click **Animate...** from the shortcut menu.



- Select the field overlay plot of interest, select the **View** tab of the ribbon, and click the Animate icon.

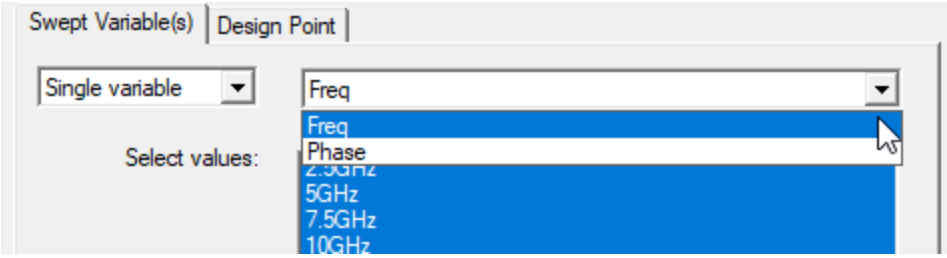


4. If you already created an animation, the **Select Animation** dialog box appears

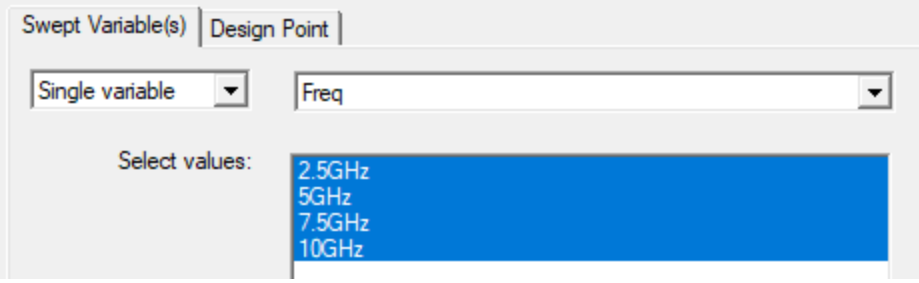


Selecting an existing animation from that list starts it when you click **OK**. Click **Edit...** to open the **Modify Animation Setup** dialog. To create a new animation, click **New** and the **Create Animation Setup** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



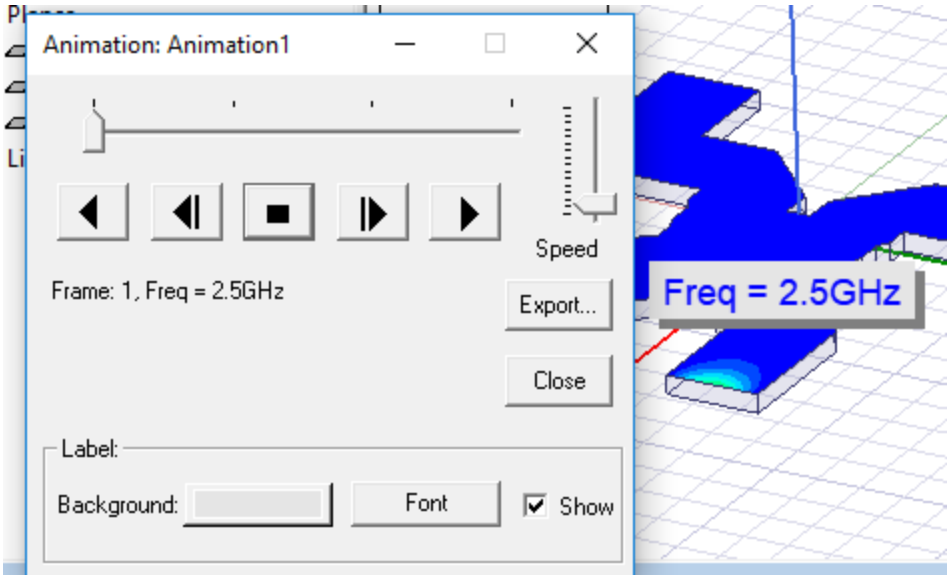
5. Under the **Swept Variable** tab, select **Frequency** from the **Swept Variable** list.
6. Select the frequency values you want to include in the animation from the **Select values** list.



   Use the **Shift** key to select a series of values, and the **Ctrl** key to select values that are not in sequence.
7. If the design has multiple project or intrinsic variables, click the **Design Point** tab to set the values of the non-animated variables.
8. Click the **Design Point** tab.
9. Deselect the **Use defaults** check box. In the table, select the row corresponding to the variable setting of interest.
10. Click **OK** and the animation begins in the view window. It will display one frame for each frequency value you selected. The play panel appears in the upper-left corner of the

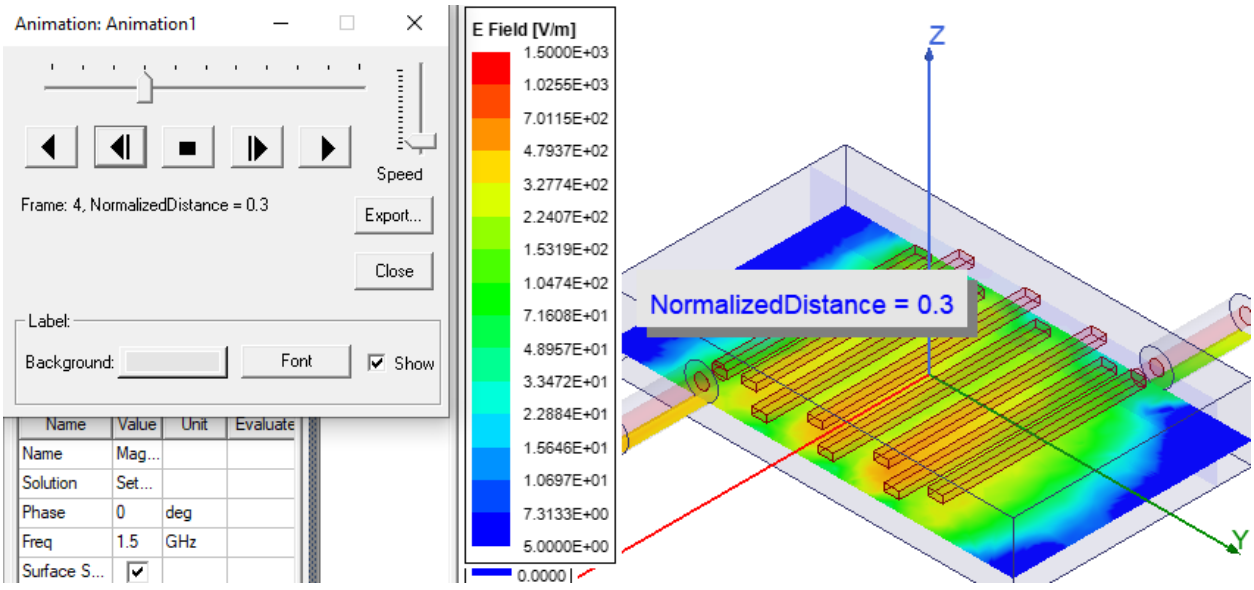
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

desktop, enabling you to stop, restart, and control the speed and sequence of the frames.



Creating Normalized Distance Animations

Before creating a normalized distance animation, you must solve a frequency sweep for the project. You can select Normalized Distance in the **Create Animation Setup** dialog box. This will show slices of the project relative to the plane or surface on which you have created the field plot.

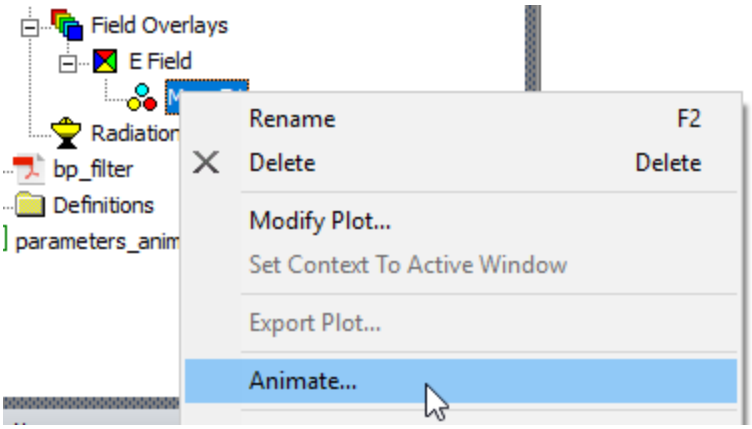


To animate a plot with respect to the normalized distance of steps through of the plotted field:

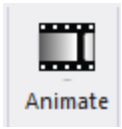
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Create a field overlay plot to animate.
2. Select the plot in the **Project** tree, and click **HFSS>Fields>Animate...**, or right-click in the **Modeler** window, and click **View>Animate...**.

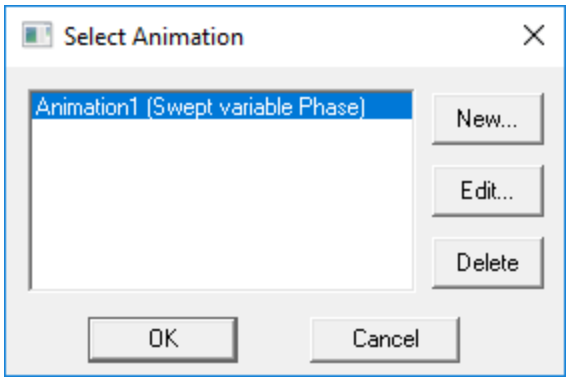
Or right-click the field overlay plot of interest and click **Animate...** from the shortcut menu,



Or select the field overlay plot of interest, select the **View** tab of the ribbon, and click the **Animate** icon.



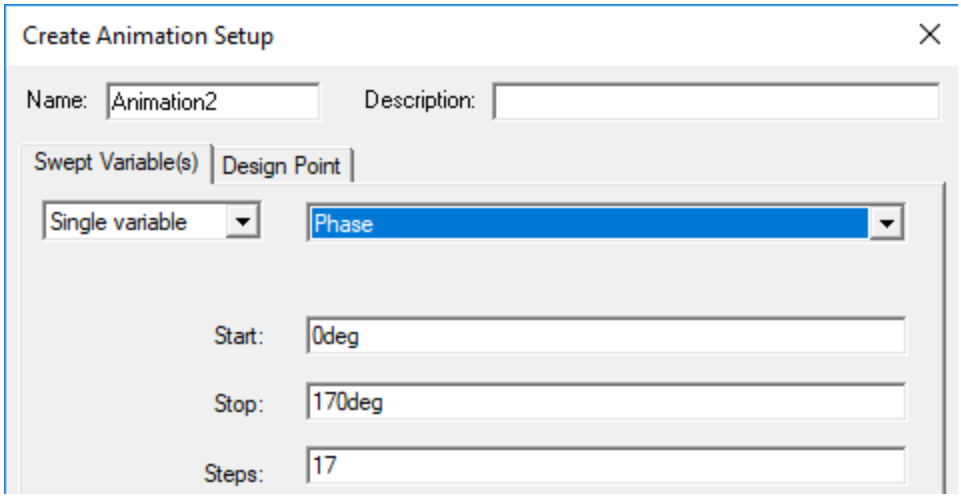
If you already created an animation, the **Select Animation** dialog box appears. Selecting an existing animation from that list starts it.



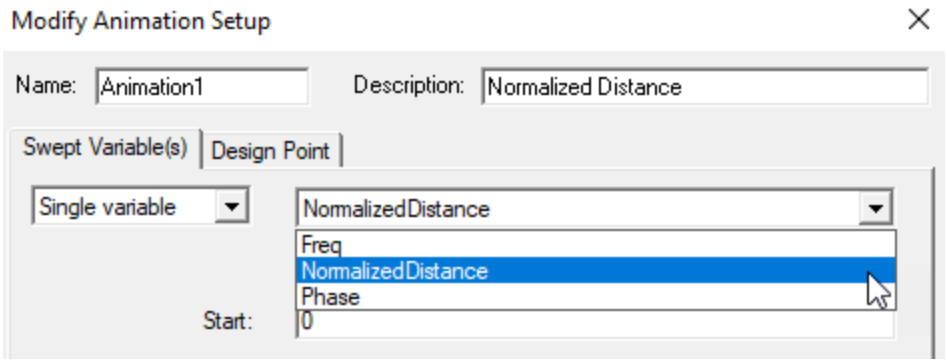
Selecting an existing animation from that list starts it when you click **OK**. Click **Edit...** to open the **Modify Animation Setup** dialog.To create a new animation, click **New**.

The **Create Animation Setup** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Under the **Swept Variable** tab, select **Single Variable** and Select Normalized Distance from the drop-down menu.



6. Specify the Start, Stop, and Steps.

   For example, if the **Start** value is **0**, the **Stop** value is **10**, and the number of steps is **10**, the animation will display the plot at 10 normalized steps between 1 and 10. The start value will be the first frame displayed, resulting in a total of 11 frames in the animation.

If the design has multiple project or intrinsic variables, click the **Design Point** tab to set the values of the non-animated variables.

a. Click the **Design Point** tab.
b. Deselect the **Use defaults** check box.

   In the table, select the row corresponding to the variable setting of interest.

7. Click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The animation begins in the view window. The play panel appears, enabling you to stop, restart, and control the speed and sequence of the frames.

Creating Geometry Animations

You can create geometry animations to evaluate the effect of varying geometry variables on the model. You can also create more complex animations by creating time variables associated with object locations. This section describes the general procedure for geometry animation, and shows a geometry animation with a field overlay, and a second animation for an HFSS Antenna Example project that includes multiple objects moving with respect to a time variable.

Prerequisites

You must define at least one variable associated with the geometry before you create a geometry animation.

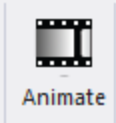
If you want to overlay a Field Solution on a geometry animation you must have solved fields for an Optimetrics setup using the same variable.

Procedure:

Following is the general procedure for creating an animation that varies a part of the model geometry.

1. Right-click in the *Modeler* window, and click to **View** > **Animate...** or

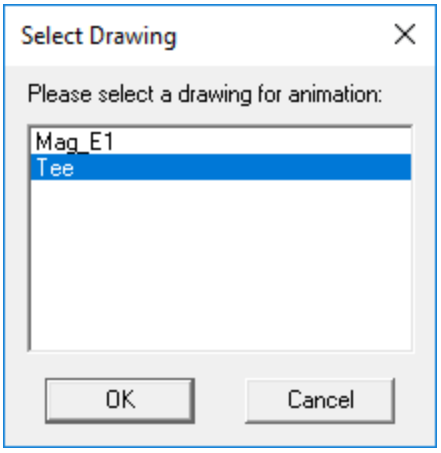
   Select the **View** tab of the ribbon, and click the **Animate** icon.

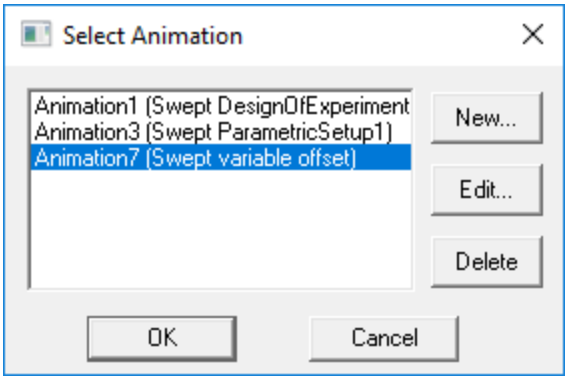
   If multiple geometries can be varied in the design, the *Select Drawing* dialog box appears; proceed to step 2. If only one geometry is variable, proceed to step 3.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the *Select Drawing* dialog box, select the object that you want to animate.

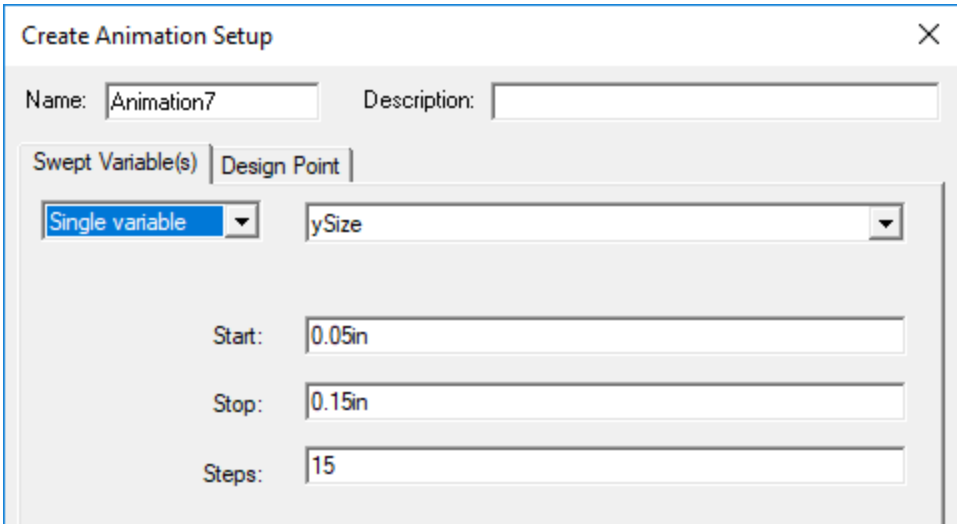


If previous animations have been created for this project, the *Select Animation* dialog box will appear. Selecting an existing animation from that list starts it when you click **OK**. Click **Edit...** to open the *Modify Animation Setup* dialog box. You may choose an animation setup from the list if one is associated with the geometry variable of interest, and the animation will start. If no existing animation setup is acceptable, select **New** and continue at Step 3 below.

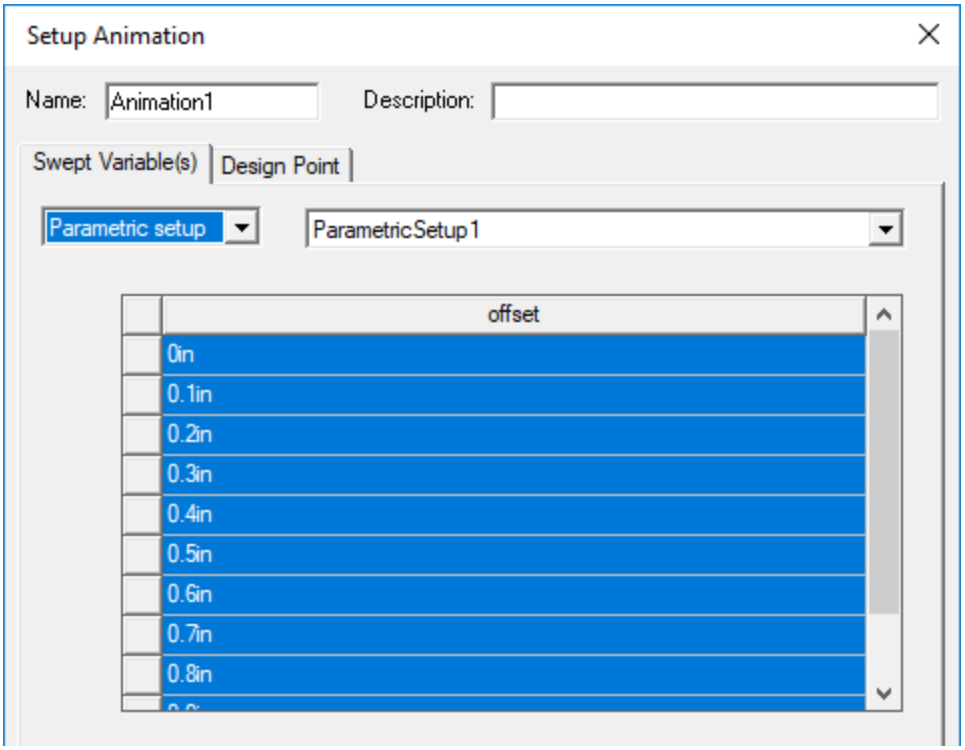


The *Create Animation Setup* dialog box appears:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. For single variable:
   a. Select the geometry variable that you want to animate from the **Swept Variable** list.
   b. Specify the Start, Stop, and Steps values to include in the animation.
4. If your have created one or more Parametric Sweeps for one or more geometric variable, you can select, **Parametric setup**, the setup to use, and the variable values to animate.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. If you have created a **Design of Experiments** setup, you can select to define an animation based on that setup and values.
6. If the design has multiple project or intrinsic variables, you use the **Design Point** tab to set the values of the non-animated variables.
7. Click **OK**.

   The animation begins in the view window. It will display one frame for each variable value.

Creating Parametric Animations

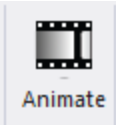
Parametric animations can be based on parametric sweeps. Following is the general procedure for creating an animation of a plot based a parametric sweep.

Prerequisites

- For animation of a field plot/overlay plot over a geometric variable, you must define and solve at least one optimetric sweep with same variable values and Save Fields selected before you can create a parametric animation of a field solution. See Creating Geometry Animations.
- For animation of geometry and VRT plots, solving parametric setup is not required.
- Animations can be based on time variables and objects whose location is linked to the time variable. See the animated example below.

Procedure:

1. Right-click in the **Modeler** window, and click **View**> **Animate...**, or select the **View** tab of the ribbon, and click the **Animate** icon.

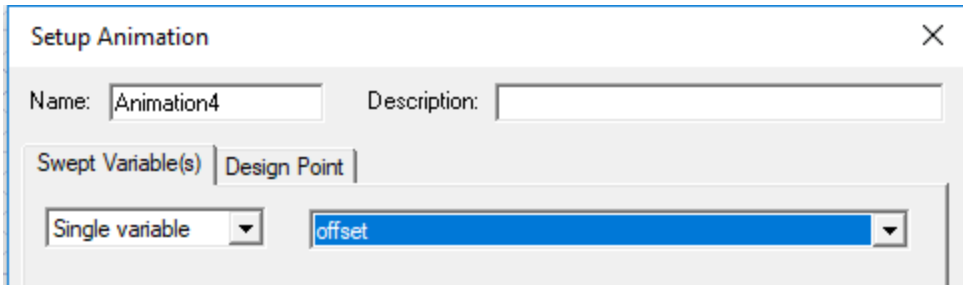
   If multiple geometries can be varied in the design, the *Select Drawing* dialog box appears; proceed to step 2. If only one geometry is variable, proceed to step 3.

2. In the *Select Drawing* dialog box:
   a. Select the geometry variable to vary in the animation.
   b. Select the object you want to animate.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

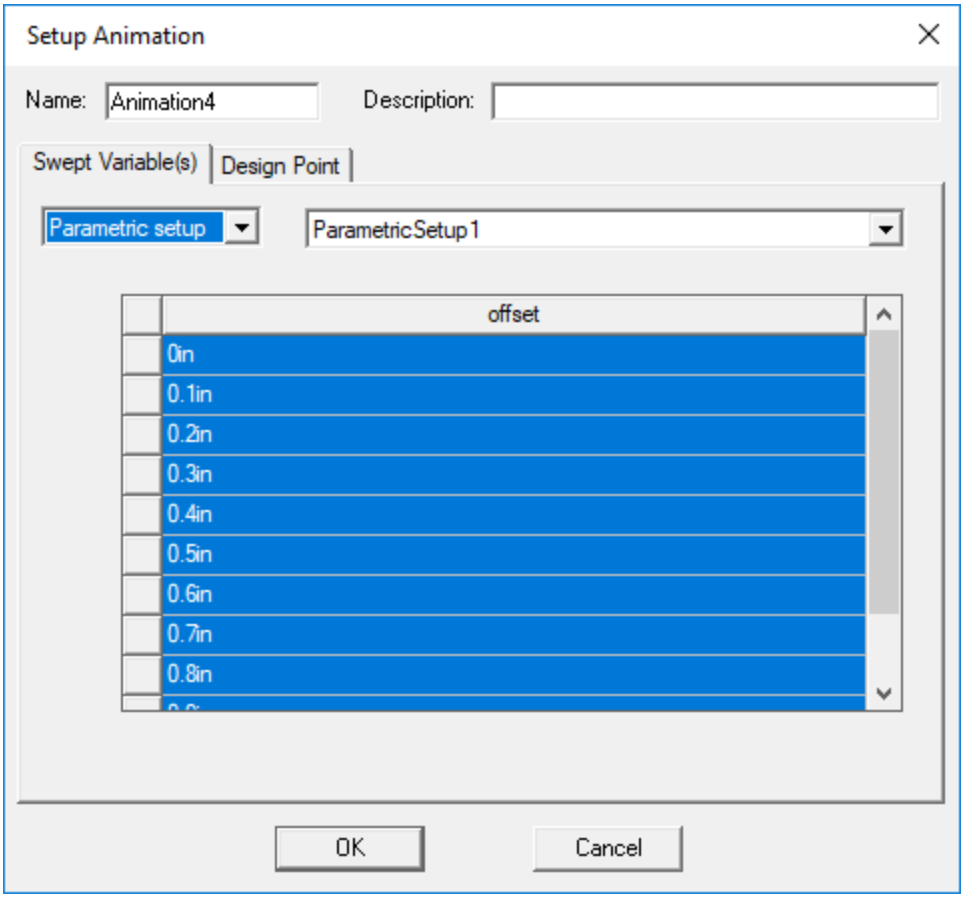
> **Note:**
>
> If previous animations have been created for this project, the *Select Animation* dialog box will appear. You may choose an animation setup from the list if one is associated with the geometry variable of interest, and the animation will start. If no existing animation setup is acceptable, select **New** and continue at Step 3 below.

The *Setup Animation* dialog box appears.



3. If your have created one or more Parametric Sweeps for one or more geometric variables, you can select **Parametric setup**, the setup to use, and the variable values to animate.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

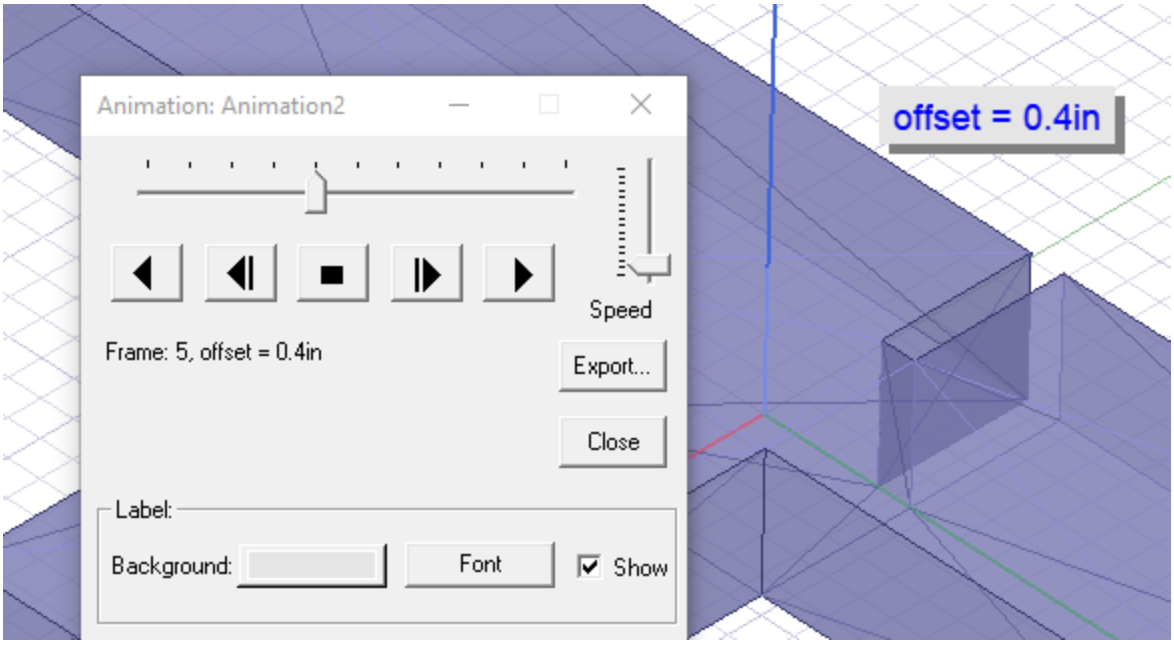


4. Click **OK**.

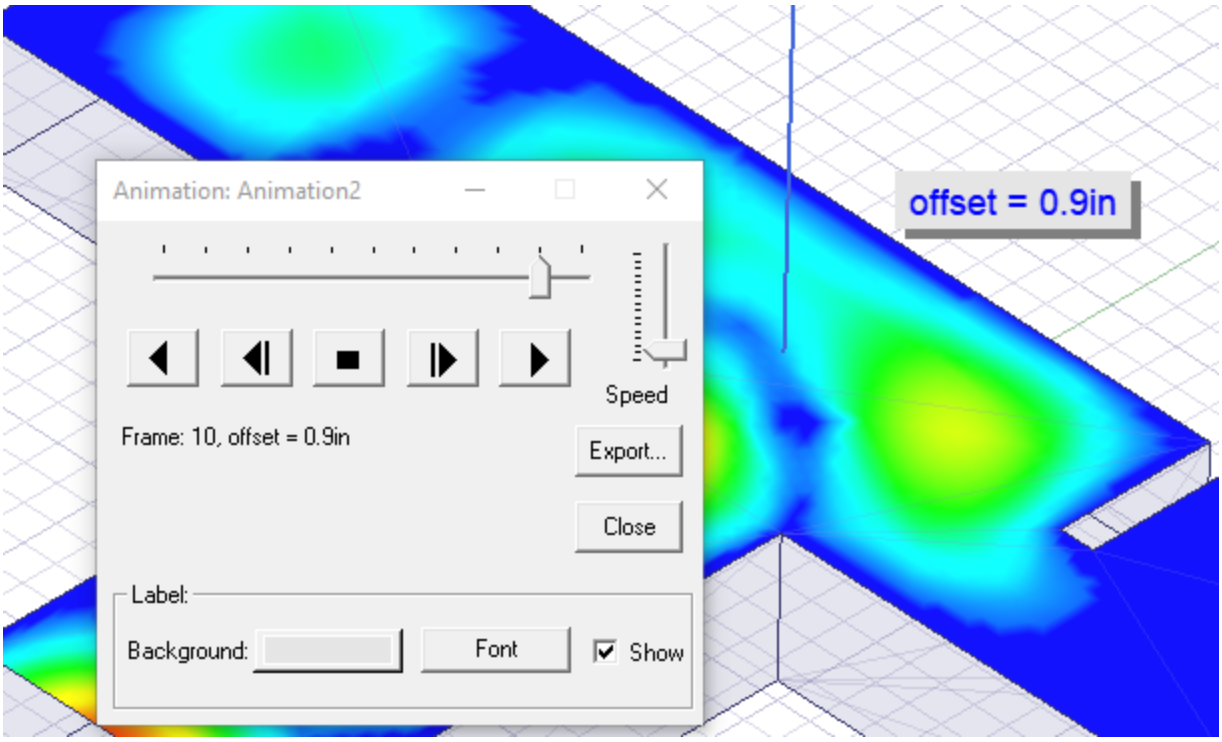
   The animation begins in the view window. It will display one frame for each variable value.

   The **Animation** control panel lets you to stop, restart, and control the speed and sequence of the frames.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



While a parametric animation is running, you can also turn on other field plots.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating Mesh Animations

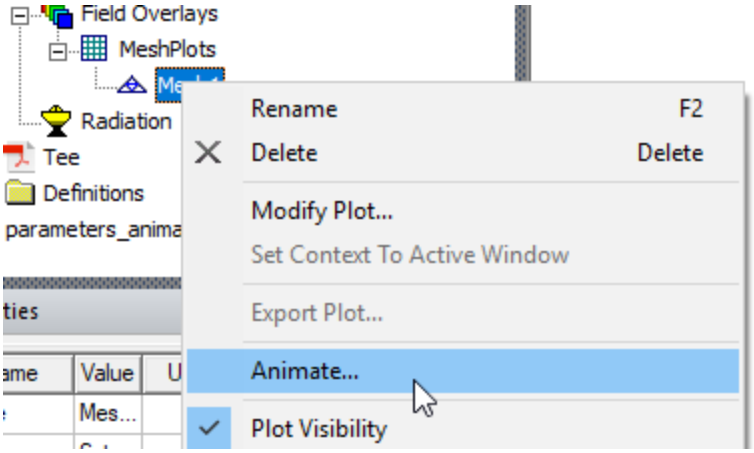
You can create mesh animations to evaluate the effect of single variables or parametric sweep variables on the model. Following is the general procedure for creating a mesh animation that varies a part of the model geometry.

Prerequisites:

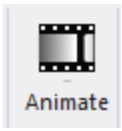
- You must define at least one variable associated with the geometry before you create a mesh animation.
- You must create a Mesh plot.

Procedure:

1. Select the Mesh plot in the Project Manager, right-click in the *Modeler* window, and click **View**> **Animate...**, or right-click the Mesh plot, and click **Animate...**



   Select the **View** tab of the ribbon, and click the **Animate** icon.



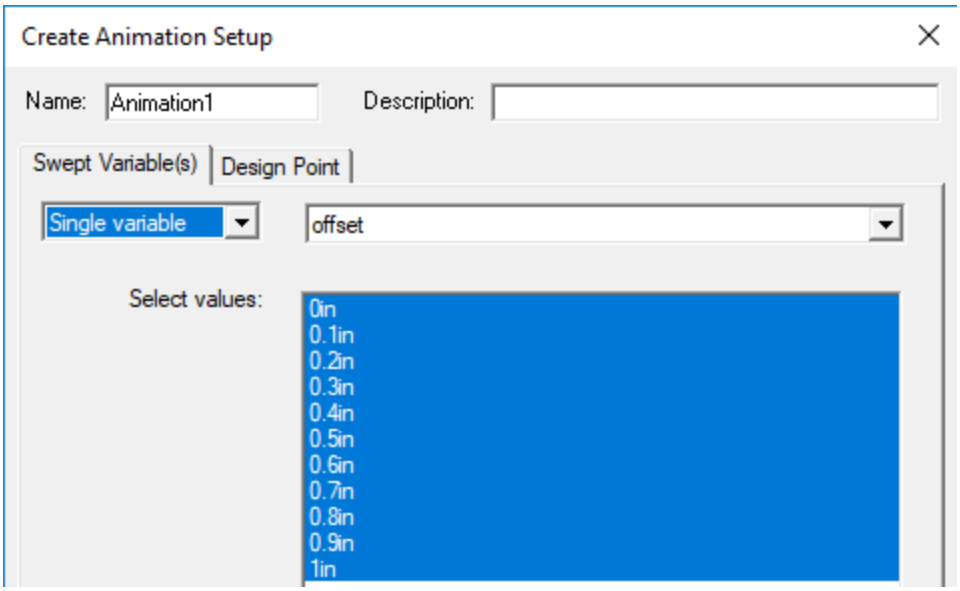
   If multiple geometries can be varied in the design, the *Select Drawing* dialog box appears; proceed to step 2. If only one geometry is variable, proceed to step 3.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the *Select Drawing* dialog box, select the object that you want to animate.

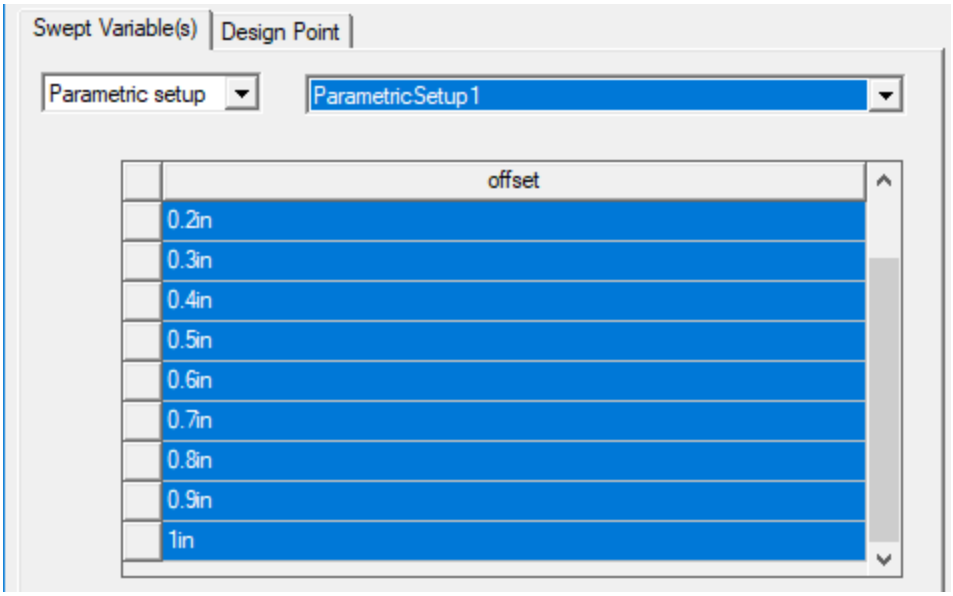
> **Note:**
>
> If previous animations have been created for this project, the *Select Animation* dialog box will appear. Selecting an existing animation from that list starts it when you click **OK**. Click **Edit...** to open the *Modify Animation Setup* dialog box. If no existing animation setup is acceptable, select **New** and continue below.

The *Create Animation Setup* dialog box appears.

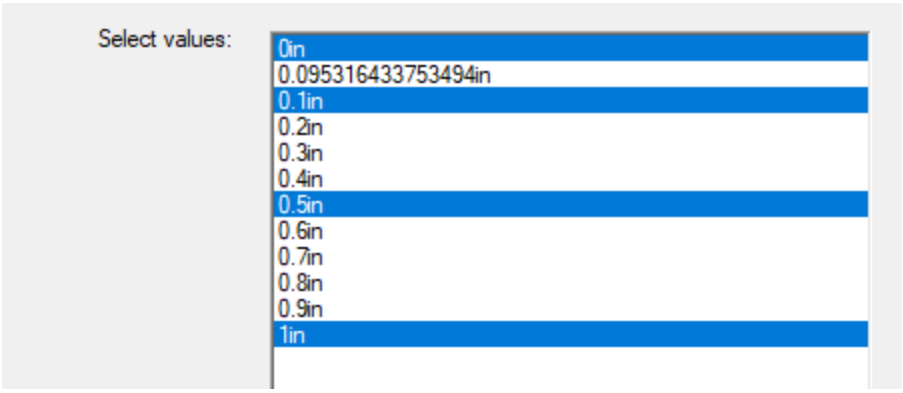


3. In the *Create Animation Setup* dialog box:
   a. Select the Single variable, or Parametric setup, or DOE setup if available. If multiple single variables are available, you can select from a drop-down menu. If one or more Parametric setups exist, you can choose from the drop-down menu. The values for the selected parametric setup are displayed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. If desired, specify the values to include in the animation. Selected values are highlighted.

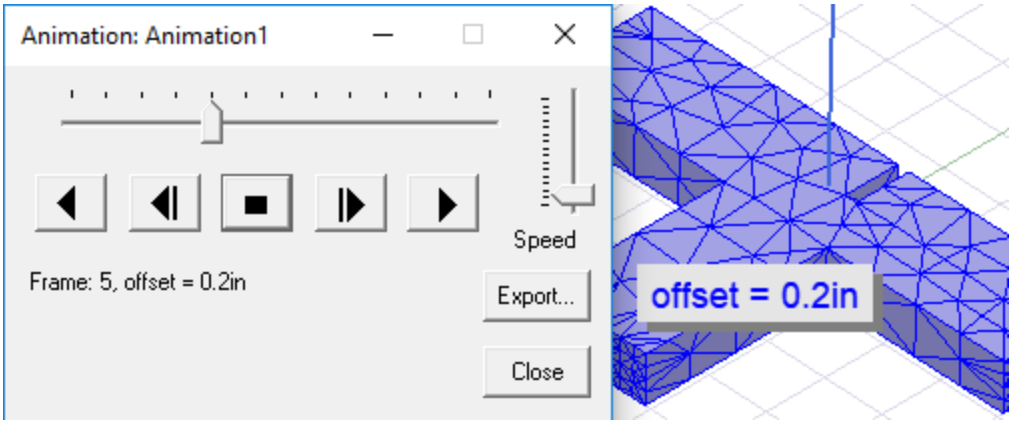


c. If the design has multiple project or intrinsic variables, you use the **Design Point** tab to set the values of the non-animated variables.

4. Click **OK**.

The animation begins in the view window. It will display one frame for each variable value.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

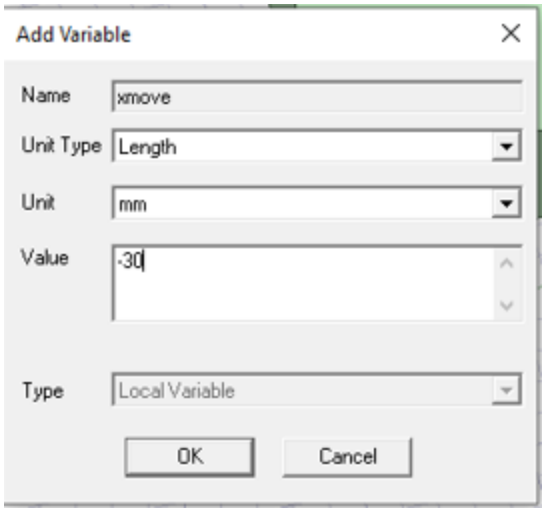


The *Animation* control panel lets you to stop, restart, and control the speed and sequence of the frames.

Animating a Sampled Near Field Plot

When a design or project variable is used to create variations of the design for use with Optimetrics (for example), we can animate the field solutions for the variations in a Sampled Near Field Plot:

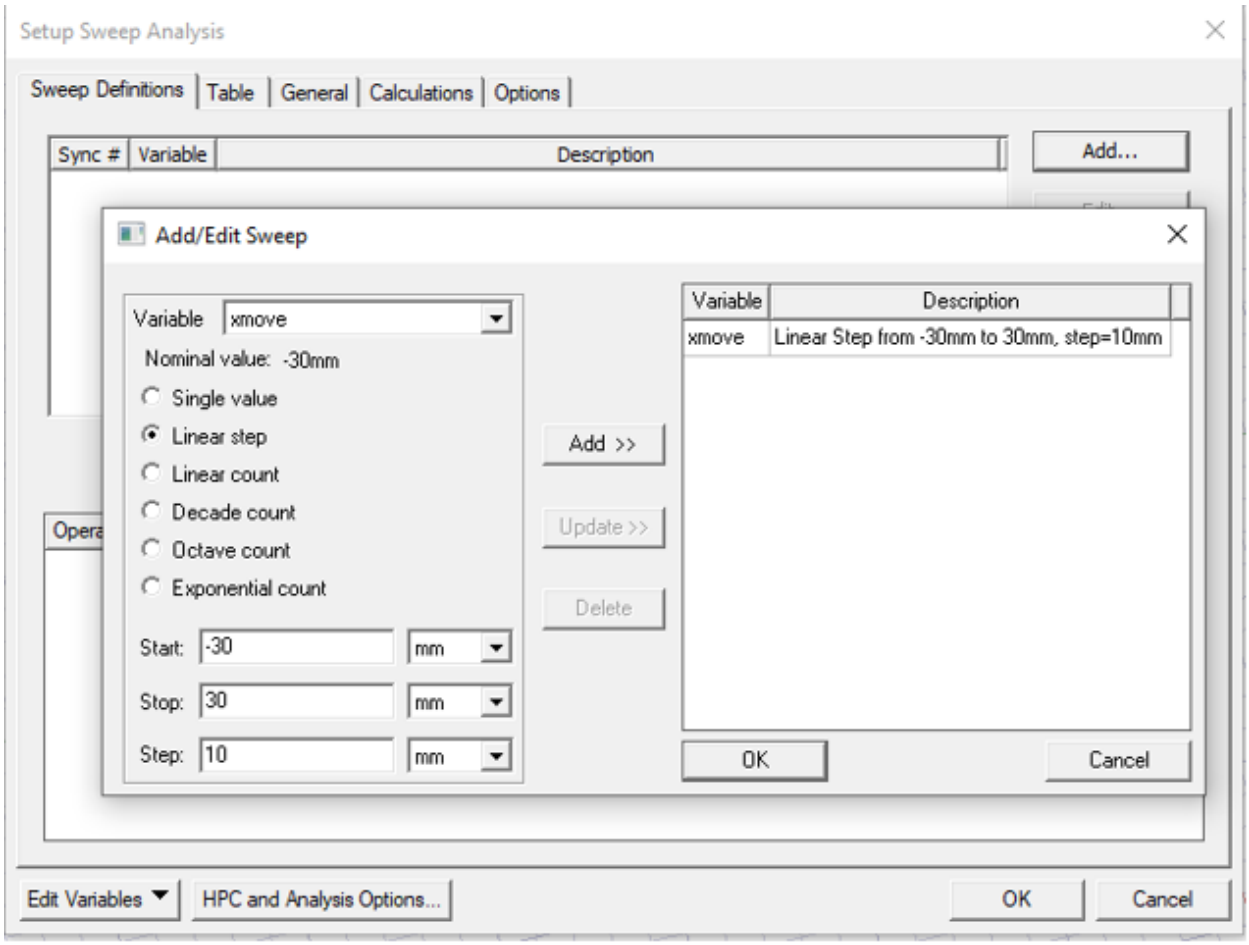
1. Create a design or project variable for the position of a geometry object in the 3D model. You can do this by performing an **Edit** > **Arrange** > **Move** operation on the object, or by creating a new Coordinate System containing the design/project variable and assigning it as the Target Coordinate System for a geometry object.



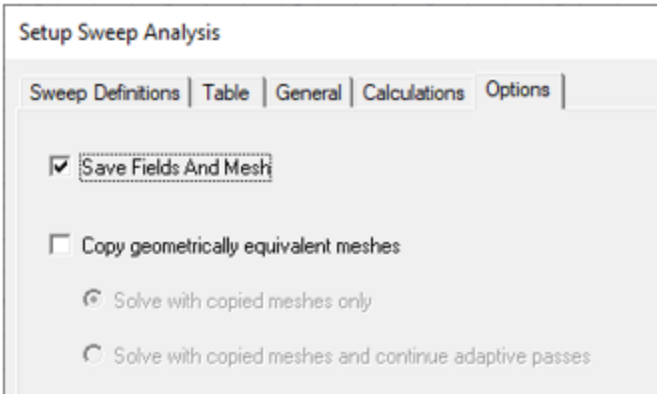
2. Then add an Optimetrics Parametric sweep setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use the design/project variable that you created for the parametric sweep definition.

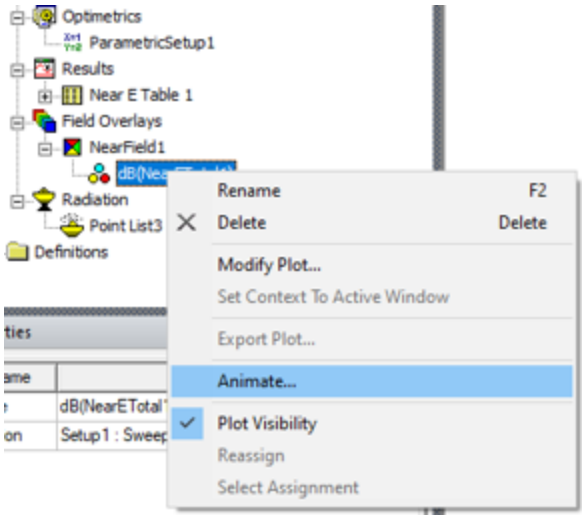


3. Check “Save Fields and Mesh” on the Options page of the sweep setup.



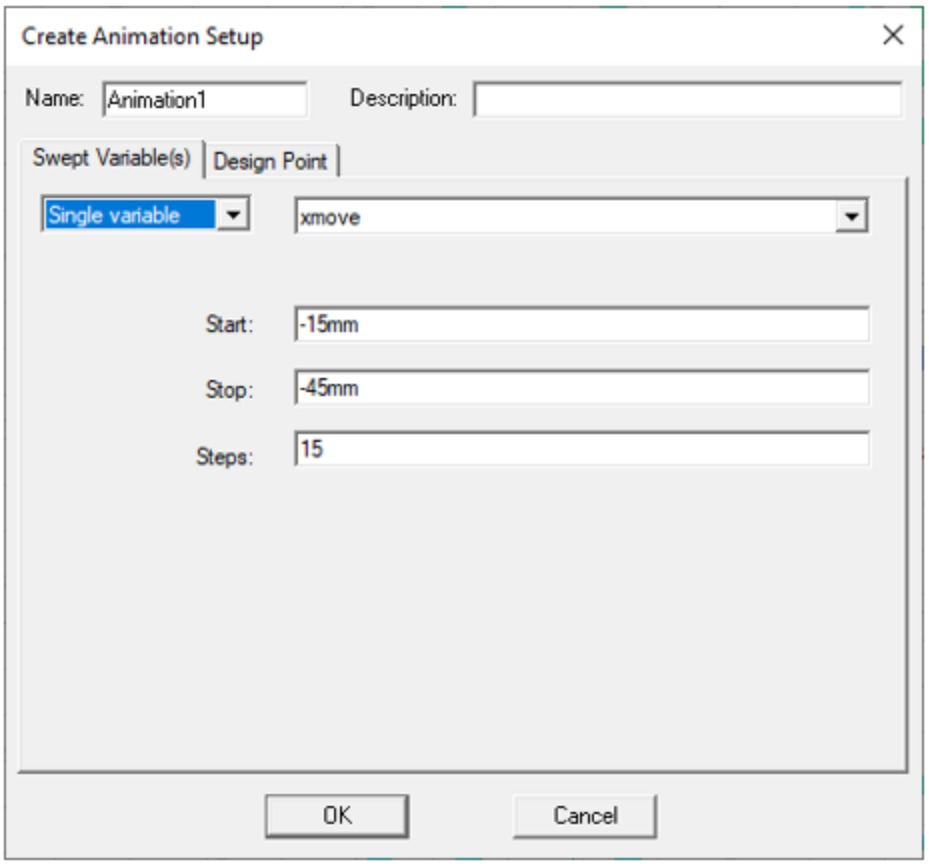
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Run the Optimetrics simulation.
5. After creating a Sampled Near Field Plot as described previously, right-click on the plot object and select **Animate...** to open the **Create Animation Setup**.



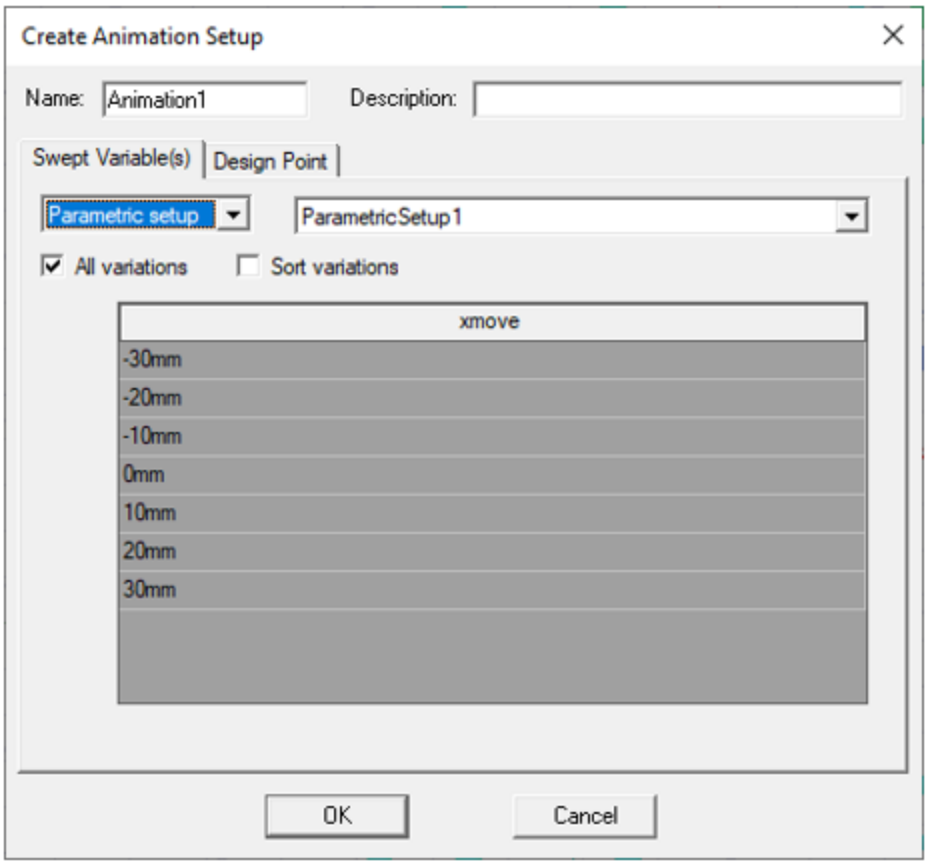
You can select the design/project variable from the list of available sweep variables and create a list of variable values to sweep for the animation. These values should be from the available simulation sweep values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



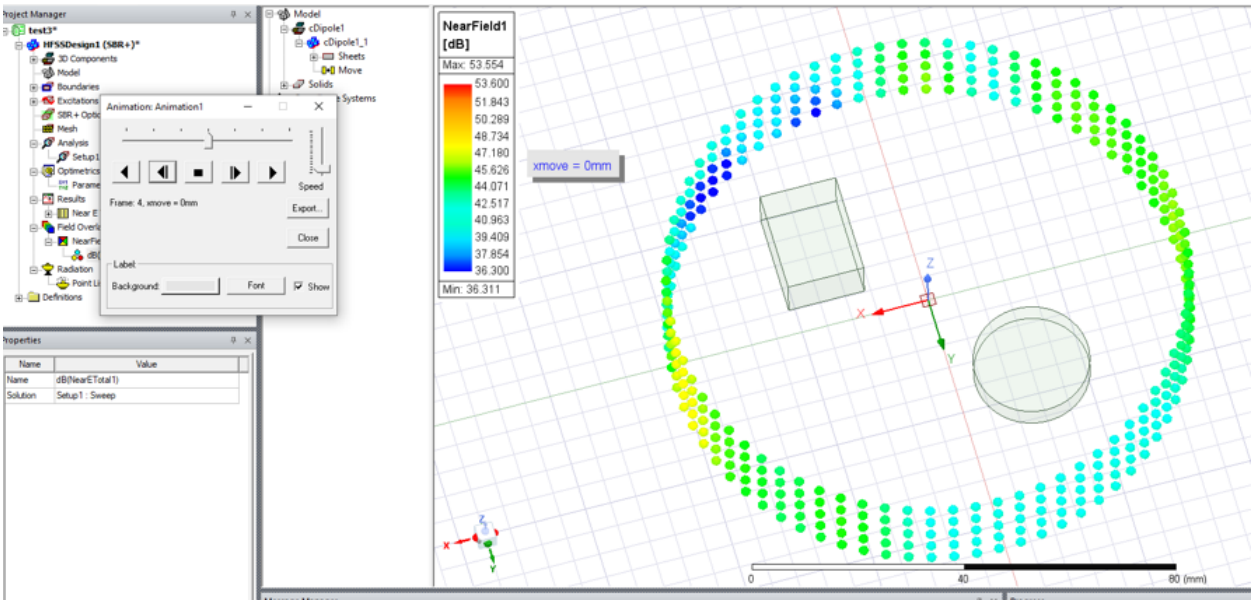
Or you can select the Parametric Setup to use the variable and all values from the specified Optimetric setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. Select the created animation sweep setup and click **OK** to generate the animation. The progress bar monitor updates status as all the animation frames are generated.

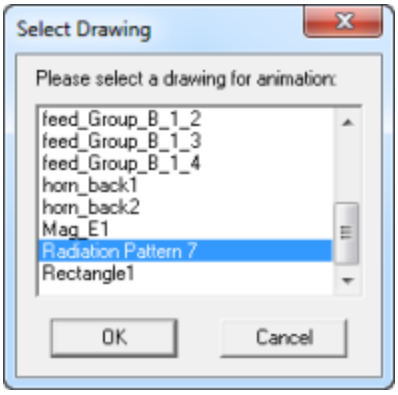
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Creating 2D Radiation Field Plot Overlay Animations

In order for a Radiation Pattern plot to be valid for overlay animation, either theta or phi must be set to a single value, so that the pattern lies in a single plane in 3D space. Note that for animation, the plot must contain at least two traces within this plane - for example traces for multiple frequencies of a discrete sweep. You create the 2D Radiation Field plot as before, but must also prepare it to be suitable for animation. See Overlaying 2D Radiation Field Plots on Models.

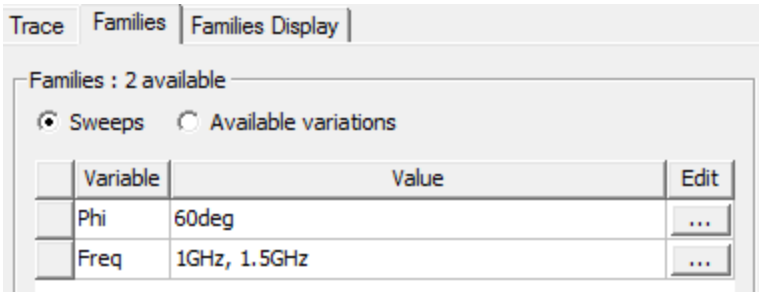
If an existing 2D Plot is a candidate for animation, you will see it listed when you click **View>Animation** to open the **Select Drawing** dialog.



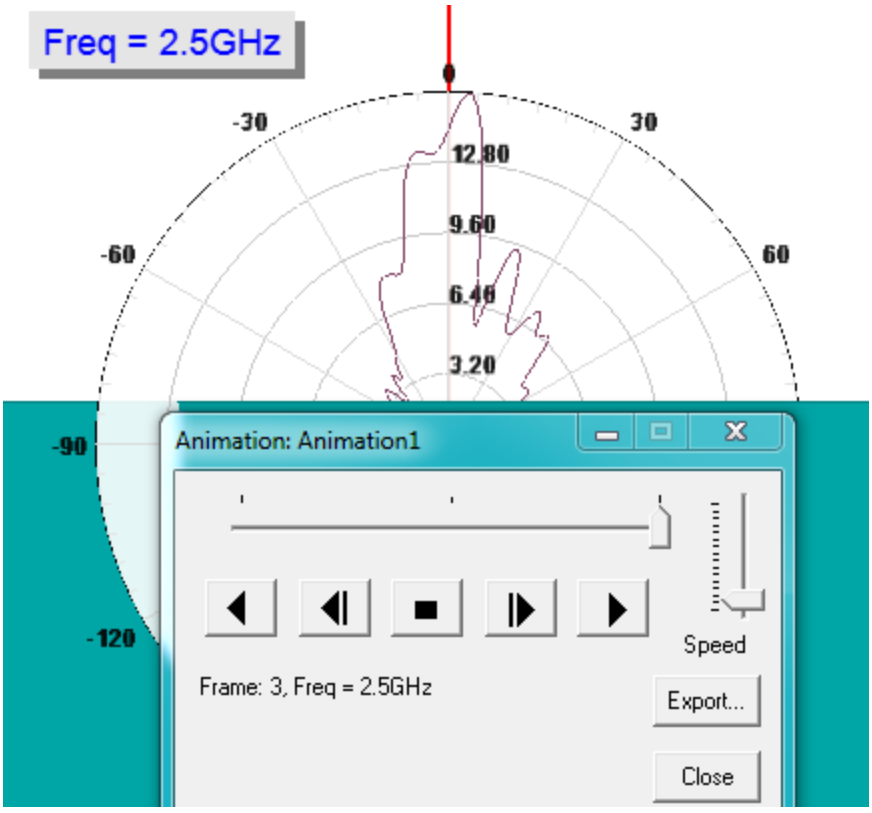
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

However, if the Radiation Pattern plot does not have at least two traces, when you try to set up the animation, you will receive a notice that you need to Modify the plot or simulation setup appropriate or create a new one.

Recall that the Context section on **Modify Plot** may let you select Setup*n*: Sweep, which will permit you to select additional frequencies. Also the Trace tab and Families tab lets you specify a single Phi and/or Theta and select multiple values for other available Variables such as frequency.



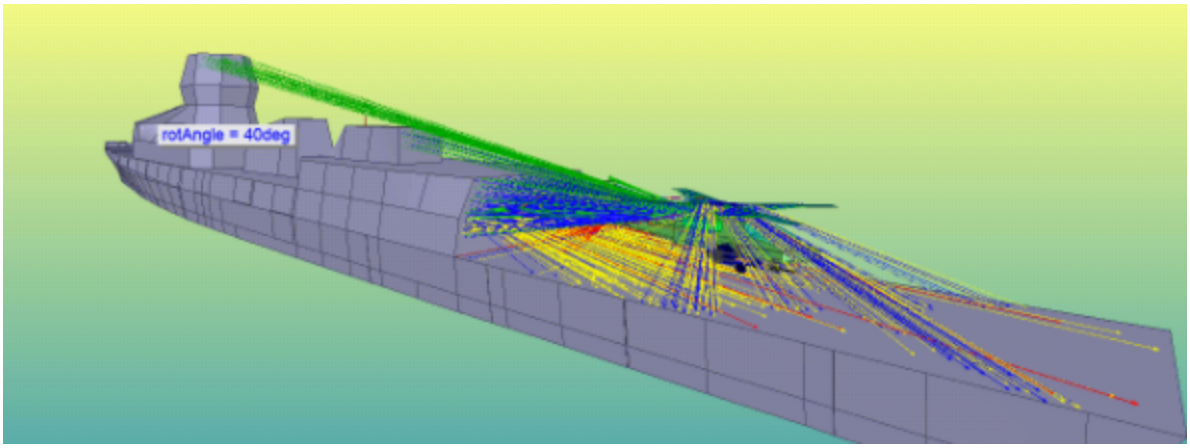
For plot with multiple traces, the animation proceeds through the existing traces.



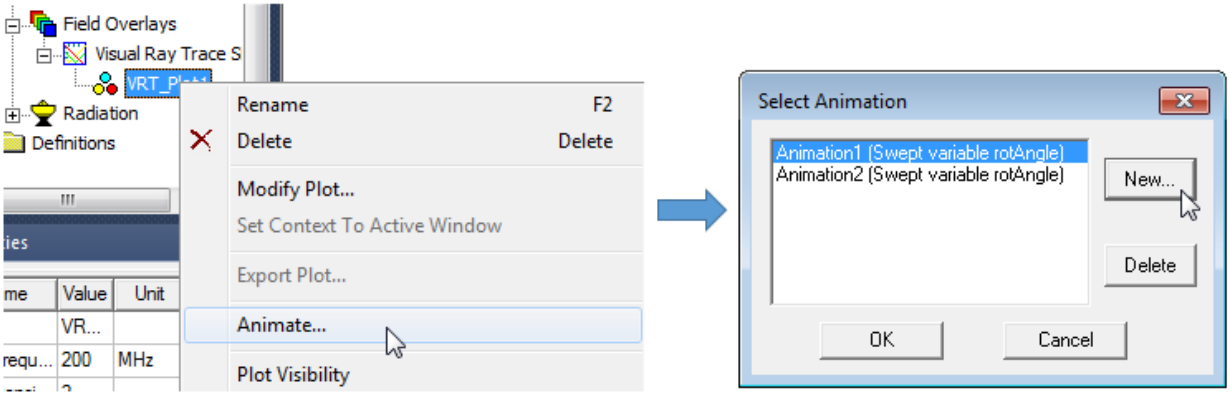
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Animating VRT Plots

Visual Ray Trace (VRT) is a powerful feature available in HFSS for visualizing the ray geometry and interactions for an SBR+ simulation. VRT plot animation provides a means to dynamically explore how geometry changes or ray launch point changes affect ray generation. Compared to regular field or mesh plot animation, VRT plot animations differ in that you do not need to solve or generate a mesh before animating a VRT plot.

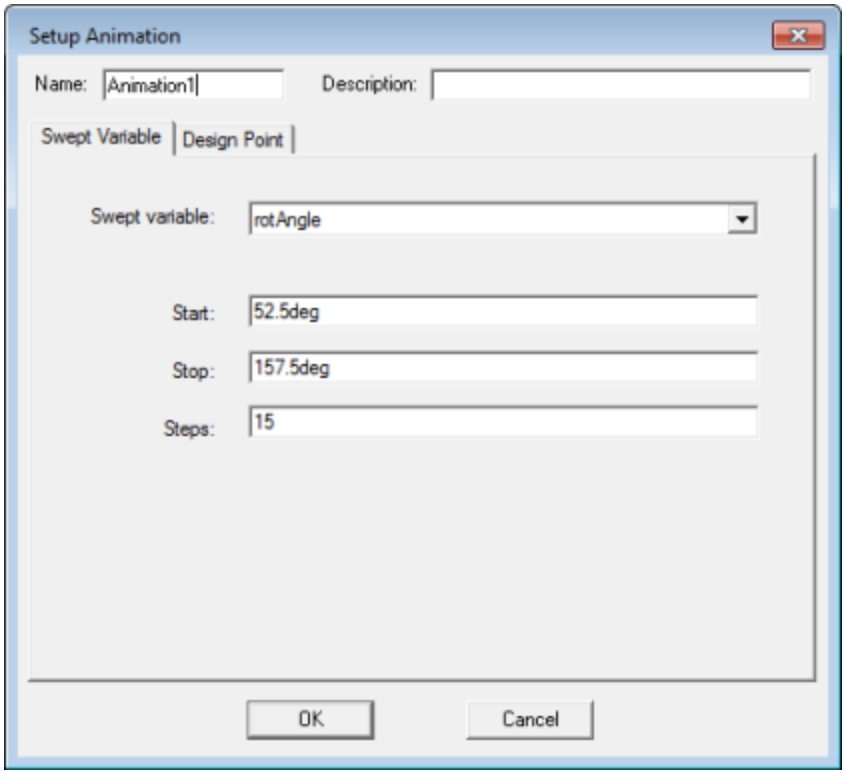


1. To create a new VRT animation, either:
   - Right-click a VRT icon in the Project tree and select **Animate...** from the short-cut menu and select **New** in the **Select Animation** dialog box.
   - Click **HFSS>Fields>Animate...** and select **New...** in the **Select Animation** dialog.



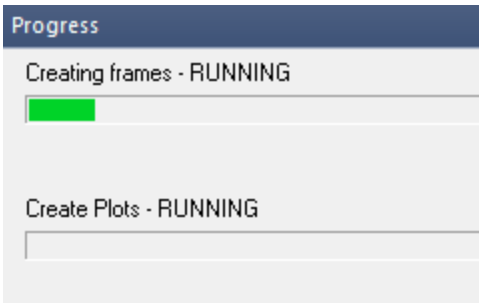
This opens the Setup Animation dialog:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



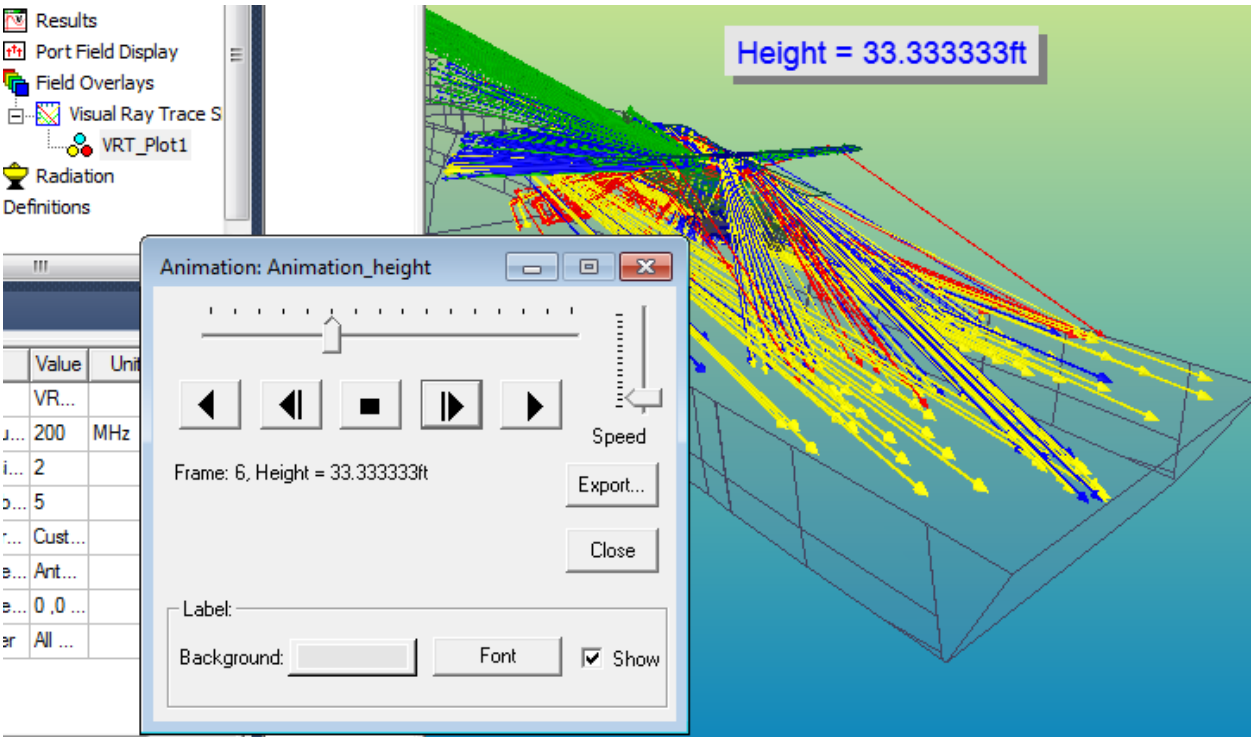
2. Select the Swept variable. If multiple swept variables are available, you can select from the drop-down menu.
3. Specify the Start, Stop and Steps for the animation.
4. The **Design Point** tab has Use defaults checked. If you uncheck it, you can see a listing of design points.
5. If desired, provide a name and description
6. Click OK to produce the animation.

You can click **Show Progress** to watch the progress bars. The arrow button on the right includes an **Abort** command.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the animation frames have been generated and the plots completed, the animation displays in the **Modeler** window, and the **Animation** control dialog appears.



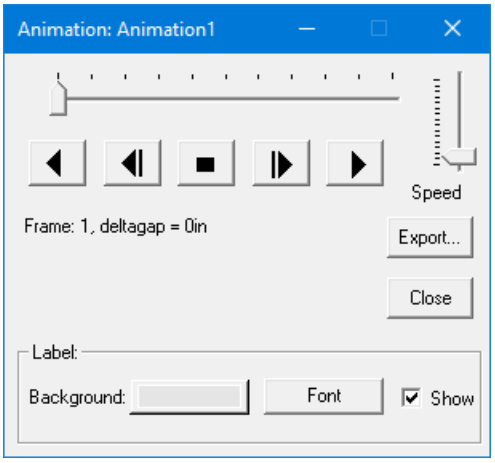
To launch an existing VRT animation, either:

- Right-click a VRT icon in the Project tree and select **Animate...** from the short-cut menu.
- Click **HFSS>Fields>Animate...** and select from any existing animations listed

Controlling the Animation's Display

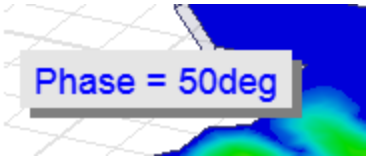
When an animation is displayed in the view window, the **Animation** window (also called the *play panel*) appears in the upper-left corner of the desktop. It has buttons that enable you to control the speed and sequence of the frames, start and stop the animation, and export the animation. Click an area of the window image below to learn its function.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



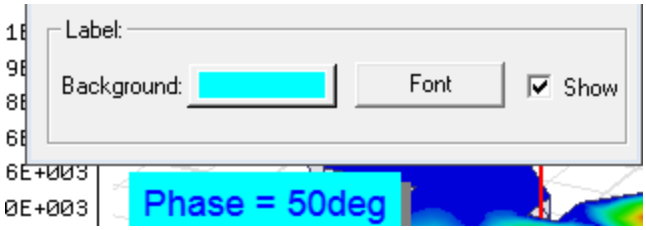
Animation slider	Each dot on the slider represents a frame in the animation. Drag the slider to the right to display the next frame in the animated plot. Drag the slider to the left to display the previous frame in the animation.
	Plays the plot's animation sequence backwards.
	Steps backward through the animated plot one frame at a time.
	Stops the animation.
	Steps forward through the animated plot one frame at a time.
	Plays the plot's animation sequence forwards.
	Drag the **Speed** slider to the top to increase the speed of the animation. Drag the **Speed** slider to the bottom to decrease its speed.
Frame information	The current frame and phase at which the plot is being displayed is listed below the control buttons.
	Enables you to export the animation to an animated Graphics Interchange Format (GIF) or to Audio Video Interleave (AVI) format.
	Closes the animation window.
Show label check box	If you select the Show check box, a label showing the swept variable value appears in the animation. You can select the label with the mouse and drag it to a location.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Background button.

Click the background button to open a color pallet dialog that lets you set the background for the swept variable label.



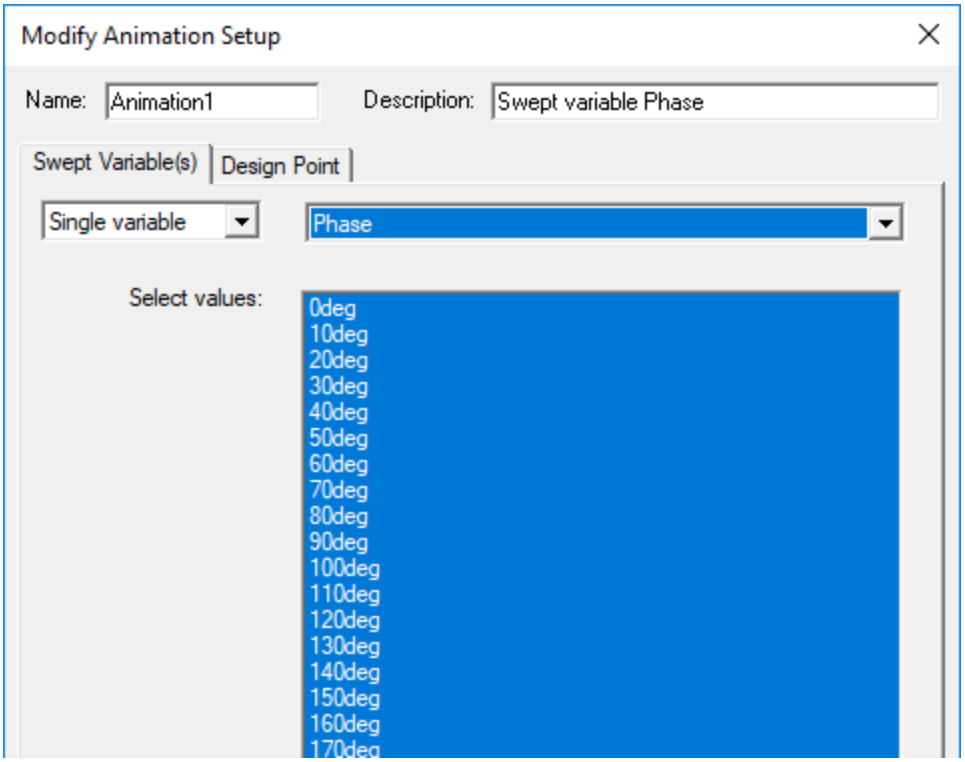
The font button opens a font selection dialog that you can use to set the Font, Font Style, and Size for the label. The Default is Arial Narrow 14pt.

Modifying an Animation Setup

The **Edit** button on the *Select Animation* dialog box opens the *Modify Animation Setup* dialog box.

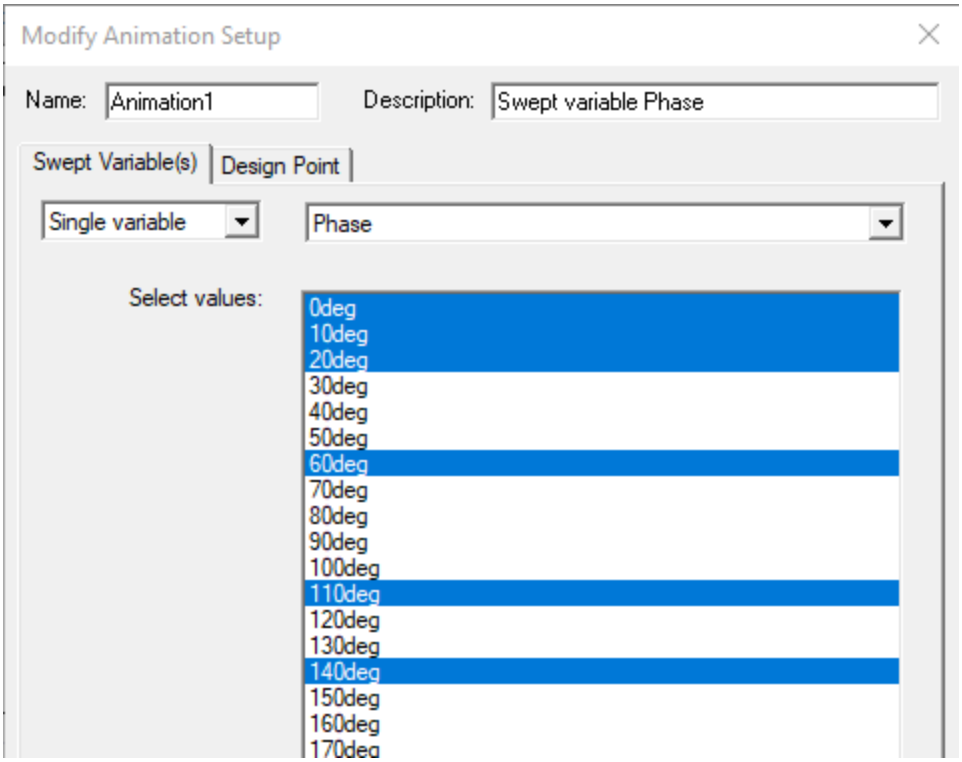
It opens showing the selections for variables and the selected values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

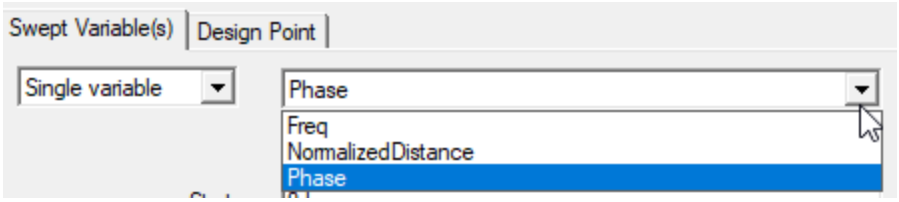


You can de-select or select any values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

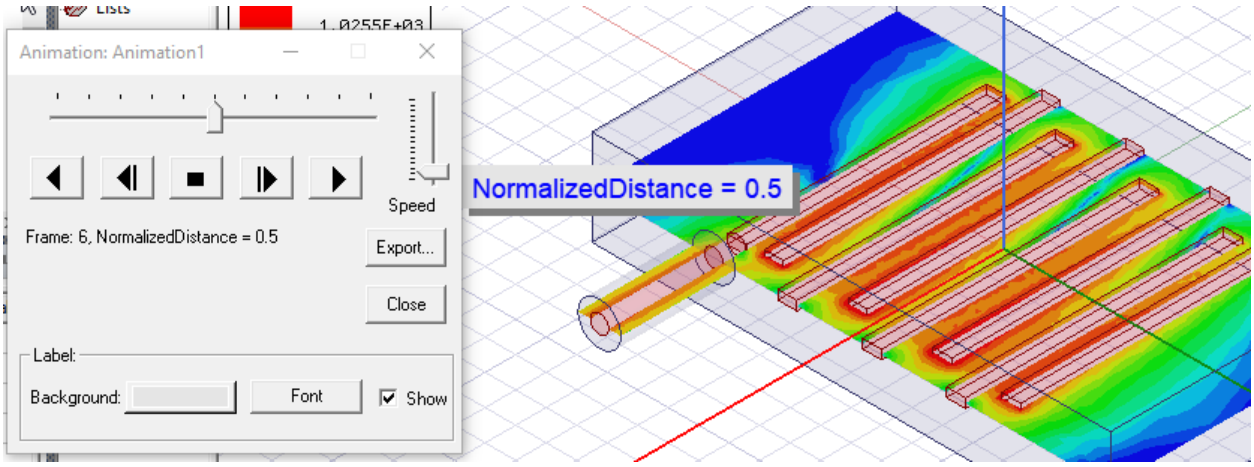


If other variables have been defined, you can select from the drop down list.

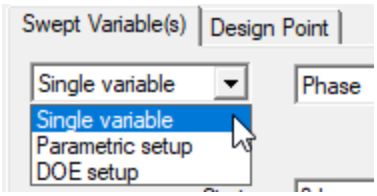


If the other variables have or can be set to appropriate values, you can modify the animation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If Parametric Setups or DOE Setups are available, you can select these.



Creating Animations

Controlling the Animation's Display

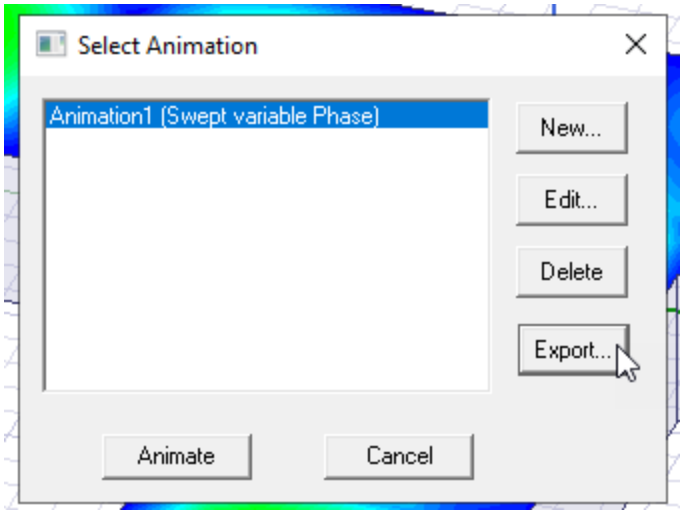
Exporting Animations

You can export animations from the **Animation** control dialog, or, for Ensight format, from the **Select Animation** dialog.

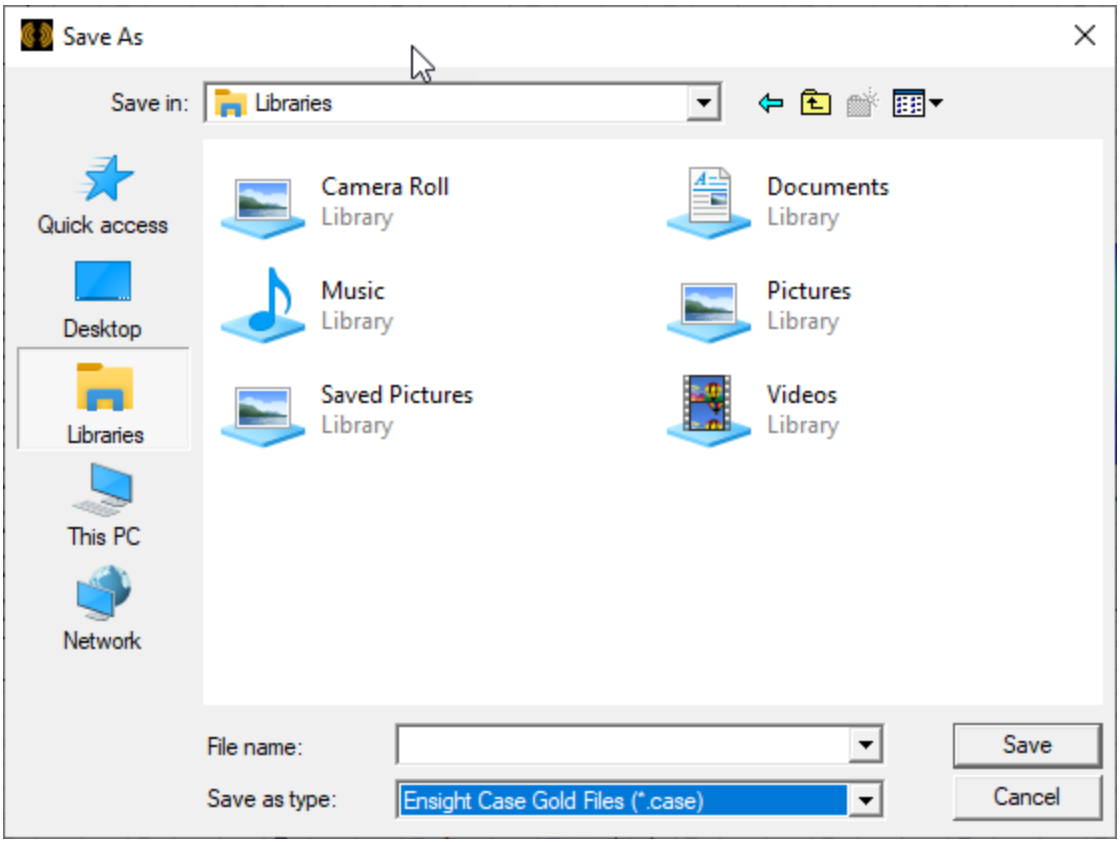
Exporting Ensight from the Select Animation Dialog

1. Create the animation you want to export.
2. In the **Select Animation** dialog, select **Export...**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



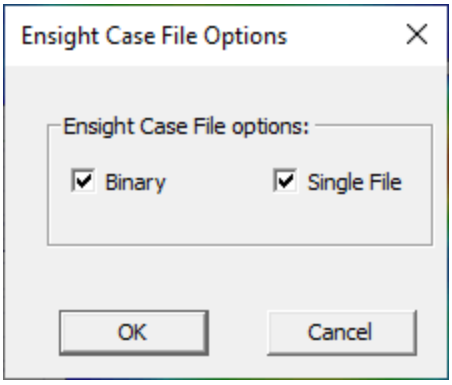
3. This opens the Browser window.



Navigate to the desired location and provide a file name. The format will be Ensight .case files.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

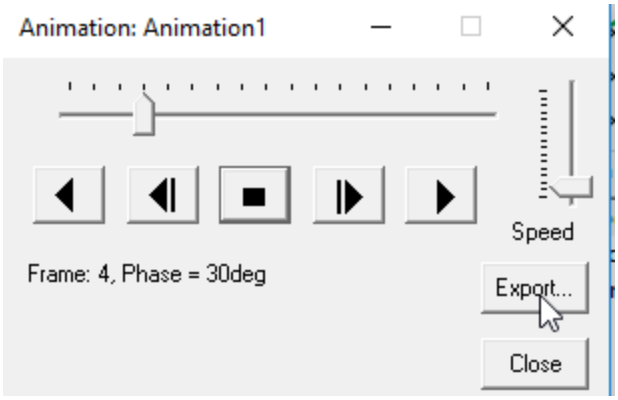
4. When you select **Save**, the **Ensight Case File Options** dialog opens.



The binary option permits a more compact and efficient animation creation process in Ensight. The Single File options leverages the Ensight transient dataset capability to export all frames of field/geometry data to a single case file. On **OK**, every visible and animatable field plot should generate a case file with name in the format of "UserTypedName_fieldPlotName.case"

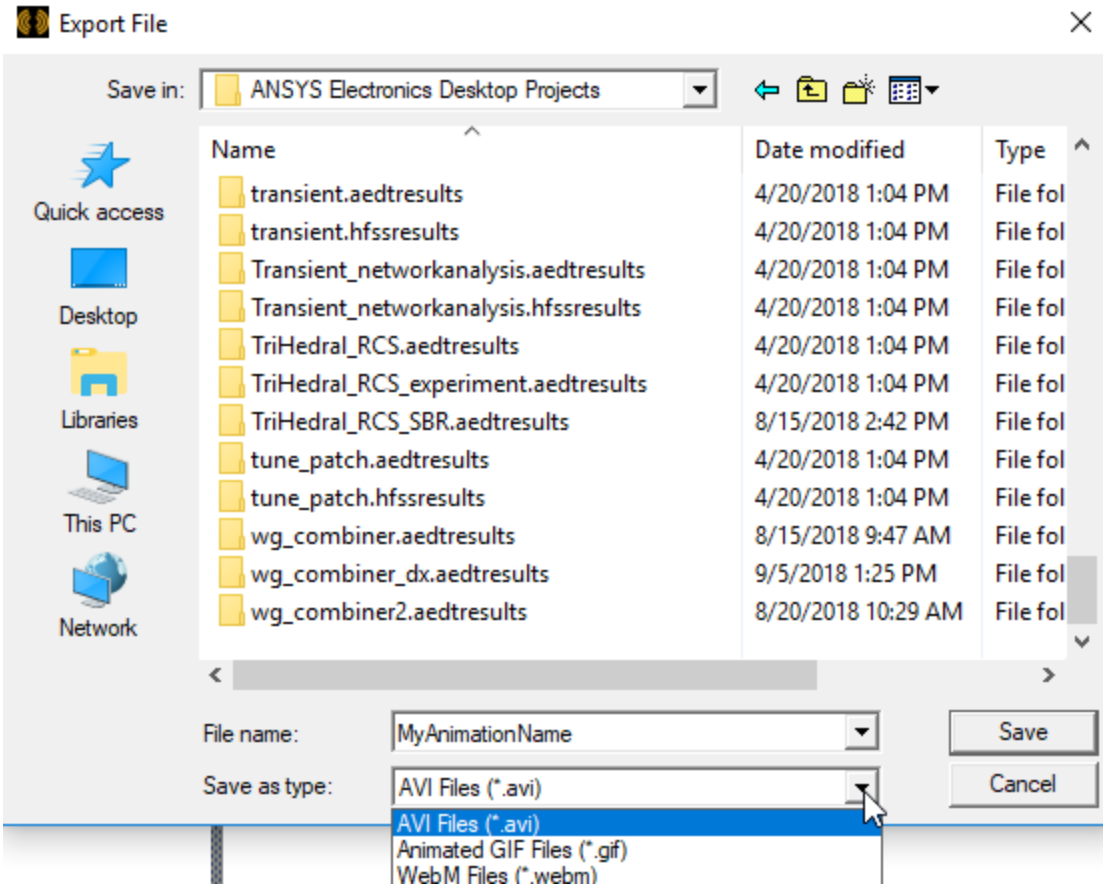
Exporting from the Animation Control Dialog

1. Create the animation you want to export.
2. In the play panel, click **Export...**.



The *Export File* dialog box appears.

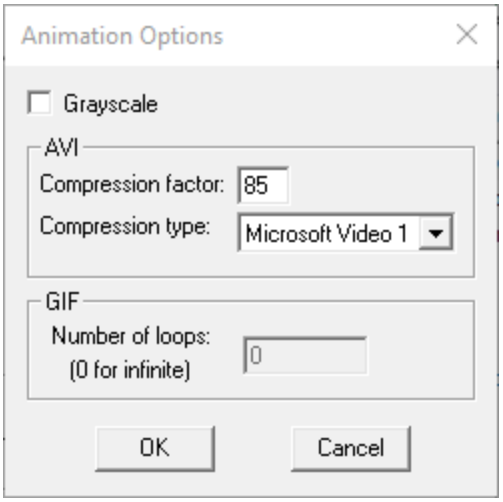
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Specify the directory to **Save in**, the **File name**, and use the **Save as type** drop-down menu to select **Animated GIF File (.gif), AVI File (.avi),** or **WebM File (.webm)**.

   The *Animation Options* dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. To replace colors in the file with 256 shades of gray, select **Grayscale**.

   Grayscale animations tend to use less memory than full color animations.

5. For AVI format export, specify the **Compression factor** (the default is 85) and one of the following **Compression types**:
   - INTEL Indeo
   - Cinepak
   - Microsoft Video 1
   - None
6. For GIF format export, specify the number of loops. The default "0" denotes infinite loops.
7. Click **OK** to close the *Animation Options* dialog box.

   The animation is exported to the file format you specified.

Modifying SAR Settings

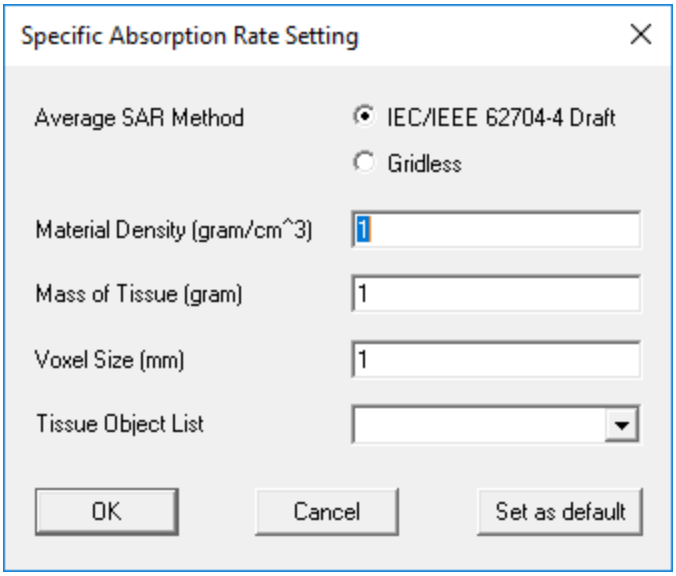
HFSS uses default specific absorption rate (SAR) settings when creating a local SAR or average SAR field overlay plot. It does use mass density for each material if defined for each material. Otherwise (that is, if the mass density for that library material is 0), the density in the **Specific Absorption Rate Setting** dialog will be used.

To change the default settings:

1. Click **HFSS**>**Fields**>**SAR Setting**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Specific Absorption Rate Setting** dialog box appears.



2. To select the Average SAR Method, select the radio buttons for IEC/IEEE 62704-4 Draft or Gridless, which is the legacy Average SAR algorithm. Selecting Gridless disables the Voxel Size and Tissue Object list fields.
3. In the **Material Density** text box, enter the mass density of the dielectric material in g/cm^3. This provides a default mass density if not specified in the material definition.
4. In the **Mass of Tissue** text box, enter the mass of the material in grams that surrounds each mesh point. The standard values are 1 and 10, corresponding to US or European standards; however, for research purposes you can enter values outside this range.
5. In the **Voxel Size** field, specify a size. The units are millimeters.

The voxelization process takes all elements as rectilinear. So it is better to solve projects with curvilinear off. See Specifying Initial Mesh Settings.

Since the number of voxels depends on the size of the model and the size of the voxel you can expect that computational complexity increases in $O(n^3)$ or more as the model size increases and the voxel size decreases. Below are a few suggestion:

- For large models you can set up an average SAR plot before you solve so that the data from the first phase could be generated at solve time. With the tissue setting you can also perform SAR calculation over partial model.
- Set the solver order to 1 for best performance.
- The algorithm takes full advantage of multi-cores. Set the number of processors to use in the desktop performance tab in **General Options**. If your machine is hyper threaded, you should only use half the number of cores.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

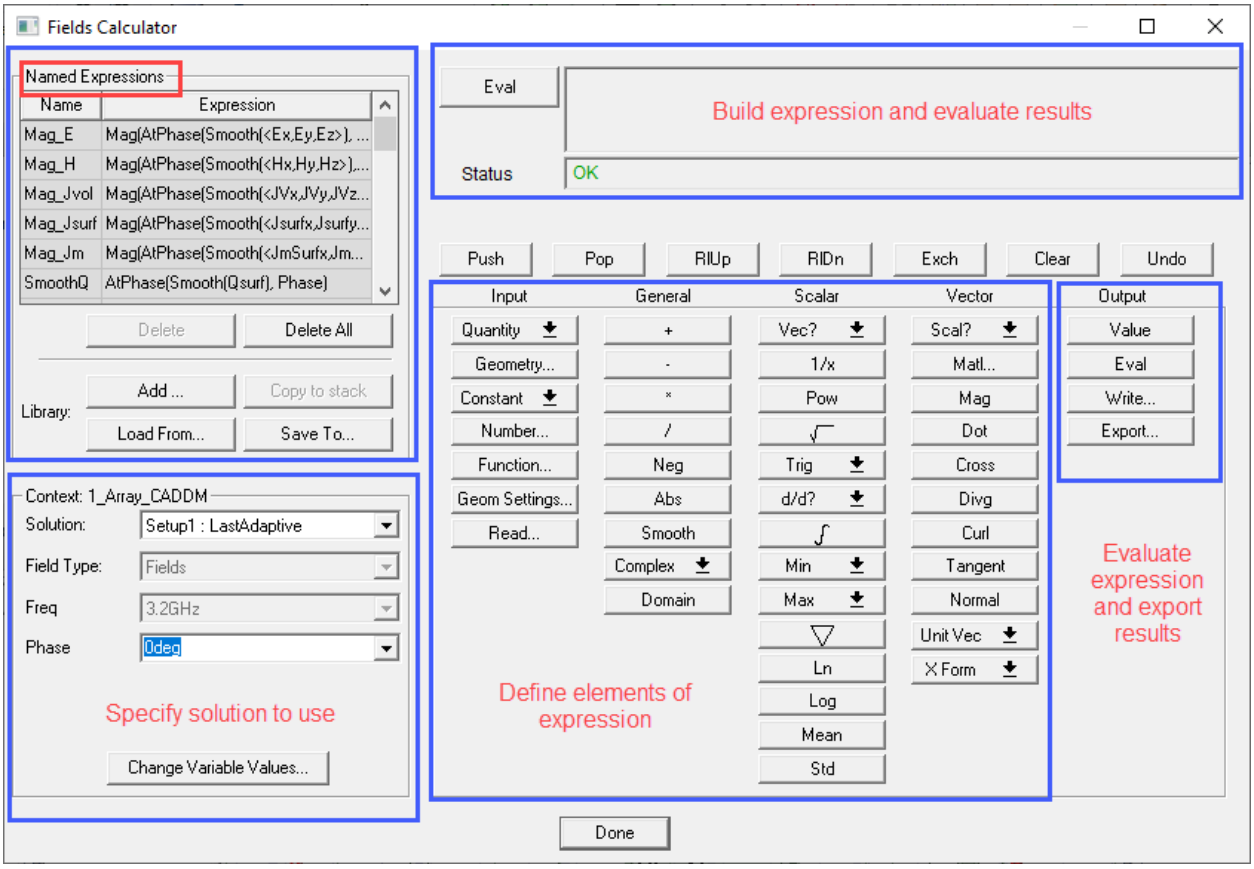
6. If the Tissue Object List is empty, all conductors are treated as tissues. It is recommended that you create an object list explicitly to be used as tissue objects. This is to ensure that you have considered which objects should be included in the SAR calculation. You can create object lists for the menu by selecting objects and clicking, **Modeler>List>Create>Object List**. (See Creating an Object List.) After saving the Project, you can then select from available object lists. When you select a list, all objects in the list are treated as tissues.
7. Click **OK**.

Using the Fields Calculator

The Fields Calculator is a very powerful tool for post processing. It is available for HFSS, Maxwell, Q3D, Icepak, and Mechanical solutions. While standard post -processed results (such as S-parameters, Y or Z matrix, animated field plots, and near and far field patterns) serve most simulation requirements, the Fields Calculator enables you to perform further computations using basic field quantities. The calculator computes derived quantities from the general electric, thermal, or displacement field solution; writes field quantities to files; locates maximum and minimum field values; and performs other operations on the field solution. Using this calculator, you can perform mathematical operations on all saved field data in the modeled geometry at a single frequency. The resulting quantities can be plotted, tabulated, or exported.

The Fields Calculator includes predefined expressions appropriate for each solver and lets you create and save additional named expressions.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



At the top-left corner of the calculator is a list of **Named Expressions**, which are standard or user-defined field quantities that are accessible from outside of the calculator. They can be added, copied to stack, saved to, or loaded from a library file using the buttons right beneath the list.

At the lower-left corner of the calculator is the **Solution Context** section, in which you can select the desired solutions, field types, frequency, and phase for the current session.

The top right of the calculator contains the **Data Stack**, in which calculator entries are held in stack registers. The data type in the Data Stack is denoted by its prefix abbreviation.

Immediately beneath the stack is the row of **Stack Command** buttons that define some basic operations for the data in the Data Stack.

The bottom half of the calculator holds the columns containing the actual calculator buttons, organized into columns, classifying them by the type of operation and the type of data upon which the operation can be performed. These columns are headed Input, General, Scalar, Vector, and Output. At the very bottom of the calculator is the button to exit, **Done**. The Fields Calculator interface is discussed in detail in subsequent sections.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The calculator does not perform the computations until a value is needed or is forced for a result. This makes it more efficient, saving computing resources and time; you can do all the calculations without regard to data storage of all the calculated points of the field. It is generally easier to do all the calculations first, then plot the results.

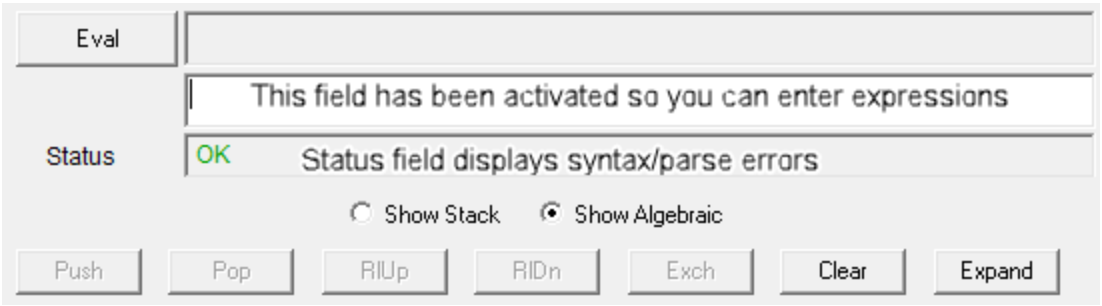
The help for the Fields Calculator includes a "Cookbook" section that provides step-by-step examples of the following operations:

- Calculating Numerical Quantities
- Calculating Quantities for 2D (Line) Plot Outputs
- Calculating Quantities for 3D (Surface or Vector) Plot Outputs
- Calculating Quantities for 3D (Volume) Plot Outputs
- Calculating Quantities for Animated Outputs
- Creating a User Defined Named Expressions Library

Working through these examples is a good way to better understand how to use the Fields Calculator and to see what it can do for you.

HFSS Support Expression Input in Fields Calculator (Beta)

With the **HFSS Support Expression Input in Fields Calculator** Beta option for HFSS, you can create custom field quantities using algebraic expressions. You can enter equations directly in the Data field, which is now editable. The text entered is in black when it parses and red when it does not. The Stack Command buttons can also be used to define an algebraic expression; the input text will be inserted at the current insert point in the Data field.



An algebraic expression can be saved in the Named Expressions list. If you select **Show Algebraic**, all the named expressions are listed in algebraic format. If **Show Stack** is selected, the named expressions will be listed in stack format.

An algebraic expression is either a `namedExpression`, a `constant`, a `name`, an `operator`, two `expressions` separated by `binaryOperator`, or an `expression` enclosed in parenthesis.

An `operator` is an *opName* followed by `arguments`.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

An `argument` is either empty parenthesis, an `expression` enclosed by parenthesis, or two `expressions` separated by a comma and enclosed by parenthesis.

A `binaryOperator` is one of the following literals: '+', '-', '*', '/'

A `name` is either a project, design, or intrinsic variable (scalar or vector), or a geometry.

A *constant* can be one of the following:

- scalar literal; for example, "1.5e-10"
- complex number; for example, "3.5-1.0i" or "<(3.5, -1.0)>"
- 3D vector; for example, "<1.0, 2.0, 3.0>"
- complex 3D vector; for example, "<1.0-2.0i, 2.0-3.0i, 3.0-4.0i>" or "<(1.0, -2.0), (2.0, -3.0) ,(3.0, -4.0)>"

The grammar for an algebraic expression is summarized below:

```
expression : namedExpression
   | constant
   | name
   | operator
   | expression binaryOperator expression
   | '(' expression ')'
operator : opName arguments
binaryOperator : + | - | * | /
arguments : '(' <empty> ')'
   | '(' expression ')'
   | '(' expression ',' expression ')'
```

Scripting Support

All calculator operations are fully scriptable. You can save the commands used in a Fields Calculator session by first clicking the **Tools** > **Record Script to File** menu and replay the same commands by clicking the **Tools** > **Run Script** menu in a later session.

Preliminary Considerations for Using the Fields Calculator

The following sections provides some consideration notes regarding use of the Post-Processor Field Calculator. Most of the statements below are fairly generalized, and may not apply to all 3D solver projects. When in doubt about the applicability of a particular consideration for a particular project, please feel free to contact your local HF Applications Engineer for further assistance.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Field Convergence and Accuracy

HFSS, Q3D, and Maxwell use finite element method (FEM) field solvers, which arrive upon a solution via adaptive meshing convergence. There are different algorithms available for determining where in each given model mesh adaptation is performed, but convergence is always evaluated by comparison of S-parameters (for driven solutions), changes in overall scattering energy (for incident wave problems) or resonant frequencies (for eigenmode solutions) from pass to pass. Since these quantities represent the results of the model as a whole, they tend to converge more rapidly than the field values. Each point in the modeled space can be said to have converged to some value. As a result, specific field quantities at each mesh point are likely to be less accurate than the overall S-parameter or Eigen frequency result of a project solution.

In order to obtain high accuracy results from calculations on field data, we recommend that you take extra precautions to assure that the model's field data is dependable. This might include:

- Running the project to a tighter than usual convergence value.
- Seeding or manually refining the mesh in the areas to be used for calculations.
- Running parametric variations to isolate sensitivity to modeling parameters (such as adaptation frequency or circular cross-section facetization).
- Specifying expressions for output convergence.

As long as the accuracy of specific field data points to be used has been assured, the results of the Field Calculator operations should provide valuable information for your electromagnetic design tasks.

Fast Sweep and Dispersive Models

If an HFSS solution includes a Fast Frequency Sweep, the Fields Post-Processor can be tuned to display field data at any point in the frequency band swept. The specific frequency selected for viewing need not even be a precise data point at which the S-parameters were calculated. While field calculator operations may be performed at any frequency to which the Fields Post-Processor is set, fast sweep solution field data (away from the center frequency of the sweep) may not be as accurate for lossy and dispersive media, within the interior of solid-meshed finite conductors, etc. For higher accuracy under these conditions, field calculator operations should be performed on a full matrix solution completed at the desired frequency.

Since materials assigned as part of a "Perfectly Matched Layer" (PML) model termination are anisotropic and highly lossy, performing field calculations on the surface of or interior to objects designated as PMLs is not recommended.

Inputs/Excitations

Use **HFSS** > **Fields** > **Edit Sources** to set the field excitation appropriate to the calculation to be performed. In some cases (e.g., FSS calculations) picking the right field solution set (incident,

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

scattered, or total) is also paramount to obtaining the intended result.

Any field calculation which has not yet been completed (such that the calculator stack still shows some form of “text” string rather than a simple numerical value) is merely a placeholder. Altering the field data loaded in the Post-Processor (by altering port excitations, changing frequency, or picking a different solution set using **HFSS** > **Fields** > **Edit Sources**) will result in subsequent evaluation of the placeholder to the newly loaded data. To preserve a placeholder's association to an existing data set before altering the excitation to a different data set, you should export the register stack by using the **Write** button. You can bring the correctly associated quantity back into the stack by using **Read** after you have changed the field data set selection.

Units

All units in Driven HFSS field solutions are expressed in the MKS system, regardless of drawing units. Therefore E-mag is always in V/m, H-mag in A/m, etc. The exception is that when plotting along a geometry (e.g., along a line) the dimension along the X axis of the graph shows the position along the line in the drawing units, while the vertical (field quantity) axis will be in the MKS system.

Eigenmode Solutions

Field values in Eigenmode solutions are normalized to a peak value of 1.0, since there is no real excitation to which to scale the internal field results. If desired, you can scale the peak value to a user-selected number using the **HFSS > Fields > Edit Sources** menu.

Opening the Fields Calculator

To open the Fields Calculator, from the Desktop Menu bar do one of the following:

- Click **HFSS**> **Fields**> **Calculator**

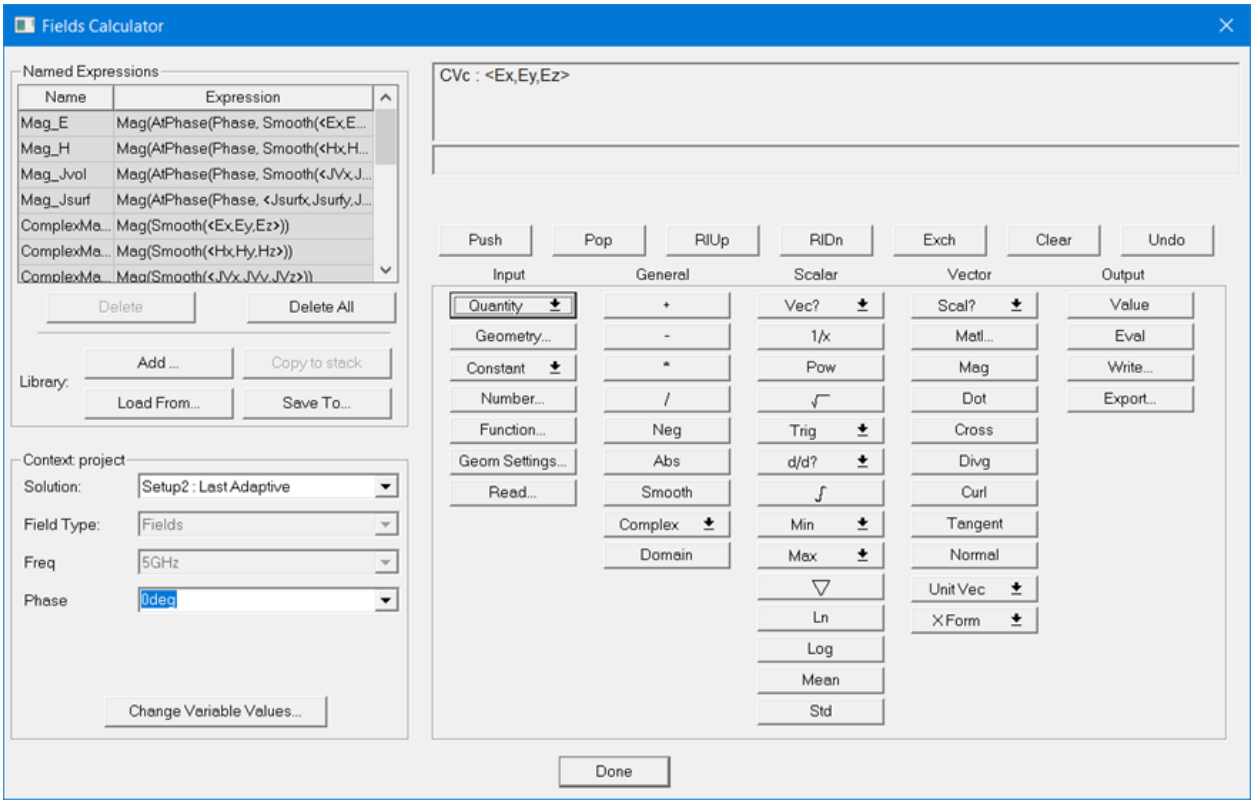
  or

- Right-click **Field Overlays** in the Project Manager and choose **Calculator** from the shortcut menu.

The *Fields Calculator* window appears.

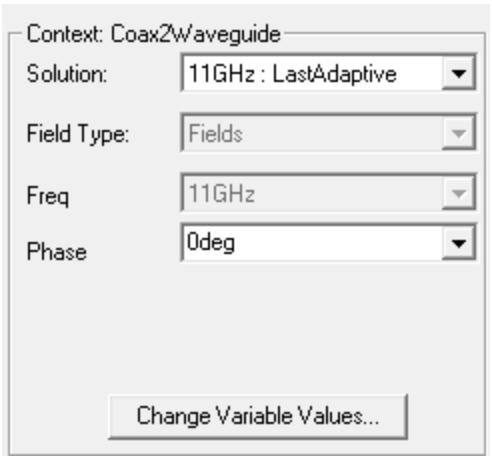
To view information on a command or screen area, click over the button or screen area on the illustration below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Context Area

The panel at the upper right of the window identifies the context to be used for the calculations. The top line identifies the design. Depending on the design, text entry boxes allow you to select a **Solution**, **Field Type**, **Freq**, **Phase**, **IWavePhi** and **IWaveTheta**. The **IWavePhi** and **IWaveTheta** are available only for incident wave projects in which the wave is defined with spherical coordinates.



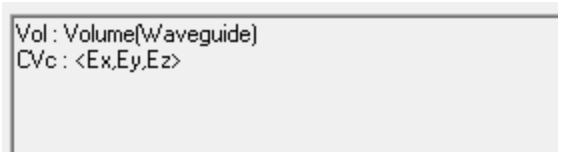
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Field Type here is not related to the edit sources. This is a general term among Ansys EM products (HFSS, Maxwell, Q3D, Icepak, and Mechanical). Some products have more than one field type for different solution types. If only one Field Type is available the drop-down menu is unavailable. For Hybrid design in HFSS R18, even if the design contains an IE Region, you can select J and Q input quantities. You no longer need to select between Fields and IE Surface Fields.

The **Change Variable Values** button opens a *Set Variable Values* dialog box. By default, it has *Use Nominal Design* checked. Unchecking the box lets you select another variable value. **OK** the dialog box to accept the selection.

The Calculator Stack

The calculator is made up of a stack of registers. Registers are displayed in the register display area at the center of the calculator window.

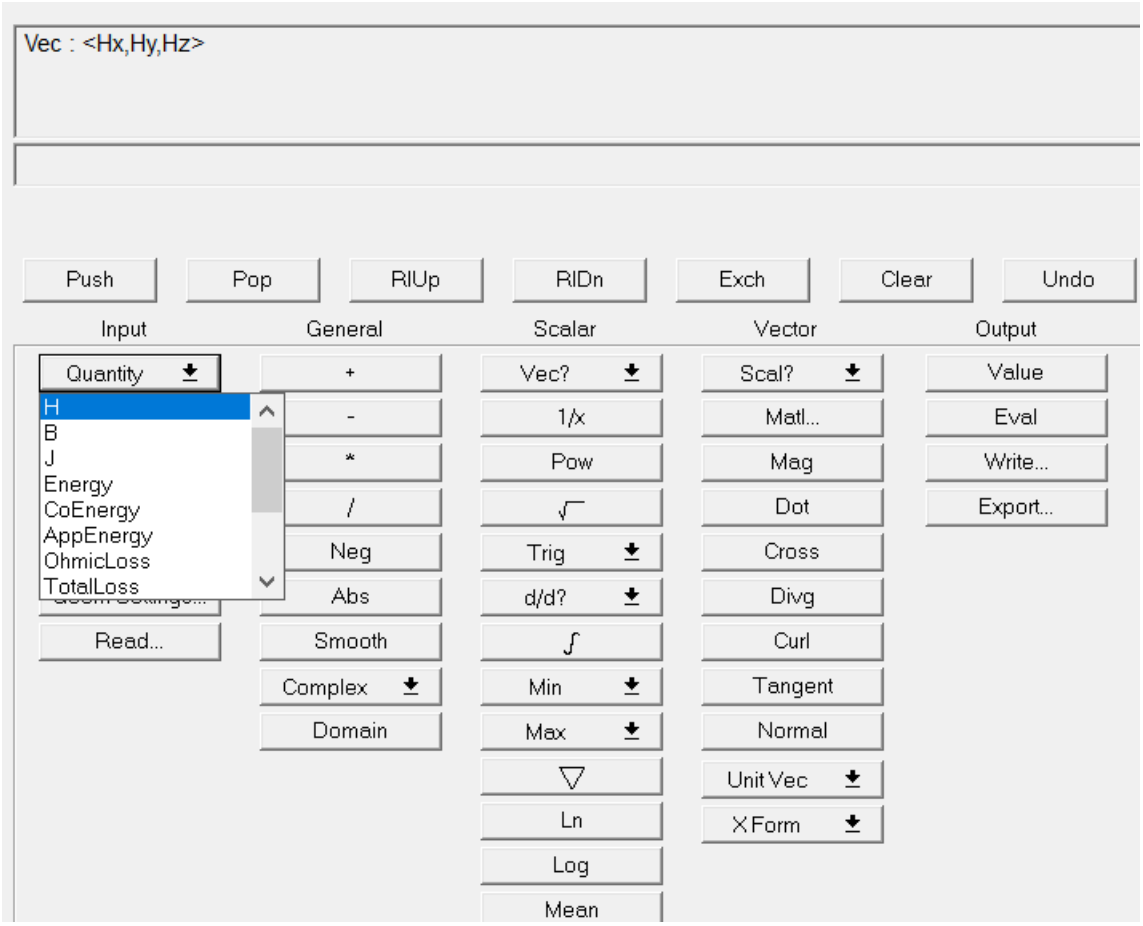


Each register can hold

- field quantities such as the H-field or E-field.
- functional or constant scalars and vectors.
- geometries — points, lines, surfaces, or volumes — on which a field quantity is to be evaluated.

To perform a computation on the field solution, you must first load a basic field quantity into a register on the stack:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once a quantity is loaded into a register, it can be

- manipulated using mathematical operations such as curls, gradients, cross products, divergences, and dot products.
- integrated over lines, surfaces, or subvolumes of the solution region — either predefined surfaces, volumes, and lists, or lines, surfaces, and volumes that were defined using the **Draw** commands.
- exported to a file, allowing you to superimpose saved solutions.

Stack Commands

Stack Commands influence the entries in the calculator stack and their position. These commands are for manipulating stack contents only, such as copy/paste, delete, or rearrange. The names for these commands match standard stack manipulation conventions. Many are self explanatory, and some can be used in pairs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Push

Reloads the quantity in the top register onto the top of the stack, creating a new register. The contents of the top two registers are identical.

Pop

Deletes the top register from the stack.

RlUp

Rolls the top register to the bottom of the stack, moving the other registers up the stack.

RlDn

Rolls the bottom register to the top of the stack, moving the other registers down the stack.

Exch

Exchanges the top two registers in the stack.

Clear

Clears the contents of the stack.

Undo

Use this command to undo the effect of the last operation you performed on the contents of the top register. Successive **Undo** commands act on any previous operations.

> **Note:**
>
> You cannot undo a simple operation such as loading a field quantity, constant, function, or geometry into the calculator. Instead, use the **Pop** or **Clear** commands to delete these items from the calculator stack.

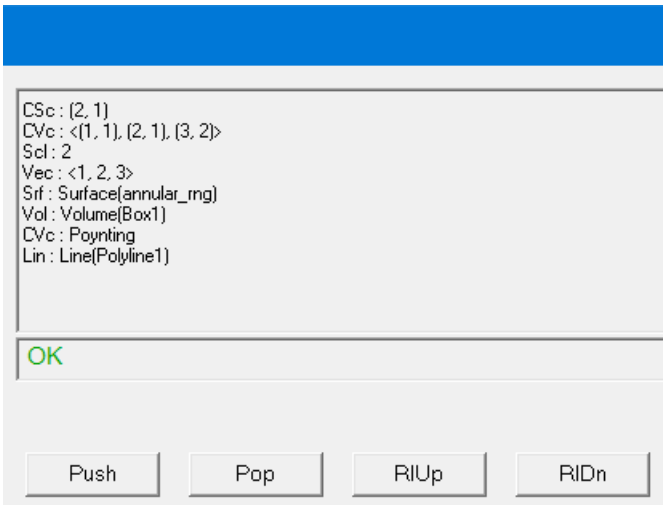
Registers

Calculator registers hold field quantities, numbers, vectors, and geometries. No registers are created until you load something into the calculator; therefore, this part of the window is initially blank. As items are loaded into the calculator, it creates new registers to hold them.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note: For Maxwell and HFSS users, make sure the Calculator is in **Show Stack** mode.

The calculator is capable of performing operations on a number of different data types. In many instances, a calculation requires certain type(s) of data to be present in the correct order in the stack register. Many operations result in a different data type than the inputs. In order to show you the type of data contained in each stack entry, calculator denotes its data type by a prefix indicator as shown in Stack Contents showing Data Type Indicators (at left), below.



The following table describes and defines each indicator, and provides guidance regarding operations which can convert data from one type to another.

Vec	Vector (non-complex) quantities, which have both direction and magnitude at each point in space. The x-, y-, and z-components of these quantities are stored in the register. Vectors are always evaluated in the coordinate system of the model. To convert a vector quantity to a scalar, use the Scal? drop-down menu from the Vector column. Suboptions ScalarX, ScalarY, and ScalarZ will take the appropriate scalar component of the vector data. Optionally, you can also Dot the vector with another vector to obtain the appropriate scalar result, or use the Tangent (return the tangential scalar component of) or Normal (return the normal scalar component of) operations to relate the vector quantity to a geometric data (Lin, Srf) stack entry. Convert to a Complex quantity using the CmplxR and CmplxI operations described in Scl, above.
Scl	Scalar quantities, which have a magnitude only. This is a simple numerical value. To convert a scalar to a vector quantity, use the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Vec? drop-down menu in the Scalar column. The choices VecX, VecY, and VecZ convert the scalar data to vector data aligned with the X, Y, or Z unit vectors, respectively. You can also multiply the scalar quantity by a desired vector direction entered manually (Num drop-down in the Input column) or obtained using the Unit Vec button from the Vector column. To convert a scalar to a complex quantity, use either CmplxR (assign the scalar value as the real component of a complex quantity) or CmplxI (assign the scalar value as the imaginary component of a complex quantity), both found under the Cmplx drop-down in the General Column.
Cvc	Complex vector quantities. This is a numerical value with real and imaginary components. Convert to a vector quantity using the same techniques described for Scl, above. Convert to a scalar using Real (take the real component), Imag (take the imaginary component), CmplxMag, (take the magnitude of the complex number) or CmplxPhase (take the phase of the complex number), all within the Cmplx dropdown in the General column.
Csc	Complex scalar quantities. This is a numerical value with real and imaginary components. Convert to a vector quantity using the same techniques described for Scl, above. Convert to a scalar using Real (take the real component), Imag (take the imaginary component), CmplxMag, (take the magnitude of the complex number) or CmplxPhase (take the phase of the complex number), all within the Cmplx drop-down in the General column.
Pnt	Points.
Lin	Lines. Lines may be straight, curved, or “polylines” in three dimensional space. Lines may also be open (have two endpoints) or closed (ending vertex same as starting vertex).
Srf	Surfaces. Surfaces need not be planar, and may actually comprise a list of object faces (faces list) as well as planar slices through the entire model space (cutplanes).
Vol	Volumes. Volumes may include sets of discontinuous object volumes created as an Object.
SclLin	Scalar value on a line.
VecLine	Vector value on a line.
SclSrf	Scalar value on a surface.
VecSrf	Vector value on a surface.
List	These indicators may exist alone, representing geometric data only, or in combination with one of the categories above, indicating a type of data applied to the geometric entity in question. For example, the notation SclSrf identifies a stack entry containing Scalar data on a

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Surface geometry set. To select only the portion of a given data entry which exists along, on, or within a given geometry quantity, use the Value button in the Output column of the calculator. Other operations (e.g., integration, the Normal button) operate when a data quantity is in the second stack register and a geometric quantity is in the top stack register. Full descriptions of the register requirements for each individual command is available in the online help.

When examining calculator registers, keep the following in mind:

- To move or delete calculator registers, use the stack commands.
- To save a register to a disk file, use the **Write** command.

Users must be cautious concerning what type of data the Fields Calculator is manipulating and whether or not it is compatible with the desired operation. For example, the integral operation is often misused. Note that the integral sign is in the *Scalar* column, implying that, to integrate complex number/quantities, the user will have to integrate the real and imaginary components separately. In other words, performing integration on complex number/quantities must be achieved by parts.

Enlarging the Register Display Area

If there are too many registers to fit into the display area, do one of the following:

- Use the scroll bars to view the hidden registers.
- Enlarge the calculator window by dragging window's borders.

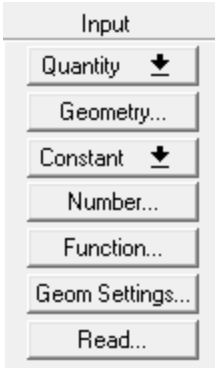
Units of Measure

Unless you are prompted specifically for the unit of measure, all measurements should be assumed to be in SI base units, not model units.

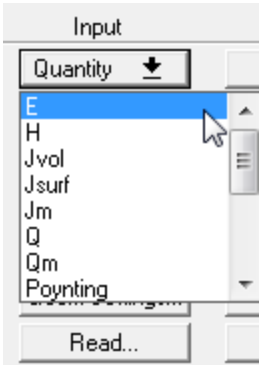
Input Commands

The *Input* Column contains all the calculator functions which place new values into the stack, such as field data, constant, geometry data, coordinate system selection, or user-entered vector or complex numbers.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Field data (e.g., E-field, H-field, Poynting vector) for the current project solution is input from the **Quantity** drop-down menu selection.



The quantities specifically available from the calculator are dependent on the solution type and include the E-field, H-field, J-vol (volume current), Poynting vector, Displacement, Temperature, Heat Flux, and Surface and Volume Loss Densities. All quantities are Peak Phasors, and not RMS quantities, with the phase information captured in the real and imaginary components and the field orientation captured in the vector components. Although the Poynting vector is automatically calculated by the interface as 0.5, it will appear in the calculator stack as a Complex Vector quantity. The imaginary portion should however be zero or ignored.

Use the following commands to load data onto the top of the calculator stack:

Quantity	Basic field quantities, such as E and H, and simple derived quantities such as volume current. For designs with IE Region, select J and Q.
Geometry	Geometries such as planes, points, polylines, 3D lines, face lists, and volumes
Constant	Predefined constants such as π, ε_0, and conversion factors between various units of measurement.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number	Vector and scalar constants, including complex numbers.
Function	User-defined or intrinsic variables
Geom Settings	Number of equally spaced points used to integrate fields and other quantities on a line.
Read	previously saved calculator registers containing field quantities.
Output Vars	This button appears only for eigenmode problems. Freq is the only value there. Evaluation for Freq returns a complex value.

These quantities can be manipulated using the **Stack** commands, **General** commands, **Scalar** commands, and **Vector** commands. The results of these calculations can then be examined using the **Output** commands.

Quantity Command

The **Quantity** command loads a specified field quantity into the top register of the calculator.

The available quantities are:

E	The electric field, E
E_t	Transient electric field, E.
H	The magnetic field, H
H_t	Transient magnetic field, H.
Jvol	The volume current density, J_{vol}
Jsurf	The surface current density, J_{surf}
Jsurf_t	Transient surface current density.
Poynting	The Poynting vector, defined as 0.5E x H*
Poynting_t	Transient Poynting vector.
LocalSAR	The local Specific Absorption Rate
AverageSAR	Peak spatial average SAR, dependent on the Average SAR Method chosen in the Specific Absorption Rate Setting.
Certification SAR	IEEE standard Specific Absorption Rate certification number. (See Modifying SAR Settings and Calculating the SAR.) To calculate certification SAR on a specific object (rather than the whole model) proceed as follows: 1. In the Calculator Input area, click the **Quantity** button and select **Certification SAR**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Certification SAR is displayed in the calculator stack.

2. In the Calculator, click the **Geometry** button.

   The *Geometry* dialog box displays with the *Volume* radio button selected and the available geometries listed.

3. Select the Geometry of interest.

   This enables the OK button.

4. Click the **OK** button.
5. This adds the selected Volume geometry to the calculator stack.
6. In the Calculator Output area, press the **Value** button.

   This prepares the calculation for the selected quantity and volume.

7. Press the **Eval** button to evaluate.

   Both the value and location will be shown on the calculator stack.

SurfaceLossDensity

This contains the surface impedance (if any) loss at every node in every triangle. This is calculated as:

$$Loss = 0.5 \times Real\left((\vec{E}\ x\ \overrightarrow{normal}) \cdot \overrightarrow{(\vec{E} \times \overrightarrow{normal})} \div \overrightarrow{Impedance}\right)$$

To export a REG file containing the surface loss density, place the SurfaceLossDensity in the top register and use the **Write...** command, selecting Reg format.

VolumeLossDensity

The volume loss density *p* is calculated as:

$$p_v = \frac{1}{2}Re(E \cdot \tilde{J} + j\omega B \cdot \tilde{H}) = \frac{1}{2}Re(E \cdot \tilde{J} - curlE \cdot \tilde{H})$$

where *E* is the electric field, $\tilde{J}$ is the conjugate of the volumetric current density, *B* is the magnetic flux density, and $\tilde{H}$ is the conjugate of the magnetic field.

To export a Reg file containing the volume loss density, place the VolumeLossDensity into the top register, and use the **Write...**.command, selecting the Reg format.

VolumeLossDensit

Transient volume loss density, calculated as:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

y_t	0.5 * *Re*(*E* * *J*)
SurfaceForceDensity	Surface Forces exist when one side is conductor, but the other is not, or finite conductivity and layered impedance boundary. This is mainly for the purpose of mapping surface force density in HFSS to Workbench Mechanical. For details on the calculations, see the technical notes.
Temp	Temperature.
Displacement	This is value is for use with Workbench when exploring stress feedback.

Geometry Command

The Fields Calculator **Geometry** command opens a dialog box for selecting a geometry to load into the top register of the calculator.

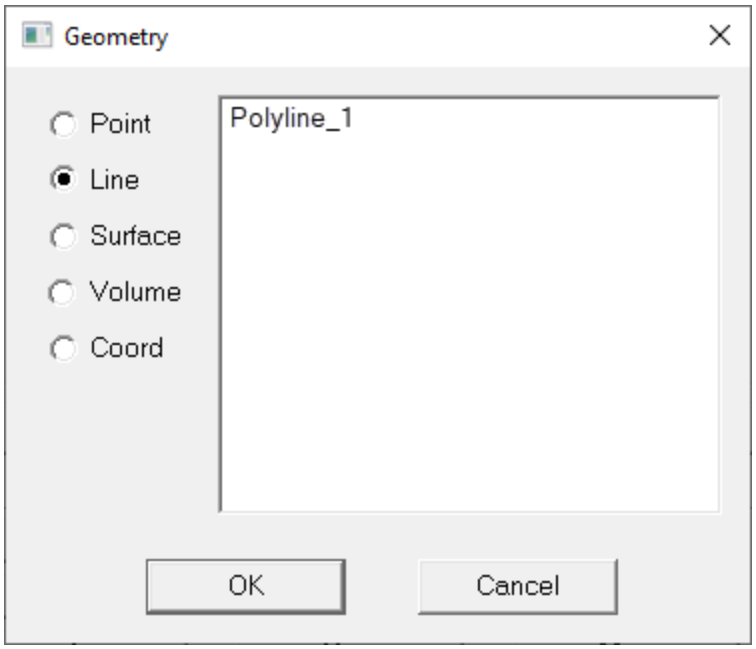
Do this to

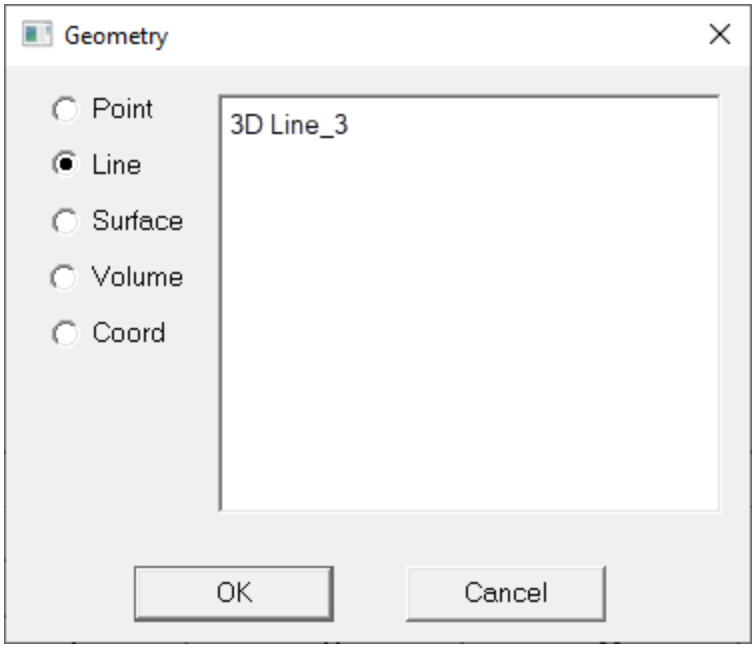
- find the value of derived field quantities on any point, line, surface, or volume.
- plot quantities directly from the calculator.
- display a previously defined isosurface, maximum or minimum field point using the **Draw** command.

The following types of geometries are available:

Point	See Drawing a Point Object. Drawn points are listed in the History Tree, and in the calculator's *Geometry* dialog box when **Point** is selected.
Line	See Drawing a Line Segment or Drawing a 3D Line Segment. Drawn lines are listed in the **History Tree**, and in the calculator's *Geometry* dialog box when **Line** is selected. To set the number of points on a line, see Geom Settings.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



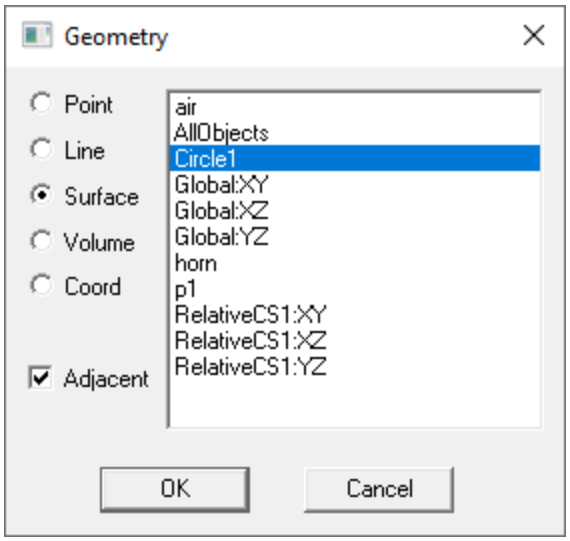


Surface

Creatable **Sheet** objects and Face Lists, (e.g., of radiation boundaries), are listed in the History Tree and in the calculator's **Geometry** dialog box when **Surface** is selected.

Due to the ambiguity of the normal vector of a sheet, the result may require a multiplication by (1) or (-1). If a Sheet object is selected from the Face list, the dialog also lists an **Adjacent** command that controls the side of the sheet on which the fields are calculated.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This check box is only available when users select a sheet object for surface geometry

- Allow operation on the adjacent side surface geometry just like regular surface geometry.
- All calculation on this geometry will be done using data from the adjacent side of the surface.
- Support scripting, saving and loading expression with adjacent side surface geometry.

```
Srf : AdjacentSurface(Circle1)
CVc : <Ex,Ey,Ez>
```

Volume — 3D objects, Regions, and Object Lists (of 3D objects including AllObjects) are available in the calculator's *Geometry* dialog box when **Volume** is selected.

Coord — Coordinate systems are available in the calculator's *Geometry* dialog box when **Coord** is selected.

To load a geometry into the calculator:

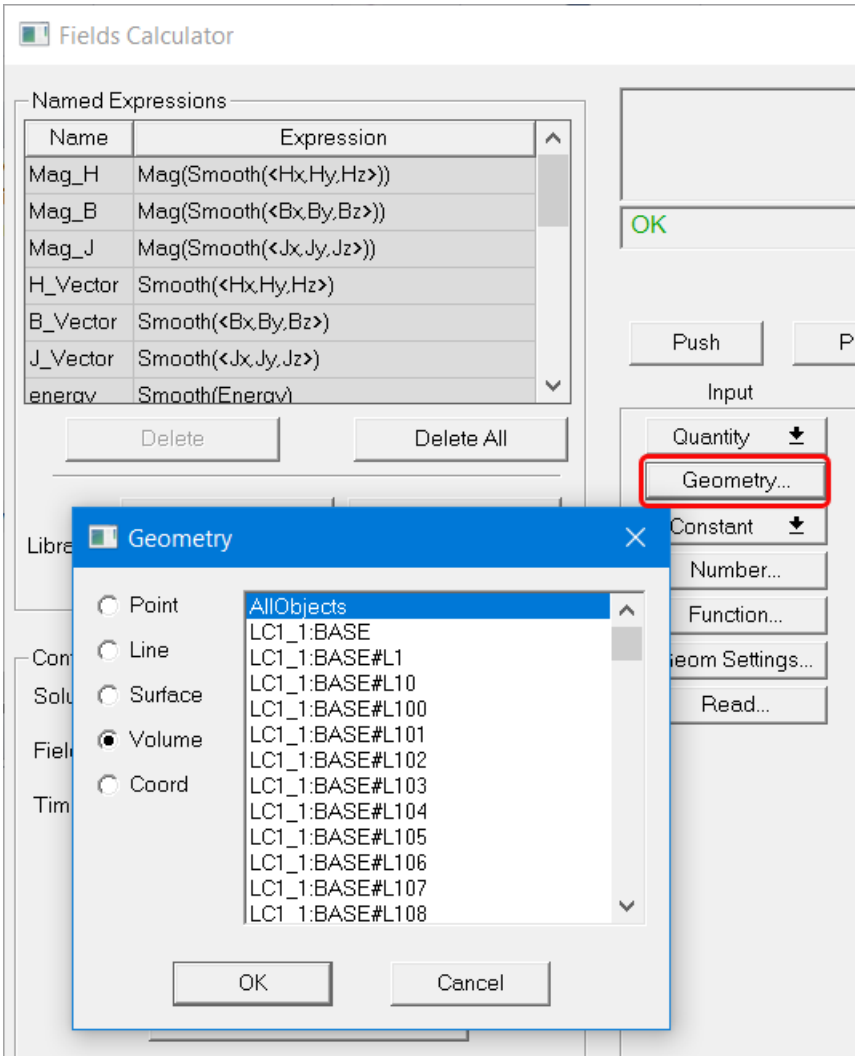
1. In the Fields Calculator, click **Geometry**.

   The **Geometry** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select a geometry type.

   A list of all applicable geometries appears. If a design includes Layout Components, the Geometry list shows the available component geometries as well.



In the modeler window, these Layout Component objects are drawn as visualizations without actual geometry. However, they can be used to calculate field quantities as shown below.

These geometry operators can be used to perform computations just like one constructed

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

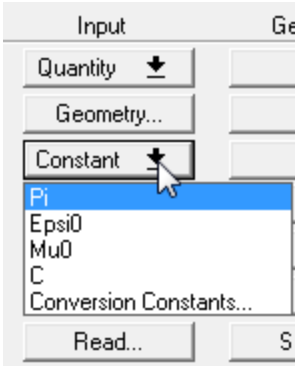
from regular 3D geometry. These additional objects are generated only after reading a mesh, so they won't show up if there is no solution.

3. Click the geometry.
4. Click **OK** to load the geometry.

Note: Consider a box (Box2) that is completely enclosed in a bigger box (Box1), so that no faces of Box2 are touching any faces of Box1. Box2 is actually implicitly subtracted from Box1 as is done in our solvers. So Box1 is used as if Box2 were already subtracted from Box1. Volume(Box1) is Box1 excluding Box2, and Surface (Box1) contains faces from both Box1 and Box2.

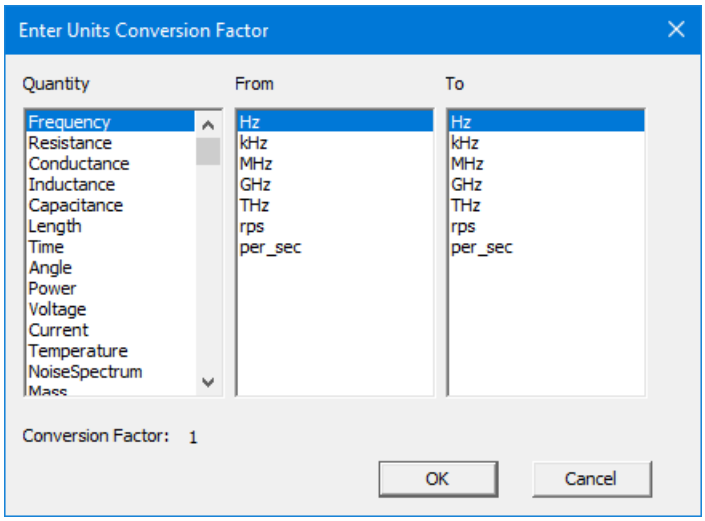
Constant Command

The **Constant** command loads one of the following four predefined constants or a selected conversion constant into the top register of the calculator:



Pi	π
Epsi0	The permittivity of free space, $\varepsilon_0 = 8.85418782 \times 10^{-12}$ C^2/Nm^2
Mu0	The permeability of free space, $\mu_0 = 4\pi \times 10^{-7}$ Wb/Am
c	The speed of light in vacuum, $c = 2.99792458 \times 10^8$ m/s
conversion constant	Displays the *Enter Units Conversion Factor* dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This lists a range of Quantities (such as frequency, resistance, and others) along with a list of Units (Hz to Thz, and rps) to convert From and To. The ratio of the Units From to the Units to is displayed for the selected values as **Conversion Factor**.

Number Command

The **Number** command enters one of the following into the top register of the calculator:

Scalar	A scalar constant. To enter a constant scalar number: 1. Click **Number**. The *Input Number* dialog box appears. 2. Select **Scalar**. 3. Type the scalar value in the **Value** text box. 4. Click **OK** to load the number into the top register.
Vector	A vector constant. To enter a constant vector: 1. Click **Number**. The *Input Number* dialog box appears. 2. Select **Vector**. 3. Enter the x-, y-, and z-components of the vector. 4. Click **OK** to load the vector into the top register.
Complex	A complex constant, entered in the form $C=A+jB$, where A represents the real part of the constant and B represents the imaginary part.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click **Number**.

   The *Input Number* dialog box appears.

2. Select **Scalar** or **Vector**.
3. Select **Complex**.
4. Enter the real and imaginary components of the number.
5. Click **OK** to load the number into the top register.

Function Command

The **Function** command enters a predefined scalar or vector function into the top register of the calculator:

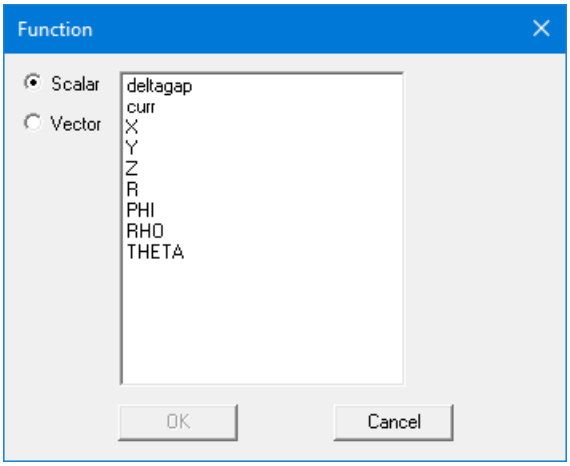
Scalar

A scalar function.

To enter a function:

1. Click **Function**.

   The *Function* dialog box appears.



2. Select **Scalar**.
3. Select the function from the list.
4. Click **OK** to load the functional scalar into the top register.

Vector

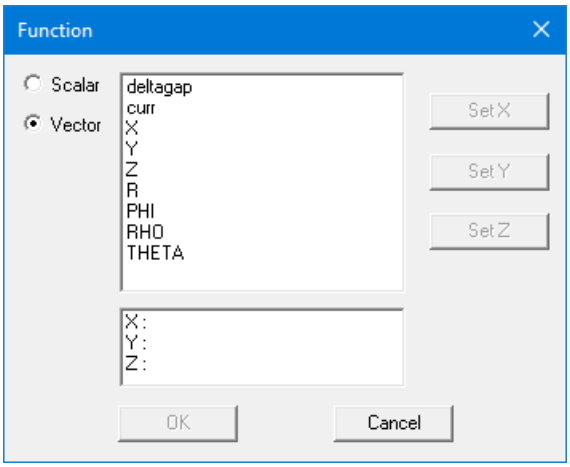
A vector function, in which the values of the vector's x-, y-, and z-components are given by functions.

To enter a functional vector:

1. Click **Function**.

   The *Function* dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Select **Vector**.
3. Select the function from the list.
4. For each component of the vector, click **SetX, SetY,** and **SetZ**.
5. Click **OK** to load the functional vector into the top register.

Note:

The predefined variables **X, Y,** and **Z**; **RHO, THETA, R,** and **PHI**; and any functions that you created can be used to define functional scalar and vector quantities.

Use of the Global Coordinate System is assumed. Local coordinate systems are not used.

Geom Settings Command

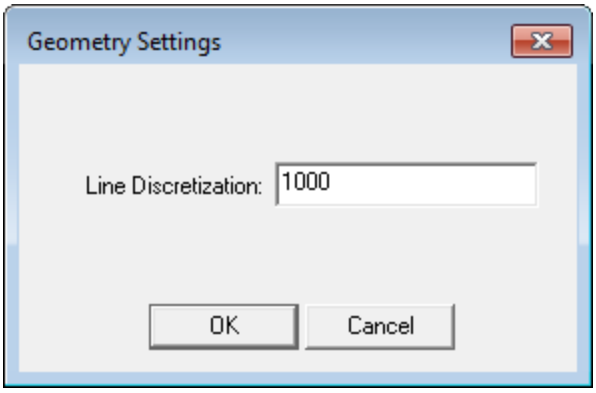
Clicking the **Geom Settings** button opens the *Geometry Settings* dialog box. This dialog box allows you to specify the line discretization, the number of equally spaced points used to integrate fields and other quantities on a line. The default is 1000 points.

To set the line spacing for geometry settings:

1. In the Fields Calculator, click **Geom Settings**.

   The *Geometry Settings* dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Enter a value in the **Line Discretization** text box and click **OK**.

Read Command

This **Read** command copies the contents of a disk file into the top register. The register must be one that has been saved using the Write output command.

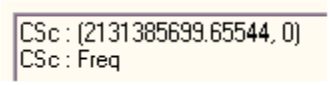
To read in a register:

1. Click **Read**.
2. Use the file browser to specify the register's file name and directory path. A .reg extension is automatically assumed for register files.
3. Click **OK**.

   The contents of the file are copied to the top register in the stack.

Output Vars [Input for Eigenmode problems]

This Output Vars button appears in the **Inputs** column of the Fields calculator only for Eigenmode problems. Freq is the only value listed. After you push Freq to the stack, you can click **Eval** to return a complex value.



General Commands

The General Column contains calculator operations which can be performed on many different data types (e.g., vector, scalar, complex). With the exception of the Complex menu, all are distinct functions. Most are self-explanatory, with the exception of Smooth which performs some data “smoothing” or statistical averaging on the top stack entry.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use these Fields Calculator commands to perform operations on both vector and scalar quantities.

+ (Add)	/ (Divide)	Smooth
-- (Subtract)	Neg	Complex
* (Multiply)	Abs	Domain

+ (Add)

Adds the quantities in the top two registers of the calculator.

- (Subtract)

Subtracts the quantity in the top register from the quantity in the second register. The two registers must hold the same type of quantity (both scalar or both vector). You cannot subtract a scalar from a vector (or vice versa).

> **Note:** Consider a box (Box2) that is completely enclosed in a bigger box (Box1), so that no faces of Box2 are touching any faces of Box1.
>
> If you explicitly subtract Box2 from Box1, any calculation on the surface (faces) of Box1 will use the 6 exterior faces and the 6 interior faces. Any calculation on the volume of Box1 will use the difference in volume between Box1 and Box2.
>
> If you do not explicitly subtract Box2 from Box1, the inner box is only implicitly subtracted. Any calculation on the surface of Box1 in this case will use only the 6 exterior faces of Box1. Any calculation on the volume of Box1 will use the entire volume without subtracting the volume of Box2.

*** (Multiply)**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Multiplies the quantity in the top register by the quantity in the second register. One of the two registers must contain a scalar value; the other register can be either a scalar or a vector.

/ (Divide)

Divides the quantity in the second register by the quantity in the top register. The second register must contain a scalar value; the top register can be either a scalar or a vector.

Neg

Changes the sign of the quantity in the top register.

Abs

Takes the absolute value of the quantity in the top register.

Smooth

Smooths the quantity in the top register. Because of the numerical solution technique used, field values are not always continuous across the boundaries of the individual elements that make up the finite-element mesh. Smoothing makes the values continuous by taking a weighted average from all of a node's neighboring elements. The weights are based on angles, so elements with larger angles provide larger contributions. In general, use smoothing before plotting a quantity.

Complex

These commands perform operations on a complex quantity in the top register. Complex quantities are indicated by a **C** at the beginning of the register label. They can be represented in terms of real and imaginary components, or in terms of magnitude and phase:

$$C = A + jB = Me^{j\phi}$$

where:

- *A* is the real part of the complex number.
- *B* is the imaginary part of the complex number.
- *M* is its magnitude, which is equal to $\sqrt{A^2 + B^2}$.
- ϕ is its phase, which is equal to **atan**(B/A).

The **Complex** commands includes a drop-down menu that let you do the following:

Real	Takes the real part of the complex quantity (A).
Imag	Takes the imaginary part of the complex quantity (B).
CmplxMag	Takes the magnitude of the complex quantity (M). Due to interpolation issues, the sequence of calculations may cause a loss of accuracy. It is best to define the points , separately obtain the value of the real part, then the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	imaginary part, and use those values to calculate the magnitude and and phase. For the sequence for using the Fields Calculator to obtain the real and imaginary parts, see the procedure here.
CmplxPhase	Takes the phase of the complex quantity(ϕ).
Conj	Takes the complex conjugate of the quantity in the top register. If a complex number is given by $C = A + jB$, its complex conjugate is given by $C^* = A - jB$.
AtPhase	Lets you specify the phase angle, θ, at which an field quantity is evaluated. These quantities can be represented in the form $A(x,y,z,t) = A(x,y,z)\cos(\omega t + \theta(x,y,z))$. where • ω is the angular frequency at which the quantities are oscillating, specified during the solution. • θ(x,y,z) is the phase angle (the offset from a cosine wave that peaks at t=0). Entering the phase angle lets you compute the real part of the field's magnitude at different points in its cycle.
CmplxReal	Converts the real scalar of the top register to the real part of a complex number.
CmplxImag	Converts the real scalar of the top register to the imaginary part of a complex number.
CmplxPeak	Calculates the peak value of a given complex vector. Intuitively, this calculates the maximum magnitude of the equivalent real vector in a waveform.

Domain

This limits a calculation to the volume you specify. The domain filter works for scalars, vectors, complex scalars and complex vectors. This operation requires the top two entries of the stack to be a volume geometry and a numeric field quantity. To do this:

1. Load the field quantity into the top register, and perform any necessary operations on it.
2. Load the volume using the **Geometry** command.
3. Click **Domain**.

The **Domain** command is often used to limit a calculation or plot to the intersection of a surface and an object or group of objects. If you export a domain filtered numeric, points that are filtered out by the domain will not be written out.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Steps for Calculating the Complex Vector Electric Field

These are the field calculator steps to obtain the real part, the imaginary part, and the magnitude of the x-directed, y-directed, and z-directed components of the phasor electric field. For each of these vector components, the magnitude should be equal to sqrt(real^2+imag^2), but the need to interpolate values and the calculation sequence means that HFSS does not give this value unless the specified location is directly on a mesh element node.

1. Calculate real part of complex vector electric field (in x, y, and z directions):
    a. Qty > E
    b. Complex > Real
    c. Geometry > Point > fieldcalc_point
    d. Value
    e. Eval
2. Calculate imaginary part of complex vector electric field (in x, y, and z directions):
    a. Qty > E
    b. Complex > Imag
    c. Geometry > Point > fieldcalc_point
    d. Value
    e. Eval

Use the real and imaginary components to manually calculate the magnitude as the sqrt (real^2+imag^2).

Scalar Commands

The Scalar Column contains calculator operations which can only be performed on scalar stack entries.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Drop-down menus in this column include Vec? (convert scalar to vector), Trig (trigonometric, containing sin, cos, etc. functions), d/d? (derivative with respect to...), Max and Min (self-explanatory), Gradient, ln (natural logarithm base e), log (logarithm base 10), mean, and Std (standard deviation). Note that the calculator's Integrate function is located in the Scalar column. The implication is that integration can only be performed on scalar quantities. To perform integration upon complex quantities, you must perform the integration separately on real and imaginary subcomponents.

Use these commands to perform operations on scalar quantities.

Vec?	Makes the scalar quantity in the top register a vector component.
1/x	Takes the inverse of the scalar quantity in the top register.
Pow	Raises a scalar quantity to the power you specify.
(Square Root)	Takes the square root of the quantity in the top register.
Trig	Performs a selected trigonometric operation on the value in the top register of the calculator stack (Sin, Cos, Tan, ASin, ACos, ATan). Angles are in radians. Additionally, the ATan2 operation is available, which requires two arguments. This operator calculates the arc tangent of the following ratio: The second register in the stack divided by the top register.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Example: If the top register value is 1.5, and the second register value is 2, the ATan2 function returns ATan(2/1.5) = 0.5880 radians. All other Trig functions operate on a single argument (top register only).
d/d?	Takes the partial derivative of the quantity in the top register.
∫? (Integral)	Takes the integral of a scalar quantity over a volume, surface, or line. To perform integration upon complex quantities, you must perform the integration separately on real and imaginary subcomponents.
Min	Computes the minimum of a scalar field quantity on a line, surface, or volume relative to your choice of Value or Position.
Max	Computes the maximum of a scalar field quantity on a line, surface, or volume relative to your choice of Value or Position.
∇ (Gradient)	Takes the gradient of the scalar quantity in the top register.
ln	Takes the natural logarithm (base e) of the scalar quantity in the top register.
log	Takes the logarithm (base 10) of the scalar quantity in the top register.
Mean	Takes the mean of the quantity in the top register.
Std	Takes the standard deviation of the quantity in the top register.

Vec? Command

Makes the scalar quantity in the top register a vector component. Choose from the following:

VecX	The x-component of a vector.
VecY	The y-component of a vector.
VecZ	The z-component of a vector.

1/x (Inverse) Command

Takes the inverse of the scalar quantity in the top register.

Pow Command

Raises a scalar quantity to the power you specify.

To raise a scalar quantity to a power:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Enter the quantity into the calculator.
2. Enter the exponent to which it is to be raised into the calculator.
3. Click **Pow**.

   The results are displayed in the top register.

(Square Root) Command

Takes the square root of the quantity in the top register.

Trigonometry Commands

Takes one of the following trigonometric values of the value in the top register of the calculator stack:

Sin	Sine.
Cos	Cosine.
Tan	Tangent.
Asin	Arcsine.
Acos	Arccosine.
Atan	Arctangent.
Atan2	A two-argument version of the At.an function. Takes the y and x coordinates of the tangent point (in the top two stack registers)

d/d? (Partial Derivative) Command

Takes the partial derivative of the quantity in the top register:

d/dx	Takes the partial derivative of the quantity with respect to x.
d/dy	Takes the partial derivative of the quantity with respect to y.
d/dz	Takes the partial derivative of the quantity with respect to z.

(Integral) Command

Takes the integral of a scalar quantity over a volume, surface, or line. The top register must contain a geometry and the second register must contain the scalar quantity to be integrated.

To perform an integration:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Load a quantity into the top register of the calculator, and perform any required operations on it.
2. Use one of the **Geometry** commands to load the line, surface, or volume over which the quantity is to be integrated.

   > **Note:**
   >
   > If you computed the tangent or normal of the quantity to be integrated, you do not have to load any geometry selection onto the calculator stack. HFSS integrates the tangential or normal component of the quantity over the line on which you computed its tangent, or the surface on which you computed its normal.

3. Choose the command to integrate the scalar quantity over the geometry.

To find the numerical results of an integration, use the **Eval** command.

Min Command

Computes the minimum of a scalar field quantity on a line, surface, or volume. Two options are available:

Value	Finds the magnitude of the minimum value of the field.
Position	Finds the point where the minimum field value occurs. You can then: • Plot the minimum field value at the point. • Plot basic field quantities at the point. • Load the point into the calculator. • Change the point's location.

These commands operate in the same way as the **Max** commands. Use the **Eval** command to display the actual minimum field value or the coordinates of the point where it occurs.

Max Command

Computes the maximum of a scalar field quantity on a line, surface, or volume. Two options are available:

Value	Finds the magnitude of the maximum value of the field.
Position	Finds the point where the maximum field value occurs. You can then:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Plot the maximum field at the point.
- Plot field quantities at the point.
- Load the point into the calculator.
- Change the point's location.

To compute the maximum field value:

1. Load a field quantity into the calculator, and perform any necessary operations on it. Keep the following in mind:
   - You cannot find the maximum value of a vector quantity. Therefore, make sure that the result is a scalar.
   - Before computing the maximum value of a complex quantity, you must find the real part of the quantity using the **Cmplx/Real** or **Cmplx/AtPhase** commands.
2. Load a point, line, or volume into the calculator using one of the **Geometry** commands.
3. Do one of the following:
   - Choose **Max/Value** to compute the maximum field value on the geometry.
   - Choose **Max/Position** to identify the point at which this value occurs.

Use the **Eval** command to display the actual maximum field value or the coordinates of the point where it occurs.

∇ (Gradient) Command

Takes the gradient of the scalar quantity in the top register.

Ln Command

Takes the natural logarithm (base e) of the scalar quantity in the top register.

Log Command

Takes the logarithm (base 10) of the scalar quantity in the top register.

Takes the mean of the quantity in the top register.

Std Command

Takes the standard deviation of the quantity in the top register.

Vector Commands

The Vector Column contains calculator operations which can only be performed on vector stack entries. Drop-down menus in this column are Scal? (convert vector to scalar) and Unit Vec (create unit vector). Standard vector algebra operations (Dot, Cross, etc.) are also present.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use these commands to perform operations on vector quantities.

Scal?	Replaces the vector in the top register with a scalar quantity whose value is a component of the vector, your choice of ScalarX, Scalar Y, or ScalarZ.
Matl	Multiplies or divides the vector field quantity in the top register by a material property, or if you select MassDensity as the material property, produces a scalar that operates like a named variable.
Mag	Takes the magnitude of the vector quantity in the top register. The magnitude of a complex vector is defined to be the length of the real vector resulting from taking the modulus of each component of the original complex vector.
Dot	Takes the dot product of the vector quantities in the top two registers.
Cross	Takes the cross product of the vector quantities in the top two registers.
Divg	Takes the divergence of the vector quantity in the top register.
Curl	Takes the curl of the vector quantity in the top register.
Tangent	Computes the tangential component of a vector quantity along a line
Normal	Computes the normal component of a vector quantity on a surface such as a cutplane or object surface.
Unit Vec	Computes the normal or tangent unit vector. The unit vector is a "wild card" entry. The context is specified at the time of plotting, integrating, or report generation. The drop-down lets you select from Tangent, Normal, or CoordSys[X, or Y, or Z]
X Form	Transforms to your choice of Cylindrical or Spherical.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scal? Command

Replaces the vector in the top register with a scalar quantity whose value is a component of the vector. Choose from the following:

ScalarX	Returns the x-component of the vector.
ScalarY	Returns the y-component of the vector.
ScalarZ	Returns the z-component of the vector.

Matl Command

Operates on the vector field quantity in the top register of the Fields Calculator based on a material property. At each tetrahedron, the field quantity is operated on by the value of the selected material property — taking the different material attributes of each object into account.

To operate on a vector quantity by a material property:

1. Click **Matl**.

   The *Material Operation* window appears.

2. Select a material property. Available properties are

Permittivity (epsi)	The permittivity, ε.
Permeability (mu)	The permeability, μ.
Conductivity	The conductivity, σ.
Omega (w)	The angular frequency, ω. The angular frequency is equal to $2\pi f$, where f is the frequency at which the solution was generated.
MassDensity	This is based on the value of the Mass Density material property. MassDensity is treated like a named expression. Selecting MassDensity disables the Operation radio buttons for Multiply or Divide in the *Material Operations* dialog box.

3. For Permittivity, Permeability, Conductivity or Omega, select an operation — **Multiply** or **Divide**. Selecting MassDensity disables these operations. The MassDensity scalar can be used like any other named expression.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Choose **OK** to operate on the field quantity by a material property or **Cancel** to stop the operation. If you selected MassDensity and click **OK**, a scalar named expression MassDensity is pushed onto the stack.

Mag Command

Takes the magnitude of the vector quantity in the top register. The magnitude of a complex vector is defined to be the length of the real vector resulting from taking the modulus of each component of the original complex vector.

With a complex vector on the calculator stack, the **Mag** button returns a non-negative scalar. In previous software versions, this command returned a complex scalar.

Dot Command

Takes the dot product of the vector quantities in the top two registers.

Cross Command

Takes the cross product of the vector quantities in the top two registers.

Divg Command

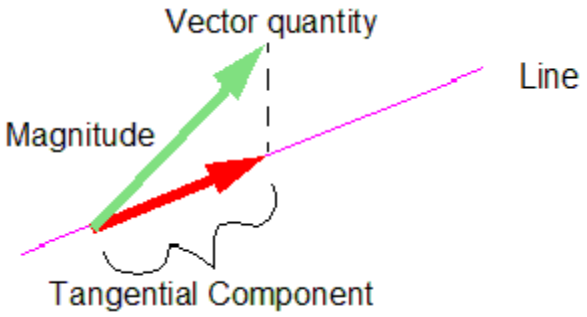
Takes the divergence of the vector quantity in the top register.

Curl Command

Takes the curl of the vector quantity in the top register.

Tangent Command

Computes the tangential component of a vector quantity along a line.



To take the tangent of a vector:

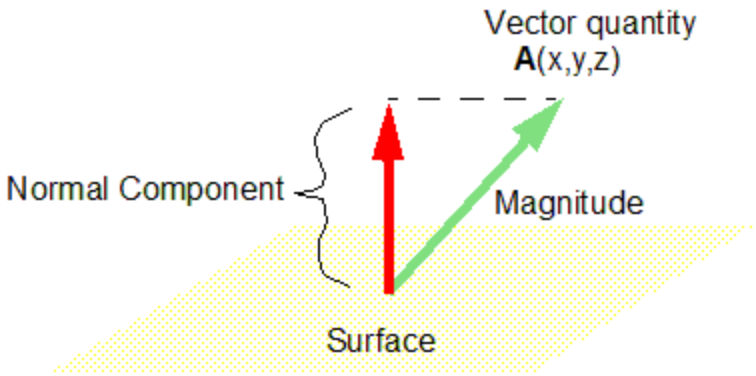
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Load a vector quantity into the top register.
2. Load a line into the top register using the **Geometry/Line** command.
3. Click **Tangent**.

Normal Command

Computes the normal component of a vector quantity on a surface such as a cutplane or object surface. This is the equivalent of taking the dot product of the quantity with the surface's unit normal vector:

$$Normal = A(x, y, z) \cdot \hat{n}$$



To take the normal of a vector:

1. Load a vector quantity into the top register.
2. Load a surface into the top register using the **Geometry/Surface** command.
3. Click **Normal**.

> **Note:**
>
> Because surface normals of sheets are not well defined the fields calculator can produce incorrect results if an expression is evaluated on a sheet. To enforce the correct direction of the surface normal of a sheet, a faceted 3D object (such as a box) can be defined such that one of its planar faces is coincident with the sheet. Because surface normals of a valid object are always defined in an outward direction in HFSS, the fields calculator uses the surface normal of the face of the 3D object that is coincident with the sheet.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Unit Vec Command

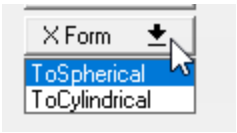
Computes the normal or tangent unit vector. The unit vector is a "wild card" entry. The context is specified at the time of plotting, integrating, or report generation.

Select from the following:

Tangent	Computes the unit vector tangent to the line specified at the time of plotting, integrating, or report generation based on the context.
Normal	Computes the unit vector normal to the surface specified at the time of plotting, integrating, or report generation based on the context.
CoordSys (X)	Computes the unit vector in the X-dimension of the relative coordinate system in the top register of the calculator stack. Add the relative CS as a geometric object using the **Geometry/Coord** command.
CoordSys (Y)	Computes the unit vector in the Y-dimension of the relative coordinate system in the top register of the calculator stack. Add the relative CS as a geometric object using the **Geometry/Coord** command.
CoordSys (Z)	Computes the unit vector in the Z-dimension of the relative coordinate system in the top register of the calculator stack. Add the relative CS as a geometric object using the **Geometry/Coord** command.

X Form

Fields are defined in the Cartesian coordinate system. In many applications it is much more convenient to use a cylindrical or spherical coordinate system – for example, when the object is of cylindrical or spherical shape. The **XForm** command allows the user to transform vectors into either the cylindrical or spherical coordinate system. Selecting **XForm** displays a selection menu for the coordinate system (**ToCylindrical** or **ToSpherical**):

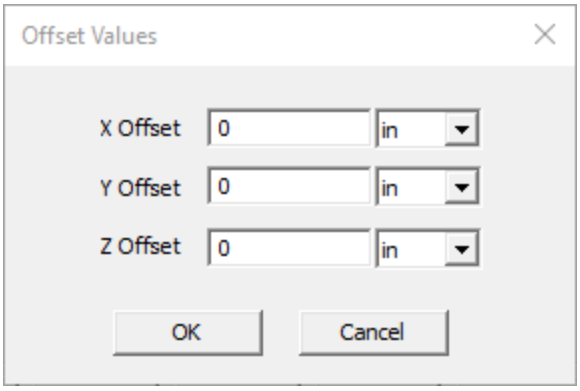


ToSpherical – Transforms the vector in the top register of the calculator stack into the spherical coordinate system.

ToCylindrical – Transforms the vector in the top register of the calculator stack into the cylindrical coordinate system.

After you select the coordinate system you want, the **Offset Values** dialog displays.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Here you enter X, Y, and Z offsets to be used as the origin of the new coordinate system.

Click **OK** to compute the offset coordinate transformation to apply to the (complex) vector quantity in the top register on the calculator stack based on your selection of Cylindrical or Spherical coordinates and your input of X, Y, and Z offsets. The new expression will be pushed onto the stack. This provides the origin of the new coordinate system. It will be something like

CVc : ToCylindrical(<Ex,Ey,Ez>,offset<0mil,0mil,0mil>)

or

CVc : ToSpherical(<Ex,Ey,Ez>,offset<0mil,0mil,0mil>)

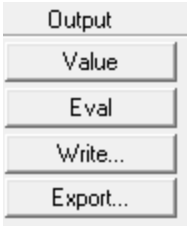
After you transform the quantities, you can then access the vector components in the Fields Calculator.

You will then use the ScalarX/ScalarY/ScalarZ commands under Scal?. For a vector converted to cylindrical coordinates, ScalarX gives the radial component, ScalarY gives the phi (polar) component, and ScalarZ gives the Z component. For a vector in spherical coordinates, ScalarX gives the radial component, ScalarY gives the phi (polar) component and ScalarZ gives the theta (azimuthal) component.

Output Commands

The Output Column contains those calculator operations that result in final data outputs from calculations. The Eval button obtains final numerical results from the last stack placeholder (such as integrations).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

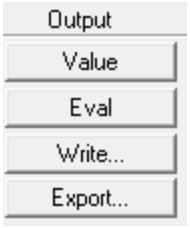


Use these commands to compute or evaluate expressions and to output the data in the calculator.

Value command	Computes the value of a field quantity on a geometry.
Eval command	Numerically evaluates and displays the results of calculator operations from the last stack placeholder, such as integrations.
Write command	Saves the contents of the top register to a disk file for future re-use.
Export command	Saves field quantities in a format that can be read by other modeling or post-processing software packages.

To plot/tabulate the extracted data using create report or create field overlays, you should use a named expression.

Value Command



This computes the value of a field quantity at a point. Use it to find:

- The magnitude of a scalar field quantity at that point.
- The x-, y-, and z-components of a vector field quantity at that point.

The **Value** command applies the value of the next-to-top stack entry (a field quantity) on the geometry entry at the top stack entry. The field quantity entry may be scalars, real or complex vectors. The geometry entry may be of any type (point, surface, line, or volume).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, if the top stack entry in the Field Calculator is a surface and the second stack register is for the real part of the E Vector ("Real(<Ex, Ey, Ez>)"), the resulting quantity will be an expression for the real part of the E vector on the surface (a VecSrf quantity). Again, the result of this command is still a placeholder expression, not the final numerical values.

Strictly speaking, the Value command provides access to the field quantity, not data outputting. It simply provides a handle to the (numerical) values of a field quantity on a given geometry, and thus the outcome of the command is not numerical values themselves. If the numerical value of a field quantity is desired, you will need to follow it up with one of the three other output commands, or define a named expression for the quantity and plot it with the Reporter.

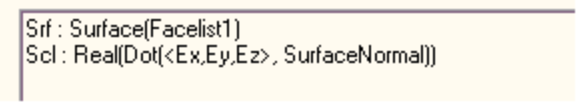
To find the value of a field quantity at a point:

1. Load the field quantity into the top register, and perform any needed operations on it.
2. Load the appropriate point into the calculator using the **Geometry/Point** command.
3. Click **Value**.

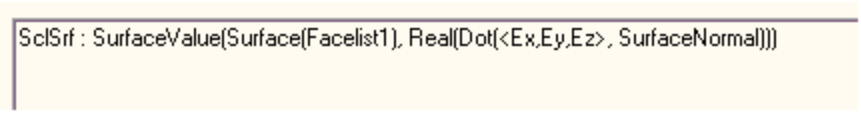
To view the numerical results of this operation, use the **Eval** command.

You can also use the **Value** command to access the intermediate SurfaceValue function.

For example, after inputting an expression for a quantity, such as an E field, and then selecting a surface geometry, the calculator stack displays something like this:



Clicking the **Value** command changes the display to the following, showing the intermediate SurfaceValue function.



In this case, SurfaceValue provides the x, y, z, coordinates of the FEM mesh and Lagrangian points so you can use **Write** to generate an .fld file containing an evaluated scalar quantity at those points.

In general for **Value**:

1. Enter any quantity onto the stack.
2. Enter a volume / surface / line / point onto the stack.
3. Press the **Value** button and you will get an appropriate geometry value on the stack.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

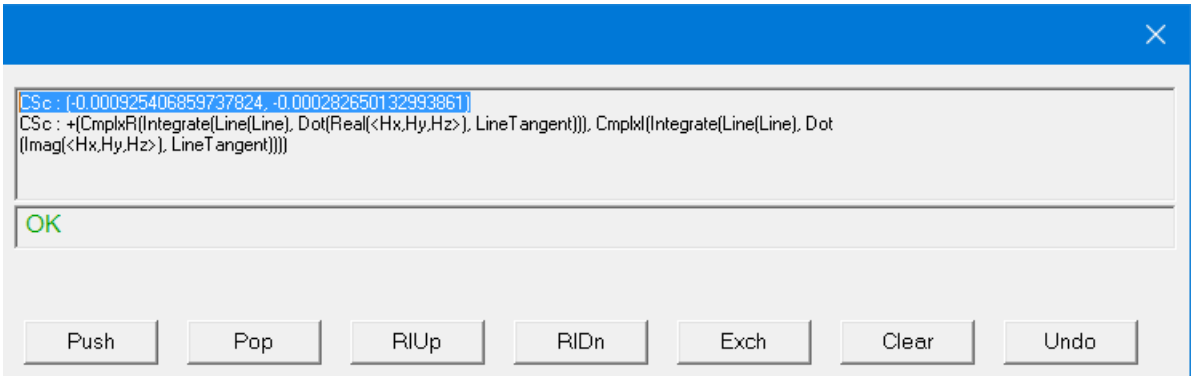
Now you can perform suitable operations such as Write, Integrate, etc. For PointValue you can also do **Eval**.

Eval Command

The **Eval** command finalizes computations and converts the text-string indicating the computation being performed to an actual numerical value. This command numerically evaluates and displays the results of calculator operations such as integrations, maximum or minimum field computations, field values at points, and so forth. For this command to work properly, the stack entry must be reduced to a single-value entry which can be a scalar, complex, or vector (real or complex). The quantity to be evaluated must be in the top register. The **Eval** command computes the numerical results of the operation, which replace the contents of the register. Units of the numerical value are not provided.

For a calculator expression that evaluates to a single value, the **Eval** command offers a quicker way to obtaining the final numerical result than defining a named expression and then trying to plot/tabulate it in the reporter. But for many expressions that don't evaluate to single values, such as the wave impedance along a line mentioned earlier, **Eval** command does not apply. As such, use of named expression and the Report Editor will be the only option.

Evaluate the expression for voltage along a line shows the definition of an expression for the voltage between two conductors by integrating the electric field along a line using the recipe given for Calculating the Current along a Wire or Trace. Then, by clicking the Evaluate command in the Output column, we obtain the numerical value for the voltage as shown in the top of the stack. It can then be either written down or copied/pasted to other programs.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

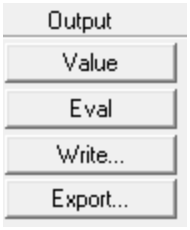
$$I = \oint H \cdot dl$$

Because **H** and **I** are complex quantities, you will need to evaluate the real part of **H** to obtain the real part of **I**, then evaluate the imaginary part of **H** to obtain the imaginary part of **I**. To do this:

1. Load **H** into the calculator using the **Qty** command.
2. Take the real part of **H** using the **Cmplx/Real** command.
3. Load the rectangular loop using the **Geom/Line** command. Create the loop, a closed polyline, to integrate over.
4. Click **Tangent** to get the component of **H** along the line.
5. Take the integral around the loop using the ∫ command.
6. Click **Eval** to evaluate the integral. The real part of **I** appears in the top register.
7. Repeat this process using the imaginary part of **H** (found with the **Cmplx/Imag** command) to obtain the imaginary part of **I**.

Write Command

If the calculator operations performed have obtained a stack entry that is intended for use in still other calculator operations, the stack entry can be saved for this purpose by using the **Write** button in the calculator Output column.



Note that in this case no name will be requested for the expression; only a save filename location will be required. This function will not work for field values derived upon a specific geometric quantity (those containing either Lin, Srf, or Vol in the stack data type indicator) as the calculator cannot know that these geometric quantities exist in identical forms in other postprocessing sessions.

When you try to read a file written with the **Write** command by clicking the **Read** button in the Input column, you must ensure that the mesh used for generating the current solution is the same as the one with which the saved field data is generated. Since the two results must be generated on the same mesh, not only the geometries of the designs on which the **Write/Read** commands were applied must be the same, they must also have gone through the same adaptive meshing process. In practice, this limits the use of Write/Read command to solutions for different frequency sweeps of the same analysis (adaptive) setup of the same design.

This command saves the contents of the top register to a disk file. Use this command to:

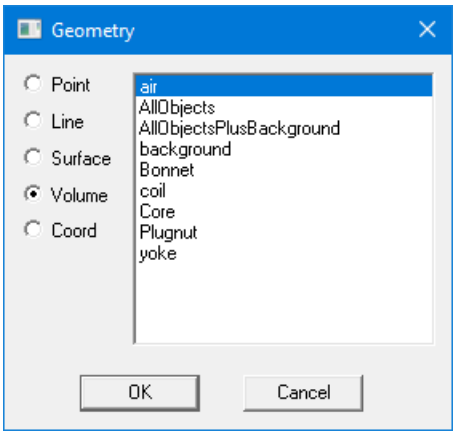
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Save registers for use during a later postprocessing session.
- Save a field quantity for use when postprocessing a different model.

> **Note:**
>
> Use of the Global Coordinate System is assumed. Local coordinate systems are not used.

To save a register:

1. Click **Write**.
2. If the register includes numeric data with a constrained quantity (such as jsurf), you see a dialog box that gives a choice of constraining geometries. For example:



3. Select the geometry of interest, and select **OK**.

   This displays a file browser.

4. Use the file browser to specify the register's file name and directory path. A .reg extension is automatically assigned to register files and a .fld extension is assigned to field files. You can choose to save both .reg and .fld files, or either one.
5. Click **OK**.

   The contents of the register are saved to the file you specified.

Export Command

To output a field quantity or calculation result for use by some third-party post-processors, use the **Export** button in the calculator Output column. You can map the field quantity to either a customized grid of points specified by a points file or a three-dimensional Cartesian, Cylindrical, or Spherical grid specified interactively through a dialog box. In the latter case, you must specify

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the dimensions and spacing of the grid in the coordinate system with units. For Cylindrical and Spherical coordinate systems, you can also specify an offset from the origin.

The **Export** command opens the **Export Solution** dialog box, from which you can export the field quantity in the top register to a file, mapping it to a grid of points.

Use this command to save field quantities in a format that can be read by other modeling or post-processing software packages. Two options are available for defining the grid points on which to export:

Input grid points from file	Maps the field quantity to a customized grid of points. Before using this command, you must create a file containing the points and units.
Calculate grid points	Maps the field quantity to a three-dimensional Cartesian grid. You specify the dimensions and spacing of the grid in the Cartesian, cylindrical, or spherical coordinates, with units that you specify. The initial units are taken from the model.

> **Note:**
>
> Use of the Global Coordinate System is assumed. Local coordinate systems are not used.

To export a field quantity to a customized grid:

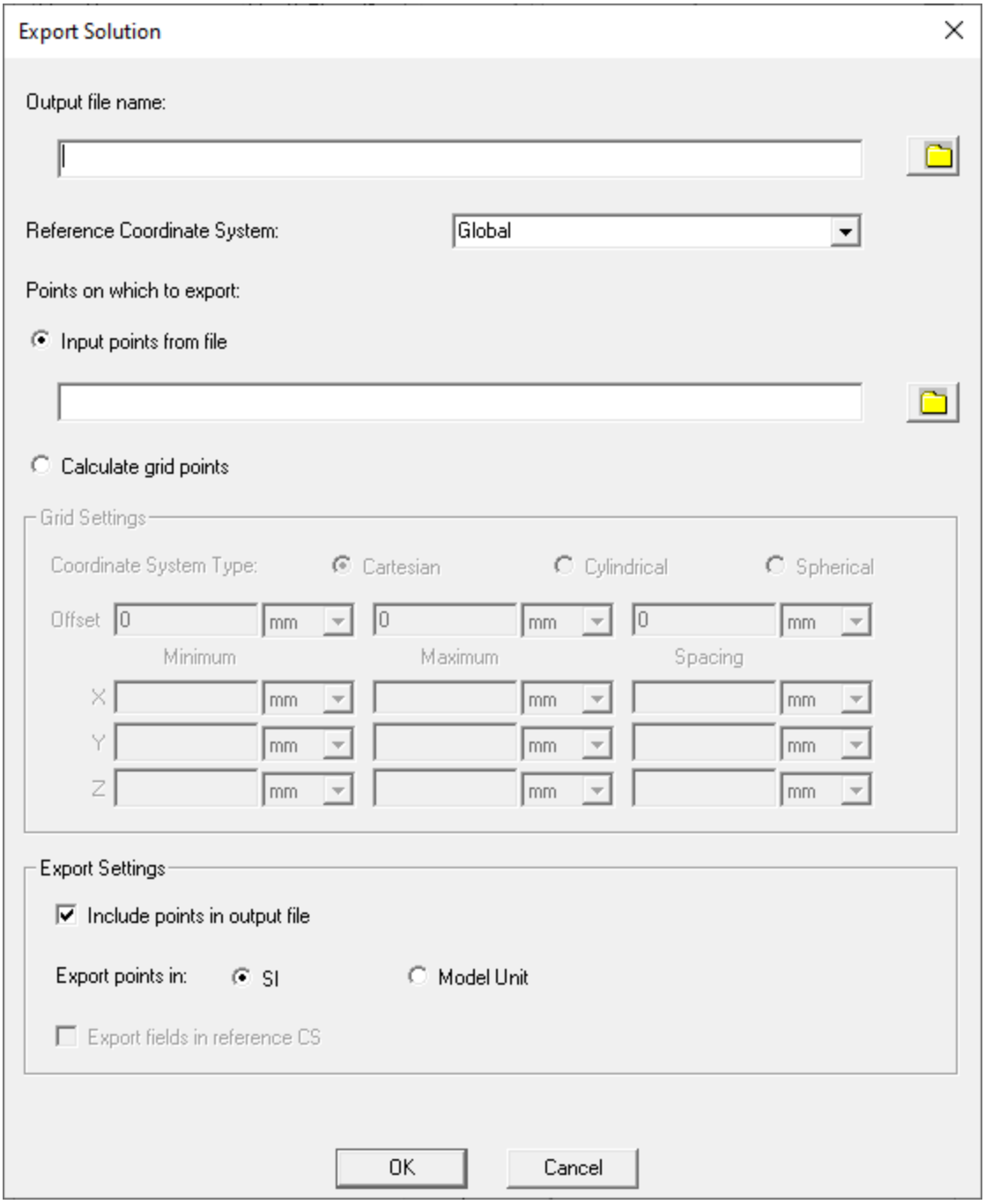
1. Load the quantity into the top register for the fields calculator, and perform any operations on it.
2. If desired, load a volume using the **Geometry** command.

   You can use the Domain command to limit the calculation to the volume you specify. If you export a Domain filtered numeric, points that are filtered out by the domain will not be written out.

3. Click the **Export** button in the Fields Calculator.

   This opens the **Export Solution** dialog box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Type or select the name of the file in which the field quantity is to be saved in the **Output File Name** text box. You can use the file icon to open the file browser to specify the file name and directory path. The **.fld** extension is automatically assigned to this file.
5. If the design contains multiple coordinate systems, a drop-down menu lets you select the coordinate system to use. This can be helpful in evaluating the fields output when relative

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

coordinate systems used and unitized in the design.

6. Click either the **Input grid points from file** radio button if you have a created a .pts file containing the grid points, or click the **Calculate grid points** radio button.
   - If you select **Input grid points from file**, either type the name and directory of the file containing the points on which the field is to be mapped, or, click on the file icon and use the file browser to locate the point file (**.pts** extension).

     Note:

     The **.pts** file should contain the units to use for the export as shown in this file stub:

     Unit=mm

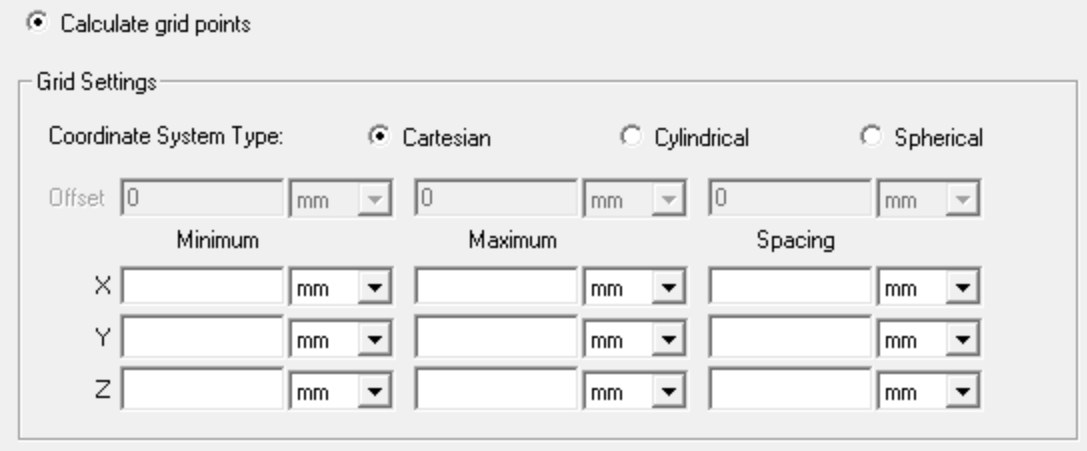
     -5.5 -5.5 -5.21475

     -5.5 -5.5 -5.14425

     -5.5 -5.5 -5.07375

     -5.5 -5.5 -5.021

   - If you select **Calculate grid points** button, you can specify the coordinate system as Cartesian, Cylindrical, or Spherical.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cartesian: for each grid dimension on **X**, **Y**, and **Z**, enter the Minimum, Maximum, and grid point spacing.

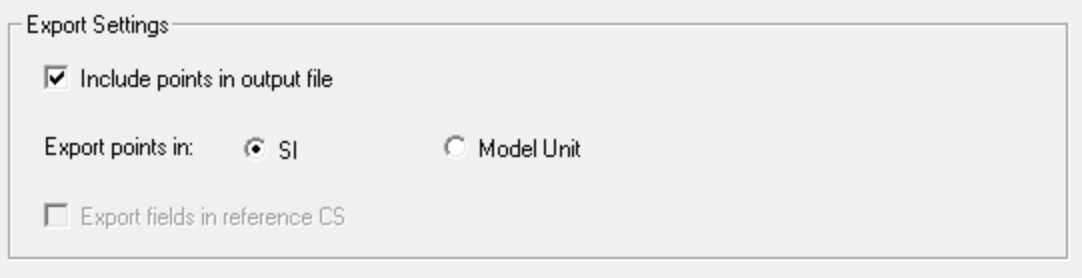
Cylindrical: for each dimension Rho, Phi, and Z, enter the Minimum, Maximum, and grid point Spacing. You can also specify an origin of Offset.

Spherical: for each dimension R, Theta, and Phi, enter the Minimum, Maximum, and grid point Spacing. You can also specify an origin of Offset.

> **Note:** When you export fields on a 1D or 2D line/surface from the field calculator, the start and stop values must be the same for one or two of the coordinate system start/stop ranges. If you specify a zero spacing for a dimension, the export uses only the minimum value.

The default coordinate system will be Cartesian. The default offset will be all zeroes. The length units will default to model unit and default angle unit will be degree. At the start the minimum/maximum/Spacing entries are blank. The user entered values are not remembered when the dialog is closed.

7. For larger files, for the Export Settings, you may want to uncheck the **Include points in output file** box. If you uncheck the box, the file header will include minimum, maximum and spacing information from which you can recalculate the grid points.
8. You can also specify SI or Model Units. Model Units can be helpful to evaluate the fields output when relative coordinate systems are unitized in the design.



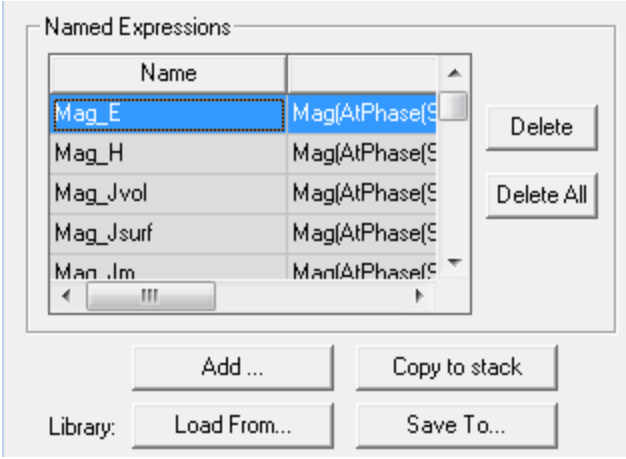
9. If field to be exported has type vector or complex vector, the 'Export Field in reference CS' checkbox is enabled and you can specify that field to be exported in reference CS. It can be helpful to evaluate the fields output when rotated coordinate systems are unitized in the design
10. Click **OK** to export the file.

    The field quantity is mapped to the grid and saved to the file you specified (**.fld** extension).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculating Derived Field Quantities Using Named Expressions

The **Named Expressions** panel displays expressions that can be included in register definitions by name. You can add additional expressions to the Named expression list by creating the expression in the register display area and then clicking the Add button. This lets you add to the Named expression library.



Click on a named expression to select it. When a named expression has been selected, the **Copy to Stack** button is activated. Click **Copy to Stack** to push the expression on the top of the stack.

When an HFSS design is open and a Solution Setup has been performed, the following predefined named expressions are available.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Expression Name	Expression Definition
Mag_E	Mag(AtPhase(Smooth (<Ex,Ey,Ez>),Phase))
Mag_E_t	Mag(Smooth(<E_tx,E_ty,Etz>)). Used with Transient projects.
Mag_H	Mag(AtPhase(Smooth (<Hx,Hy,Hz>),Phase))
Mag_H_t	Mag(Smooth(<Hx,Hy,Hz>). Used with Transient projects.
Mag_Jvol	Mag(AtPhase(Smooth (<JVx,JVy,JVz>),Phase))
Mag_Jsurf	Mag(AtPhase(Smooth (<Jsurfx,Jsurfy,Jsurfz>),Phase))
Mag_Jsurf_t	Mag(<Jsurf_tx,Jsurf_ty,Jsruf_ tz>). Used with Transient projects.
ComplexMag_E	Mag(CmplxMag(Smooth (<Ex,Ey,Ez>))
ComplexMag_H	Mag(CmplxMag(Smooth (<Hx,Hy,Hz>))
ComplexMag_Jvol	Mag(CmplxMag(Smooth (<JVx,JVy,JVz>))
ComplexMag_Jsurf	Mag(CmplxMag(Smooth (<Jsurfx,Jsurfy,Jsurfz>))
Vector_E	AtPhase(Smooth (<Ex,Ey,Ez>),Phase)
Vector_E_t	(Smooth(<E_tx,E_ty,E_tz>). Used in Transient projects.
Vector_H	AtPhase(Smooth (<Hx,Hy,Hz>),Phase)
Vector_H_t	(Smooth(<H_tx,H_ty,H_tz>). Used in Transient projects.
Vector_Jvol	AtPhase(Smooth (<JVx,JVy,JVz>),Phase)
Vector_Jsurf	AtPhase(Smooth(<Jsurf_ x,Jsurf_y,Jsurf_z>),Phase)
Vector_Jsurf_t	(Smooth(<Jsurf_tx,Jsurf_ ty,Jsurf_tz>). Used in Transient projects.

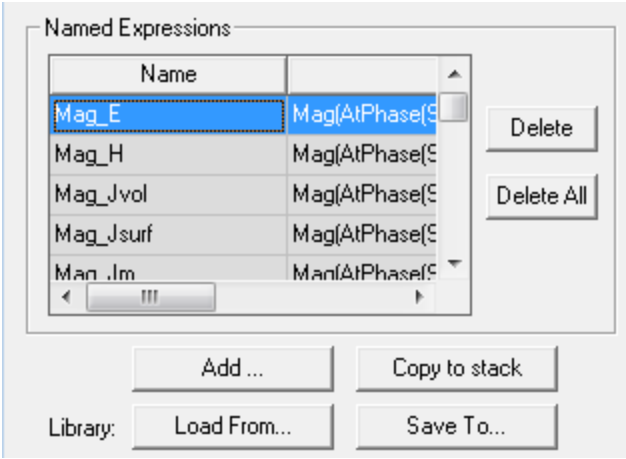
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Vector_RealPoynting	Real(Poynting)
VectorReal_Poynting_t	Poynting_t. Used in Transient projects.
Local_SAR	LocalSAR
Average_SAR	AverageSAR
Surface_Loss_Density	SurfaceLossDensity. See further discussion here.
Volume_Loss_Density	VolumeLossDensity. See further discussion here.
Volume_Loss_Density_t	VolumeLossDensity_t. Used in Transient projects.
Surface_Force_Density	Surface Forces exist when one side is conductor, but the other is not, or finite conductivity and layered impedance boundary. This is mainly for the purpose of mapping surface force density in HFSS to Workbench Mechanical. For details on the calculations, see the technical notes.
Mag_Displacement	Mag(Smooth(<Ux,Uy,Uz>)). Magnitude of displacement, used with Workbench in projects exploring stress feedback.
Displacement_Vector	Smooth(<Ux,Uy,Uz>). Used with Workbench in projects exploring stress feedback.

Named Expression Library

The named expression library in the Fields Calculator provides a way to conveniently calculate frequently used quantities. Some standard field quantities have been predefined and are always accessible. These field quantities have a gray background color and cannot be deleted. You can assign a name to the top entry of Data Stack and add it in the Named Expression list. The library comes with several predefined expressions. Once these new expressions are added/loaded, they are available through HFSS's standard post-process capabilities. For example, they can be copied back to the Data Stack in calculator by **Copy to Stack** for further operations, or be plotted using Report Editor and Field Overlay.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can combine calculator Input commands in any legal fashion, including complex quantities, to produce new named expressions.

- Adding named expressions to the Fields Calculator expression library
- Copying named expressions to the Calculator Stack
- Saving named expressions to a Personal Library
- Loading named expressions
- Deleting named expressions that you added

To add a named expression of your own to the Fields Calculator list:

1. In the register display area, create the expression by using the calculator Input commands.

   You can combine input commands in any legal fashion, including the use of complex quantities. If you select an input command that is not legal for a current operation, you receive an error message.

2. When you finish creating the expression, click **Add** in the *Named Expressions* panel.

   The *Named Expression* dialog box appears.

3. Type a name for the expression in the **Name** text box.

   The new expression is added to the list of named expressions.

To copy a named expressions to the Calculator Stack:

- You can scroll through the list, select any desired named expression, and click **Copy to Stack** to move it to the calculator stack, where you can use it to generate calculated outputs.

To delete named expressions that you added:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the *Named Expression* list contains one or more user-defined expressions, the **Delete** and **Clear All** buttons are active. (You cannot delete or clear the predefined named expressions.)

- To delete the selected user-defined named expression, click **Delete**. To delete all user-defined named expressions, click **Clear All**.

To save one or more named expressions for the Fields Calculator to a personal Library:

1. Click **Save To** on the Fields Calculator.

   The *Select Expressions for Saving* dialog box displays.

2. If any new named expressions exist, you can select one or more to save to a file.
3. Specify a file name and click **OK** to save the file.

To load named expressions for the Fields Calculator from a personal library:

1. From the Fields Calculator, click **Load From**.

   This displays a file browser that you can use to search for existing .clc files.

2. Select the library to load and click **OK**.

   This loads the expression file you have selected.

Exiting the Fields Calculator

Click **Done** to exit the Fields Calculator.

Fields Calculator Recipes

The following pages contain calculator recipes for deriving a number of commonly used output parameters from solved HFSS projects.

Calculating Numerical Quantities

Calculating the Current along a Wire or Trace

"Calculating the Voltage Drop along a Line" on page 23-467

"Calculating Net Power Flow through a Surface" on page 23-468

"Calculating the Average of a Field Quantity on a Surface" on page 23-469

"Calculating the Peak Electrical Energy in a Volume" on page 23-471

"Calculating the Q of a Resonant Cavity" on page 23-472

"Calculating the Max Value of a Field Quantity in a Volume" on page 23-474

"Calculating Dielectric (or Conduction) Loss in a Volume" on page 23-475

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculating Quantities for 2D (Line) Plot Outputs

"Plotting Wave Impedance along a Line" on page 23-476

"Plotting the Phase of E Tangential to a Line/Curve" on page 23-477

"Plotting the Maximum Magnitude of E Tangential to a Line/Curve" on page 23-478

Calculating Quantities for 3D (Surface or Vector) Plot Outputs

"Plotting the E-Field Magnitude Normal to a Surface" on page 23-480

Calculating Quantities for 3D (Volume) Plot Outputs

"Generating an Iso-Surface Contour for a Given Field Value" on page 23-481

Calculating Quantities for Animated Outputs

"Generating an Animation on Multiple Planes with a Positional Variable" on page 23-482

Creating User Defined Named Expressions Library

"Generating Cartesian/Cylindrical/Spherical-Coordinate Field-Components Library" on page 23-483

Recipe Format

Each calculator recipe will be provided in the format shown below:

EXAMPLE: Title of Current Calculation

Description

The first paragraph will give a brief description of the calculation's intent.

Usage Examples

The second paragraph will give an example of a project type on which the calculation might be useful. It may also comment upon the reasons such a calculation might be of interest.

Prerequisites

The third (optional) paragraph will indicate what must be present before doing the calculator operations, e.g., if certain geometry need to be generated to use in calculations.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Each button click shown as a step	resulting stack entry/type
drop-down menu>Submenu pick also shown as a single step	Scl: {placeholders for numerical results shown in brackets}
Button steps requiring data entry will have entry quantity shown in {brackets}	resulting stack entry/type (*notes follow in italics)*

Calculating the Current along a Wire or Trace

Description

Obtains the full complex current in a wire or trace conductor (e.g., microstrip, stripline) at a specific location by integrating the magnetic field along a closed path encircling the conductor. .

$$I = \oint_l \vec{H} \cdot \vec{dl}$$

where *l* is a closed path, which could be a circled line object

Usage Examples

To find the current distribution along a wire (dipole, monopole, etc.) antenna, this calculation could be repeated at periodic positions along the length of the antenna.

Prerequisites

You must create a closed line for the integration path using **Draw>Line** before beginning calculator operations. The line must be orthogonal to the direction of current flow, should not intersect the wire/trace, and should not be too much bigger than the wire/trace.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Quantity>H	CVc : <Hx,Hy,Hz>
Complex>Real	Vec : Real(<Hx,Hy,Hz>)
Geometry>Line...> {select line}	Lin : Line (line1) (user line name may differ from example)
Tangent	SclLin: LineValue(Line(...),Dot(Real<Hx,Hy,Hz>), LineTangent))

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
∫	Scl : Integrate(Line(....
Complex>CmplxReal	CSc : CmplxR(Integrate(Line(Line1),Dot(…)))
Quantity>H	CVc : <Hx,Hy,Hz>
Complex>Imag	Vec : Imag(<Hx,Hy,Hz>)
Geometry>Line...> {select line}	Lin : Line (line1) (user line name may differ from example)
Tangent	ScLin: LineValue(Line(...),Dot(Imag<Hx,Hy,Hz>), LineTangent))
∫	Scl : Integrate(Line(....
Complex>CmplxImag	CSc : CmplxI(Integrate(Line(Line1,Dot(…)))
+	CSc: (CmplxR(Integrate(Line(Line1),Dot(…))), CmplxI(Integrate (Line(Line1),Dot(…))))
Eval	CSc : {complex numerical value} *(Final complex current result)*

Calculating the Voltage Drop along a Line

Description

Provide the complex voltage drop, in volts between two points by integrating the E-field along a line.

$$V = \int_{l} \vec{E} \cdot \vec{dl}$$

where *l* is a path between two points on which voltage difference are measured. Usually it is a straight line object.

Usage Examples

To find the voltage excited across the width of a slot antenna element; to test whether a voltage exceeds breakdown in a particular dielectric media.

Prerequisites

You must create the line along which the E-field is to be integrated using **Draw>Line** before you can complete the calculator routine.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Quantity>E	CVc : <Ex,Ey,Ez>
Complex>Real	Vec : Real(<Ex,Ey,Ez>)
Geometry>Line...> {select line}	Lin : Line (line1) (*user line name may differ from example*)
Tangent	SclLin: LineValue(Line(...),Dot(Real<Ex,Ey,Ez>), LineTangent))
∫	Scl : Integrate(Line(....
Complex>CmplxReal	CSc : CmplxR(Integrate(Line(Line1),Dot(…)))
Quantity>E	CVc : <Ex,Ey,Ez>
Complex>Imag	Vec : Imag(<Ex,Ey,Ez>)
Geometry>Line...> {select line}	Lin : Line (line1) (*user line name may differ from example*)
Tangent	ScLin: LineValue(Line(...),Dot(Imag<Ex,Ey,Ez>), LineTangent))
∫	Scl : Integrate(Line(....
Complex>CmplxImag	CSc : CmplxI(Integrate(Line(Line1),Dot(…)))
+	CSc: (CmplxR(Integrate(Line(Line1),Dot(…))),CmplxI(Integrate (Line(Line1),Dot(…))))
Eval	CSc : {complex numerical value} (*Final complex voltage result*)

Calculating Net Power Flow through a Surface

Description

This recipe allows calculation of power flow through an open or closed surface by integrating the Poynting vector normal to that surface.

$$W = \int_S Re(\vec{P}) \cdot \vec{n} dS$$

where *S* is the surface that is used to calculate the power, and $\vec{n}$ is the normal vector to the surface *S.*

Usage Examples

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This calculation could be used on scattered field data resulting from an incident wave excited HFSS project to evaluate reflection from a radome filter or FSS (frequency selective surface). It might also be used on the closed exterior surface of a solid volume to determine power dissipation within the volume (due to conservation of energy, what goes in a closed surface must come out, unless there is a loss or storage [e.g., standing wave, resonance] mechanism).

Prerequisites

The surface on which the integration is to be performed must exist prior you can complete the calculation. If the surface is the exterior of a solid object, no customer geometry creation is necessary. If the surface is only a subset of an object's faces, or a slice through the entire plane of the model not already defined by a separate 2D entity, then you must create a Faces List and/or Cutplane to represent the integration location.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Quantity>Poynting	CVc : Poynting
Complex>Real	Vec : Real(Poynting) (*discards the unneeded imaginary component*)
Geometry>Surface...> {select surface}	Srf : Surface(Facelist1) (*above is example; user surface shown may vary*)
Normal	SclSrf : SurfaceValue(Surface(Facelist1), Dot(Real(Poynting), SurfaceNormal) *(takes the dot product of the vector data with the normal to the surface(s) selected*)
∫	Scl : Integrate(Surface..)
Eval	Scl : {numerical value} (*final answer in **watts***)

Calculating the Average of a Field Quantity on a Surface

Description

This recipe permits you to calculate the average of a field quantity on a Surface geometry, by dividing the Integration of the field value on the surface by the surface area.

Usage Examples

This calculation could be used to determine the average phase of the E-field at a given cutplane through a project, to find the average current on a trace surface, or to calculate the average H-field tangential to a 2D object used as an aperture. The specific example steps below will be for

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the first usage example mentioned (average phase of an E-field on a surface), but the format for integration on a surface and for finding the area of the surface is identical for the other applications as well.

Prerequisites

The surface on which the integration is to be performed must exist. If the surface is the exterior of a solid object, no customer geometry creation is necessary. If the surface is only a subset of an object's faces, or a slice through the entire plane of the model not already defined by a separate 2D entity, then you must create a Faces List and/or Cutplane to represent the integration location.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Quantity>{select field quantity}	CVc : <Ex, Ey, Ez> (E-field used as example)
{Derive desired scalar field data For example: Scal?>ScalarX Complex>CmplxPhase}	CSc : ScalarX(<Ex, Ey, Ez>) *(first operation result)* Scl : Phase(ScalarX(<Ex, Ey, Ez>)) *(second operation result)*
Geometry>Surface...>{select surface}	Srf: Surface(plane1) (*user surface shown may vary*)
Value	SclSrf : SurfaceValue(Surface(plane1), Phase(ScalarX (<Ex, Ey, Ez>)))
∫	Scl : Integrate(...)
Geometry>Surface...>{select surface}	Srf : Surface(plane1)
Unit Vec>Normal	Vec : SurfaceNormal
Geometry>Surface...>{select surface}	Srf : Surface(plane1)
Normal	SclSrf: SurfaceValue(Surface(plane1)... (*takes the dot product of the surface with its own normal*)
∫	Scl : Integrate(Surface(...
/	Scl : /(Integrate(SurfaceValue(...
Eval	Scl : {numerical value} (*for this example units are in **deg** or **radians***)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculating the Peak Electrical Energy in a Volume

Description

This recipe permits you to calculate the peak electrical energy in a volume object. The solution is achieved by integrating **E E** within the volume.

$$W = \int_V \frac{\epsilon_r \epsilon_0 (\vec{E} \cdot \vec{E}^*) dV}{2}$$

where *V* is the volume.

Usage Examples

This calculation could be used to determine the average total energy with respect to time in a terminating resonant cavity. (In a sealed, one-port structure at resonance, energy is converted back and forth between the electrical and magnetic fields, but maintains the same total quantity; therefore the peak electrical energy is equal to the average total energy.)

Prerequisites

The volume object which the integration is to be performed must exist before the computation can be completed. If the volume for integration consists of the volume of several drawing objects, you must create a single list entry representing their combined volumes using **Modeler>Lis>Create>Object List**.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Quantity>E	CVc : <Ex, Ey, Ez>
Complex>Conj	CVc : Conj(<Ex, Ey, Ez>)
Quantity>E	CVc : <Ex, Ey, Ez>
Dot	CSc : Dot(Conj(<Ex, Ey, Ez>), <Ex...
Complex>Real	Scl : Real(Dot(Conj(<Ex, Ey, Ez>), ... (*note: the dot product of the E with its conjugate should lead to a real quantity, but the calculator still assume as complex*)
Geometry>Volume...> {select volume}	Vol : Volume(box1) (*above is example, user entry may differ*)
∫	Scl : Integrate(Volume(...
Eval	Scl : {numerical quantity}
Constant>Epsi0	Scl : 8.854187817E-012

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Number>Scalar> {enter r for volume}	Scl : {numerical quantity}
*	Scl : {numerical quantity} *(stack entry is volume)*
Number>Scalar>0.5	Scl : 0.5
*	Scl : {numerical quantity}
*	Scl : {numerical quantity} *above is electrical energy in* ***joules***

Calculating the Q of a Resonant Cavity

Description

This recipe permits you to calculate the Q in a homogeneous dielectric-filled cavity with uniform wall losses, using the equation:

$$Q_u = \frac{\int_\Omega |\mathrm{H}|^2 d\Omega}{\frac{s}{2}\oint_\Gamma |\mathrm{n} \times \mathrm{H}|^2 d\Gamma + tg\delta\left(\int_\Omega |\mathrm{H}|^2\right) d\Omega}$$

where *s* is skin depth, *tg* is dielectric loss tangent, *n* is the surface normal for the cavity wall faces, and and represent wall surface area and cavity volume, respectively.

Usage Examples

To calculate the Q of an air- or solid-dielectric filled cavity, fed with a below-cutoff port aperture, or obtained via an eigen solution.

Prerequisites

The Object (or Object List) representing the cavity total volume must already exist, as must the Face List corresponding to the total wall surface area of the cavity. You can create both via the Modeler menu. The solution should be tuned to the desired resonant frequency for evaluation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Quantity>H	CVc : <Hx, Hy, Hz>
Push	*(above entry duplicated)*
Complex>Conj	CVc : Conj(<Hx, Hy, Hz>)
Dot	CSc : Dot(<Hx, Hy, Hz>, Conj(...
Complex>Real	Scl : Real(Dot(<Hx, Hy, ...
Geometry>Volume>{select cavity volume}	Vol : Volume(cav_total) (*above is example; user entry may differ*)
∫	Scl : Integrate(Volume(cav... (*above represents energy stored in cavity*)
Push	*(above entry duplicated)*
Number>Scalar>{enter loss tan for volume}	Scl : {numerical value} (*loss tangent for dielectric fill in cavity*)

The above equation is ***only*** valid for cavities filled with one dielectric material across the entire volume. For cavities with different dielectric fills (e.g., a dielectric resonator within a larger metal cavity), dielectric loss must be evaluated using integration by parts for each dielectric material volume. The equation also assumes the same conductivity for all walls, and no nonreciprocal (e.g., ferrite) property to either walls or fill.

*	Scl : *(Integrate(Volume(... (*above represents energy lost in dielectric material losses*)
Quantity>H	CVc : <Hx, Hy, Hz>
Geometry>Surface>{select cavity surfaces}	Srf : Surface(cav_tot_faces) (*above is example; user entry may differ*)
Unit Vec>Normal	Vec : NormalSurfaceNormal
Cross :	Cross(<Hx, Hy, Hz>, SurfaceNormal)
Push	*(above entry duplicated)*
Complex>Conj	CVc : Conj(Cross(<Hx, Hy, Hz>, ...
Dot	CSc : Dot(Cross(<Hx, Hy, Hz>, ...
Complex>Real	Scl : Real(Dot(Cross(<Hx, ...
Geometry>Surface>{select cavity surfaces}	Srf : Surface(cav_tot_faces)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*	Scl : *(Integrate(Volume(... (*above represents energy lost in dielectric material losses*)
∫	Scl : Integrate(SurFace(...
Number>Scalar>2	Scl : 2
Constant>Pi	Scl : 3.14159265358979
Function>Scalar>Freq	Scl : {current freq, in Hz}
*	Scl : {numerical result, pi*f}
Number>Scalar>{enter r for walls}	Scl : {entered value, unitless}
*	Scl : {numerical result, pi*f*mur}
Constant>Mu0	Scl : 1.25663706143592E-006
*	Scl : {numerical, pi*f*mur*mu0}
Number>Scalar>{enter wall conductivity}	Scl : {entered value, s/meter}
*	Scl : {numerical, pi*f*mur*mu0* }
	Scl : {numerical, sqrt of above}
*	Scl : {numerical result, 2*above}
1/x	Scl : {numerical result} (*above is skin depth/2*)
*	Scl : *(Integrate(... (*above is energy lost in walls*)
+	Scl : +(*(Integrate(...
/	Scl : /(+(*(Integrate(...
Eval	Scl : {numerical result} (*above is Q of homogeneous fill and wall conductivity cavity**, unitless***)

Calculating the Max Value of a Field Quantity in a Volume

Description

This recipe permits you to calculate the Max electrical field (value or position) in a volume object.

Usage Examples

This calculation could be used to determine the Max (or Min) value (or position) of electrical field in a resonant cavity or dielectric object. Find out the hot (or quiet) spot value and location.

Prerequisites

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The volume (or surface) object within which the Max function is to be performed must already exist before the computation can be completed. If the volume (or surface) for Max function consists of the volume of several drawing objects, you first create a single list entry representing their combined volumes using **Modeler>List>Create>Object (or Face) List**.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Named Expressions>Mag_E>Copy to stack	Scl : Mag_E (Mag_E *is used as an example*)
Geometry>Volume..>{select volume}	Vol : Volume(box1) (*above is example, user entry may differ*)
Max>Value	Scl:Maximum(Volume(box1),Mag_E) *(user can enter Max→Position to find out Max E location. Also Min→Value (Position) leads to* Min E value and location)
Eval	Scl : {numerical quantity}

Calculating Dielectric (or Conduction) Loss in a Volume

Description

This recipe shows you to calculate the dielectric/conduction loss in a volume object.

Usage Examples

This calculation could be used to determine the loss in a dielectric or metal object. For loss dielectric or finite conductivity metal object, its loss may have significant impact on user"s design performance. Find out the key loss factors is critical for a successful design.

Prerequisites

The volume object within which the loss calculation is to be performed must already exist before the computation can be completed. If the volume for loss calculation consists of the volume of several drawing objects, you must create a single list entry representing their combined volumes using **Modeler>List>Create>Object List**.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Named Expressions>Volume_Loss_Density> Copy to stack	Scl : Volume_Loss_Density *(Volume Loss Density is used as an example)*
Geometry>Volume...>{select volume}	Vol : Volume(box1)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Operation	Resulting Stack Display **(top entry only unless noted)** (*above is example, user entry may differ*)
∫	Scl: Integrate(Volume(box1), Volume_ Loss_...)
Eval	Scl : {numerical quantity} (*above is loss from box1, either dielectric or conductional, **Watt***)

Plotting Wave Impedance along a Line

Description

This recipe generates a 2D plot of wave impedance in ohms vs. length for a line geometry. Wave impedance is obtained directly by taking the ratio of the transverse components of the electric field to the ratio of the transverse components of the magnetic field.

$$Z = \frac{E_o^-(x)}{H_{\bar{o}}(x)}$$

Where $E_o^-(x)$ is the transverse component of the electric field, and $H_{\bar{o}}(x)$ is transverse component of the magnetic field.

Usage Examples

This calculation could be used to display wave impedance vs. position along a length of waveguide with a changing cross-section. It could also be used to display the changes in wave impedance in free space at some boundary (e.g., frequency selective surface, radome) when performed on an incident wave problem.

Prerequisites

The line along which the impedance is to be plotted should be defined before performing this calculation. You can generate a line using **Modeler>Draw>Line**.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Quantity>E	CVc : <Ex, Ey, Ez>

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Smooth	CVc : Smooth(<Ex, Ey, Ez>)
Complex>CmplxMag	Vec : CmplxMag(Smooth(<Ex, Ey, ...
Number>Vector>{enter unit vector in direction of propagation}	Vec : <0, 0, 1> *(Z-directed unit vector used for example)*
Cross	Vec : Cross(CmplxMag(Smooth(<...
Mag	Scl : Mag(Cross(CmplxMag(Smooth...
Quantity>H	CVc : <Hx, Hy, Hz>
Smooth	CVc : Smooth(<Hx, Hy, Hz>)
Complex>CmplxMag	Vec : CmplxMag(Smooth(<Hx, Hy,...
Number>Vector>{enter unit vector in direction of propagation}	Vec : <0, 0, 1> *(Z-directed unit vector used for example)*
Cross	Vec : Cross(CmplxMag(Smooth(<...
Mag	Scl : Mag(Cross(CmplxMag(Smooth...
/	Scl : /(Mag(Cross(CmplxMag(Sm...
Add	Zwave : /(Mag(Cross(CmplxMag(…
Type: Zwave	(*User defined Named Expression Zwave is added*)
OK	
Done	{*exit field calculator window*}
HFSS>Results> **Create Fields Report>** **Rectangular Plot>** Geometry: Line1 Category: Calculator Expressions Quantity: Zwave New Report	{2D graph displayed} *(y axis is wave impedance in **ohms** and x axis is position along line in drawing units)*

Plotting the Phase of E Tangential to a Line/Curve

Description

This recipe generates a 2D plot of the phase of an E-field whose vector component is tangential to a line. The line may also be a curve (faceted polyline).

Usage Examples

This calculation could be used to display the change in phase of the E field tangential to a circular path within a cylindrical dielectric resonator, when used on either a driven or

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

eigensolution problem. Identifying the phase change along this curved path is often necessary to determine the mode index (e.g., Mode 10) which a particular eigensolution or S-parameter resonance represents.

Prerequisites

You must define the line along which the phase is to be plotted before performing this calculation. Use the **Modeler>Draw>Line** command.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Quantity>E	CVc : <Ex, Ey, Ez>
Geometry >Line >{select desired line}	Lin : Line(line1) (*above line is example; user's may vary*)
Unit Vec>Tangent	Vec : LineTangent
Complex>CmplxReal	CVc : CmplxR(LineTangent) (*converts unit vector to complex vector*)
Dot	CSc : Dot(<Ex,Ey,Ez>),CmplxR(...
Complex>CmplxPhase	Scl : Phase(Dot(<Ex,Ey,Ez>), ...
Add	Ephase : Phase(Dot(<Ex,Ey,Ez>),…
Type: Ephase	(*User defined Named Expression Ephase is added*)
OK	
Done	{*exit field calculator window*}
HFSS>Results> **Create Fields Report>** **Rectangular Plot** Geometry: Line1 Category: Calculator Expressions Quantity: Ephase New Report	{2D graph displayed} *(y axis is E field phase in **deg** and x axis is position along line in drawing units)*

Plotting the Maximum Magnitude of E Tangential to a Line/Curve

Description:

This recipe generates a 2D plot of the maximum magnitude of an E-field tangential to a line. The line may also be a faceted curve. The maximum magnitude is not necessarily tied to the same input phase value along the length of the line.

Usage Example(s):

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This calculation could be used to display the maximum magnitude of an E-field at all points along a line or curve in a transmission line structure, where it is the maximum magnitude and not the magnitude along the line corresponding to a single „snapshot in time" (single port excitation phase) that is of interest. Such data could be used to determine whether the present design might exceed dielectric breakdown voltage in a particular location.

Prerequisites:

You should define the line along which the field data is to be plotted before performing this calculation. Use the **Modeler>Draw>Line** command.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Quantity>E	CVc : <Ex, Ey, Ez>
Geometry>Line>{select desired line}	Lin : Line(line1) (*above line is example; user's may vary*)
Unit Vec>Tangent	Vec : LineTangent
Complex>CmplxReal	CVc : CmplxR(LineTangent) (*converts unit vector to complex vector*)
Dot	CSc : Dot(<Ex,Ey,Ez>),CmplxR(...
Complex>CmplxPeak	Scl : CmplxMag(Dot(<Ex,Ey,Ex ... (a*bove quantity is the **maximum** magnitude of the E-field tang. to the line. To obtain the mag. associated with a particular port phase excitation, enter a number into the stack and use the* Complex>AtPhase *operation instead.*)
Add	Et_max : Phase(Dot(<Ex,Ey,Ez>),…
Type: Et_max	(*User defined Named Expression Et_max is added*)
OK	
Done	{exit field calculator window}
HFSS>Results>Create Fields Report>Rectangular Plot	{2D graph displayed} *(y axis is E field mag in **v/m** and x axis is position along line in drawing units)*
Geometry: Line1 Category: Calculator Expressions Quantity: Et_max	
New Report	

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plotting the E-Field Magnitude Normal to a Surface

Description

This recipe generates a scalar intensity plot of the E-field magnitude normal to a particular surface (or group of object surfaces, list of object faces), relative to a given input phase excitation. where *S* is the surface geometry, and is the normal vector to the surface *S*.

Usage Examples

This calculation could be used instead of the automatic Plot→Fields→MagE upon surface, when only the magnitude of the E-field with a particular vector orientation is desired. For example, to evaluate the field available for coupling to a probe structure with a particular orientation.

Prerequisites

You should create the plane to which the desired field component should be normal before you begin the following steps. Use **Modeler>Draw>Plane**, or **Modeler>List>Create>Faces List**, or **Modeler>Draw>Rectangl**e (or other 2D sheet).

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Quantity>E	CVc : <Ex, Ey, Ez>
Function>Phase	Scl : Phase
Complex>AtPhase	Vec : AtPhase(<Ex,Ey,Ez>, Phase)
Geometry>Surface>{select desired cutplane, faces list, or surface list}	Srf : Surface(faces1) *(faces1 used as example)*
Unit Vec>Normal	Vec : SurfaceNormal
Dot	Scl : Dot(AtPhase (<Ex,Ey,Ez>,Phase),…)
Add Type: E_normal OK	E_normal : Dot(AtPhase (<Ex,Ey,Ez>,0),… (*User defined Named Expression E_ normal is added*)
Done	{*exit field calculator window*}
Select the surface (or list) **HFSS>Fields>Plot Fields>** **Named Expression** Select E_normal	{Scalar Plot on faces displayed} *(E-field normal component value in **v/m**)*

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Generating an Iso-Surface Contour for a Given Field Value

Description

This recipe generates a geometry entry called an IsoSurface which represents the surface upon which a selected scalar field quantity has a single value. This surface can be displayed, or used in later operations (to plot other quantities upon, etc.).

Usage Examples

This calculation could be used to locate regions of excessive field magnitudes for voltage breakdown or ohmic heating analysis. It could also be used to generate a desired isosurface to be used as an integration surface for another quantity.

Prerequisites

You should plot of the field quantity of interest to determine the isovalue to use. Isovalues should be entered in MKS units (e.g., V/m, A/m) unless the problem is an eigen solution, in which case all field values are normalized to a peak of 1.0.

Calculator Operation	**Resulting Stack Display (top entry only unless noted)**
Quantity>E	CVc : <Ex, Ey, Ez> (*IsoSurfaces for other quantities can also be created; E used as example*.)
Smooth	CVc : Smooth(<Ex,Ey,Ez>) (*as this routine generates a surface geometry object, data smoothing is recommended*)
Function>Phase	Scl : Phase
Complex>AtPhase	Vec : AtPhase(Smooth(<Ex,Ey,Ez>),Phase)
Scal>ScalarX	Scl : ScalarX(AtPhase(Smooth(<Ex,...
Add... Type: E_x OK	E_x: ScalarX(AtPhase(Smooth<Ex,… (E_x *is used here as an example; you can apply other quantity*)
Done	{*exit field calculator window*}
Highlight the geometry	{*Scalar Plot in the geometry displayed*}
HFSS>Fields>Plot Fields>Named Expression Select E_x	*(E-field x-component value in **v/m**)*
HFSS>Fields>Modify Plot Attribute… Select the plot	{*IsoSurface contour is displayed*}

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
OK **Scale** tab: Num. Division: *1* Use Limits: *100 (as an example)* **Plots** tab IsoValSurface checked Apply	

Generating an Animation on Multiple Planes with a Positional Variable

Description

This recipe generates animated field output in which each frame is a snapshot of the fields on a different plane of the modeled volume. Any derived field quantity could be plotted in this manner, but this example will simply use the E-field magnitude at zero degrees input excitation.

Usage Examples

This calculation permits you to generate animated output results in addition to those automatically available from the post-processor. For example, peak E field (E dot E conjugate) could be plotted at multiple planes in sequence.

Prerequisites

This operation will only work in the global coordinate system if you are using X, Y, or Z positions as the animation variable.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Quantity>E	CVc : <Ex, Ey, Ez> (*Animations for other quantities can also be created; E used as example*.)
Smooth	CVc : Smooth(<Ex,Ey,Ez>)
Number>Scalar>”0”	Scl : 0
Complex>AtPhase	Vec : AtPhase(Smooth(<Ex,Ey,Ez>,.
Mag	Scl : Mag(AtPhase(Smooth(<Ex,...
Add Type: E_mag0 OK	E_mag0 : Dot(AtPhase(<Ex,Ey,Ez>,0),… (*User defined Named Expression E_mag0 is added*)
Done	{*exit field calculator window*}

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Planes>Global:YZ *(user can choose other planes Under modeler tree/Planes)*	(*YZ plane that can vary with X position)*
HFSS>Fields>Plot Fields>Named Expression (E_mag0)	*(Plot named expression on YZ plane)*
HFSS>Fields>Plot Animate>New	{*launches Animation Plot Settings*}
Swept variable Normalized Distance OK	{*displays animation*}

Generating Cartesian/Cylindrical/Spherical-Coordinate Field-Components Library

Description

This recipe demonstrates the steps to export user-defined named expressions into a library which can be loaded into and reused in other designs or projects.

Usage Examples

This calculation allows you to generate named expressions in addition to those automatically available ones in the field calculator, and save them as a user library, and reload into other designs/projects for use. For example, Cartesian components of E field are used for demonstration.

Prerequisites

Since this recipe is intended to generate generalized user-defined named-expressions, this operation should not be geometry-related.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Named Expressions>Vector_E	Vec : Vector_E (*E used as example*)
Scal?>ScalarX	Scl : ScalarX(Vector_E)
Add... Type inEx OK	

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculator Operation	Resulting Stack Display (top entry only unless noted)
Named Expressions>Vector_E	Vec : Vector_E
Scal?>ScalarY	Scl : ScalarY(Vector_E)
Add...	
Type in Ey	
OK	
Named Expressions>Vector_E	Vec : Vector_E
Scal?> ScalarZ	Scl : ScalarZ(Vector_E)
Add...	
Type in Ez	
OK	
Save To…	(*Exyz used as an library name example*)
Select (**Ex, Ey, Ez**)	
OK	
Type in **Exyz** (Library Name)	
Save	
Load From…	*(Named Expressions of Ex, Ey and Ez are loaded into Named Expressions)*
(Find the predefined library)	
Open	
Select (**Ex, Ey, Ez**)	
OK	

Radiated Fields Observation Regions

To analyze the radiated fields associated with a design, define a radiation surface over which the fields will be calculated. Such a surface can be a boundary radiation surface, a geometry, or a custom radiation surface which you define as a face list. The values of the fields over this surface are used to compute the fields in the space surrounding the device. This space is typically split into two regions — the near-field region and the far-field region. The near-field region exists at less than a wave length from an energy source. The far field is where radiation occurs. See Radiated Fields for the specific equations used in HFSS for calculating the near and far field regions.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can define a **spherical surface** over which to analyze the **near** or **far** fields by specifying a range and step size for phi and theta, or as Az over El or El over Az. This defines the spherical direction in which radiated fields will be evaluated. If you manually enter Phi or Theta component of a far field quantity when you using Az/El or El/Az far field definition, then the Report uses the Z axis of the Az/El or El/Az coordinate system definition as pole to calculate theta/phi component for rE. You can define a **box** over which to analyze **near fields** by specifying a coordinate system, length, width, height, and samples. You can define a **rectangle** over which to analyze **near fields** by specifying sample number for length and width. You can also draw a **line** along which to calculate the **near fields**.

You can also specify arbitrary near field sample point locations in the Global (or a specified Local) XYZ coordinate system as an external **point list**.

You also may need to edit the Global Material Environment in consideration of the far fields calculation.

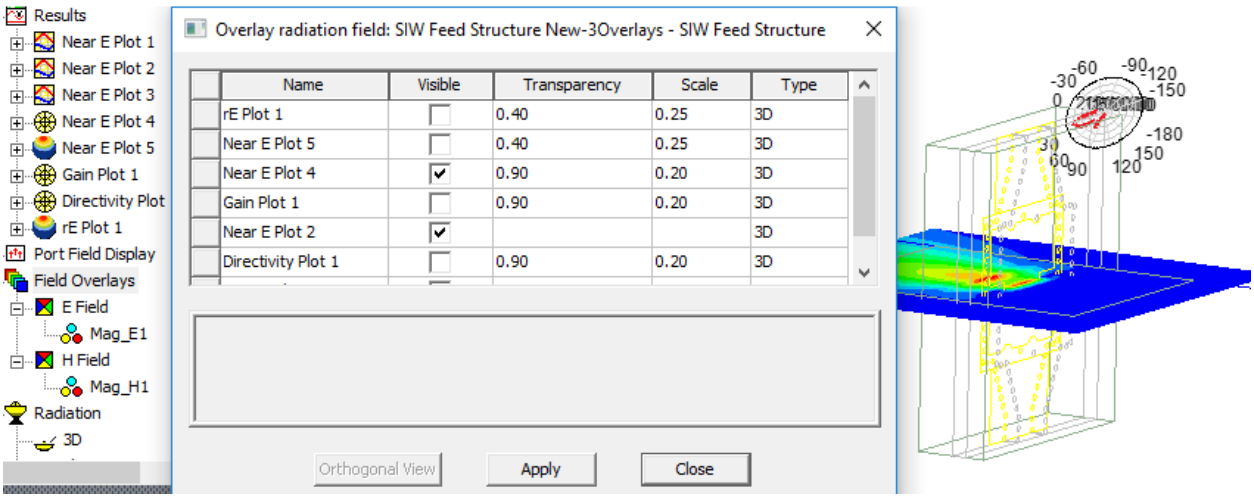
Optionally, after defining the radiation surface, HFSS can compute antenna array radiation patterns and antenna parameters for designs that have analyzed a single array element. HFSS models the array radiation pattern by applying an "array factor" to the single element's pattern when far fields are calculated. You set up the array factor information by defining either a finite, 2D array geometry of uniformly spaced, equal-amplitude elements (a regular array) or an arbitrary array of identical elements distributed in 3D space with individual complex weights (a custom array.)

If the design contains finite arrays, you have the option to create a Source group under the Radiation folder in the Project tree. The Source Group lets you activate a subset of the virtual excitations (e.g. array[5,5]port1:2 …) instead of the physical excitations associated with the unit cell(s).

HFSS can also compute antenna parameters, such as the maximum intensity, peak directivity, peak gain, and radiation efficiency. For near-field analysis, HFSS can also compute maximum parameters, such as the maximum of the total E-field and the maximum E-field in the x-direction. For FEM-only designs, you can optionally set the Radiated Power Calculation method.

Once your have created reports, you can overlay appropriate reports on the modeler window. You can overlay multiple Results plots for radiation fields at the same time and in conjunction with other Field Overlays,and antenna parameter overlays.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

When computing near and far fields, keep in mind that you must have defined at least one radiation or PML boundary in the design. You may change the radiation surfaces that HFSS uses when calculating the radiated fields at any time without needing to re-solve the problem, but the radiation-type boundary is still required.

Setting up a Near-Field Sphere

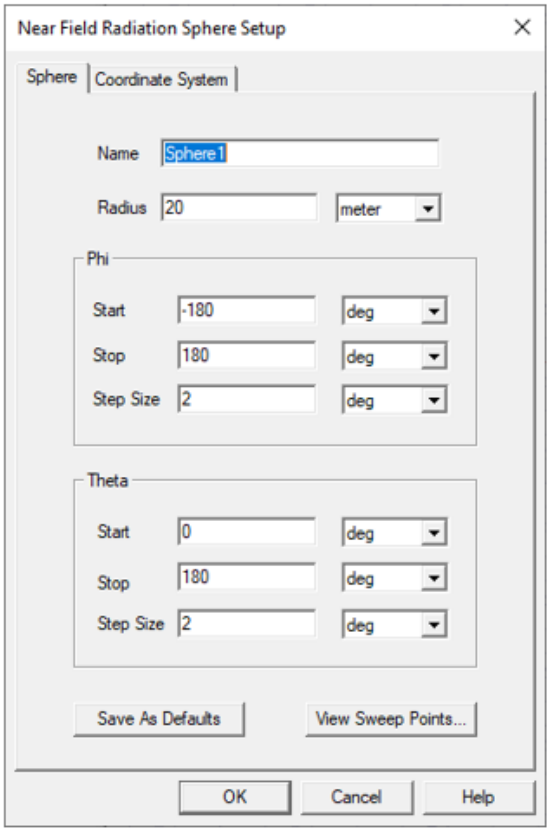
To evaluate near fields on a spherical surface, set up a near-field sphere. Near fields are evaluated at a fixed finite distance from the radiating device. You can use the near-field sphere setup to create near-field reports.

1. To create a near-field sphere setup, click **HFSS > Radiation>Insert Near Field Setup>Sphere** to bring up the **Near Field Radiation Sphere Setup** dialog box. If the design is Modal/Terminal Solution type the dialogue will include a **Radiation Surface** tab. If the design is HFSS SBR+ or includes SBR+ Hybrid Regions, the dialogue does not include the **Radiation Surface** tab because with SBR+ designs, the Radiation surface is defined or selected before simulation using the Field Observation Domain pane in the **SBR+ Solution** setup **Options** tab or the **Hybrid** tab of an HFSS Modal or Terminal setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modal/Terminal

SBR+



2. Use the **Sphere** tab define the **Name**, **Radius**, and sampling of **Phi** and **Theta** for the near-field sphere. The radius is measured from the origin of the sphere's coordinate system, which you specify under the **Coordinate System** tab. You can assign a variable to the radius, and a post-processing variable will often make sense in this context.

   Specify the sphere's sampling in terms of **Start**, **Stop**, and **Step Size** angles given in radians or degrees. To verify your settings, use the **View Sweep Points** button to display a list of the theta and phi sweep points. See Spherical Cross-Sections in the **Technical Notes** for guidelines for setting phi and theta.

   You can use **Save as Defaults** to set the current values as the default for new near-field sphere setups.

3. Use the **Coordinate Systems** tab to specify the orientation of the sphere.

   Use global coordinate system is selected by default, but in some cases the orientation of the antenna requires the use of a local coordinate system. In this case, select **Use local coordinate system**, and choose a local coordinate system that you created previously in the modeler.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. For HFSS Terminal or Modal without Hybrid regions, use the **Radiation Surface** tab to select the solved surface from which to calculate radiated fields.

   Use Boundary Radiation Surfaces is selected by default, indicating that the radiated fields will be calculated using the assigned radiation or PML surface. For some models you may find it it more efficient and/or accurate to use an interior surface. In this case, select **Use Custom Radiation Surface**, and choose a face list that you previously created in the modeler.

> **Note:**
>
> You must have defined at least one radiation or PML boundary in the design for HFSS to compute near-field quantities, regardless of which radiation surfaces you instruct HFSS to use when calculating the near fields. You do not need to re-solve the problem if you modify radiation surfaces in the **Near Field Radiation Sphere Setup** dialog.

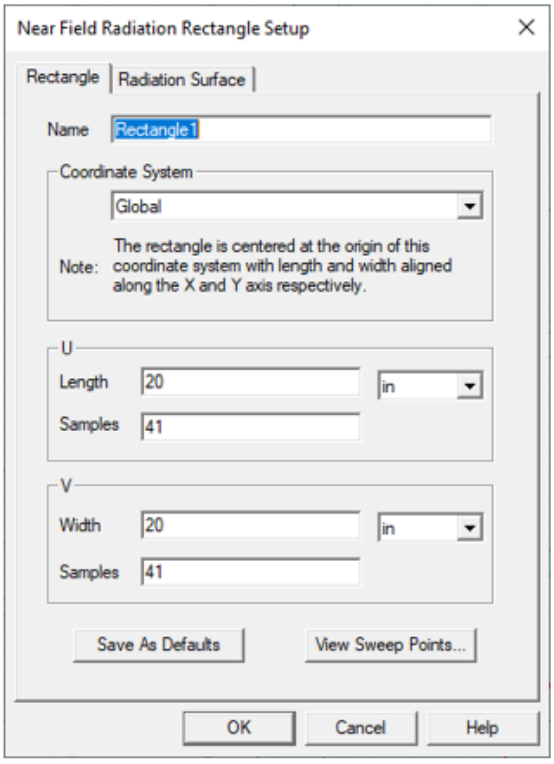
Setting up a Near-Field Rectangle

To evaluate near fields on a rectangle, set up a near-field rectangle. Make sure contour plot on sweep of _u and _v can be overlaid in model geometry. Near fields are evaluated on the rectangle centered at the origin of the defined coordinate system of the rectangle with length and width aligned along the X and Y axis of that coordinate system. The plot should have the same size of the rectangle in model units. You can use the near-field rectangle setup to create near-field reports. A near field plot setup in this way can also be overlaid on the geometry and animated.

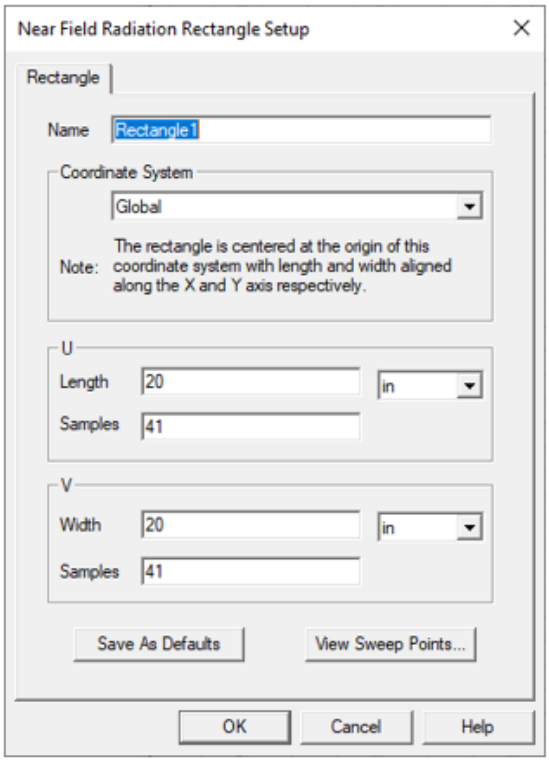
1. To create a near-field rectangle setup, click **HFSS** > **Radiation** > **Insert Near Field Setup** > **Rectangle** to bring up the **Near Field Radiation Rectangle Setup** dialog. If the design is Modal/Terminal Solution type the dialogue will include a **Radiation Surface** tab. If the design is HFSS SBR+ or includes SBR+ Hybrid Regions, the dialogue does not include the **Radiation Surface** tab because with SBR+ designs, the Radiation surface is defined or selected before simulation using the Field Observation Domain pane in the **SBR+ Solution** setup **Options** tab or the **Hybrid** tab of an HFSS Modal or Terminal setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modal/Terminal



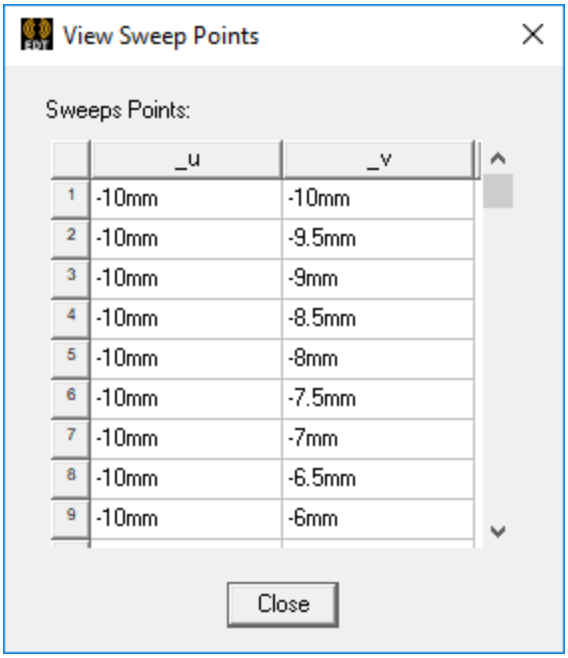
SBR+



2. Use the **Rectangle** tab define the **Name**, Relative Coordinate System, and Length, units, number of samples for U and V for the near-field rectangle. The rectangle is centered at the origin of the coordinate system you specify. You can assign a variable to the length and width, and a post-processing variable will often make sense in this context.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specify the sphere's sampling as any integer greater than 1. To verify your settings, use the **View Sweep Points** button to display a list of the U and V sweep points.



You can use **Save as Defaults** to set the current values as the default for new near-field rectangle setups.

3. Use the **Coordinate Systems** field and drop down menu to specify the location and orientation of the rectangle. Refer to the Note in the **Setup** dialog.

   The drop down menu consists of the Global CS and all local coordinate systems that you created previously in the modeler

4. For a Terminal or Modal design, use the **Radiation Surface** tab to select the solved surface from which to calculate radiated fields.

   Use Boundary Radiation Surfaces is selected by default, indicating that the radiated fields will be calculated using the assigned radiation or PML surface. For some models you may find it it more efficient and/or accurate to use an interior surface. In this case, select **Use Custom Radiation Surface**, and choose a face list that you previously created in the modeler.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

You must have defined at least one radiation or PML boundary in the design for HFSS to compute near-field quantities, regardless of which radiation surfaces you instruct HFSS to use when calculating the near fields. You do not need to re-solve the problem if you modify radiation surfaces in the **Near Field Radiation Sphere Setup** dialog.

Setting up a Near-Field Box

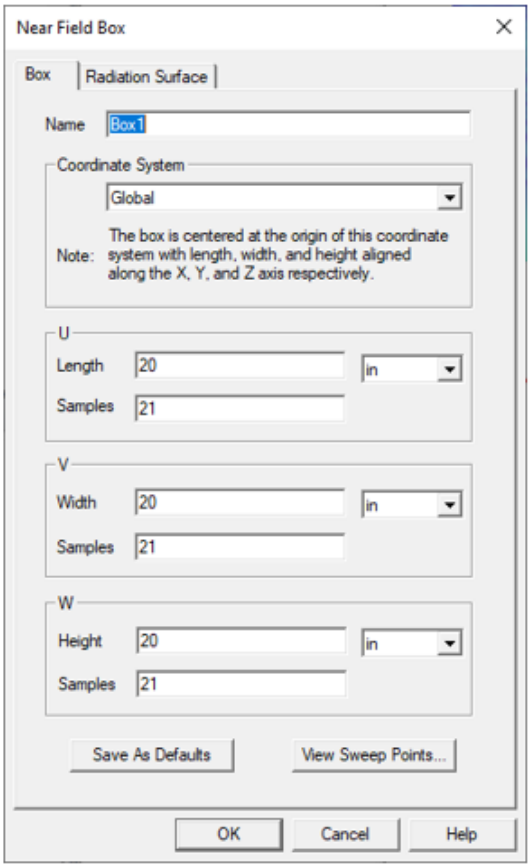
To evaluate the near field in a box, set up a near-field box. The near-field box is defined length, height, and width parameters, and samples parameters for each dimension. To plot near-field values in the box, you will select the box object from the **Geometry** list in the **Traces** dialog box when you create a report.

1. Draw a polyline in post-processing mode.
2. Click **HFSS >Radiation**>**Insert Near Field Setup>Box**.

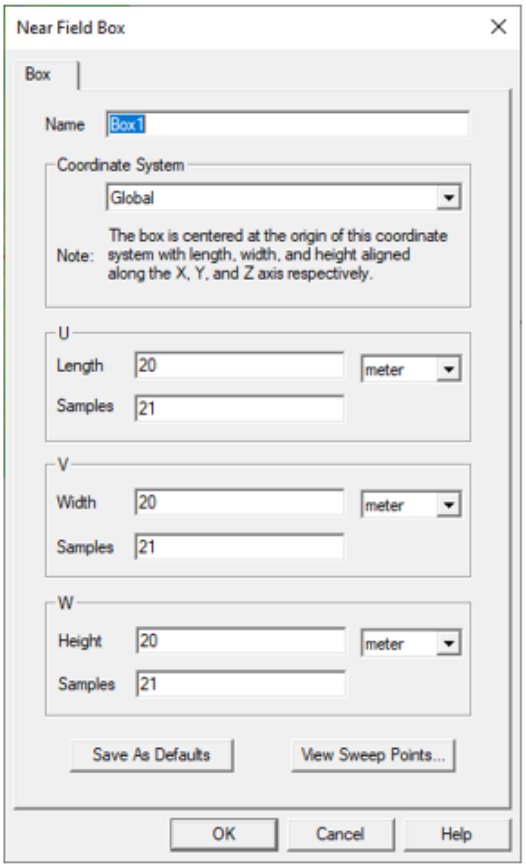
The **Near Field Line Setup** dialog box appears. If the design is Modal/Terminal Solution type the dialogue will include a **Radiation Surface** tab. If the design is HFSS SBR+ or includes SBR+ Hybrid Regions, the dialogue does not include the **Radiation Surface** tab because with SBR+ designs, the Radiation surface is defined or selected before simulation using the Field Observation Domain pane in the **SBR+ Solution** setup **Options** tab or the **Hybrid** tab of an HFSS Modal or Terminal setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modal/Terminal



SBR+



3. Type a name for the box in the **Name** text box.
4. Select a Relative Coordinate system to use for the box.
5. Specify the Length for the U, V, and W, dimension, the units, and the number of samples.

This defines the box and gives the total number of equally spaced sample points in the box.

6. For a Terminal or Modal design, use the **Radiation Surface** tab to select the solved surface from which to calculate radiated fields.

   Use Boundary Radiation Surfaces is selected by default, indicating that the radiated fields will be calculated using the assigned radiation or PML surface. For some models you may find it it more efficient and/or accurate to use an interior surface. In this case, select **Use Custom Radiation Surface**, and choose a face list that you previously created in the modeler.

7. Click **OK**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You must have defined at least one radiation or PML boundary in the design for HFSS to compute near-field quantities, regardless of which radiation surfaces you instruct HFSS to use when calculating the near fields. You do not need to re-solve the problem if you modify radiation surfaces in the **Near Field Box Setup** window.

> **Note:**
>
> For parts of the near-field box lying outside of the model region, near-field approximation is calculated. However, if parts of the line lie inside the model region, the model fields are used to compute interpolated values. A section of the near-field line is considered to overlap the model if it lies in the enlarged model region after accounting for symmetry planes.

Setting up a Near-Field Line

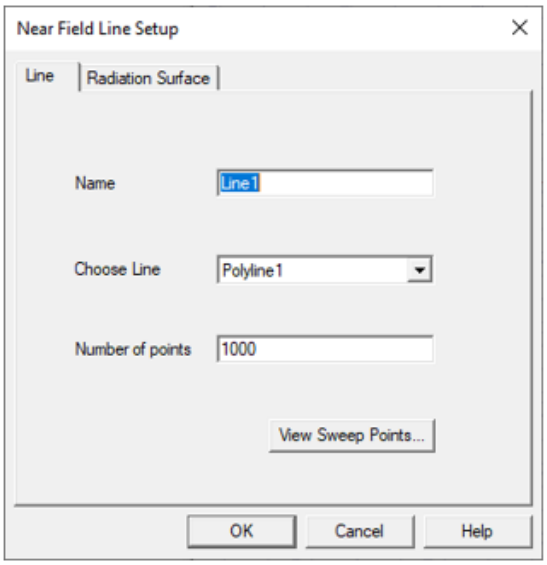
To evaluate the near field along a line, set up a near-field line. The near-field line can be a polyline with one or more segments. To plot near-field values along the line, you will select the line object from the **Geometry** list in the **Traces** dialog box when you create a report.

1. Draw a polyline in post-processing mode.
2. Click **HFSS >Radiation**>**Insert Near Field Setup>Line**.

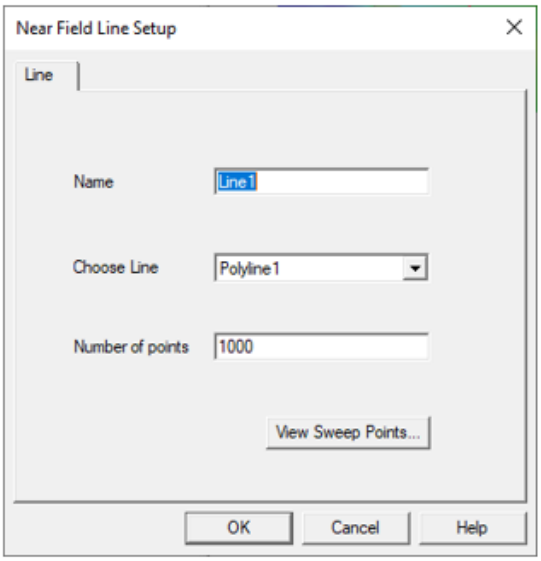
The **Near Field Line Setup** dialog box appears. If the design is Modal/Terminal Solution type the dialogue will include a **Radiation Surface** tab. If the design is HFSS SBR+ or includes SBR+ Hybrid Regions, the dialogue does not include the **Radiation Surface** tab because with SBR+ designs, the Radiation surface is defined or selected before simulation using the Field Observation Domain pane in the **SBR+ Solution** setup **Options** tab or the **Hybrid** tab of an HFSS Modal or Terminal setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modal/Terminal



SBR+



3. Under the **Near Field Line Setup** tab, type a name for the line in the **Name** text box.
4. Select the polyline along which you want to evaluate the near fields from the **Choose Line** list.
5. Specify the **Number of points** in the line.

This is the total number of equally spaced points on the line. Specifying points on the line will enable you to plot the near-field values across a normalized distance, that is, to create a value versus distance plot of a near-field quantity on the line.

You can click **View Sweep Points** to view a dialog that lists the points.

6. For a Modal or Terminal design, click the **Radiation Surface** tab.

By default, the **Use Boundary Radiation Surfaces** radio button is selected.

To specify a surface other than an assigned radiation or PML boundary over which to integrate the radiated fields, you must first create a face list. To create a facelist see Creating a Face List. The face list cannot include a face that lies on a PML object.

If you have created one or more face lists, the **Use Custom Radiation Surface** radio button is enabled.

a. Select **Use Custom Radiation Surface**.

This enables the **Choose from existing face list** field.

b. Select a defined face list from drop down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HFSS will use the surfaces in the face list as the radiating surfaces when calculating the near fields.

7. Click **OK**.

You must have defined at least one radiation or PML boundary in the design for HFSS to compute near-field quantities, regardless of which radiation surfaces you instruct HFSS to use when calculating the near fields. You do not need to re-solve the problem if you modify radiation surfaces in the **Near Field Line Setup** window.

> **Note:**
>
> For parts of the near-field line lying outside of the model region, near-field approximation is calculated. However, if parts of the line lie inside the model region, the model fields are used to compute interpolated values. A section of the near-field line is considered to overlap the model if it lies in the enlarged model region after accounting for symmetry planes.

Setting up an Arbitrary Near Fields Point List

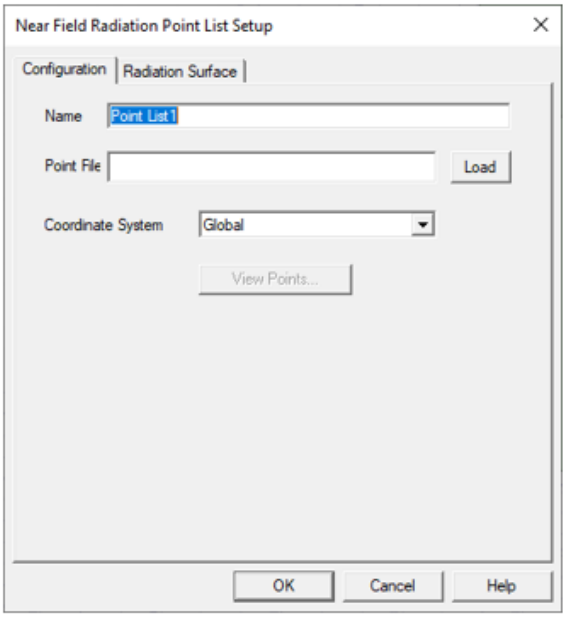
To evaluate the near fields using an arbitrary points, set up an external points list using a global or local coordinate system. The *.pts file format (which is used for exporting field data and specifying field point locations in the Fields Calculator – Export dialog) is also used for specifying 3D near field point locations. Be sure to check “Compute Fields” on the **General** tab on the **Solution Setup**.

1. Prepare a *.pts file format file.
2. Click **HFSS >Radiation**>**Insert Near Field Setup>Points List**.

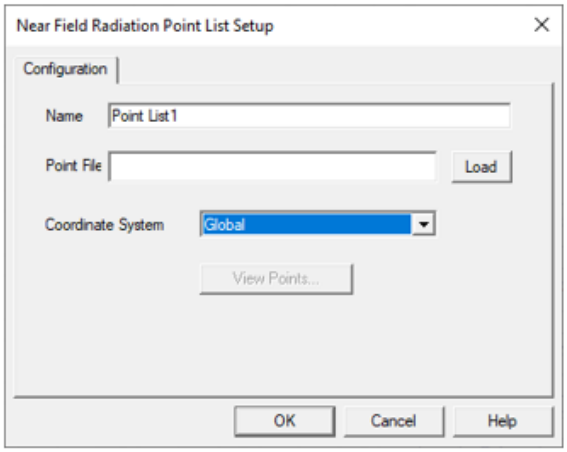
The **Near Field Radiation Points List Setup** dialog box appears. If the HFSS design is Modal/Terminal Solution type the dialogue includes a **Radiation Surface** tab. If the design is SBR+ the dialogue does not include the **Radiation Surface** tab

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HFSS with Radiation Boundary Defined



HFSS SBR+



With SBR+ designs, a Near Field Domain Setup is defined or selected *before simulation* using the Field Observation Domain pane in the **SBR+ Solution** setup **Options** tab or the **Hybrid** tab of an HFSS Modal or Terminal setup. (For a solution type other than hybrid or SBR+, you can create and configure a near field radiation setup using File-Based Point List after simulation).

3. Type a name for the point list in the **Name** text box.
4. Select the points file for which you want to evaluate the near fields. Pressing the **Load** button opens a browser.
6. For a Modal or Terminal design, click the **Radiation Surface** tab.

By default, the **Use Boundary Radiation Surfaces** radio button is selected.

To specify a surface other than an assigned radiation or PML boundary over which to integrate the radiated fields, you must first create a face list. To create a facelist see Creating a Face List. The face list cannot include a face that lies on a PML object.

If you have created one or more face lists, the **Use Custom Radiation Surface** radio button is enabled.

a. Select **Use Custom Radiation Surface**.

This enables the **Choose from existing face list** field.

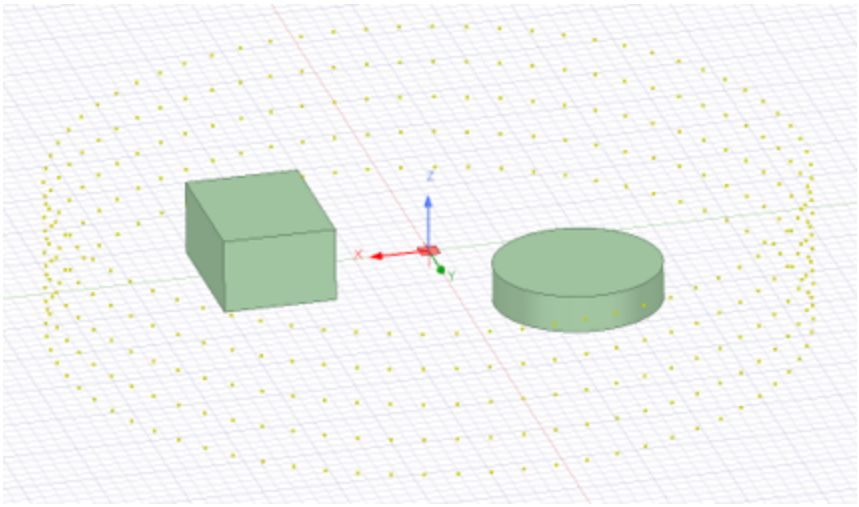
b. Select a defined face list from drop down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

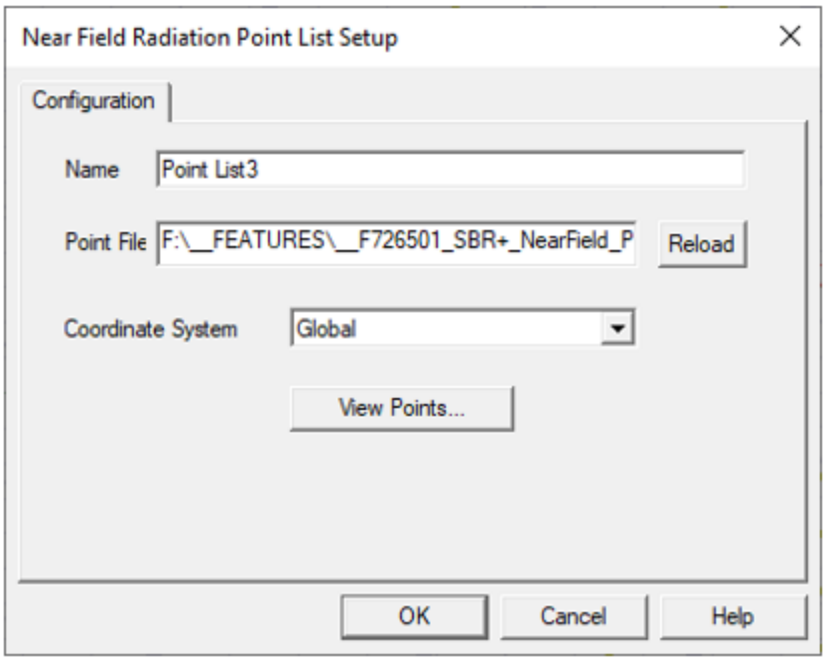
HFSS will use the surfaces in the face list as the radiating surfaces when calculating the near fields.

7. Click **OK**.

HFSS renders the near field points in the 3D model window.

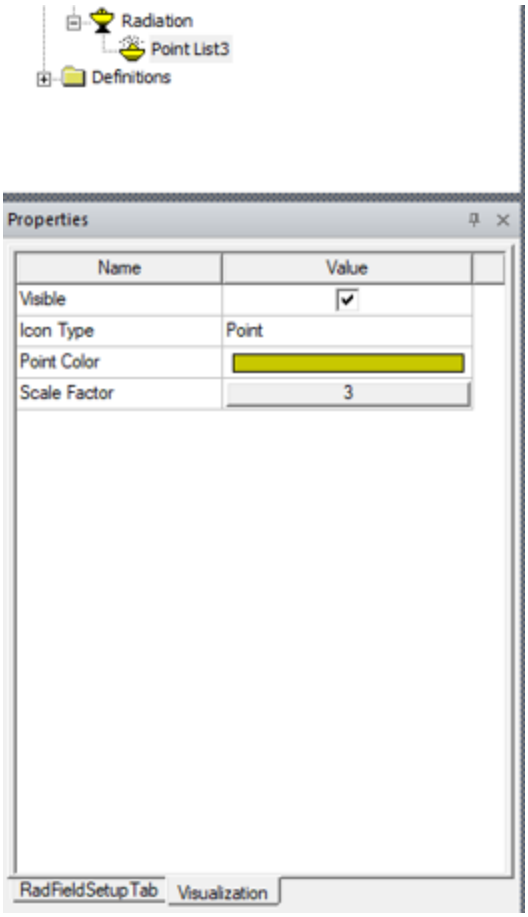


The **View Points** button is enabled and provides a data table view of the XYZ point coordinates loaded from the specified *.pts file.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you select the Radiation Near Field setup in the **Project** tree, the **Properties** includes a **Visualization** tab.



- Visible [On/Off]
- Icon Type [Point, Sphere, Box, Dot, Plus]
- Point Color (colored button, brings up color chooser)
- Scale Factor (sets size of marker)

When a radiation surface is used, you must have defined at least one radiation or PML boundary in the design for HFSS to compute near-field quantities, regardless of which radiation surfaces you instruct HFSS to use when calculating the near fields.

Generating a Report from Point List

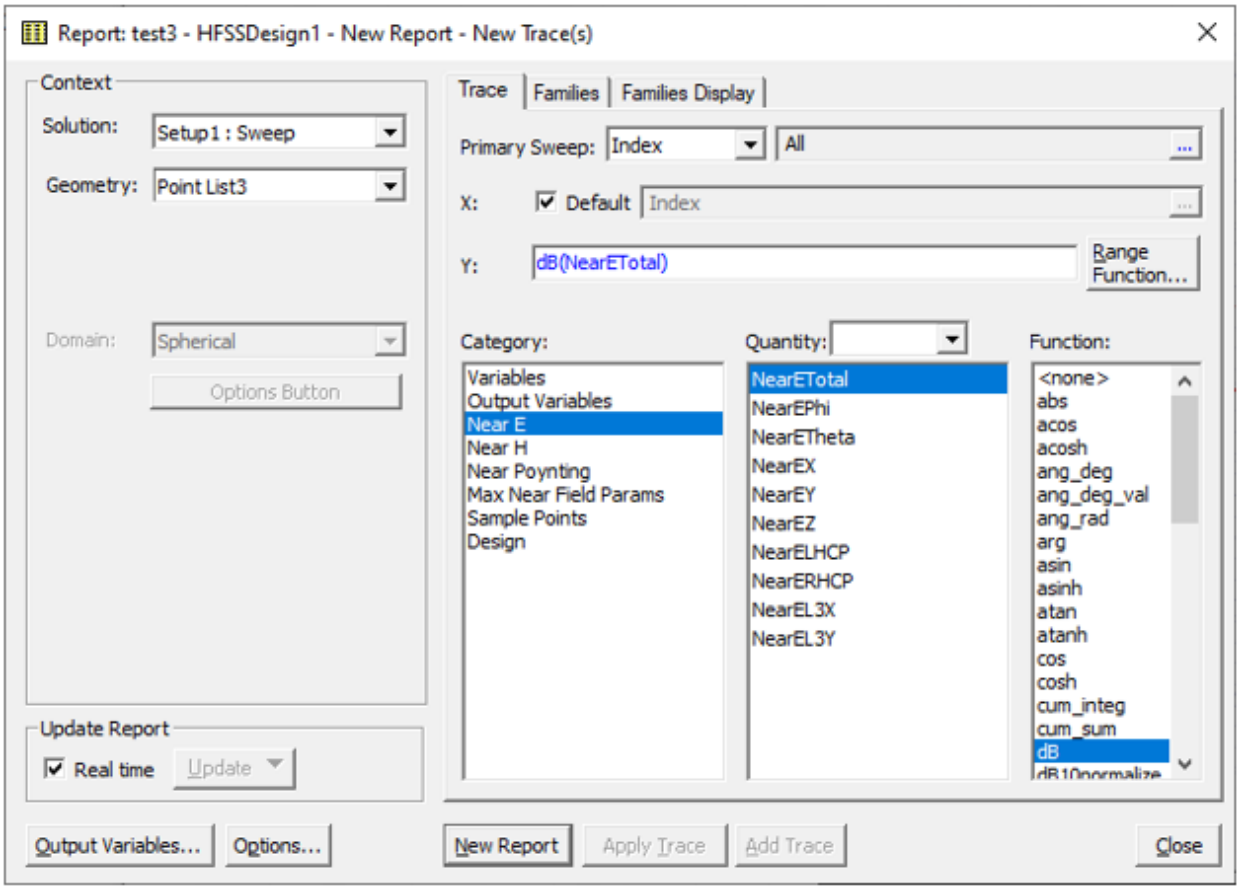
To Generate a Report from a Point List simulation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Under Results node in the **Project** tree select **Near Fields Report > Data Table**.

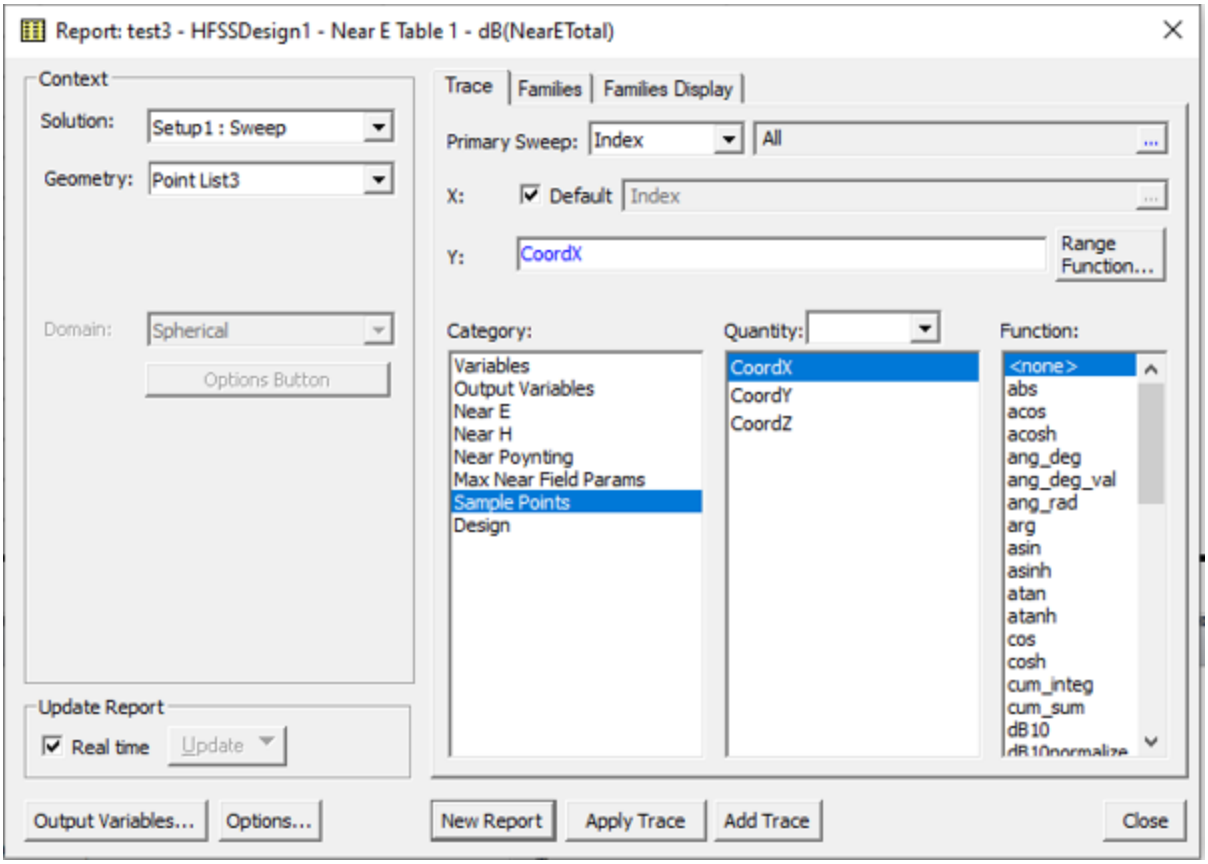
   This opens the **Report** dialog.

2. Under the Context setup in the **Report** dialog, set the Geometry selection to the created near field Point List radiation setup; notice the only Primary Sweep available is the point list Index.



Select the desired Category, Quantity, and Function (such as Near E, NearETotal, dB). Notice that the Category: list includes Sample Points.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



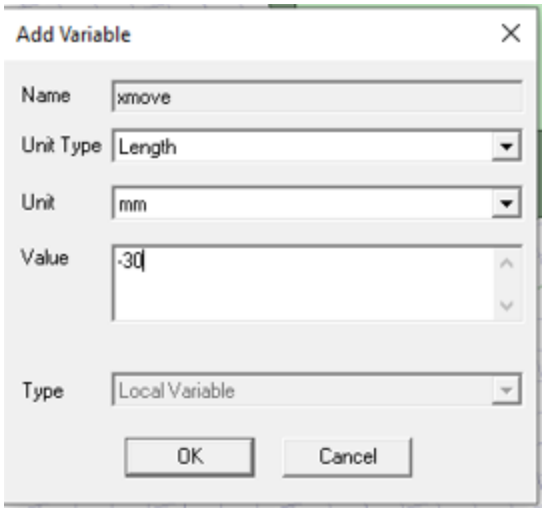
Selecting Sample Points as the Category then include the Quantities for CoordX, CoordY, and CoordZ.

Animating an Near Field Points List Plot

To animate a near field points list plot:

1. Create a design or project variable for the position of a geometry object in the 3D model. You can do this This can be done by performing an **Edit > Arrange > Move** operation on the object, or by creating a new Coordinate System containing the design/project variable and assigning it as the Target Coordinate System for a geometry object.

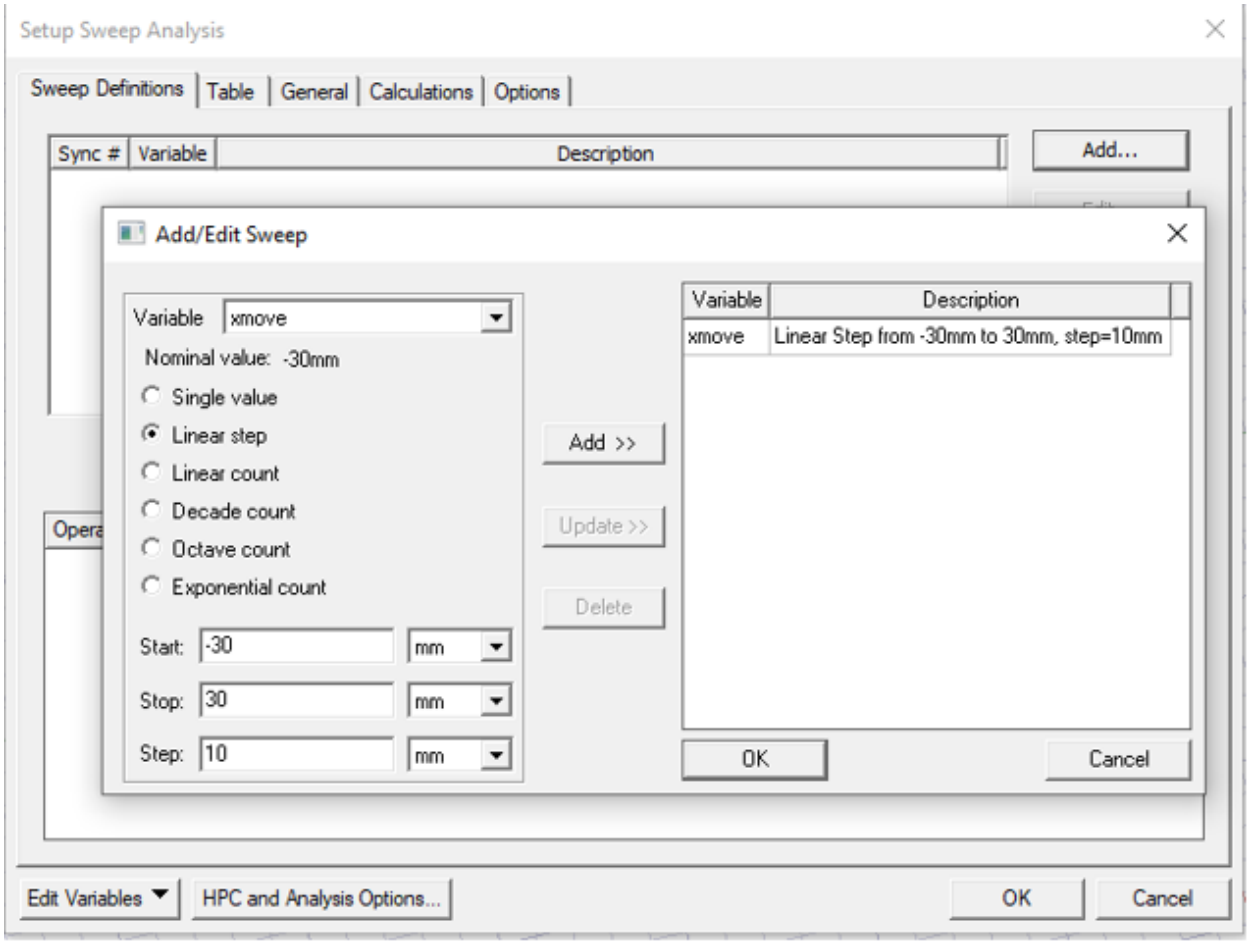
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



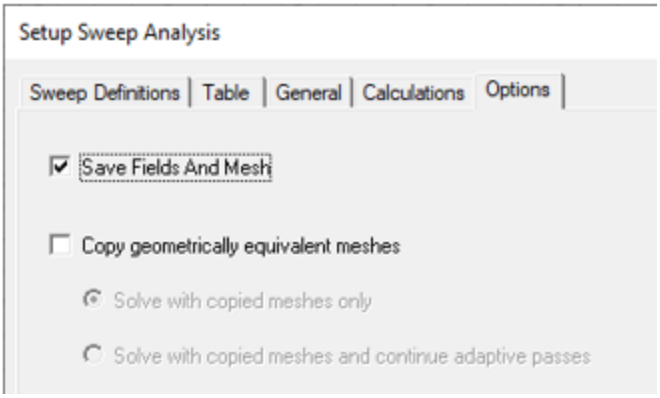
2. Then add an Optimetrics Parametric sweep setup.

   Use the design/project variable that you created for the parametric sweep definition.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



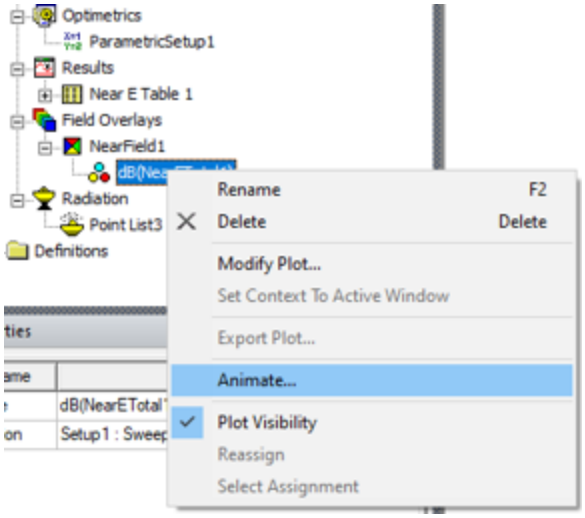
3. Check “Save Fields and Mesh” on the Options page of the sweep setup.



4. Run the Optimetrics simulation.

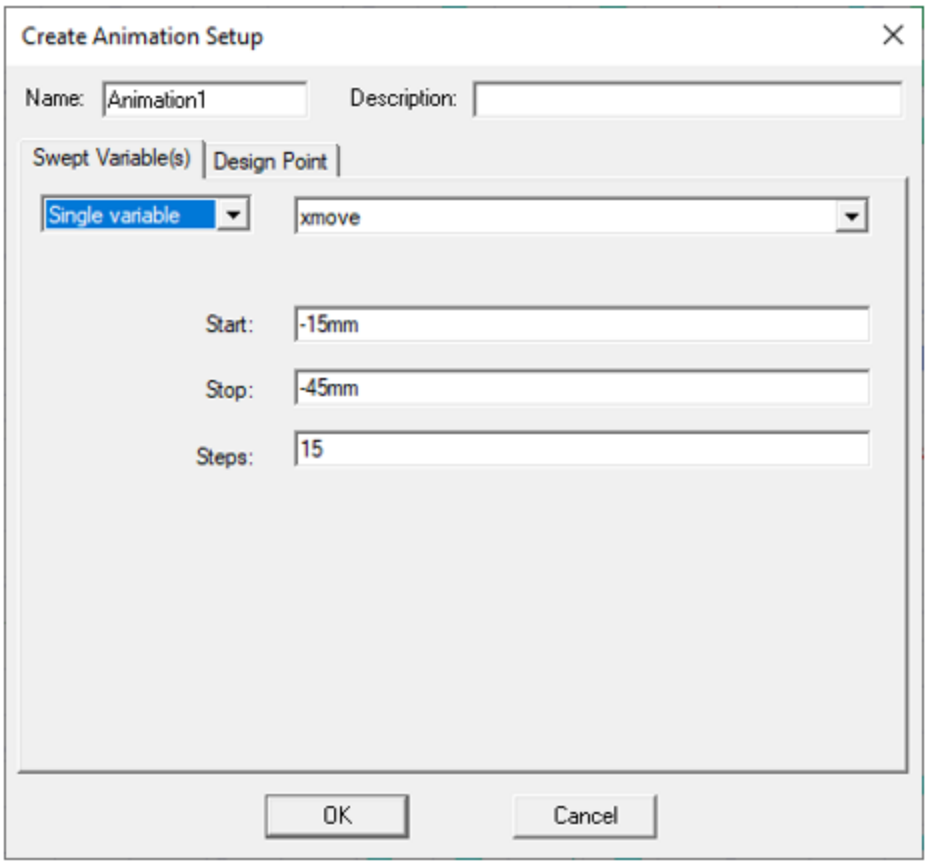
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. After creating a Sampled Near Field Plot as described previously, right-click on the plot object and select **Animate...** to open the **Create Animation Setup**.



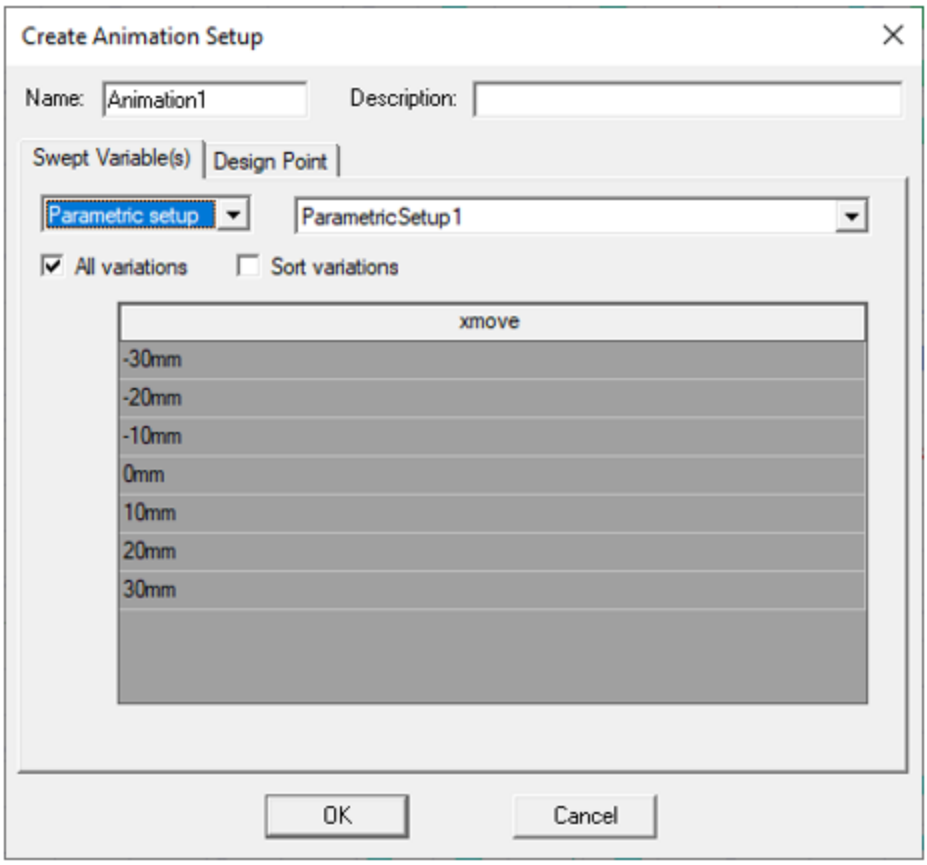
You can select the design/project variable from the list of available sweep variables and create a list of variable values to sweep for the animation. These values should be from the available simulation sweep values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

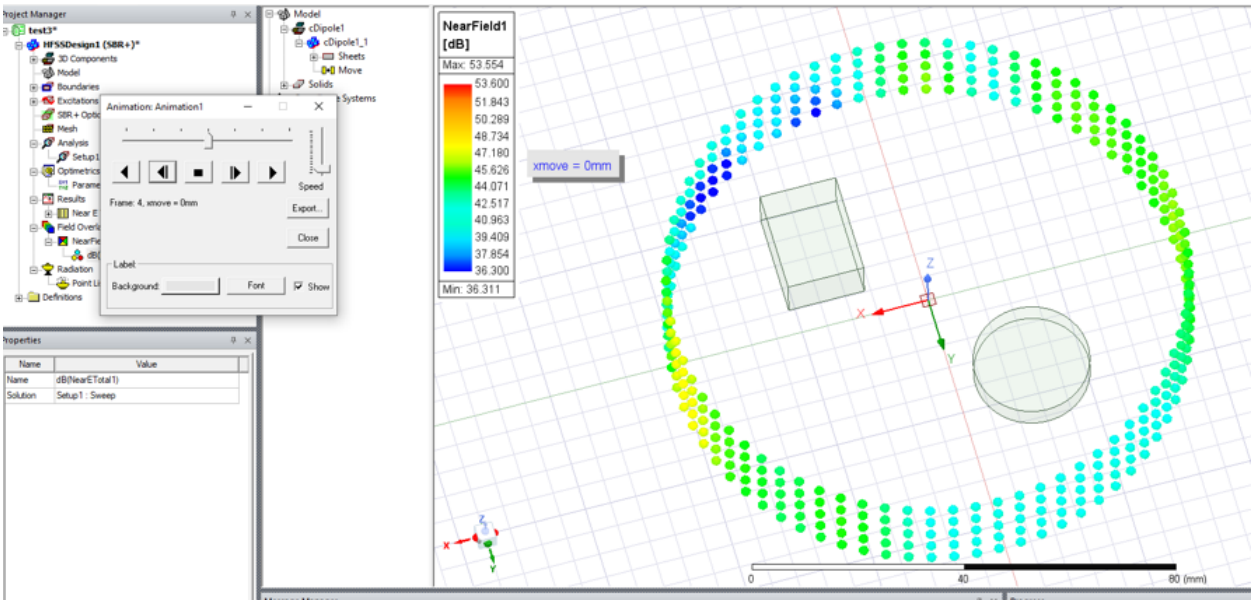


Or you can select the Parametric Setup to use the variable and all values from the specified Optimetric setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. Select the created animation sweep setup and click **OK** to generate the animation.

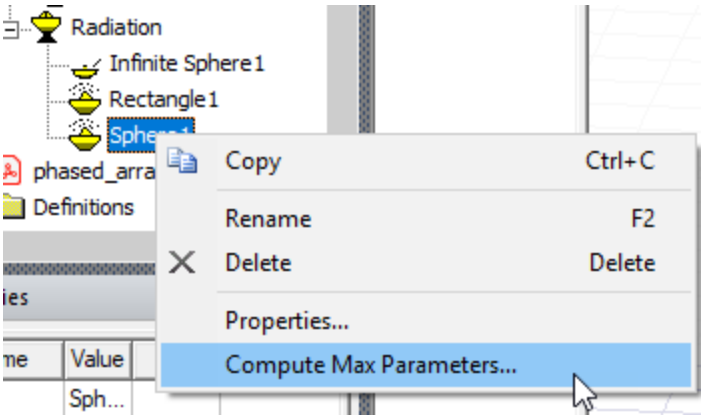


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Computing Maximum Near-Field Parameters

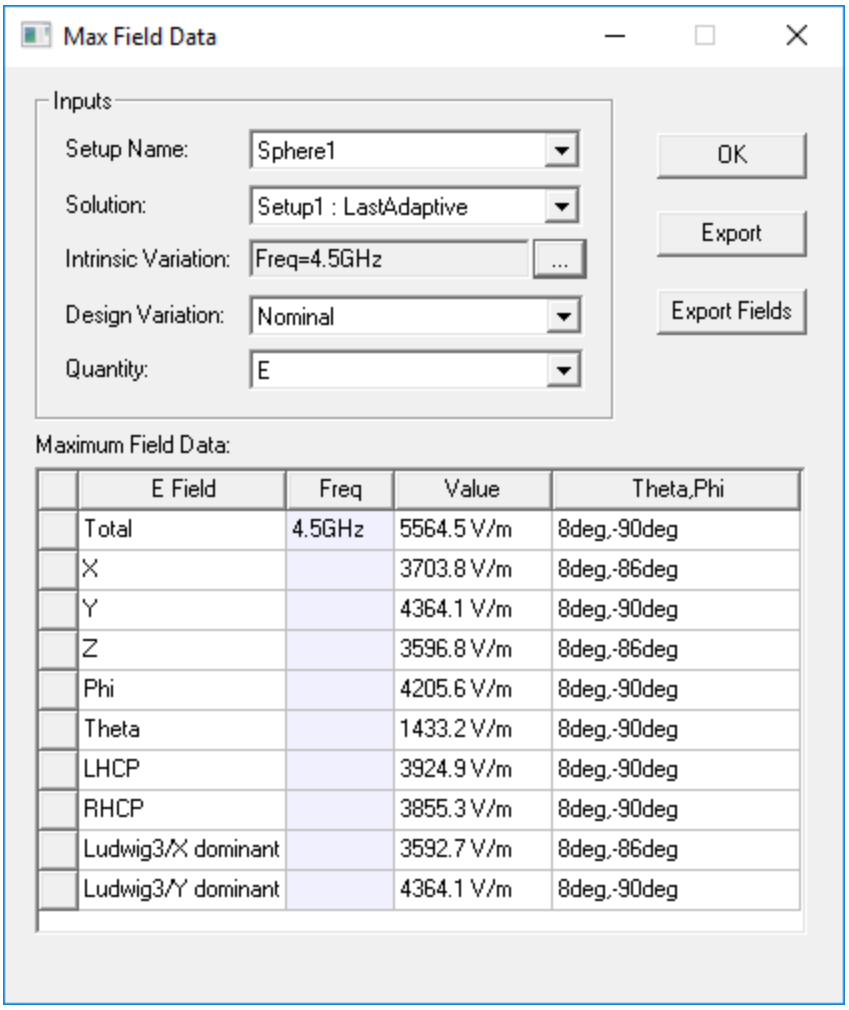
You must have defined at least one radiation or PML boundary in the design for HFSS to compute maximum field data for the near-field region, and you must have defined a near field sphere, rectangle, or sphere.

1. Right-click the **Sphere**, **Rectangle**, or **Line** icon in the project tree, and then click **Compute Max Parameters** on the shortcut menu.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. The Max Field Data window appears.



E Field	Freq	Value	Theta,Phi
Total	4.5GHz	5564.5 V/m	8deg,-90deg
X		3703.8 V/m	8deg,-86deg
Y		4364.1 V/m	8deg,-90deg
Z		3596.8 V/m	8deg,-86deg
Phi		4205.6 V/m	8deg,-90deg
Theta		1433.2 V/m	8deg,-90deg
LHCP		3924.9 V/m	8deg,-90deg
RHCP		3855.3 V/m	8deg,-90deg
Ludwig3/X dominant		3592.7 V/m	8deg,-86deg
Ludwig3/Y dominant		4364.1 V/m	8deg,-90deg

3. For **Setup Name**, you can select from available Near Field setups.
4. For Solution, you can select from available solutions.
5. For Intrinsic Variation and Design Variation, you can select from any available variations.
6. For Quantity, you can select From E, H, or Poynting.

The **Max Field Data** window lists the following information for E for a sphere:

- **Total**
- **X**
- **Y**
- **Z**
- **Phi**
- **Theta**
- **LHCP**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **RHCP**
- **Ludwig 3/X dominant**
- **Ludwig 3/Y dominant**

> **Note:**
>
> When calculating the maximum far-field values, the distance *r* is factored out of the E-field. Therefore, the units for the maximum field data values are given in volts.

The **Export** button lets you export to the parameters to an exportparams.txt file.

The **Export Fields** button lets you export the fields to an exportfields.csv file.

Note: A Beta feature for HFSS, Hfss Interactive Dialogs, allows you run the **Compute Max Parameters**dialog in modeless fashion, that is, if you **Edit Sources** for magnitide or renormalization, affected Max Parameters update. If you edit excitations or ports in the Far Field Setup, or create design variations in Optimetrics, the **Edit Source** dialog will close.

Case that update in this mode:

- Infinite sphere setup created, deleted or renamed: Setup Name combo box changes radiation setup.
- Solution setup created, deleted or renamed.
- Antenna array setup created or deleted.
- Source group created, deleted or renamed.

Note: Intrinsic Variation updates as the solution setup changes. Grids of “Maximum Field Data” update data as any of the combo box selections changes. The dialog closes if solution type is changed. Selections in Quantity combo box will not change.

Related Topics

Radiated Fields Post Processing

Assigning PML Boundaries

Assigning Radiation Boundaries

Technical Notes: Maximum Near-Field Data

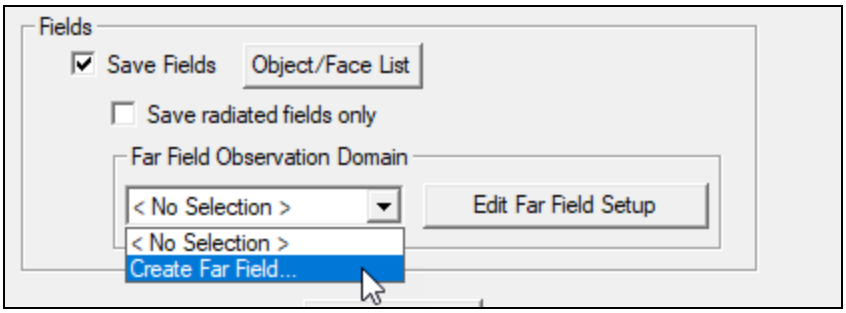
Setting up a Far-Field Infinite Sphere

To evaluate radiated fields in the far-field region, you must have Saved Fields in the solution and sweep setups, and set up an infinite sphere that surrounds the radiating object. To plot far-field

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

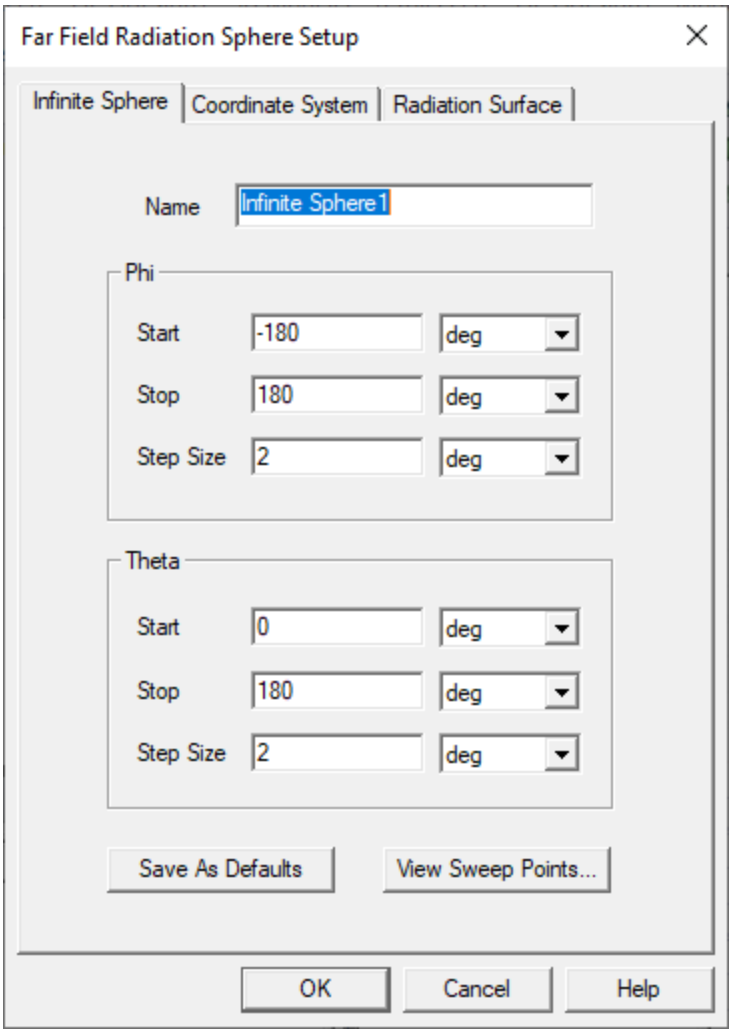
values across the sphere, you will select the sphere object from the **Geometry** list in the **Traces** dialog box when you create a report. To evaluate RCS using a design with an incident plane wave and an SBR+ region, you can also setup an infinite sphere.

1. **Click HFSS** > **Radiation**> **Insert Far Field Setup**>**Infinite Sphere** or, on an Advanced Solution Setup, **Advanced** options tab for an HFSS design without SBR+ Hybrid regions, or for use with **SBR+ Solver options,** Click the **Create...** button under on the **SBR+ Solver options** on **Hybrid** tab of the **Solution setup.**



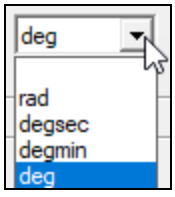
The **Far Field Radiation Sphere Setup** window appears. Depending on your selections on the **Coordinate System** tab, the basis will be displayed as Phi Theta, Azimuth Elevation, or Elevation Azimuth. The Default is Phi and Theta. The selection there will also affect display of Maximum Far Field Data if you Computer Antenna Parameters.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



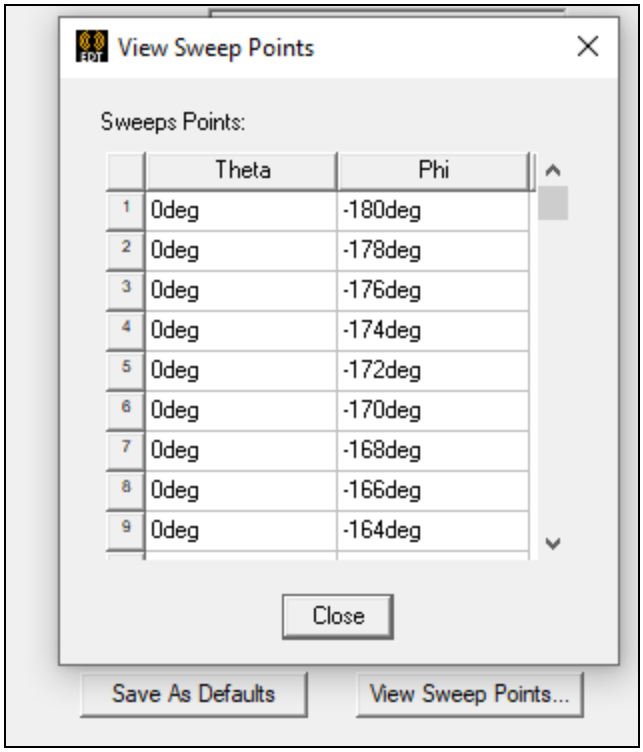
2. Use the **Sphere** tab define the **Name** and sampling in the selected for the far-field sphere.

Specify the sphere's sampling in terms of **Start**, **Stop**, and **Step Size** angles given in radians, degrees, degree seconds or degree minutes.



To verify your settings, use the **View Sweep Points** button to display a list of the theta and phi sweep points. See Spherical Cross-Sections in the **Technical Notes** for guidelines for setting phi and theta.

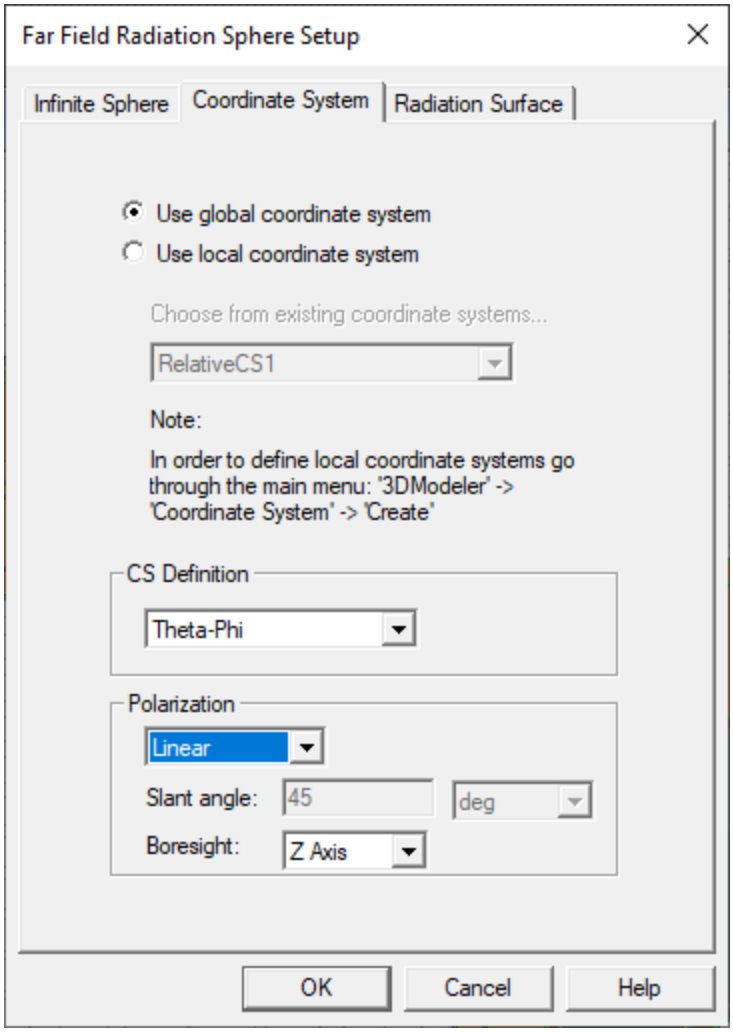
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



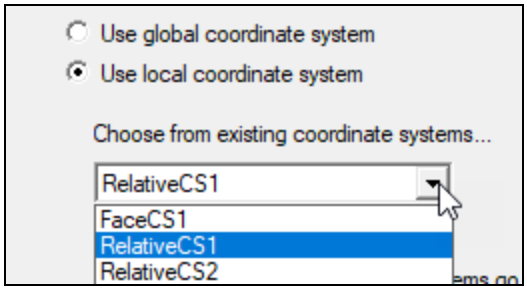
You can use **Save as Defaults** to set the current values as the default for new near-field sphere setups.

3. Use the **Coordinate Systems** tab to specify the orientation of the sphere as well as the CS System Definition, and the Polarization.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

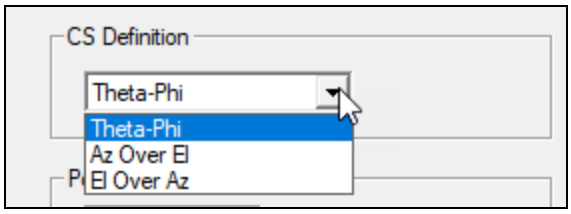


Use global coordinate system is selected by default, but in some cases the orientation of the antenna requires the use of a local coordinate system. In this case, select **Use local coordinate system**, and choose a local coordinate system that you created previously in the modeler.

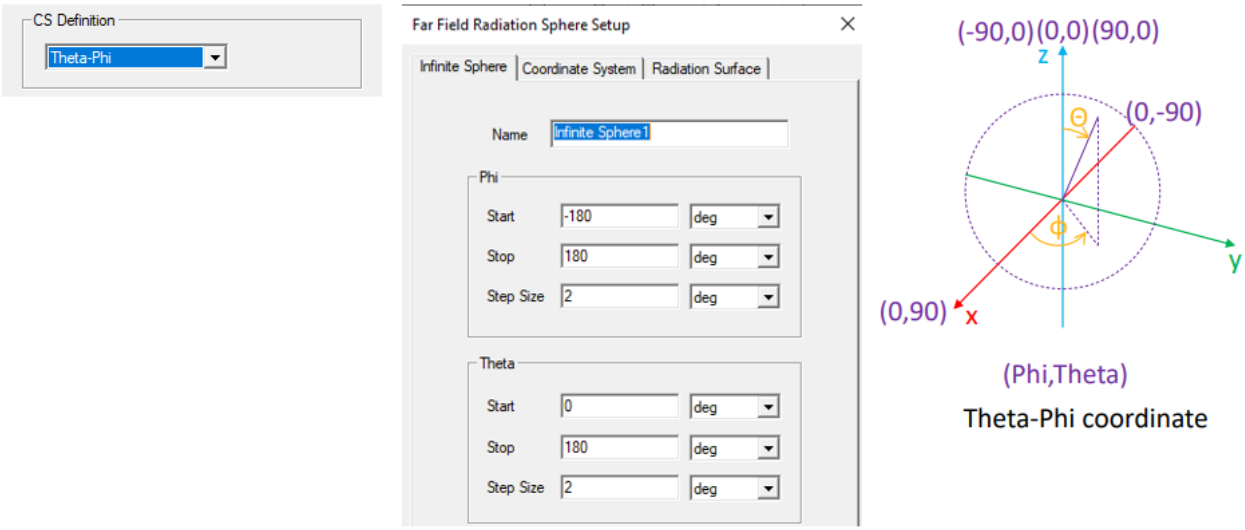


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CS Definition: In addition to support Theta-Phi spherical coordinate system, you can select Azimuth over Elevation definition and Elevation over Azimuth definition. Your selection here affects the display on the **Infinite Sphere** tab. The selection there will also affect display of Maximum Far Field Data if you calculate Computer Antenna Parameters.

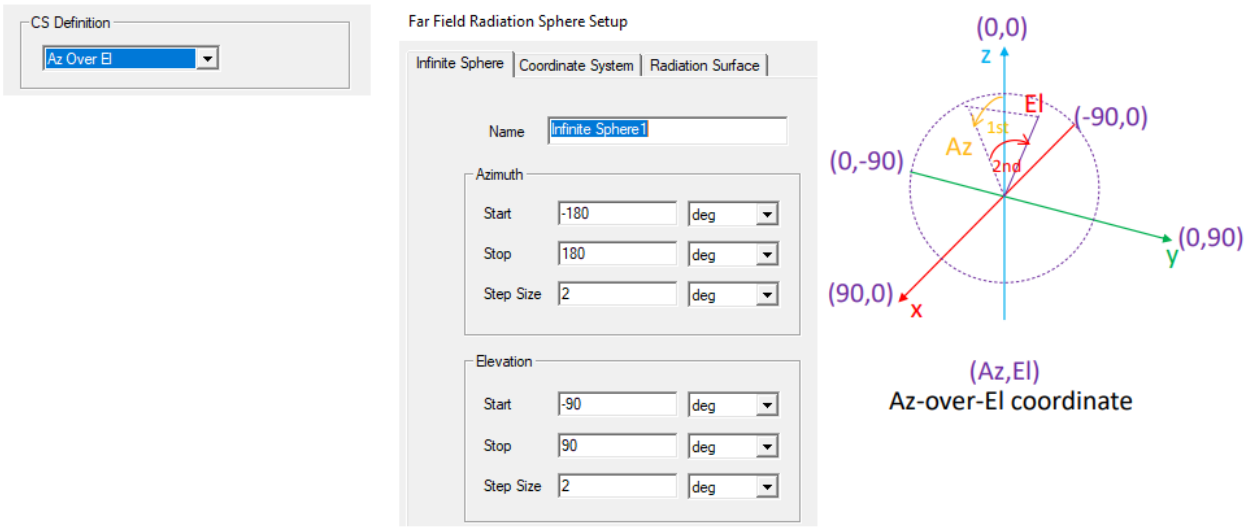


For Theta-Phi selection as Coordinate System, the Infinite Sphere setup is as follows.

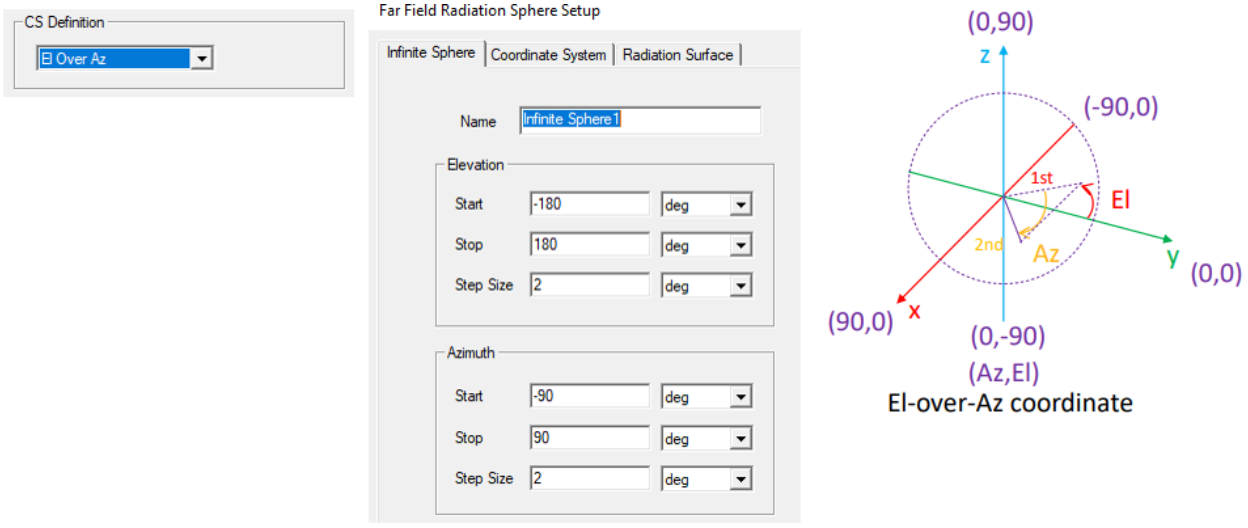


Azimuth over elevation (**Az Over El**) has the pole around Y axis while both azimuth and elevation start their 0 deg angle from Z axis. The pole is at elevation angle of –90 deg and 90 deg.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

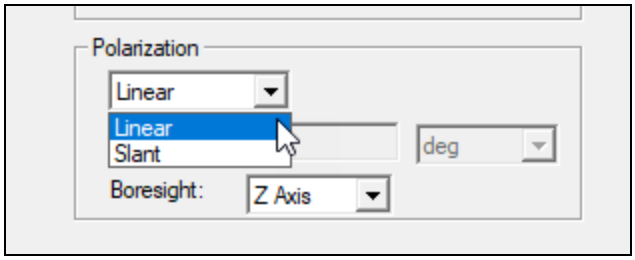


Elevation over azimuth (**El Over Az**) has the pole around X axis while both azimuth and elevation start their 0 deg angle from X axis. The pole is at azimuth angle of –90 deg and 90 deg.

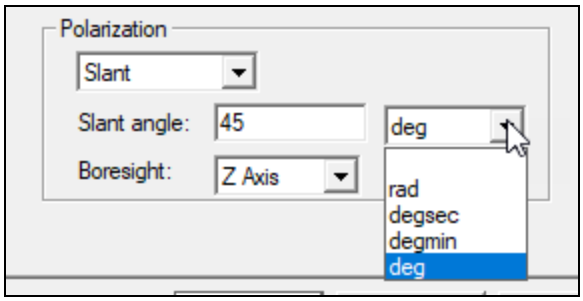


Polarization: specify the field polarization as Linear, or Slant which decomposes the field into axes rotated with slant angle around the Z axis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

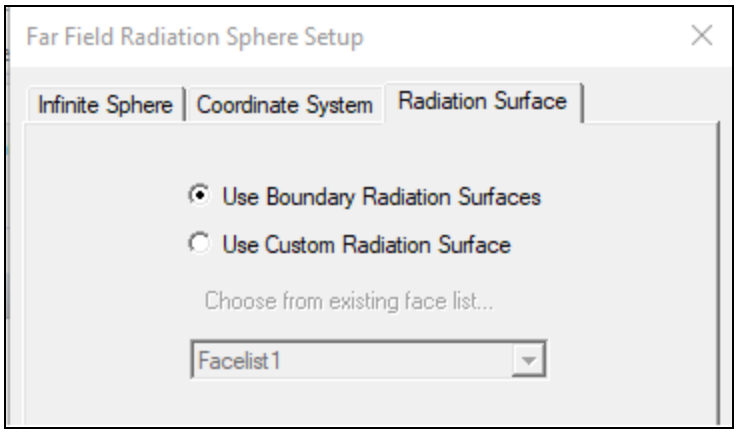


If you select Slant, the field for value and unit menu are enabled. Only CoPolar and CrossPolar values (see Selecting a Far Field Quantity to Plot, Step 4) are affected if Slant polarization is chosen vs. Linear option.



4. Use the **Radiation Surface** tab to select the solved surface from which to calculate radiated fields.

Use Boundary Radiation Surfaces is selected by default, indicating that the radiated fields will be calculated using the assigned radiation or PML surface. For some models you may find it it more efficient and/or accurate to use an interior surface. In this case, select **Use Custom Radiation Surface**, and choose a face list that you previously created in the modeler.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Do not use a sheet-object based face list as the radiation computation surface.

You can use the **Save as Default** to set the current values as a default, and the **Use Defaults** button to use previously saved options.

Note:

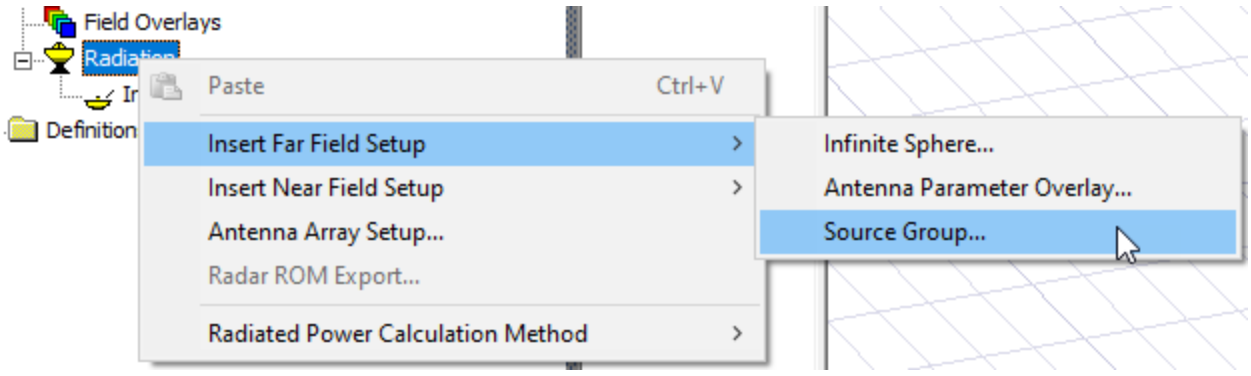
You must have defined at least one radiation or PML boundary in the design for HFSS to compute far-field quantities, regardless of which radiation surfaces you instruct HFSS to use when calculating the far fields. You do not need to re-solve the problem if you modify radiation surfaces in the **Far Field Radiation Sphere Setup** window.

Setting up a Source Group for a Finite Array

HFSS Edit Sources has had the concept of 'Source Contexts' for far field plots. Each Source Context specifies a single excitation (port) which will be active within the reporter and RadField module. All other excitations are inactive – in Edit Sources terms, they have a magnitude of zero.

This was perfectly adequate for simple antenna simulation, but it fails to meet users' needs with respect to finite arrays. For those users, sets of excitations are needed. That is to say, we need multiple 'Source Contexts' and each context must maintain its own set of active excitations.

A new item, Source Groups, will be first class objects in the project tree. They will be shown under the Radiation folder along with Infinite Sphere and Antenna Parameter overlays. Under the Radiation menu will be the option to create a new Source Group. Within the tree, the normal Edit, Rename, Copy, Paste, and Delete operations will be available to manage Source Groups.



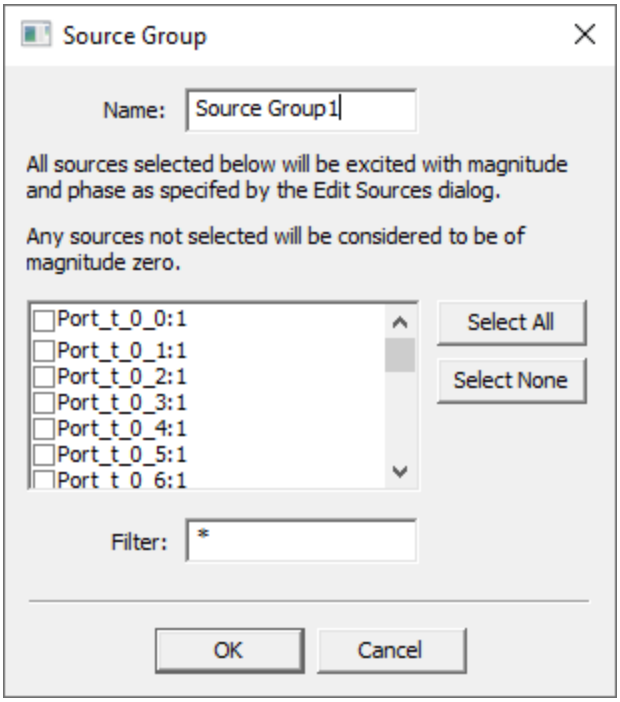
If a finite array exists in the design, the new Source Group will allow the user to activate a subset of the virtual excitations (e.g. array[5,5]port1:2 …) instead of the physical excitations associated with the unit cell(s).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To evaluate radiated fields in the far-field region, you must set up an infinite sphere that surrounds the radiating object. To plot far-field values across the sphere, you will select the sphere object from the **Geometry** list in the **Traces** dialog box when you create a report. To evaluate RCS using a design with an incident plane wave and an SBR+ region, you can also setup an infinite sphere.

1. **Click HFSS >Radiation>Insert Far Field Setup>Source Group**

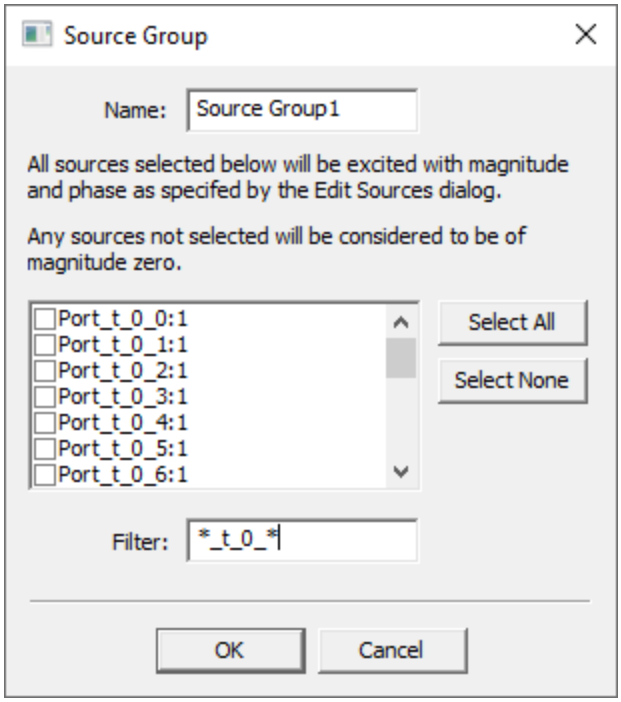
The **Source Group** window appears.



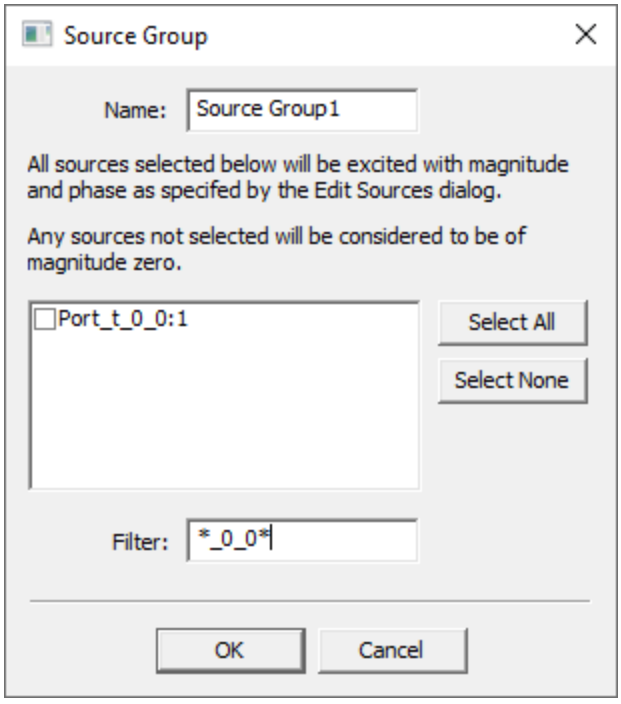
2. Use check boxes and/or **Select All/None** buttons to specify which sources to excite with the magnitude and phase as specified in the Edit Sources dialog.

The Filter: box permits you to sort ports based on regular expressions. For example, where a pattern of port names has been created by a script, the first number is the row number and the second number is a column number. When the filter is *_t_0*, all ports are in the 0th row.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

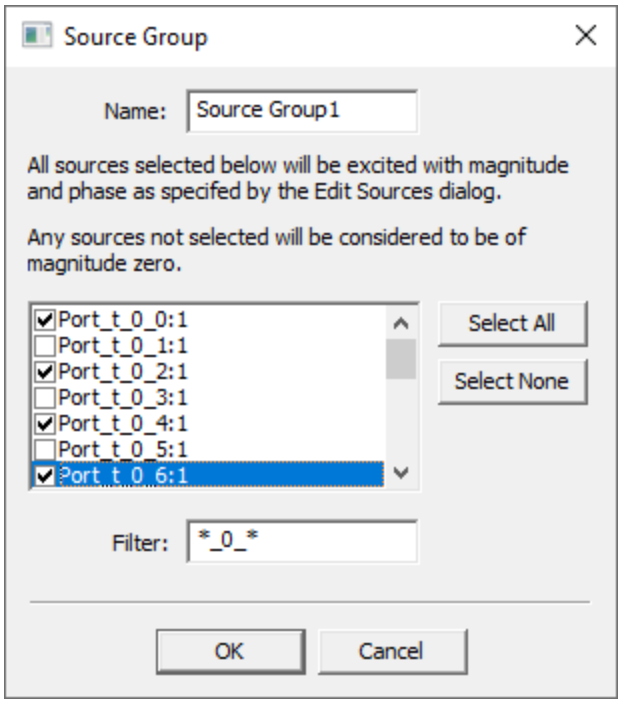


When the Filter is *0_0_* only Port_t_0_0 is selected.



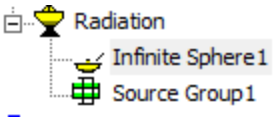
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Check which Ports to excite with a magnitude and phase as specified by the Edit Sources dialog.



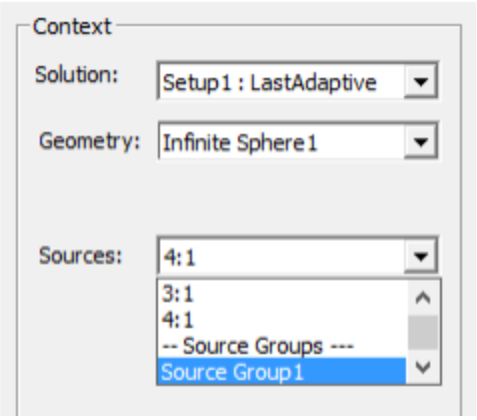
Select All can check all sources. **Select None** deselects all sources. Any sources not selected will be assigned magnitude zero.

3. Click **OK** to close the dialog box. The Source Group appears in the Project tree under the Radiation icon.



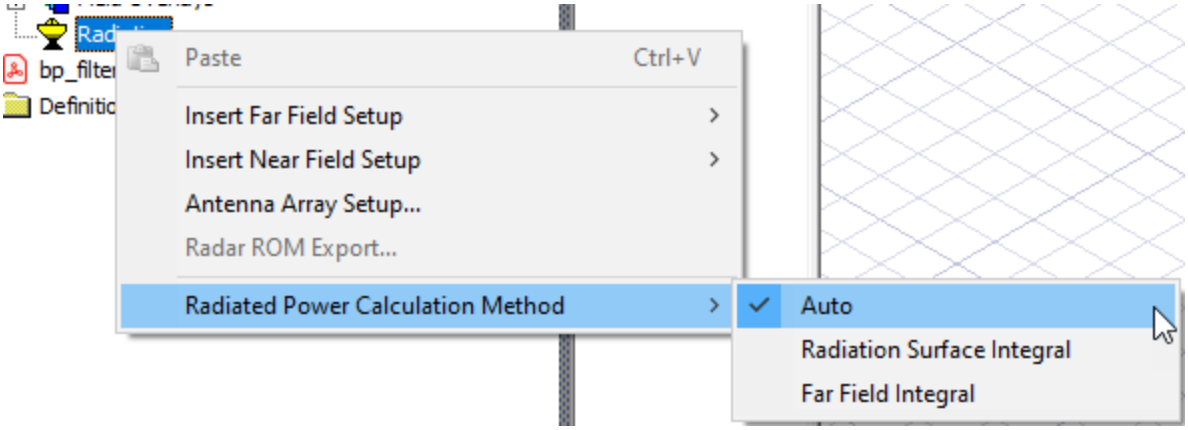
If you create a Radiated Fields report, the Source groups you have created will be listed with other sources in the Context window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Radiated Power Calculation Method

For design that contain only FEM objects, you can select the Radiated Power Calculation method. This can ensure that the radiated power result is continuous in a frequency range or parametric sweep. If the design contains surface regions (IE, PO or SBR+ regions) or linked fields (either a near field or far field link) the power calculation method commands do not appear.



The default selection is Auto.

- For **Auto** mode, the rule is: When design has a single FEM region, we use radiation surface integral. When design has multiple disjointed FEBI regions, we use far field integral. Based on our studies, this rule can ensure the best accuracy for the majority of projects.
- For **Radiation Surface Integral**, near fields are integrated over each element on the radiation surfaces to obtain their contribution of radiated power, and then added together.
- For **Far Field integral**, far field is first calculated over a full sphere, and far fields are integrated over the sphere.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

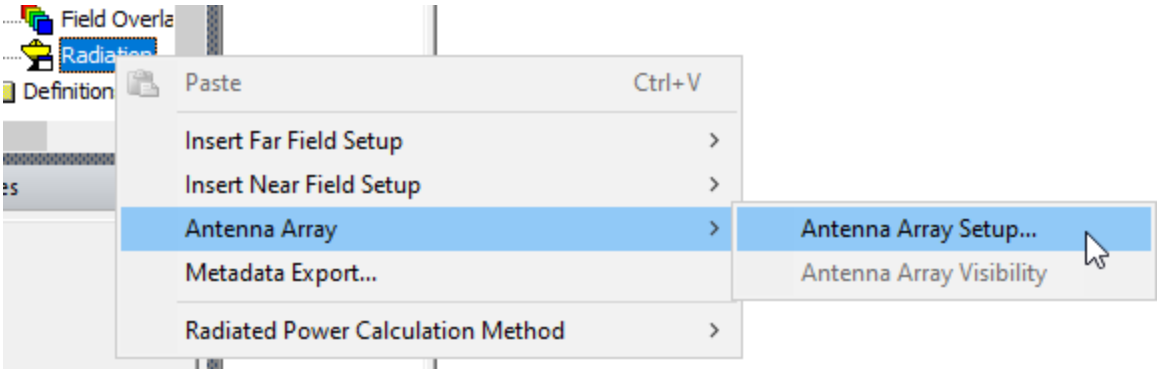
In some case, when the model is barely radiating**, Far Field integral** method may give better accuracy than **Radiation Surface Integral** method.

When **Far Field integral** is used, a correction factor is applied to Radiation Efficiency and Gain calculation (see Computing Antenna Parameters). The correction factor is equal to the ratio of the radiated power calculated from **Radiation Surface Integral** divided by the one calculated from **Far Field Integral**.

Defining Antenna Arrays

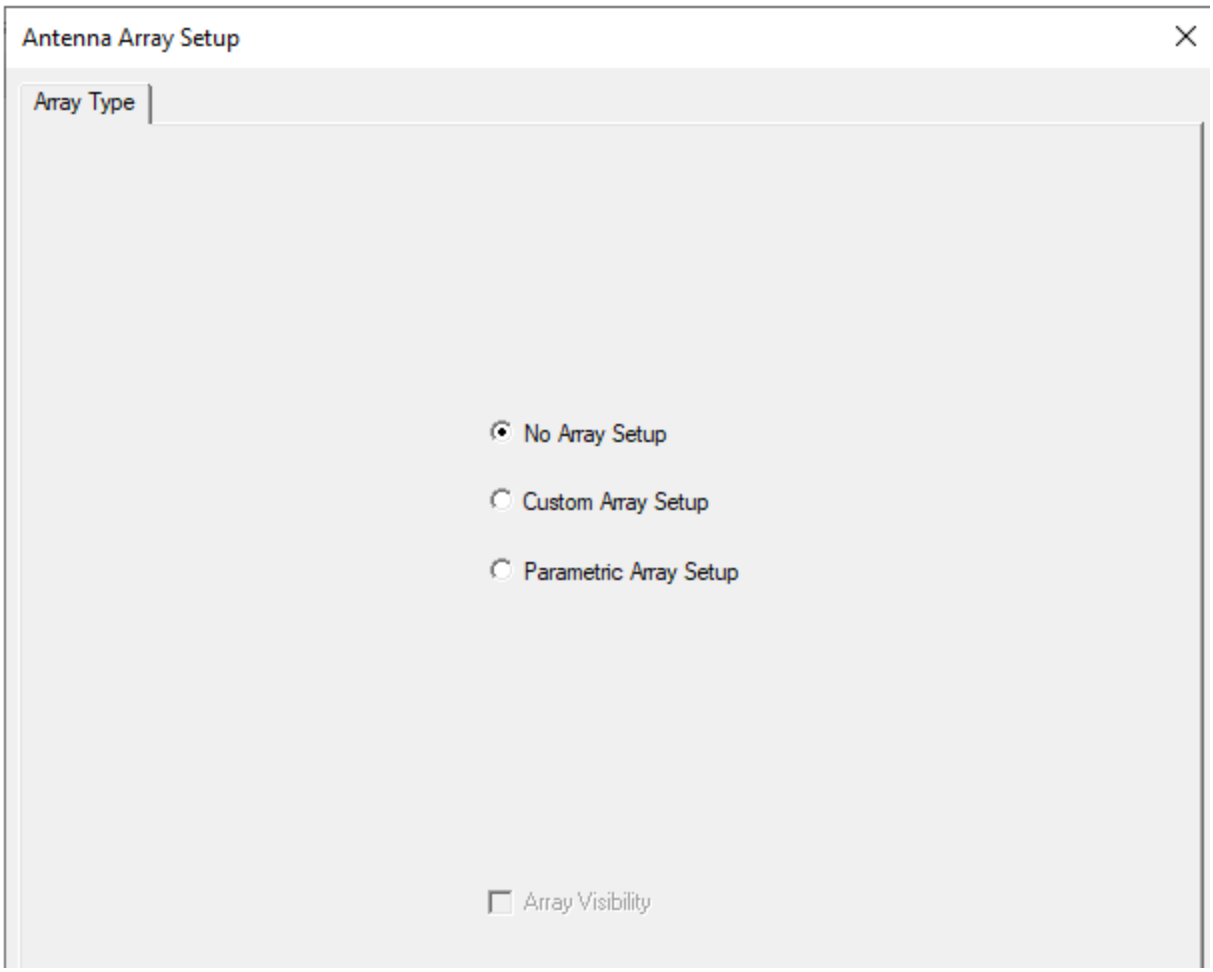
Define a parametric or custom antenna array when you want HFSS to compute antenna array radiation patterns and antenna parameters for designs that have only a single array element. HFSS models the array radiation pattern by applying user specified to the single element's pattern when far fields are calculated.

Click **HFSS >Radiation> Antenna Array>Antenna Array Setup...** or right click on the **Radiation** icon in the **Project** tree and select **Antenna Array>Antenna Array Setup...**



This displays the **Antenna Array Setup** dialog.

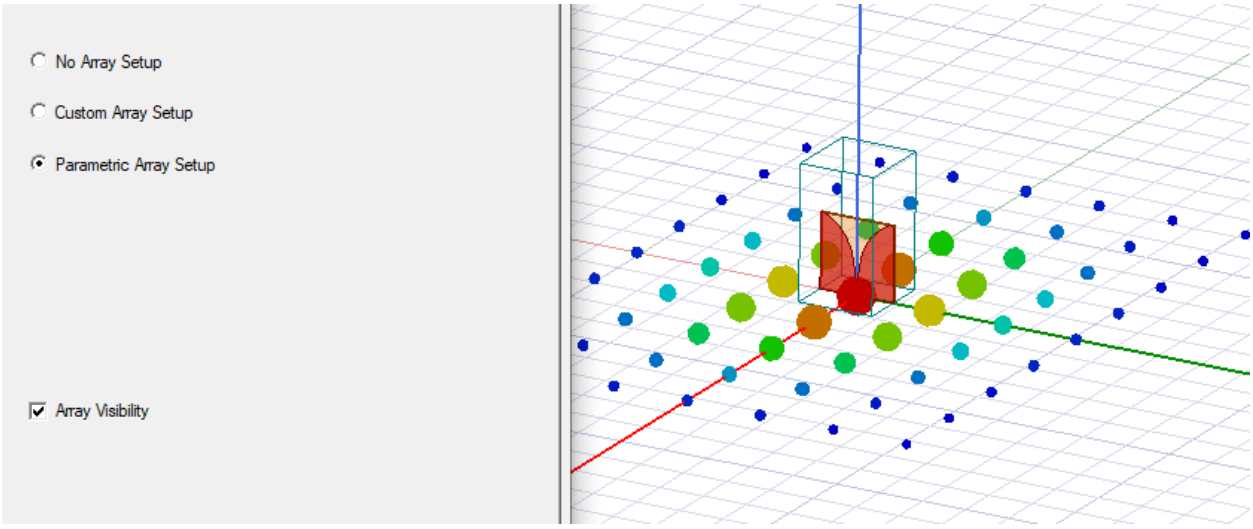
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



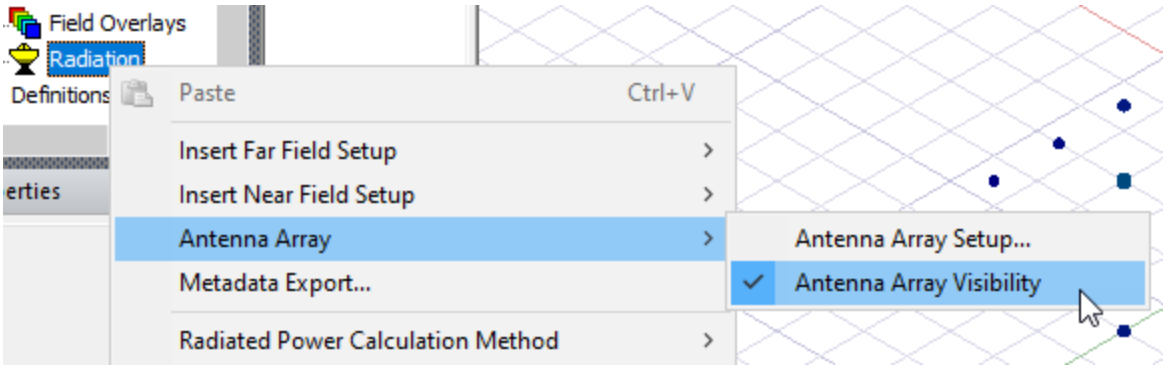
The antenna array setup defines a 2D array based on either a custom array setup based on a text file, or a parametric array setup.

After array type is selected, the **Array Visibility** checkbox is enabled to provides the array visualization based on the parameters provided.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After antenna array is defined, you can turn on/off the array visualization by toggling on/off the **Antenna Array Visibility** menu .

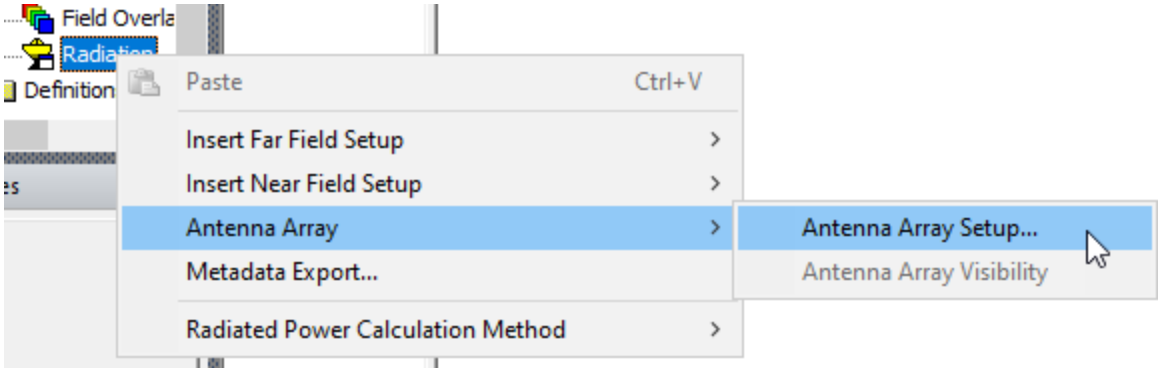


Defining a Parametric Antenna Array

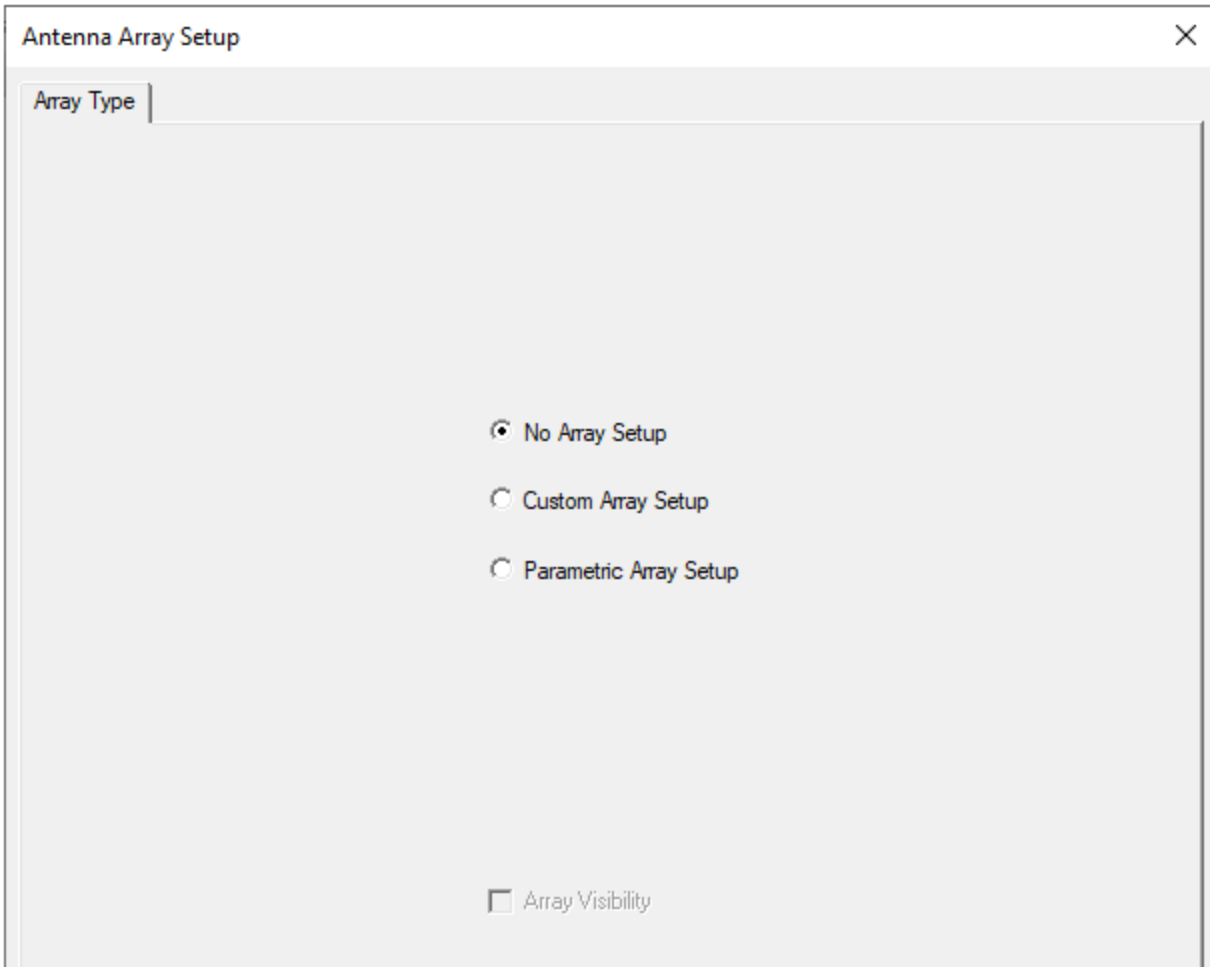
A Parametric antenna array is a finite 2D array geometry, which could easily simulate predefined layout types (rectangular, elliptical, linear), amplitude/weight types (like Uniform, Hamming) and scan definition methods (scan angles, phase shift).

1. Click **HFSS >Radiation> Antenna Array>Antenna Array Setup...** or right click on the **Radiation** icon in the **Project** tree and select **Antenna Array>Antenna Array Setup...**

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



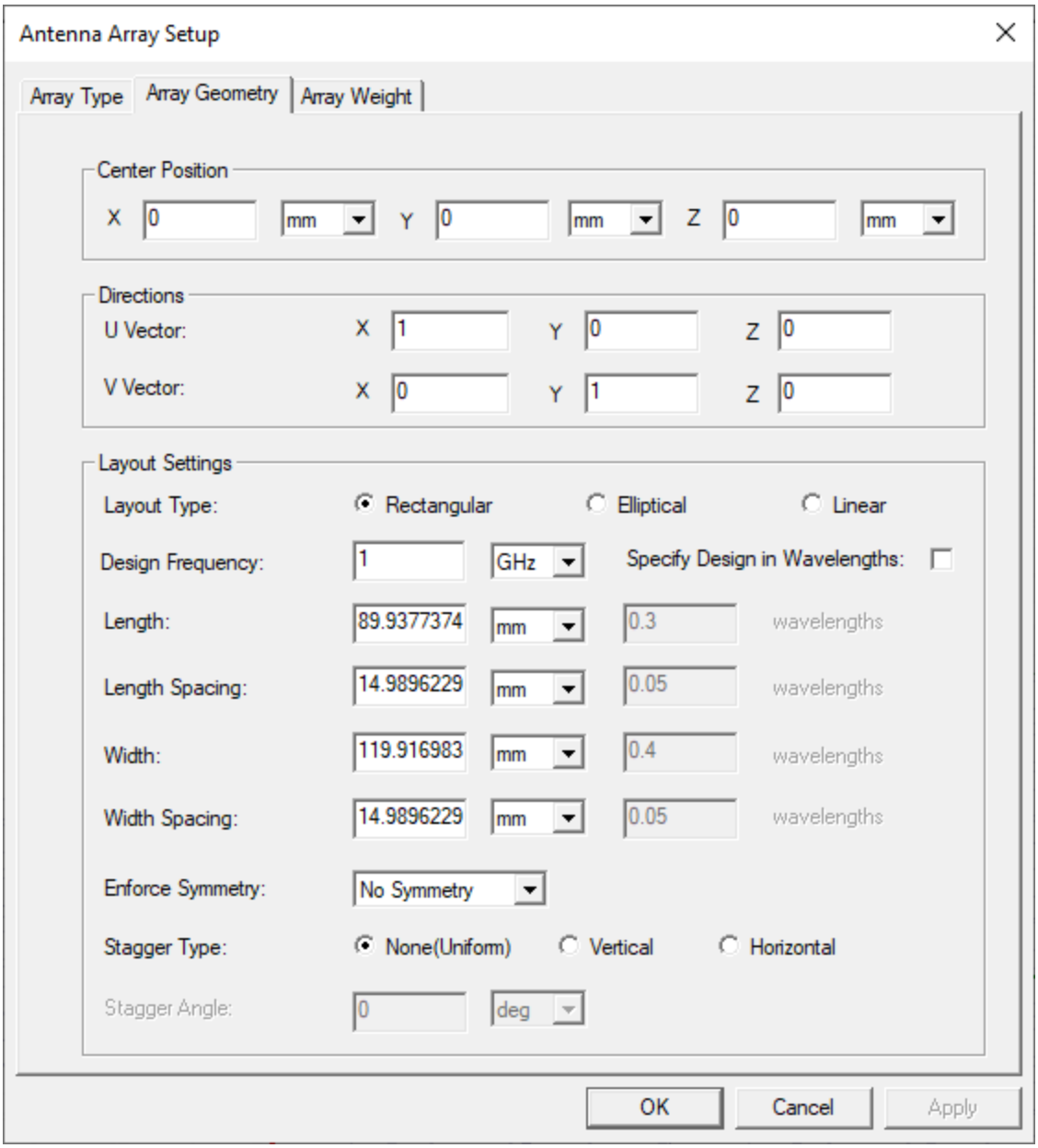
The **Antenna Array Setup** window appears.



2. Under the **Array Type** tab, select **ParametricArray Setup**. The enables tabs for **Array Geometry** and for **Array Weight**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

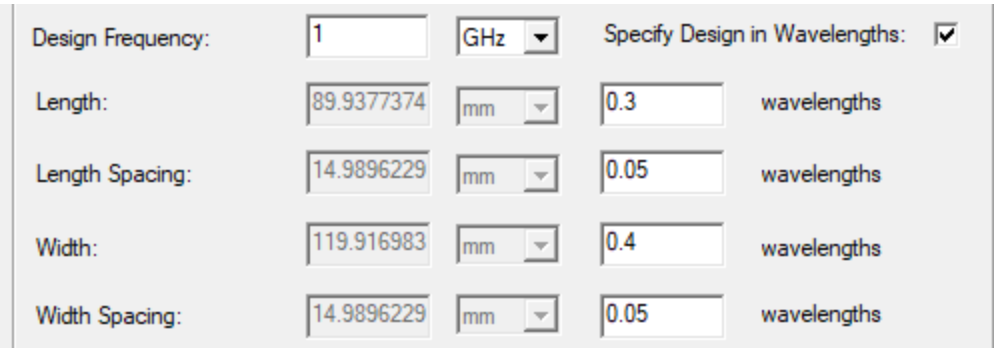
3. Click the **Array Geometry** tab.



4. Under **Center Position**, enter the xyz-coordinates where the center cell is placed.
5. Under **Directions**, define the lattice vectors:
   a. To the right of **U Vector**, enter the vector coordinates in the **X**, **Y**, and **Z** text boxes along which the cells in the U-direction are placed.
   b. To the right of **V Vector**, enter the vector coordinates in the **X**, **Y**, and **Z** text boxes along which the cells in the V-direction are placed.

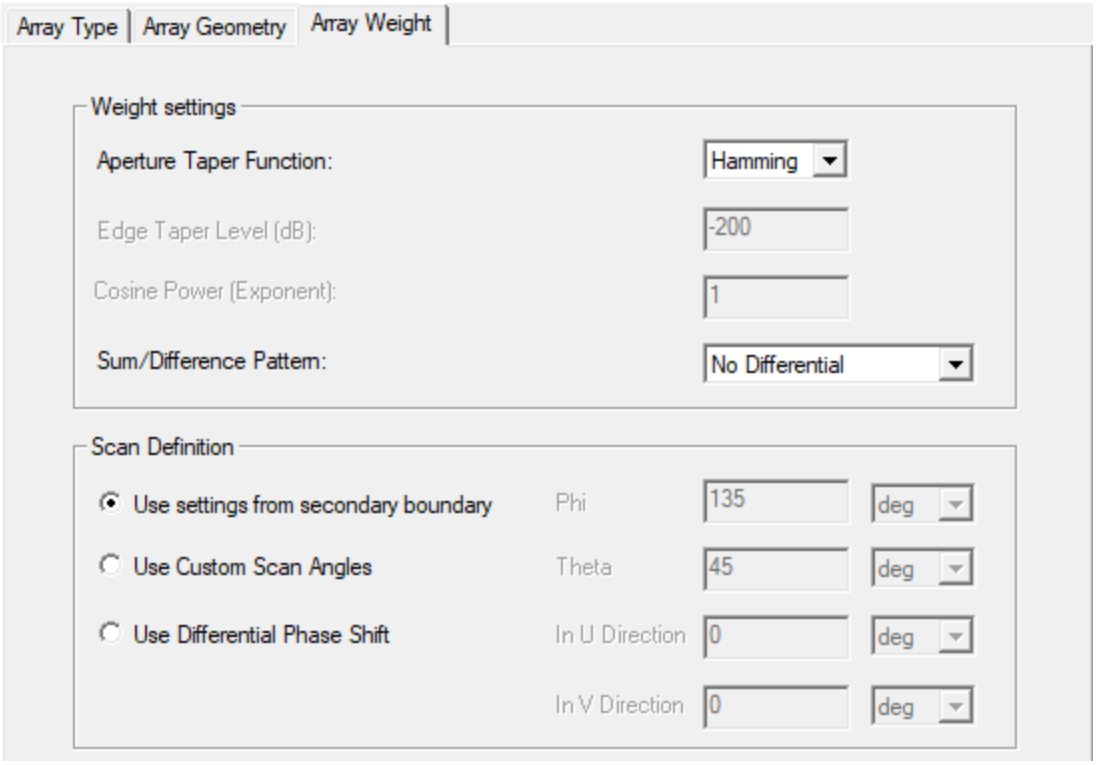
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Under **Layout Settings**, specify the layout type as Rectangular, Elliptical, or Linear.
7. For **Design Frequency**,by default, specify the Frequency, Length, Length Spacing, Width, and Width Spacing values, as or check to enable Specify Design in Wavelengths.



Note: The “Design Frequency” is only used for array setup and preview. It will not replace the solution frequency. If a Secondary boundary is used, U / V directions and length / width settings would match lattice vectors when setting up a new parametric array.

8. For **Enforce Symmetry**, use the menu to specify No Symmetry, Horizontal Symmetry, Vertical Symmetry, or Horizontal and Vertical Symmetry.
9. For **Stagger Type**, select one of None (Uniform), Vertical, or Horizontal. If you specify a Stagger type, the Stagger Angle field is enabled.
10. Click the **Array Weight** tab.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11. Under **Weight settings**, select the **Aperture Taper Function** from Hamming, Uniform, Cosine, or Triangular. Each selection enables the appropriate parameters:
    Hamming and **Uniform** require a Sum/Difference Pattern as No Differential, Horizontal Differential, Vertical Differential, or Horizontal and Vertical Differential.
    Cosine requires Edge Taper Level, and Cosine Power [Exponent], as well as Sum/Difference Pattern.
    Triangular requires Edge Taper Level, as well as Sum/Difference Pattern.
12. Under **Scan Definition**, either Use settings from secondary boundary, Use Custom Scan Angles as Phi and Theta, or Use Differential Phase Shift in U or V directions.
13. Click Apply (with Preview Array Layout checked) to view a representation in the Modeler window, and OK to close the dialog and dismiss the preview.

The Array Geometry parameters will be applied, when far fields are calculated.

Defining a Custom Antenna Array

A custom antenna array is an arbitrary array of identical elements distributed in 3D space with individual user-specified complex weights. The array is defined in a text file that includes the element positions, voltage amplitude weights, and phases. See Custom Arrays in the *Technical Notes* for examples of custom array geometry text files.

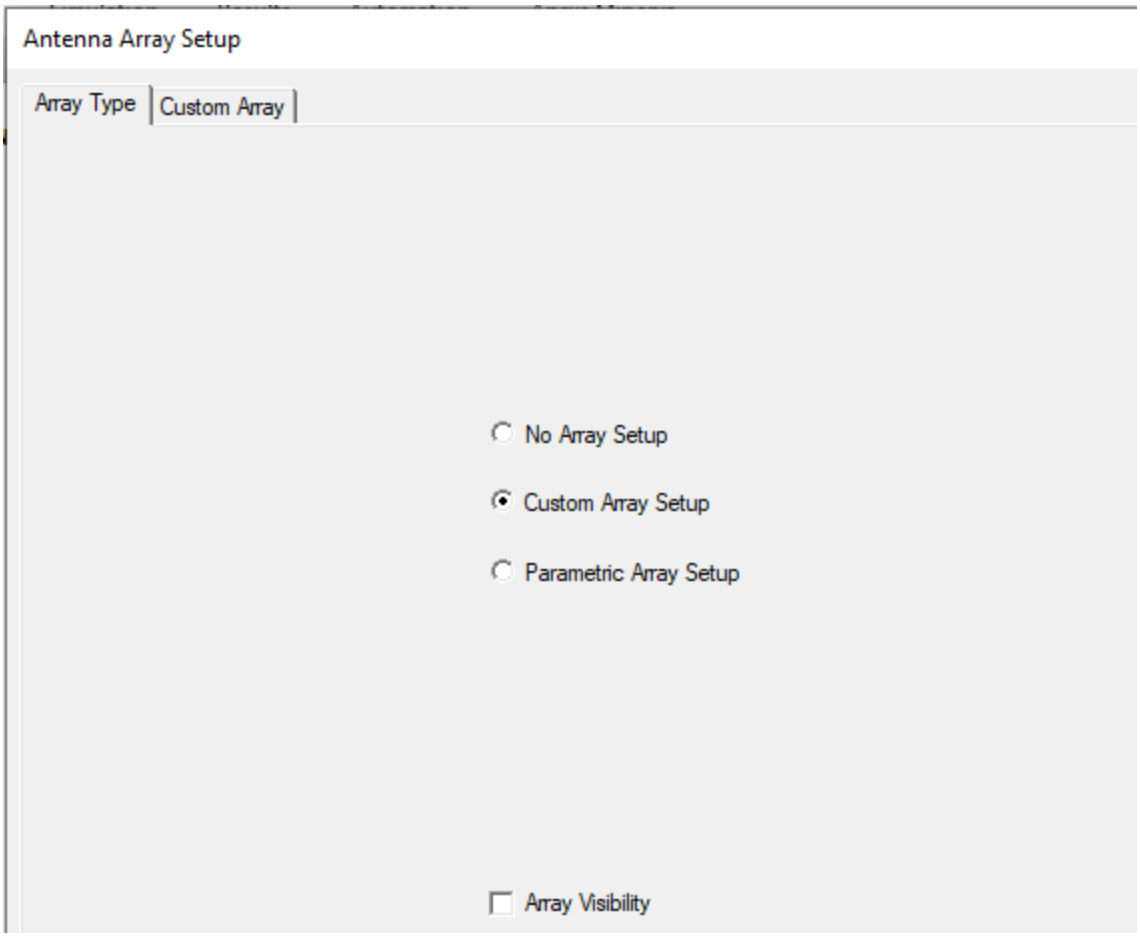
> **Note:**
>
> For custom antenna arrays, phases should be specified in radians.

1. Click **HFSS >Radiation>Antenna Array>Antenna Array Setup...** or right click on the **Radiation** icon in the **Project** tree and select **Antenna Array>Antenna Array Setup...**.

The **Antenna Array Setup** window appears.

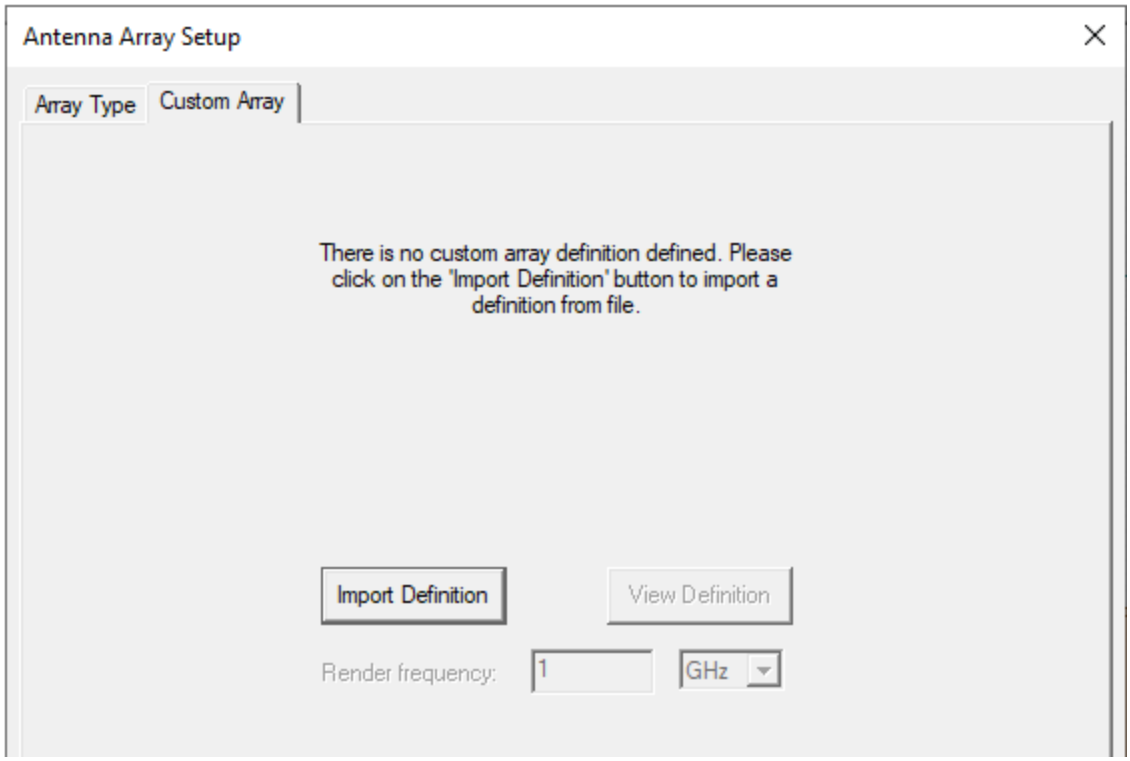
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Under the **Array Type** tab, select **Custom Array Setup**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click the **Custom Array** tab.



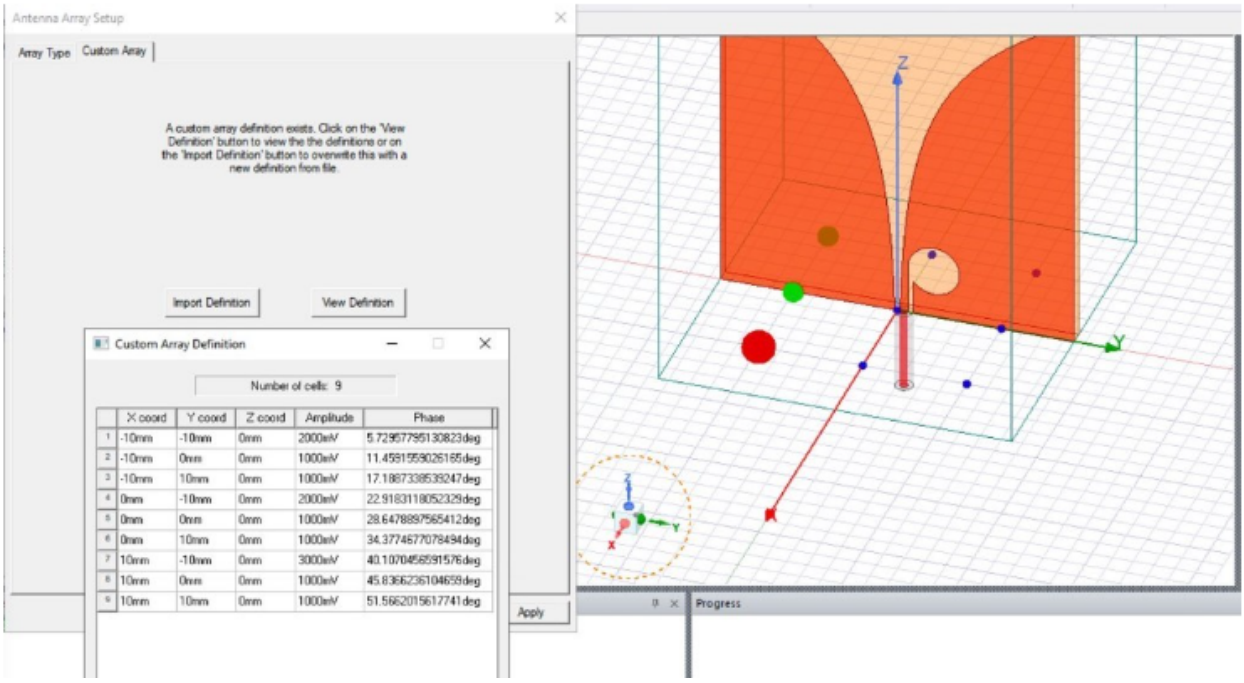
4. Click **Import Definition**.

The **Open** dialog box appears.

5. Follow the procedure for opening a file. Select **.txt** as the file type. When you are finished, click **Open**.
   Note: You can use design / project variables in a definition .txt file. Specifically, the render frequency edit box is enabled if intrinsic variable “freq” is used. As with the parametric array, this render frequency is only used for array preview and will not change the solution frequency.
6. Optionally, review the definition in the text file by clicking **View Definition** under the **Custom Array Setup** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. With Array Visibility checked, you can Preview a Custom Array Definition in the Modeler window.



8. Click **OK** to dismiss the dialog.

The array factor will be applied, using the information specified in the text file, when far fields are calculated.

Computing Antenna Parameters

You can Export the calculated antenna parameters and maximum field data to a file. You can also Export Fields for a selected setup in the .csv format or .ffd format. This can be useful for situations when want to share the fields without sharing the model geometry. You can save for use on an antenna parameters Overlay. You can also Export Element Pattern the far field relative to the specified inputs and sources

Note: A Beta feature for HFSS, Hfss Interactive Dialogs, allows you run the **Edit Sources** dialog and the **Antenna Parameters** dialog in modeless fashion, that is, if you **Edit Sources** for magnitude or renormalization, affected Antenna Parameters update. If you edit excitations or ports in the Far Field Setup, or create design variations in Optimetrics, the **Edit Source** dialog will close.

Case that update in this mode:

- Infinite sphere setup created, deleted or renamed: Setup Name combo box changes radiation setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Solution setup created, deleted or renamed.
- Antenna array setup created or deleted.
- Source group created, deleted or renamed.

Note: Intrinsic Variation updates as the solution setup changes. Grids of “Antenna Parameters” and “Maximum Field Data” update data as any of the combo box selections changes. The dialog closes if solution type is changed.

Prerequisites

You must have defined at least one radiation or PML boundary in the design for HFSS to compute antenna parameters and maximum field data for the far-field region.

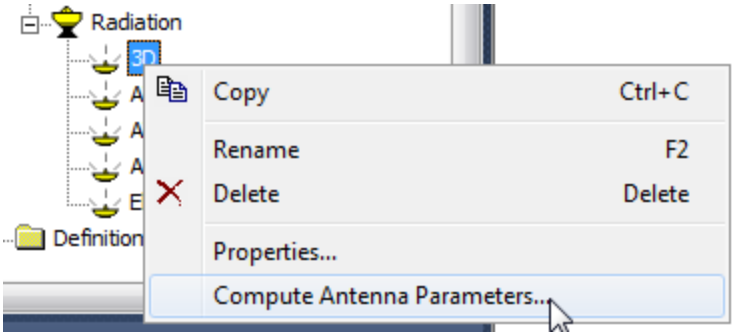
You must have selected Save Fields or Save Radiation Fields only in the Solution setup, and must have generated a solution for a Driven Modal / Driven Terminal / SBR+ / Transient design.

You must have specified and selected Radiation setup.

If the project is a Finite Array, and you want to specify Source Groups, you must previously create them.

Procedure:

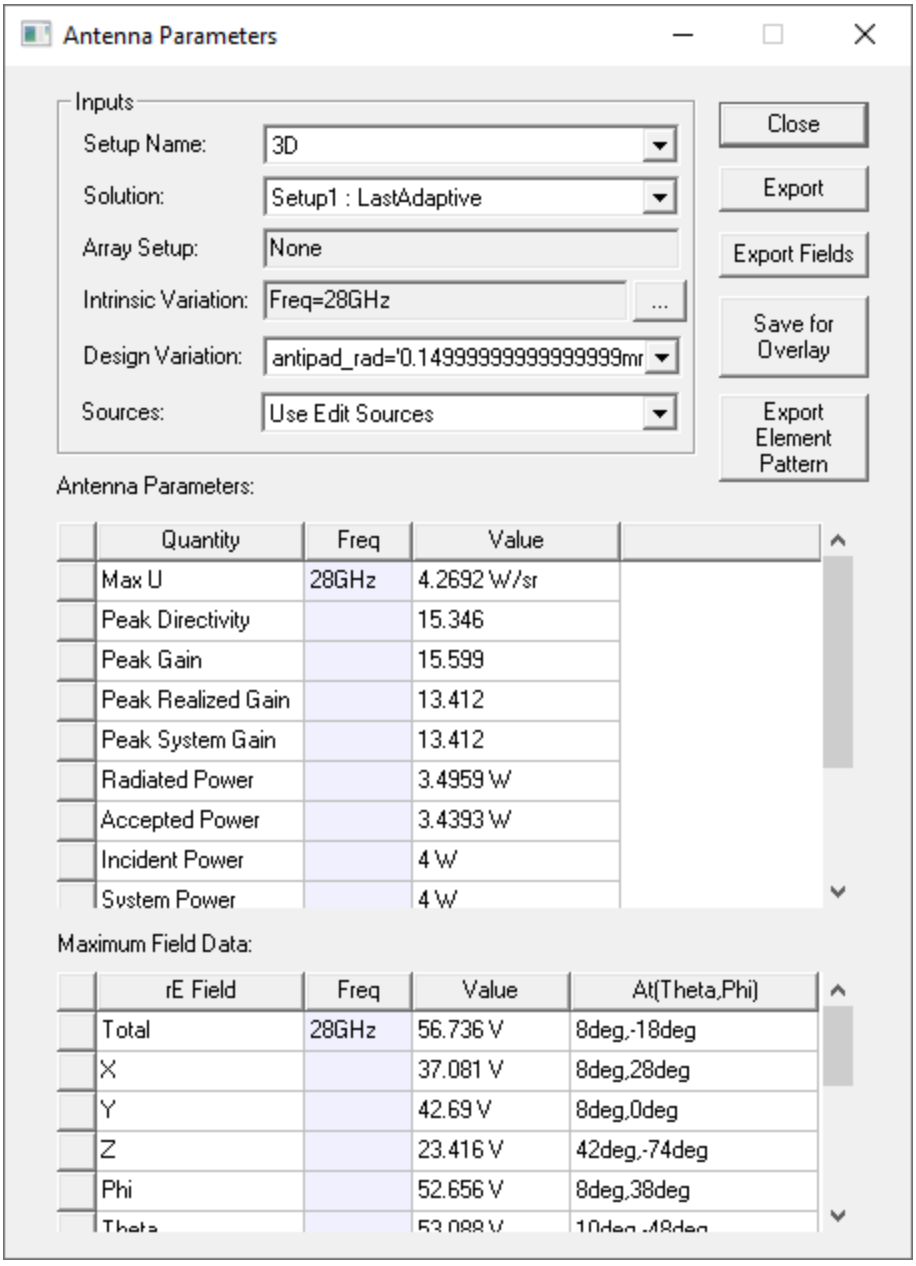
1. To select the radiation setup from the Project tree for a Driven Modal / Driven Terminal / SBR+ / Transient design, right-click the **Infinite Sphere** icon in the project tree under Radiation, and then click **Compute Antenna Parameters** on the shortcut menu.



Or, to select the radiation setup from a dialog, click **HFSS >Radiation>Compute Antenna Max/Params**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

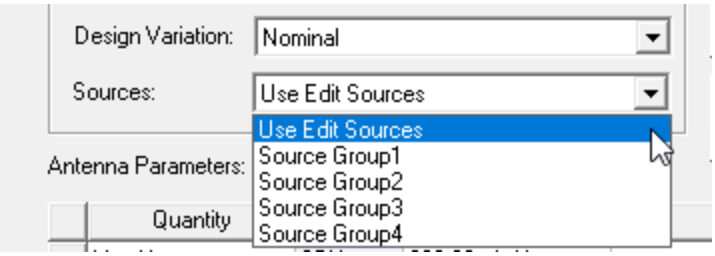
The **Antenna Parameters** dialog box appears.



2. Select the Radiation Setup Name. A dropdown menu lets you select from available setups.
3. For the **Solution**, select the solution for which you want HFSS to compute antenna parameters.
4. If there is an array, select the setup to use.
5. For the **Intrinsic Variation**, select the solved frequency point at which you want HFSS to compute antenna parameters.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. If there is more than one Design Variation available, you can select from the drop-down menu.
7. For Finite Arrays, if you have created one or more source groups, you can select from the drop-downlist.



8. The **Antenna Parameters** table lets you view computed parameters. If the design includes ports, the following antenna parameters are listed:

Maximum intensity (Max U)
- Peak directivity
- Peak gain
- Peak realized gain

System Gain (if specified in Edit Sources)
- Radiated power
- Accepted power

System Power (typically from an external circuit design and specified in Edit Sources)
- Incident power
- Radiation efficiency

System efficiency

Total Efficiency
- Front to Back Ratio
- Decay Factor

> **Warning:**
>
> The computed values of max U and peak directivity depend on the user-determined set of aspect angles chosen for the computation of the radiated fields. If this set does not encompass the actual peak intensity of the radiated pattern, the displayed results for these three parameters will be inaccurate.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> **Accepted Power** is computed from the raw S-parameter data. Post-processing operations are excluded from the calculation, for example, renormalized S-parameters.

> **Note:**
>
> It is important to understand the impact of passive ports (for Transient Network designs or models containing an Array with passive ports) on antenna parameters. For accepted power calculations, passive ports are not included when computing the total power passing through the union of all port surfaces. This means that the passive ports can be viewed as a loss mechanism for the device and it is not equivalent to viewing the passive ports as active ports with zero excitations.

Gain and realized gain are very close to each other when the antenna is matched. However, for designs with a multi-port antenna, gain and realized gain can be different if incident and accepted power are different. There can be a small reflection at the ports and accepted power may still be small if energy injected in one port exits the model through a different port. A review of the S-matrix can show this to be the case. This does not happen in a single port antenna, which is what most users base their expectations on.

When global material environment is a lossy material, antenna parameters such as gain and radiated power are no longer unique because the radiated power depends on the location of the radiation surface since power is lost as the wave travels in the background material.

If the design does not have ports, the following antenna parameters are listed:

Maximum intensity (Max U)
Peak directivity
Radiated power

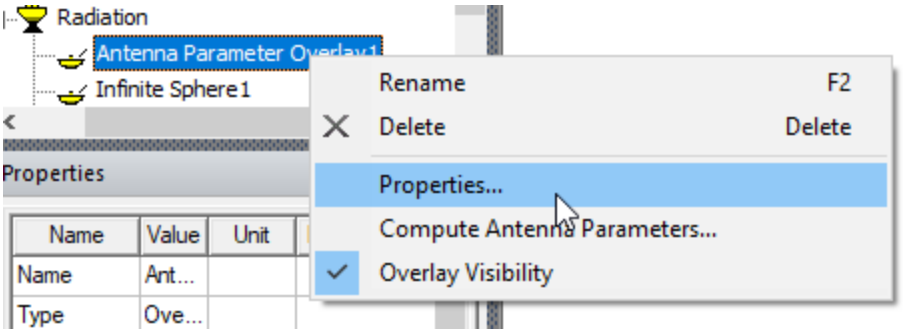
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. You can view the following maximum far-field data:

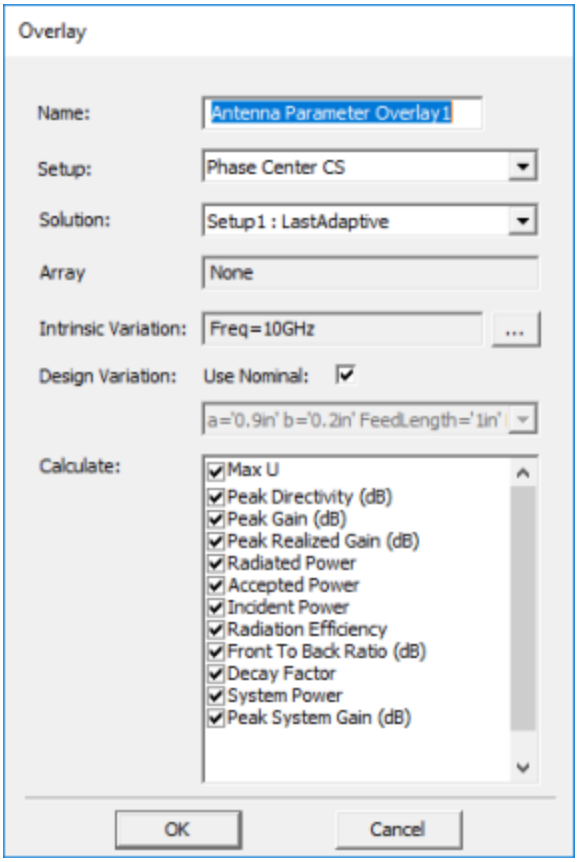
   Total
   X
   Y
   Z
   Phi (depending on CS Definition in the Far Field Infinite Sphere)
   Theta (depending on CS Definition in the Far Field Infinite Sphere)
   Azimuth(depending on CS Definition in the Far Field Infinite Sphere)
   Elevation(depending on CS Definition in the Far Field Infinite Sphere)
   LHCP
   RHCP
   Ludwig 3/X dominant
   Ludwig 3/Y dominant

> **Note:**
>
> When calculating the maximum far-field values, the distance *r* is factored out of the E-field. Therefore, the units for the maximum field data values are given in volts.

You can right click on an Antenna Parameters Overlay in the Project tree to open a Properties dialog that lets you select a set of the available parameters.



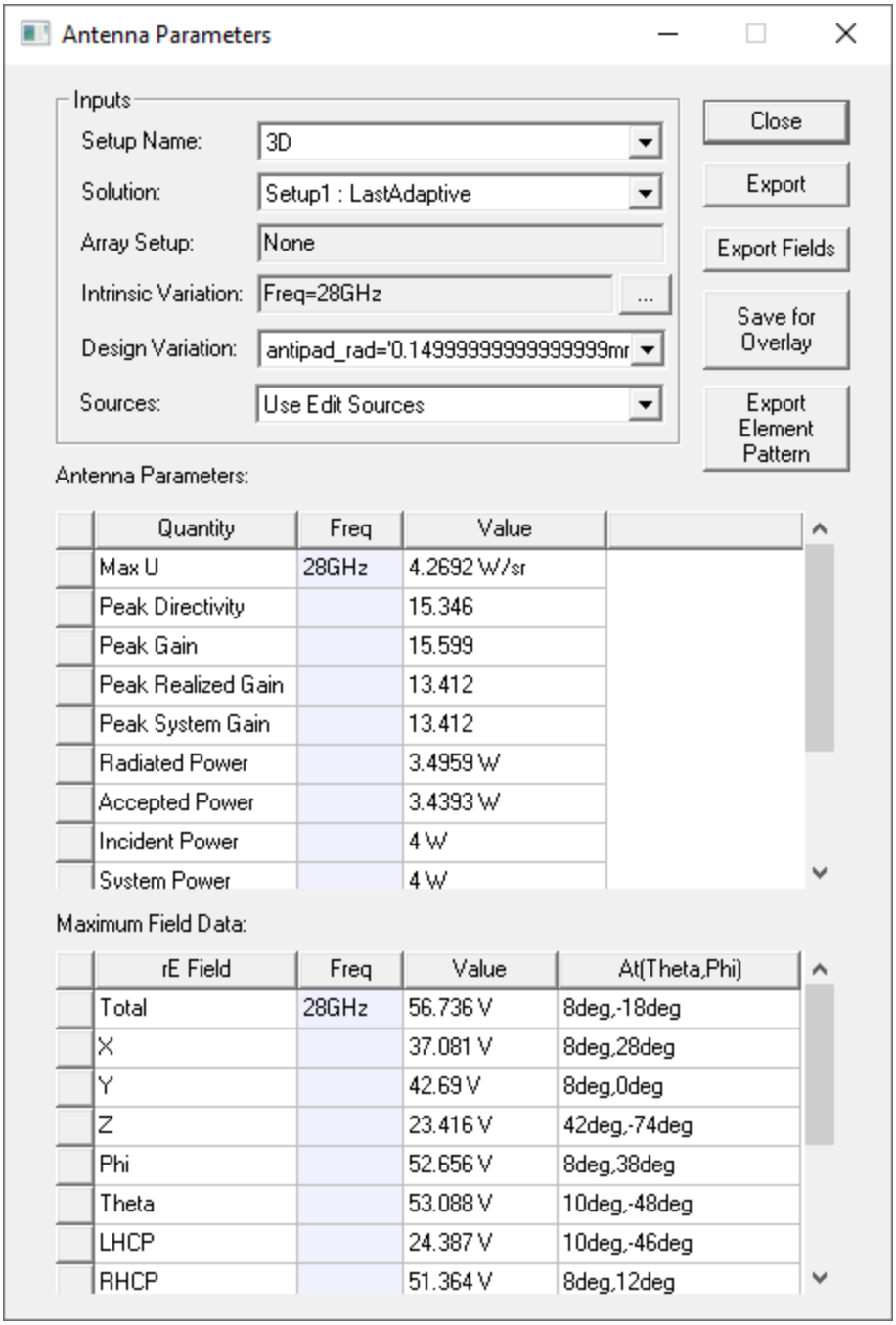
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Exporting Antenna Parameters and Maximum Field Data

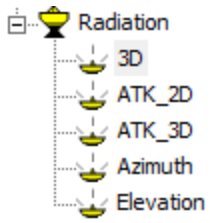
The **Antenna Parameters** dialog displays the calculated antenna parameters and Maximum Field data for a setup. The dialog also includes a buttons to **Export** antenna parameters and to **Export Fields**. The fields can be exported in ffd or .csv format and imported into reporter as a table. You can specify the Setup Name, the Solution, Array Setup (if any), Intrinsic Variation, and Design Variation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

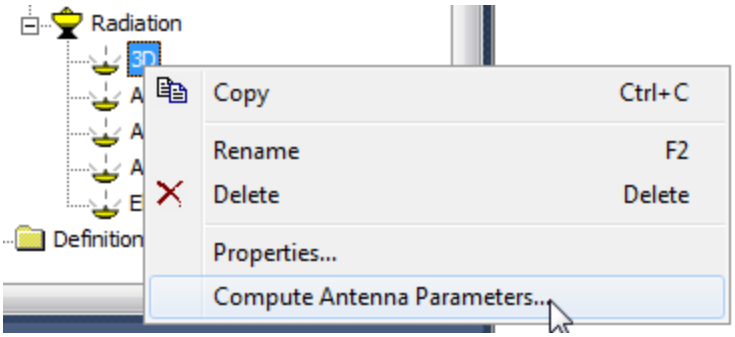


The Setup Name corresponds to the Radiation Setups you create, or that are created by the Antenna Design Kit, for example:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To open the **Antenna Parameters** dialog box, right-click the Radiation setup for which you want to export parameters or fields, and select **Compute Antenna Parameters** from the menu



To export the antenna parameters to a text file:

1. Click **Export** on the **Antenna Parameters** dialog.

   This displays a file browser.

2. Specify the file name and location (or accept the defaults).
3. Click **Save**.

   This saves the text file and closes the browser.

   Parameter Format: The parameters are exported to a text file in the following format:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Quantity	Freq	Value
Max U	0.9GHz	1.358067 mW/sr
Peak Directivity		1.745194
Peak Gain		1.752898
Peak Realized Gain		1.706638
Radiated Power		9.779070 mW
Accepted Power		9.736095 mW
Incident Power		10.000000 mW
Radiation Efficiency		1.004414
Front to Back Ratio		1.012037
Decay Factor		0.000000

rE Field	Freq	Value	(Theta, Phi)
Total	0.9GHz	1.011920 V	(90deg,-46deg)
X		456.079554 mV	(48deg,180deg)
Y		456.794982 mV	(48deg,-90deg)
Z		1.011920 V	(90deg,-46deg)
Phi		4.228478 mV	(28deg,-52deg)
Theta		1.011920 V	(90deg,-46deg)
LHCP		715.836833 mV	(90deg,-46deg)
RHCP		715.234428 mV	(90deg,-46deg)
Ludwig3/X dominant		1.008985 V	(90deg,0deg)
Ludwig3/Y dominant		1.008579 V	(90deg,-90deg)

To export the maximum field data to a comma separated format file:

1. Click **Export Fields** on the **Antenna Parameters** dialog

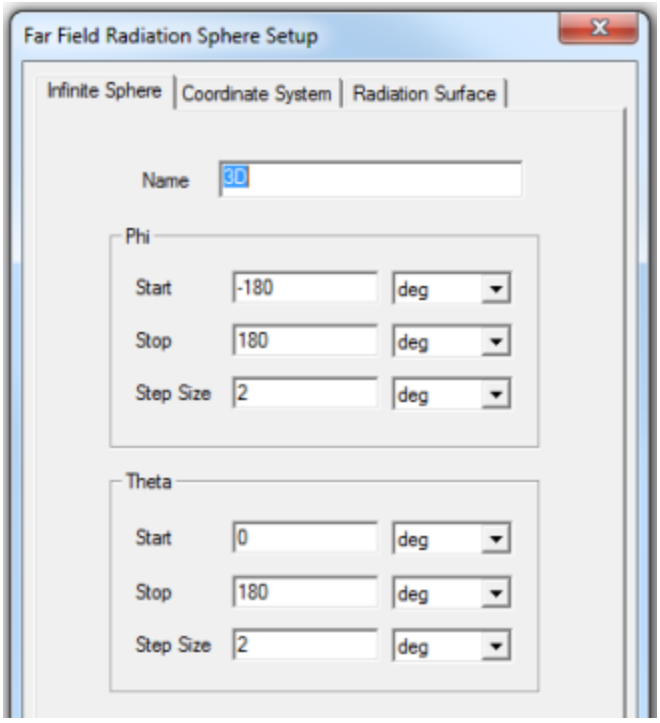
   This displays a file browser

2. Specify the file name and location (or accept the defaults)
3. Select Save as type as Far Field Data (*.ffd) or Comma Separated (*.csv).
3. Click **Save**.

   This saves a text file for the selected Radiation setup in the specified type and closes the browser.

Depending on the kind of Radiation Setup you define, and your Save as type specification, and the coordinate system of the design, the formats will differ. For example for an Infinite Sphere Radiation Setup, using the CS definition as Theta-Phi, will appear as follows:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The first lines of the exported .ffd file show the Theta start and stop, then number of steps and then the Phi start and stop values, then number of steps, followed on the next line by the Frequencies number, then on the next line, the first frequency value, followed by field values for that frequency.

```
0 180 91
-180 180 181
Frequencies 1
Frequency 9.000000000000000e+008
1.064046258878916e-003 -2.994761550861182e-003 1.730463512343075e-003 -1.912365743030195e-003
1.123790376290396e-003 -3.059677824982989e-003 1.692274681823832e-003 -1.806685110620843e-003
1.182165328234976e-003 -3.120866352982335e-003 1.652024075279754e-003 -1.698803310702743e-003
1.239099993825706e-003 -3.178252586063140e-003 1.609760731874738e-003 -1.588851780631109e-003
```

If the CS Definition is AzOverEl, or ElOverAz, the first line will state that.

After that identification the next two lines are the Azimuth over Elevation angle definitions (i.e. start, end angle and # of angles). The second angle is always the outer loop, same as for theta phi.

```
AzOverEl
-90 90 91
-180 180 181
Frequencies 1
Frequency 5.000000000000000e+09
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

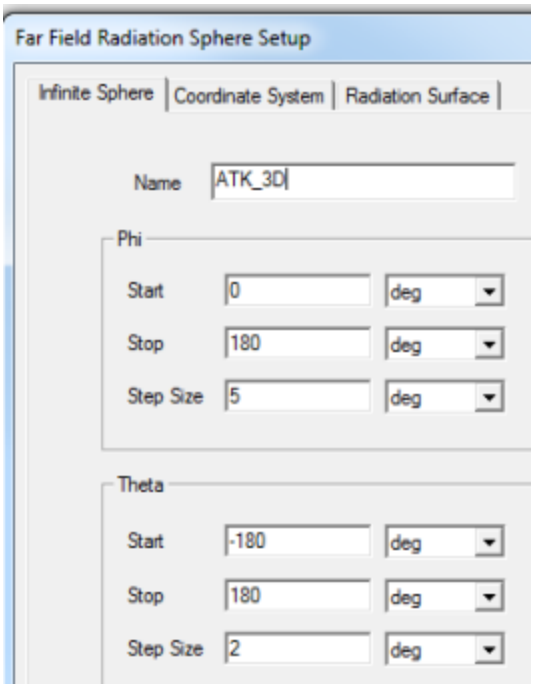
```
-1.015259849178660e+00 -1.655218649533607e-01 -3.522705268988268e-01
-3.529118467476278e-01
-1.002347316218049e+00 -1.531045876736399e-01 -3.874879909704635e-01
-3.584734921660798e-01
-9.882135774523947e-01 -1.405007760377924e-01 -4.222333606130519e-01
-3.635983928529357e-01
-9.728758526652468e-01 -1.277257858488285e-01 -4.564643039457966e-01
-3.682803049061392e-01
-9.563528285116635e-01 -1.147951814644569e-01 -4.901391158371188e-01
-3.725135241370467e-01
-9.386646357513984e-01 -1.017247168343128e-01 -5.232167687159368e-01
-3.762928930201028e-01
...
```

If the CS definition in the Infinite Sphere uses ElOverAz, the first line of the .ffd file has a line saying this. After that identification the next two lines are the Elevation over Azimuth angle definitions (i.e. start, end angle and # of angles). The second angle is always the outer loop, same as for theta phi.

```
ElOverAz
-90 90 91
-180 180 181
Frequencies 1
Frequency 5.000000000000000e+09
-4.637399324983455e-02 2.097570360902490e+00 2.541263616056640e-01 -
4.298638962513578e-01
-3.747686134725477e-02 2.081290544083335e+00 2.555899837169427e-01 -
5.028061846738747e-01
-2.853406966199131e-02 2.062474995334444e+00 2.567422088036812e-01 -
5.751358812117218e-01
-1.955651360818052e-02 2.041146638503657e+00 2.575816330570980e-01 -
6.467648632712089e-01
-1.055513095498903e-02 2.017331458908390e+00 2.581072337680435e-01 -
7.176058619712824e-01
-1.540888500621979e-03 1.991058471676533e+00 2.583183705730141e-01 -
7.875725684672551e-01
7.475231289027627e-03 1.962359686395983e+00 2.582147862343362e-01 -
8.565797391048062e-01
...
```

As another example, consider the following radiation Setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The first lines of the exported .ffd file show the Theta start and stop, then the number of steps and then the Phi start and stop values, then the number of steps, followed on the next line by the Frequencies number, then on the next line, the first frequency value, followed by field values for that frequency.

```
-180 180 181
0 180 37
Frequencies 1
Frequency 9.000000000000000e+008
2.150840721987304e-004 3.885036470901229e-004 -1.491058342690848e-003 2.645215701834250e-004
3.442199097056676e-004 3.639706994983726e-004 -1.466638603468462e-003 2.973753096699591e-004
4.707360258540648e-004 3.366677151119628e-004 -1.431056858892947e-003 3.279658434897414e-004
5.936695566075544e-004 3.068024861279506e-004 -1.384583907525629e-003 3.560603592093903e-004
7.120849035678197e-004 2.746023049721015e-004 -1.327573436587363e-003 3.814450406003258e-004
```

Near Field Rectangular Setup

For a Near Field Rectangular setup, here are the first six lines of an exported .nfd file. The first line indicates the format of each data line, the 2nd line gives the number of frequencies exported and the 3rd line starts the block for a specific frequency.

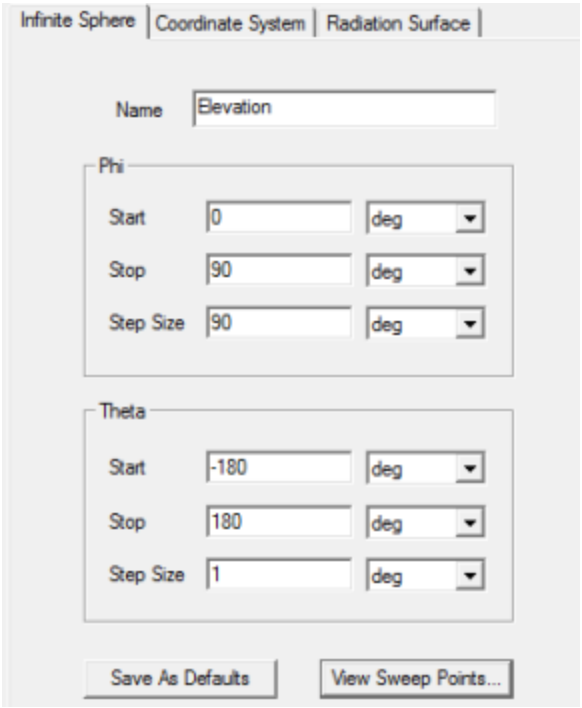
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
#Index, X, Y, Z, Ex(real, imag), Ey(real, imag), Ez(real, imag), Hx(real, imag), Hy(real, imag), Hz(real, imag)
Frequencies 1
Frequency 6.000000e+10
1, 3.250000e-04, 5.000000e-04, 1.787000e-03, 1.020152e+02, -2.134793e+02, -1.563946e+02, -6.808936e+01,
8.922556e+01, -3.982027e+02, 9.937118e-02, 1.032325e+00, 2.097039e-01, -9.222087e-01, 5.878483e-01, -1.232716e-01
2, 3.750000e-04, 5.000000e-04, 1.787000e-03, 1.102636e+02, -2.047746e+02, -1.512812e+02, -8.654425e+01,
1.108607e+02, -3.907956e+02, 3.819065e-02, 1.030528e+00, 2.156445e-01, -8.755677e-01, 5.852999e-01, -1.200639e-01
3, 4.250000e-04, 5.000000e-04, 1.787000e-03, 1.185121e+02, -1.960699e+02, -1.461171e+02, -1.049589e+02,
1.317594e+02, -3.832947e+02, -2.298989e-02, 1.028731e+00, 2.215851e-01, -8.289267e-01, 5.827515e-01, -1.168563e-01
```

For a Near Field Rectangular setup, here are two lines of a .csv file. The 1st line indicates the format of each data line, the 2nd line is a typical data line.

```
Variation, X, Y, Z, Ex(mag ang), Ey(mag ang), Ez(mag ang)
"Freq='60GHz', cd='0.75mm' cx='7.9mm' cy='7.9mm'",3.250000e-04, 5.000000e-04, 1.787000e-03, 2.366020e+02 -
1.125010e+00, 1.705738e+02 -2.730972e+00, 4.080767e+02 -1.350367e+00
```

The following Radiation Setup, combined with the .csv format, produces a different set of first lines, and a differently formatted file.



In this .csv file, the first line specifies that the following row includes variation index inclosed in double quotes, then phi value, theta value, then rEPhi (mag ang), rEtheta (mag ang).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Variation, Phi(rad), Theta(rad), rEphi(mag ang),rEtheta(mag ang)
"Freq='0.9GHz', dipole_length='14.99cm' port_gap='0.25cm' wire_rad='0.25cm'",0.000000000000000e+000,
"Freq='0.9GHz', dipole_length='14.99cm' port_gap='0.25cm' wire_rad='0.25cm'",1.570796326794897e+000,
"Freq='0.9GHz', dipole_length='14.99cm' port_gap='0.25cm' wire_rad='0.25cm'",0.000000000000000e+000,
"Freq='0.9GHz', dipole_length='14.99cm' port_gap='0.25cm' wire_rad='0.25cm'",1.570796326794897e+000,
"Freq='0.9GHz', dipole_length='14.99cm' port_gap='0.25cm' wire_rad='0.25cm'",0.000000000000000e+000,
"Freq='0.9GHz', dipole_length='14.99cm' port_gap='0.25cm' wire_rad='0.25cm'",1.570796326794897e+000,
"Freq='0.9GHz', dipole_length='14.99cm' port_gap='0.25cm' wire_rad='0.25cm'",0.000000000000000e+000,
"Freq='0.9GHz', dipole_length='14.99cm' port_gap='0.25cm' wire_rad='0.25cm'",1.570796326794897e+000,
"Freq='0.9GHz', dipole_length='14.99cm' port_gap='0.25cm' wire_rad='0.25cm'",0.000000000000000e+000,
"Freq='0.9GHz', dipole_length='14.99cm' port_gap='0.25cm' wire_rad='0.25cm'",1.570796326794897e+000,
"Freq='0.9GHz', dipole_length='14.99cm' port_gap='0.25cm' wire_rad='0.25cm'",0.000000000000000e+000,
```

Far fields format:

```
Index, Phi(rad), Theta(rad), rEPhi(mag ang), rETheta(mag ang)
```

Near fields format in Cartesian Coordinate System:

```
Index, X, Y, Z, Ex(real, imag), Ey(real, imag), Ez(real, imag), Hx
(real, imag), Hy(real, imag), Hz(real, imag)
```

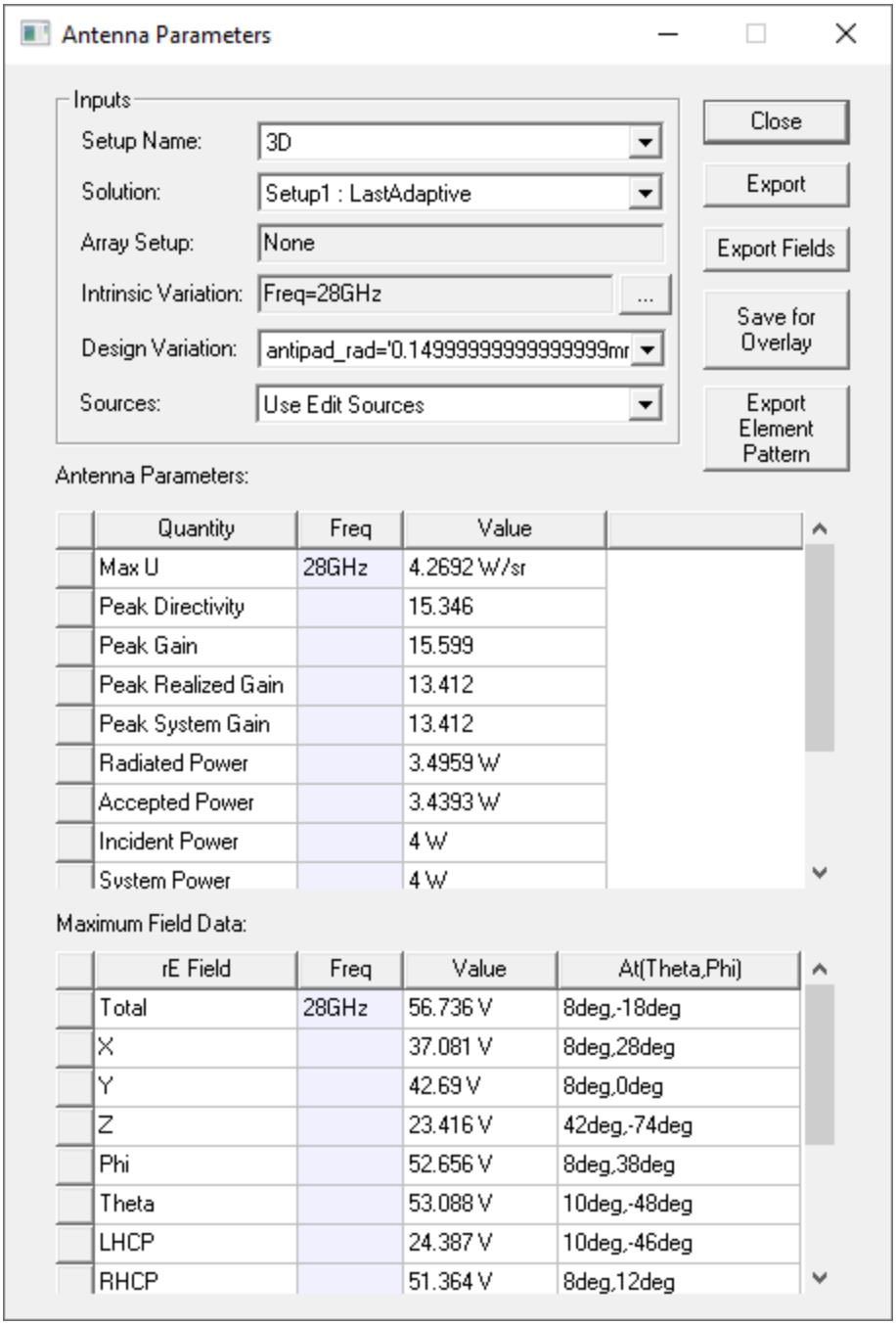
Near fields format in Spherical Coordinates System:

```
Index, Theta, Phi, Er(real, imag), Etheta(real, imag), Ephi(real,
imag), Hr(real, imag), Htheta(real, imag), Hphi(real, imag)
```

Export Element Pattern from Antenna Parameters

The **Antenna Parameters** dialog displays the calculated antenna parameters and Maximum Field data for a setup. The dialog also includes a button to **Export Element Pattern** for the far field relative to the specified inputs and sources. That is, you can specify the Setup Name, the Solution, Array Setup (if any), Intrinsic Variation, Design Variation and Sources.

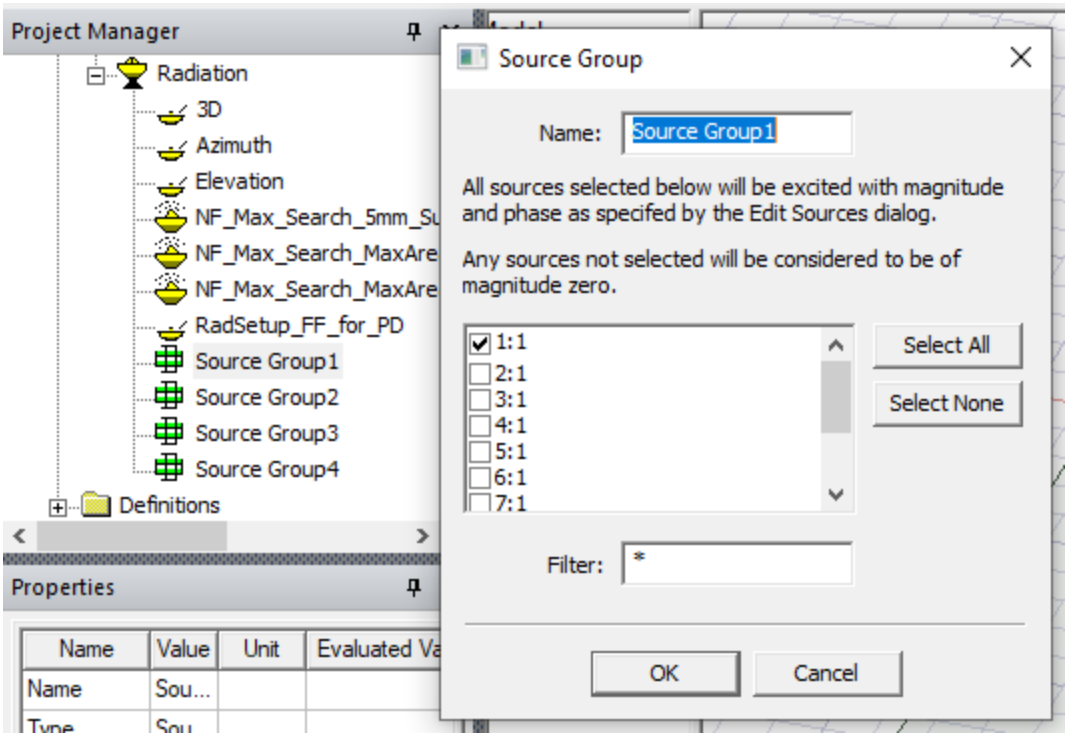
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The element pattern export functionality is based on the source group, which means only export the sources contained in the selected source group. To create Source Groups, see Setting up a Source Group. If you select “Use Edit Sources”, then the export includes the sources in Edit Sources dialog box. For each source in the source group, the export provides far field in one ffd file. And each ffd file name is based on the base file name: base file name + “_” + index, where the index is the order of the source shown in **Edit Sources** dialog box. For clarity, we also export

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

an additional txt file for user, showing the source name to its corresponding ffd file name. The fields are exported in ffd format and can be imported into Reporter as a table.



Since source groups are not supported In Transient solution type, clicking **Export Element Pattern** exports all the sources in **Edit Sources** dialog (i.e. equivalent with “Use Edit Sources”).

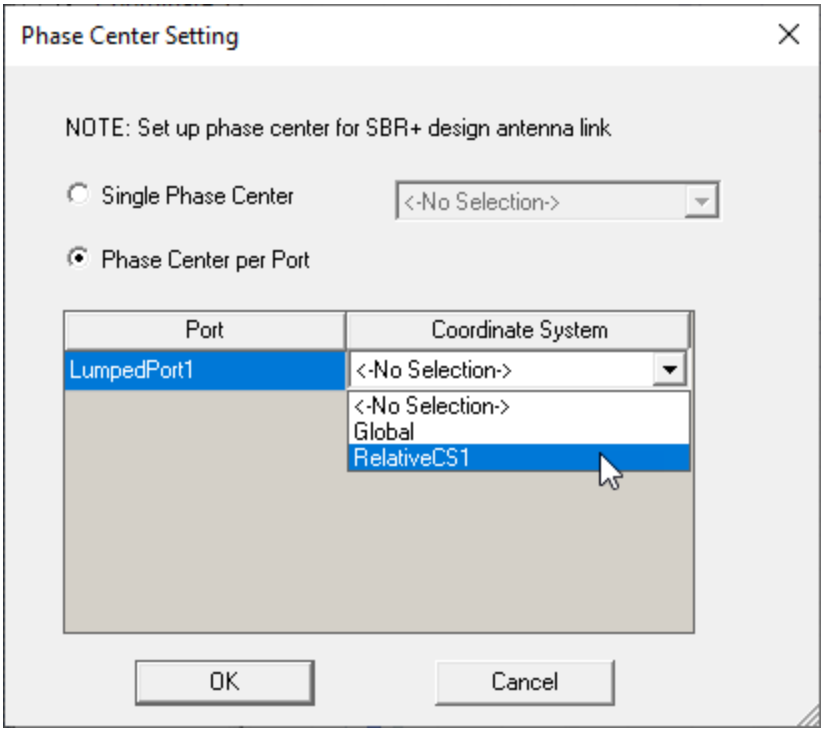
The exported far field data in ffd file is based on the conditions when activated source(s) are excited with 1.0 magnitude in SI units and 0.0 phase, and inactivated sources have 0.0 magnitude and 0.0 phase.

In Driven Terminal solution types, if a source is terminated, no far field file will be exported for it. The other exported sources are affected by the resistance and reactance of the terminated source.

Export Element Pattern respects the phase center setting, if used. (For the **Set Far Field Phase Center** command, see Link from Driven Modal Design to SBR+ Design.)

If a per-port phase center is defined, for a finite array design, the phase center coordinate system (CS) will shift according to the unit cell location (also the same to SBR+ link). If a global phase center is defined, all the ports will use that CS for export. No phase center CS shift will happen.

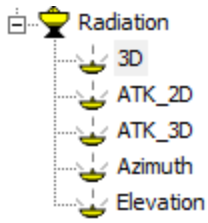
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the phase center setting is not defined, then the export will use the reference coordinate system in the infinite sphere setup, and there will not be an automatic shift for finite array design for unit cells. So basically all the ports will use the same CS for phase center.

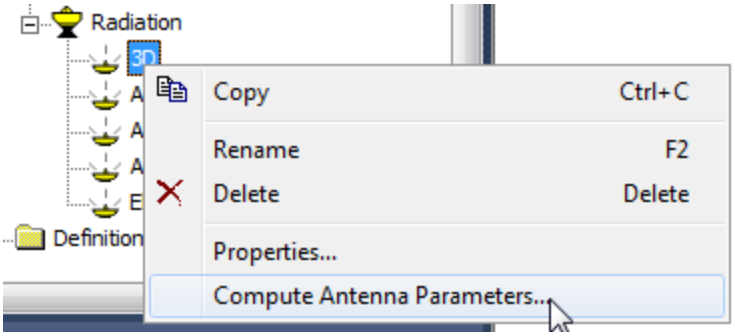
Procedure for Export Element Pattern

The Setup Name corresponds to the Radiation Setups you create, or that are created by the Antenna Design Kit, for example:

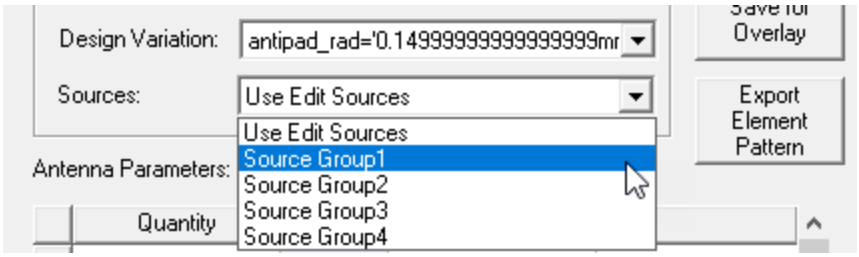


To open the **Antenna Parameters** dialog box, right-click the Radiation setup for which you want to export parameters or fields, and select **Compute Antenna Parameters** from the menu

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Select the desired source group via combo-box named as “Sources”.

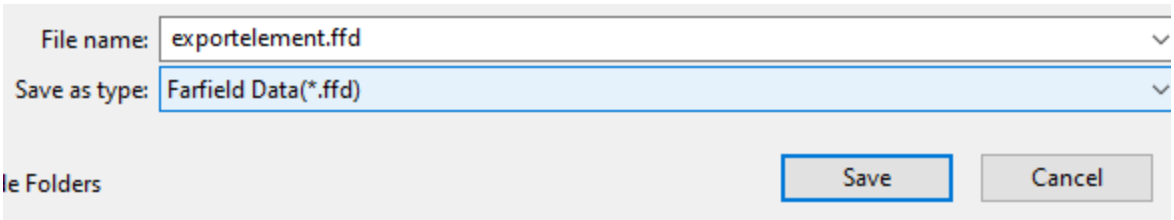


The element pattern export functionality is based on the source group, which means we only export the sources contained in the selected source group. If user selects “Use Edit Sources”, then we will export all the sources in the Edit Sources dialog.

To export the Element patterns to an .ffd text file:

1. Click **Export Element Pattern** on the **Antenna Parameters** dialog

   This displays a file browser

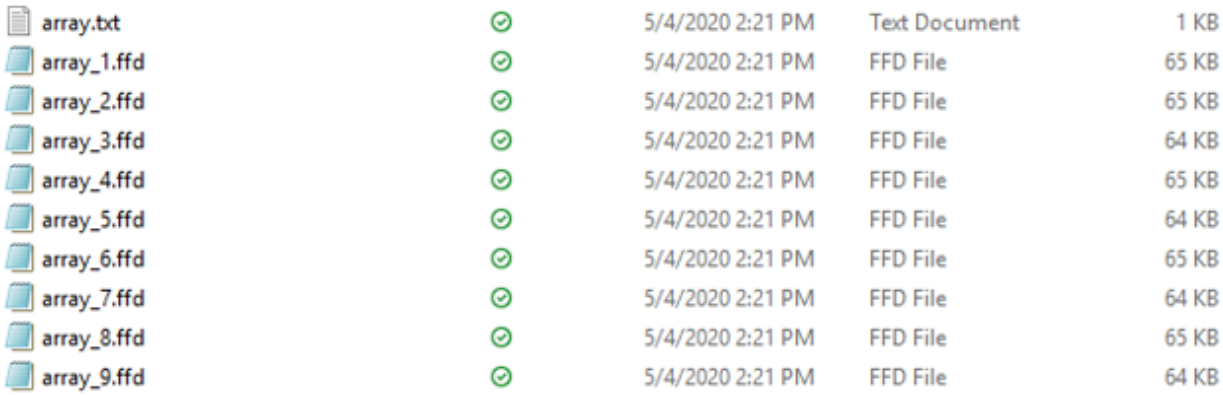


2. Specify the file name and location (or accept the defaults). Save as type is Far Field Data (*.ffd).
3. Click **Save**.

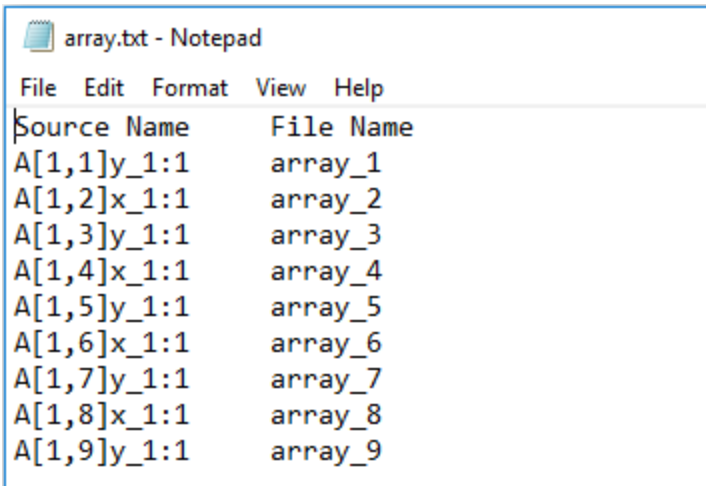
   This saves a text file for the selected Radiation setup in the specified type and closes the browser.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For each source in the source group, we export its far field in one ffd file. And each ffd file name is based on the base file name: base file name + "_" + index, where the index is the order of the source shown in Edit Sources dialog box. For clarity, we also export an additional txt file for user.



This figure shows the content of an array.txt file. It shows the source name to its corresponding ffd file name



The content in each source's ffd file is the far field data on an infinite sphere. If the infinite sphere setup is defined with the CS Definition as Theta-Phi, the first two lines are the theta, phi angle definitions (i.e. start, end angle and # of angles). Each data line contains the eTheta and ePhi in complex #. It has the same format as the far field data file produced by **Export Fields** in Antenna Parameters dialog.

```
0 180 91
-180 180 181
Frequencies 1
Frequency 5.000000000000000e+09
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
7.766368308173010e+00 9.532038867040301e+00 8.755867923644274e-01
2.563969382196918e-01
7.792194784812795e+00 9.535180330609501e+00 6.040110633865901e-01 -
7.642261087584279e-02
7.808527672402878e+00 9.526704645727785e+00 3.316994399687877e-01 -
4.091490507920261e-01
7.815347071835619e+00 9.506622138711604e+00 5.898369187776575e-02 -
7.413770056143987e-01
7.812644674723257e+00 9.474957277002284e+00 -2.138039187560649e-01 -
1.072701706755799e+00
7.800423773520401e+00 9.431748639356224e+00 -4.863310422488643e-01 -
1.402719486104554e+00
...
```

If the CS definition in the Infinite Sphere uses AzOverEl, the first line of the .ffd file has a line saying this. After that identification the next two lines are the Azimuth over Elevation angle definitions (i.e. start, end angle and # of angles). The second angle is always the outer loop, same as for theta phi.

```
AzOverEl
-90 90 91
-180 180 181
Frequencies 1
Frequency 5.000000000000000e+09
-1.015259849178660e+00 -1.655218649533607e-01 -3.522705268988268e-01
-3.529118467476278e-01
-1.002347316218049e+00 -1.531045876736399e-01 -3.874879909704635e-01
-3.584734921660798e-01
-9.882135774523947e-01 -1.405007760377924e-01 -4.222333606130519e-01
-3.635983928529357e-01
-9.728758526652468e-01 -1.277257858488285e-01 -4.564643039457966e-01
-3.682803049061392e-01
-9.563528285116635e-01 -1.147951814644569e-01 -4.901391158371188e-01
-3.725135241370467e-01
-9.386646357513984e-01 -1.017247168343128e-01 -5.232167687159368e-01
-3.762928930201028e-01
...
```

If the CS definition in the Infinite Sphere uses ElOverAz, the first line of the .ffd file has a line saying this. After that identification the next two lines are the Elevation over Azimuth angle

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

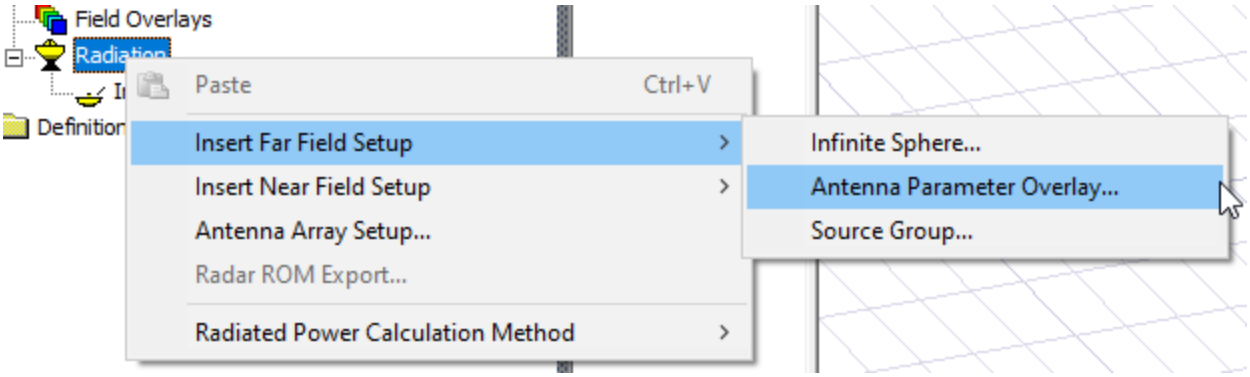
definitions (i.e. start, end angle and # of angles). The second angle is always the outer loop, same as for theta phi.

```
ElOverAz
-90 90 91
-180 180 181
Frequencies 1
Frequency 5.000000000000000e+09
-4.637399324983455e-02 2.097570360902490e+00 2.541263616056640e-01 -
4.298638962513578e-01
-3.747686134725477e-02 2.081290544083335e+00 2.555899837169427e-01 -
5.028061846738747e-01
-2.853406966199131e-02 2.062474995334444e+00 2.567422088036812e-01 -
5.751358812117218e-01
-1.955651360818052e-02 2.041146638503657e+00 2.575816330570980e-01 -
6.467648632712089e-01
-1.055513095498903e-02 2.017331458908390e+00 2.581072337680435e-01 -
7.176058619712824e-01
-1.540888500621979e-03 1.991058471676533e+00 2.583183705730141e-01 -
7.875725684672551e-01
7.475231289027627e-03 1.962359686395983e+00 2.582147862343362e-01 -
8.565797391048062e-01
```

...

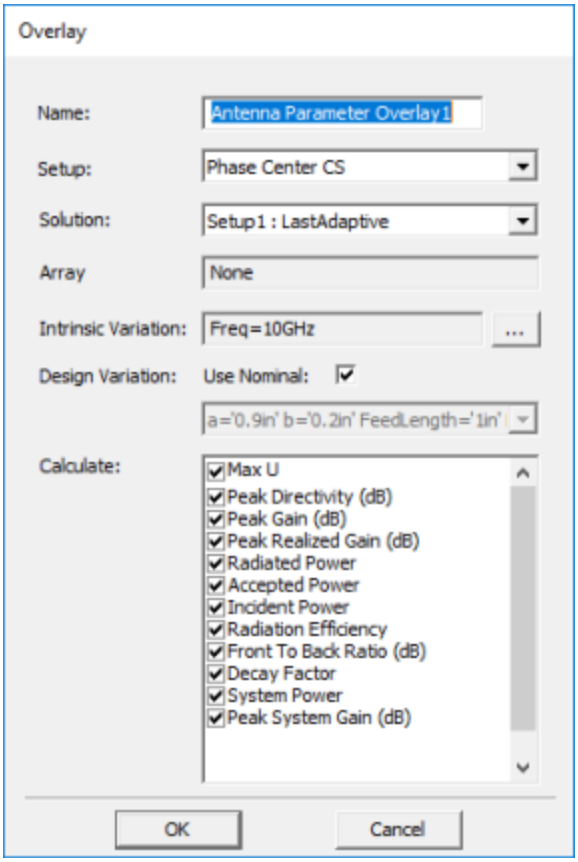
Antenna Parameters: Save for Overlay

To save antenna parameters for overlay, you can either use the Save for Overlay button in the **Antenna Parameters** dialog box, or go through the **HFSS>Radiation>Insert Far Fields Setup>Antenna Parameters Overlay** command, or right-click the Radiation icon for the shortcut menu and select **Insert Far Field Setup>Antenna Parameters Overlay.**



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

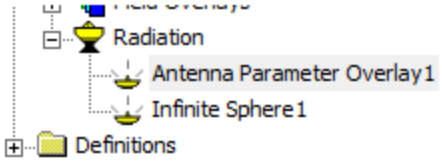
These methods display the **Overlay** dialog.



Here you can specify the Name, the Setup to use, an Array, if any, and which design variation to use. You can then check which antenna parameters to save for use in an overlay.

You can specify multiple intrinsic variations (for example, parameters at several frequencies), just as in the **Compute Antenna Parameters** dialog box. Click the ellipsis button [...] to select from available variations.

When you **OK** the dialog box, the Antenna icon appears under the Radiation icon in the Project tree.



You can rename, delete, undo, or redo just as for any other item in the Project tree.

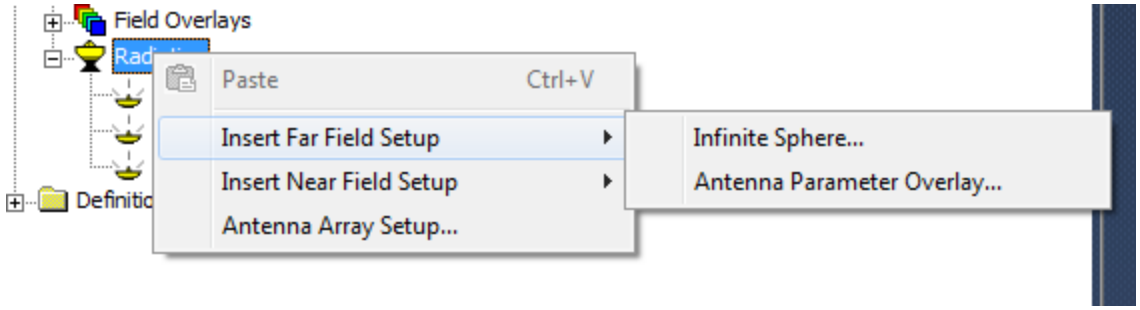
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once you have created an overlay, you can also edit the name, setup, solution, variation, and so forth, and have the changes immediately reflected in the display. To display the saved Overlay, see Creating and Displaying Antenna Parameter Overlays.

Creating and Displaying Antenna Parameter Overlays

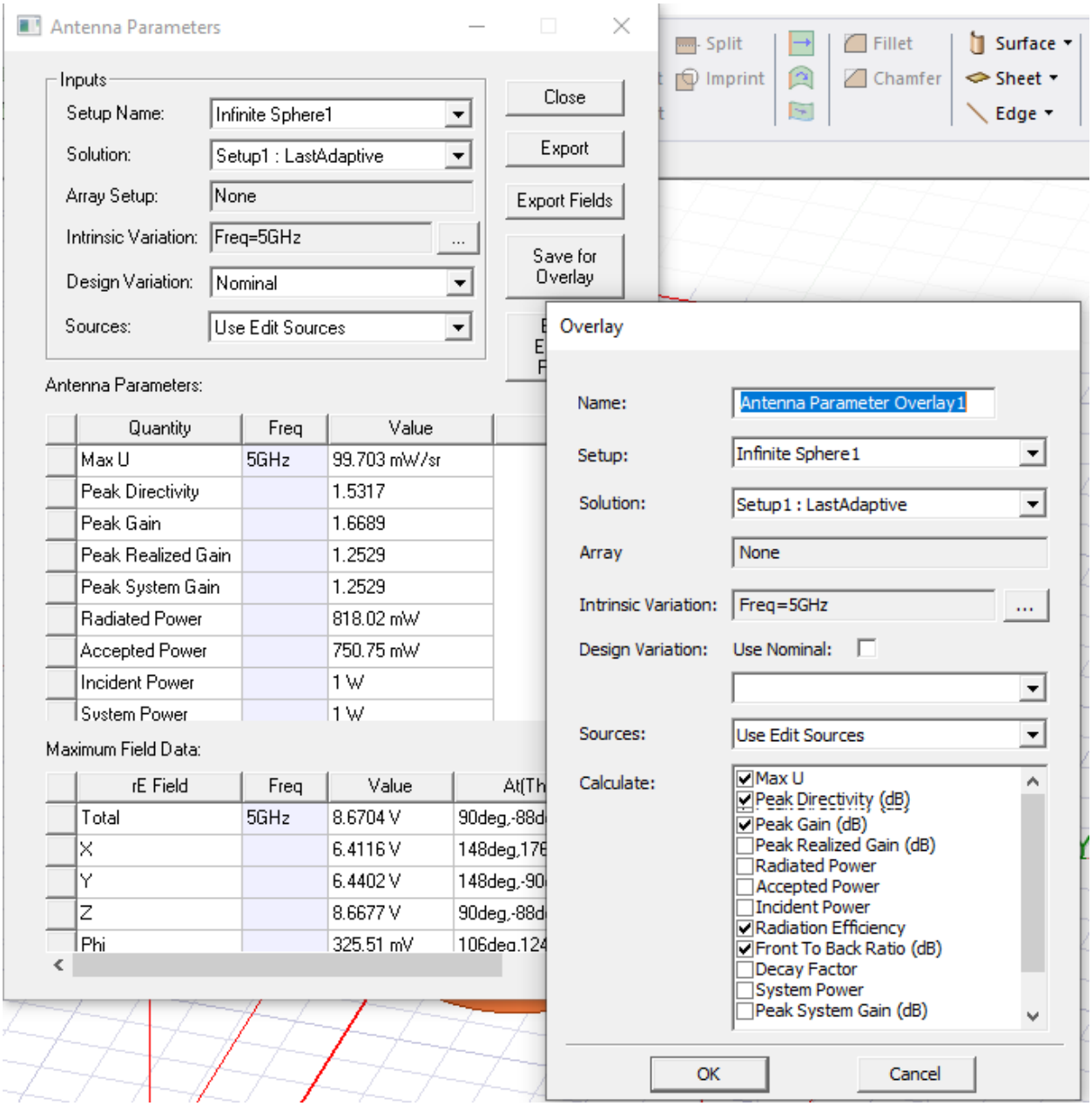
An Antenna Parameter Overlay is a table of antenna parameters that you can overlay on the 3D Modeler window for Far Field radiation plots an animations.

1. To create and display an overlay, you can either use the Save for Overlay button in the **Antenna Parameters** dialog box, or go through the **HFSS>Radiation>Insert Far Fields Setup>Antenna Parameters Overlay** command, or right-click the Radiation icon for the shortcut menu and select **Insert Far Field Setup>Antenna Parameters Overlay.**



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These methods display the **Antenna Parameters** dialog box. Click **Save for Overlay** to select which parameters to save for the overlay display.

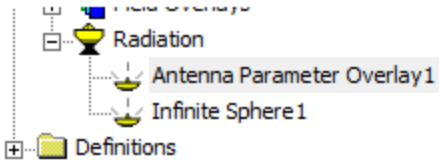


In the Antenna Parameters dialogy, you can specify the Name, the Setup to use, an Array, if any, and which design variation to use. You can then check which antenna parameters to save for use in an overlay.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can specify multiple intrinsic variations (for example, parameters at several frequencies), just as in the **Compute Antenna Parameters** dialog box. Click the ellipsis button [...] to select from available variations.

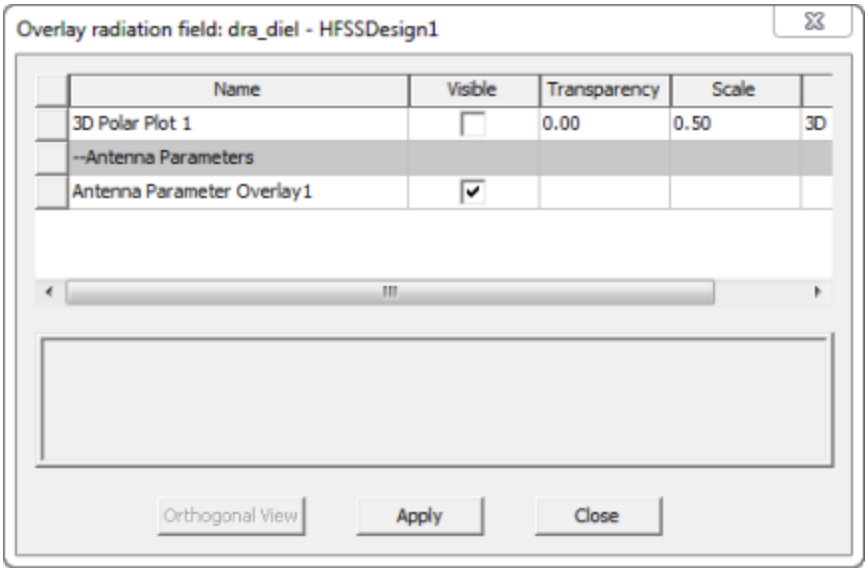
2. When you **OK** the dialog box, the Antenna icon appears under the Radiation icon in the Project tree.



You can rename, delete, undo, or redo just as for any other item in the Project tree.

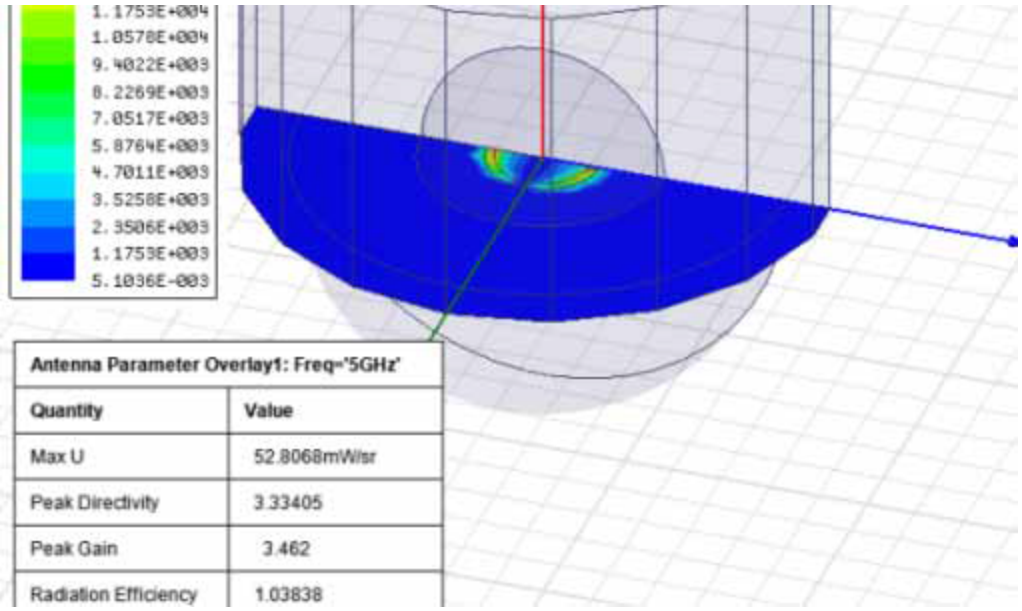
3. To display the Antenna Parameter an overlay, you either click **HFSS>Fields>Plot Fields>Radiation Fields** or right-click **Field Overlays** in the Project tree and select **Plot Fields>Radiation Field** from the short cut menu.

The Antenna parameters appear as a selection on the **Overlay radiation field** dialog.



4. Check the Visible check box and click **Apply** to cause the parameters to display as an overlay on the lower left of the 3D Modeler window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once you have created an overlay, you can also edit the name, setup, solution, variation, and so forth, and have the changes immediately reflected in the display.te and display an antenna parameters overlay:

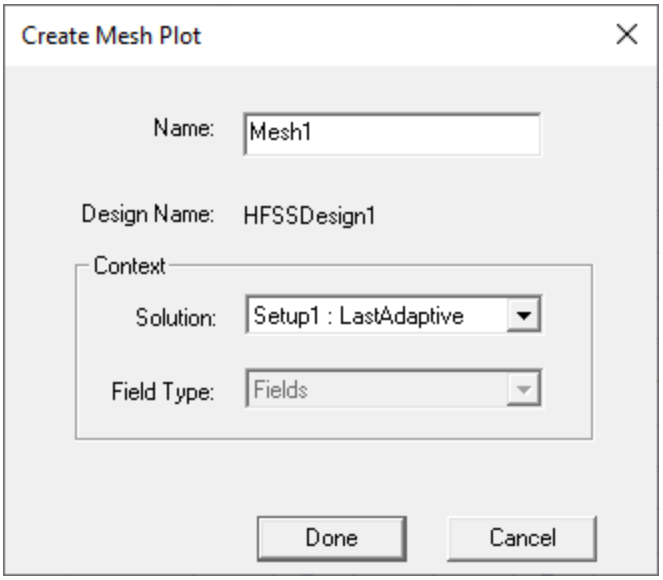
Plotting the Mesh

Before or after the solution is complete, you can plot the finite element mesh on surfaces or within 3D objects. For an HFSS or Maxwell design that includes Layout Components, you can also select nets and layers from the layout components to plot.

1. Select one or more surfaces or objects to create the mesh plot on or within. If the design is HFSS and includes Layout Components, you can select the surfaces either before or after selecting **Plot Mesh**.
2. Select **Plot Mesh** in one of two ways:
   - Click **HFSS** > **Fields**> **Plot Mesh**.
   - In the Project Manager, right-click **Field Overlays** and select **Plot Mesh...**

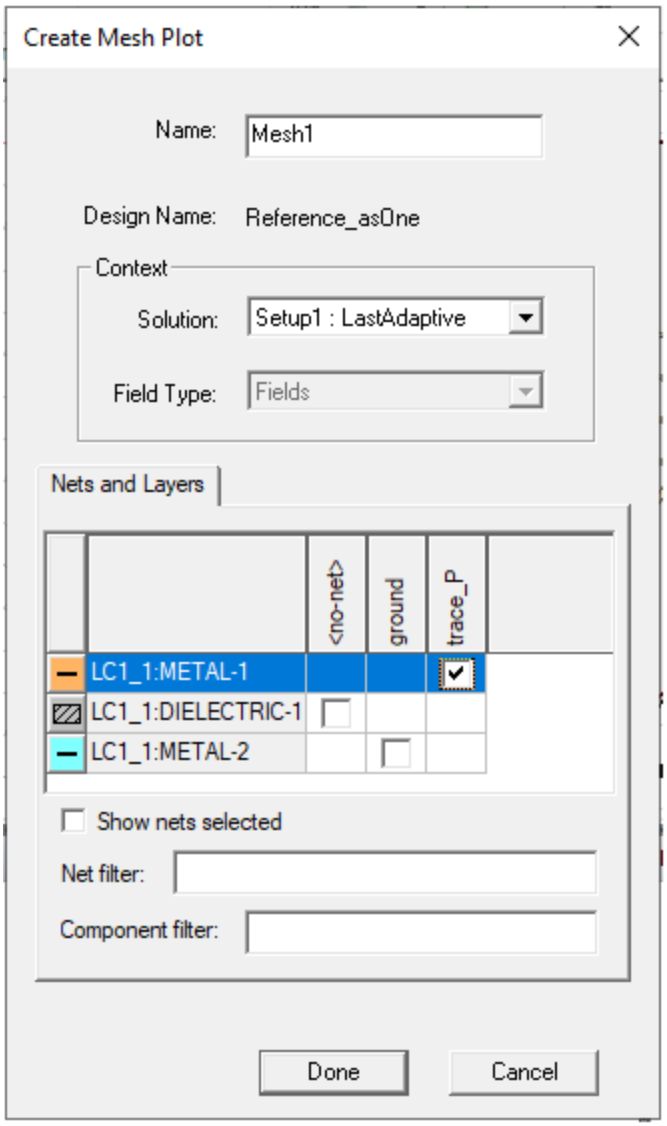
     The *Create Mesh Plot* dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the HFSS design includes Layout Components, the **Create Mesh Plot** dialog box includes a table of Nets and Layers from which you can make selections.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This **Nets and Layers** section displays all the layers and nets of every layout component in the design. Upon opening, it adjusts its size to try to show all the rows and columns for the layout components. You can resize the parent dialog. For objects in the component belonging to a specific layer and net, you can select a checkbox. By clicking the column headers, you can select or unselect checkboxes for an entire column. You can toggle the **Show nets selected** checkbox to move all the selected nets to the left side of the table. You can use the **Net filter** and **Component filter** fields to filter out specific nets and components. The net and component filters support '*' and '?' wildcards and you can delimit multiple filters with commas. The Nets and Layers table is refreshed each time you update the filter text.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

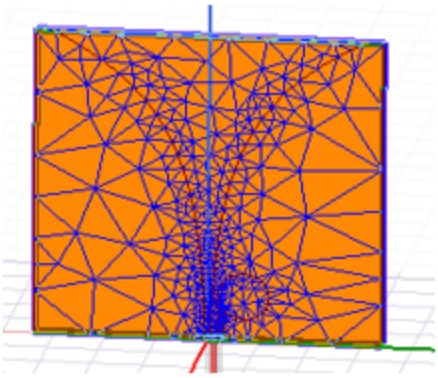
If you right click on the Nets and Layers panel, a short-cut menu appears that lets you Select This Layer Only, Select or Unselect All, Select or Unselect All Signals, or Select and Unselect All Dielectrics.

3. Select the **Solution** to use from the drop-down menu.

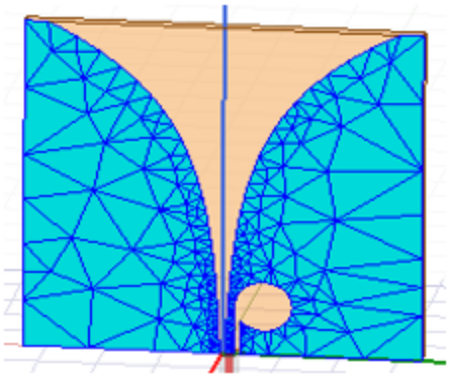
   For HFSS designs that include layout components, see a discussion with example below.

4. If **Field Type** is enabled, you can select from the drop-down menu for a mesh plot.
5. Click **Done**.

   The mesh appears on the surface or object you selected. For example:

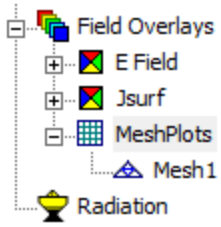


Mesh Plot of All Objects



Mesh Plot of Selected Surfaces

An icon for the mesh also appears in the Project Manager under **Field Overlays** > **Mesh Plots**.



If a solution is ongoing, you can select the **Mesh Plots** icon in the in the Project Tree, right-click to display the shortcut menu, and select **Update Plots**. This updates the mesh plot to include the latest data available. After the last adaptive pass, the mesh plot is automatically updated.

If a mesh includes seeding, these effects appear at the start of the adaptive passes. Any mesh adaptation at the start of a subsequent pass is not plotted until after that pass is completed. This delay ensures that mesh plots and actual solutions remain consistent with each other.

You can modify an existing plot by selecting the plot and changing the properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you modify the geometry of an object, the solution is invalidated and the mesh plot is removed.

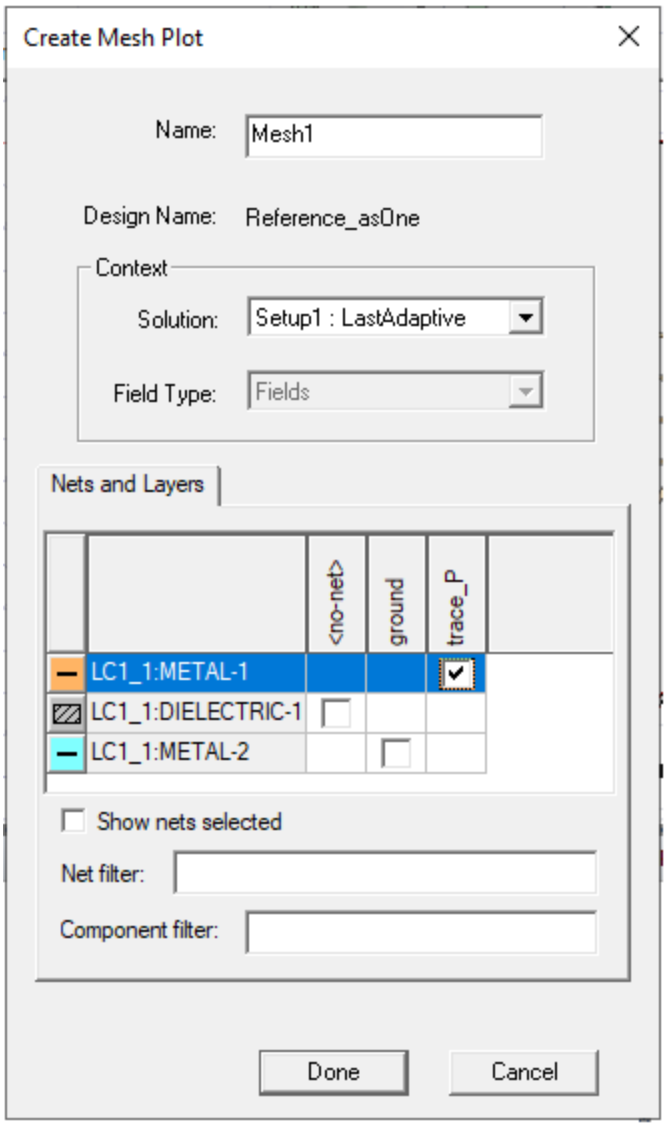
If you have plotted the mesh of a Solve-Inside object, you can view the internal mesh by using either the clip-plane feature or the Model Analysis dialog box.

Also see the discussion here, under **Visualization for Skin Depth Refinement Mesh Results**. The same techniques for visualization for internal mesh plots apply even if seeding is not used.

Example of Mesh Plots for an HFSS Design with Layout Components

First, solve the design. You can either select geometries in the Modeler window before opening the **Create Mesh Plot** dialog, or use the **Nets and Layers** area in the Create Mesh Plot dialog to select objects from the layout components to plot. These objects are represented as combinations of nets and layers and can be selected by checking the corresponding checkboxes.

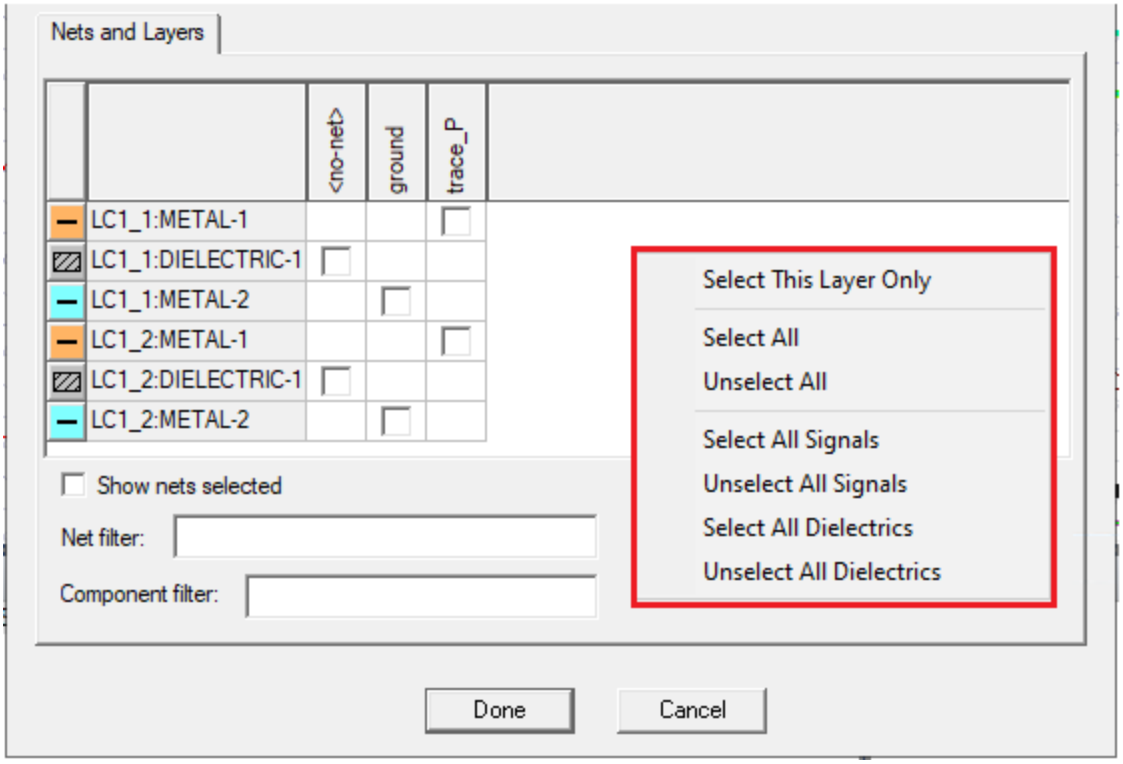
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This **Nets and Layers** section displays all the layers and nets of every layout component in the design. Upon opening, it adjusts its size to try to show all the rows and columns for the layout components. You can resize the parent dialog. For objects in the component belonging to a specific layer and net, you can select a checkbox. By clicking the column headers, you can select or un-select checkboxes for an entire column. You can toggle the “Show nets selected” checkbox to move all the selected nets to the left side of the table. You can use the **Net filter** and **Component filter** fields to filter out specific nets and components. The net and component filters support ‘*’ and ‘?’ wildcards and you can delimit multiple filters with commas. The Nets and Layers table is refreshed each time you update the filter text.

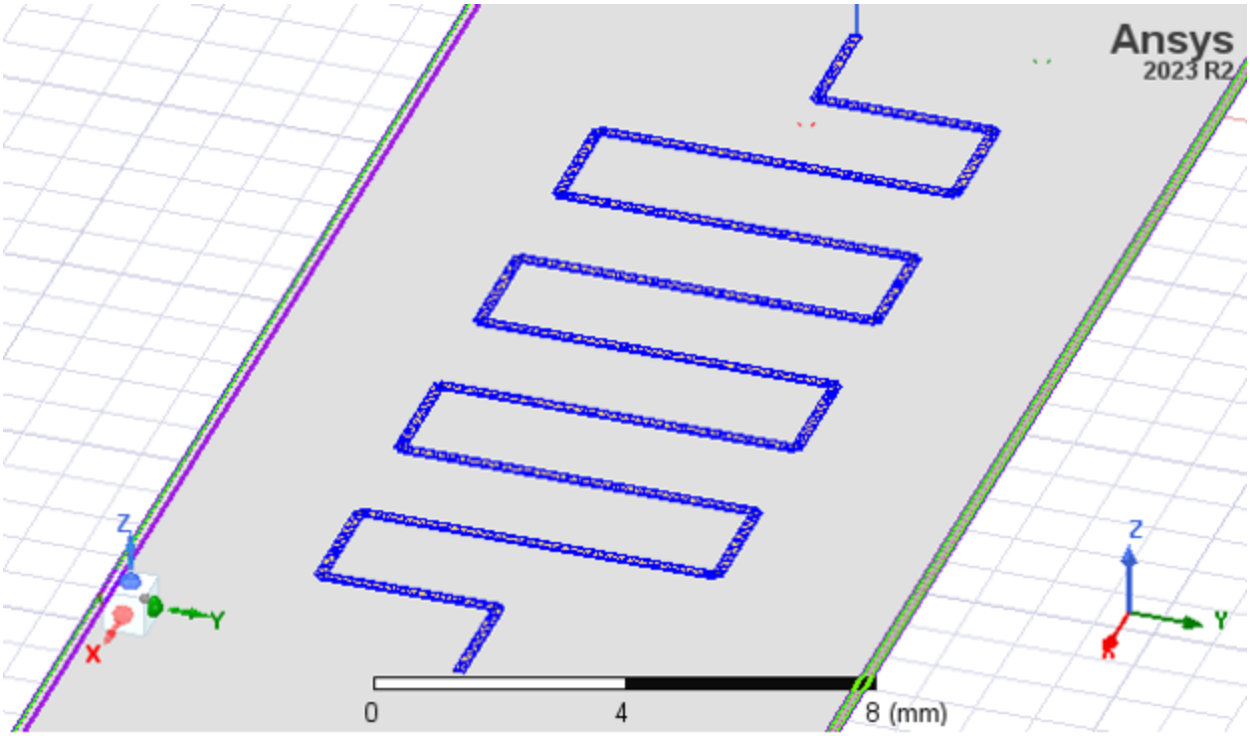
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you right click on the Nets and Layers panel, a short-cut menu appears that lets you Select This Layer Only, Select or Unselect All, Select or Unselect All Signals, or Select and Unselect All Dielectrics.

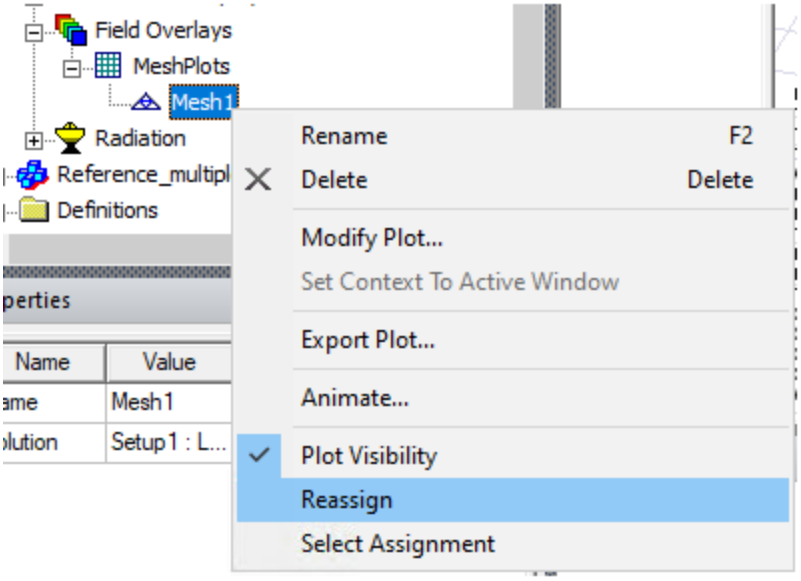


After making selections click **Done**. The mesh plot will be generated on the objects belonging to the selected layer/net combinations.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

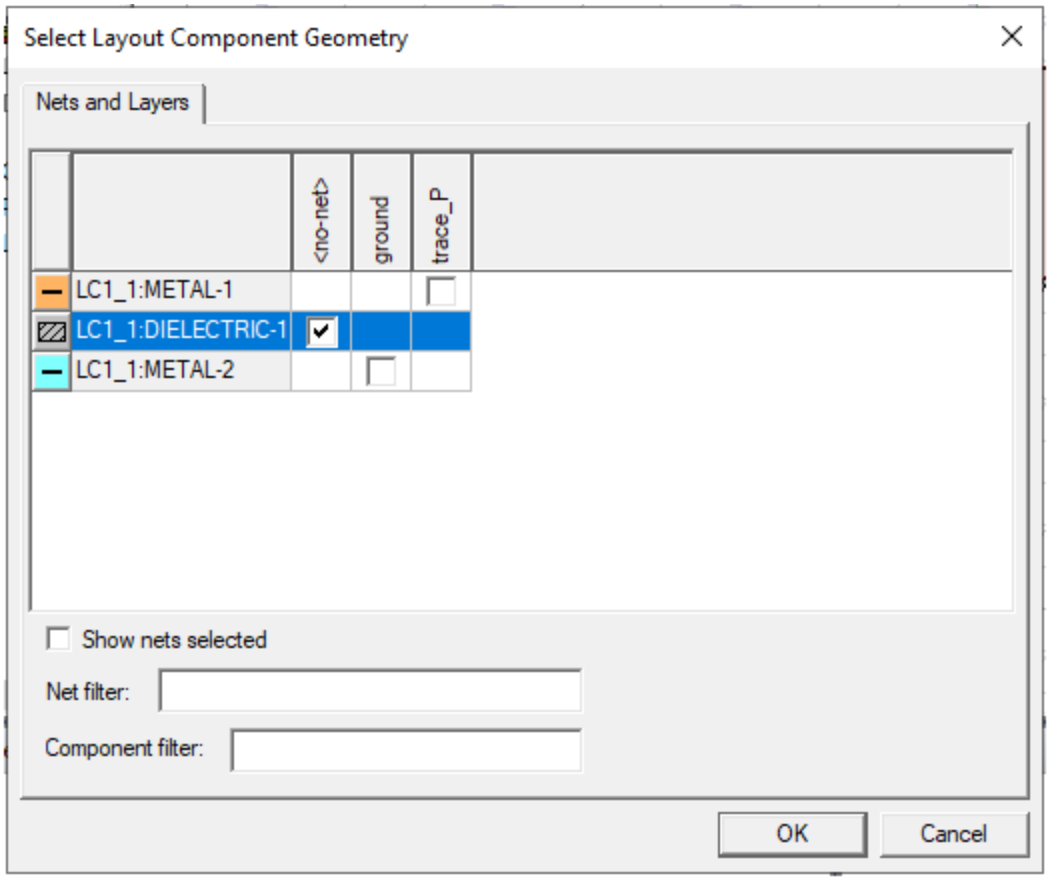


If you want to reassign a mesh or field plot, either select 3D geometries from layout components in the modeler window before selecting **Reassign** in the right-click menu, or select **Reassign**.



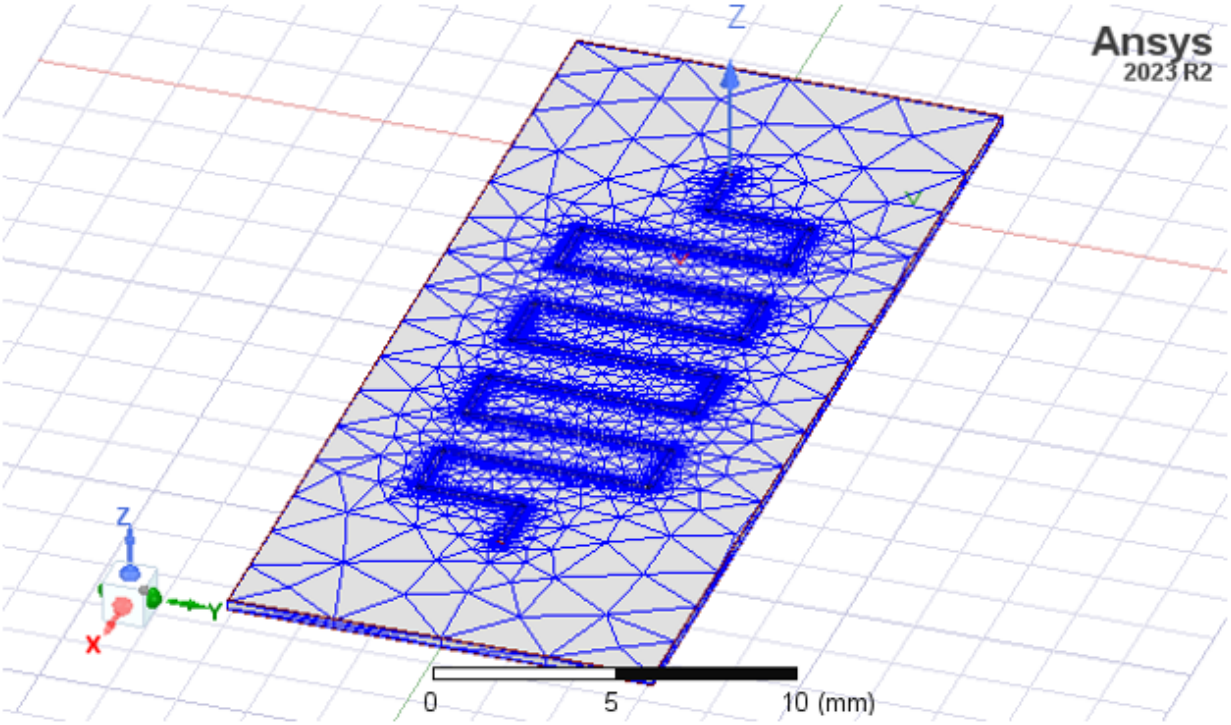
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For HFSS designs with layout components, the **Select Layout Component Geometry** dialog appears. This has the same functionality and menus as the this **Nets and Layers** section of the **Create Mesh Plot** dialog.



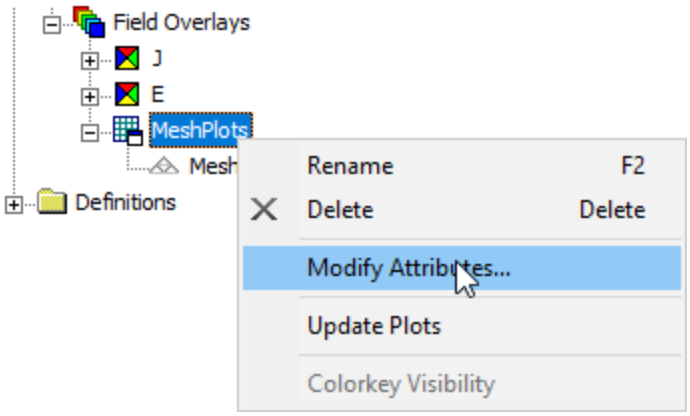
Change the selection in the dialog and click **OK**. The mesh plot will be re-assigned to the newly selected objects, in addition to any 3D geometries previously selected from the modeler window. If you click cancel, the plot is only re-assigned on 3D geometries previously selected. If there are no previously selected 3D geometries, the program issues an error message. Change the selection in the dialog and click **OK** to display the reassigned mesh plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



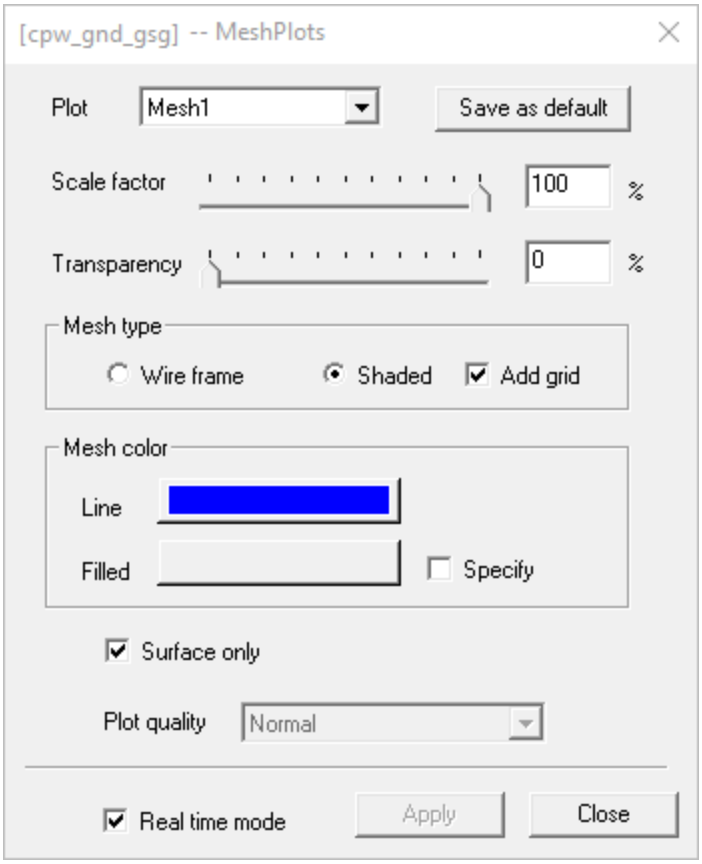
Setting Mesh Plot Attributes

1. In the Project Tree, expand **Field Overlays**.
2. Right-click **Mesh Plots** and select **Modify Attributes**.



The *MeshPlots* window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

You can also open the MeshPlots window from **HFSS** > **Fields** > **Modify Plot Attributes**. You will be prompted to select the folder containing the mesh you want to modify.

For Mesh plots, the following attributes can be modified:

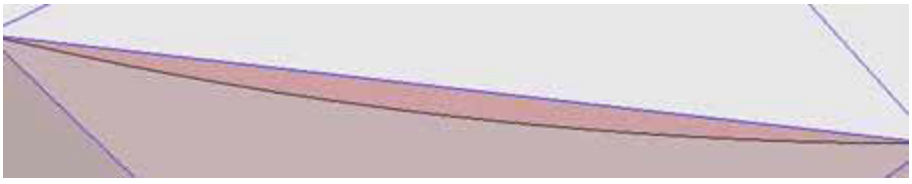
Plot	A drop-down menu of available plots.
Scale Factor	The size at which the tets are displayed. Scaling may let you analyze particular situations better. For example, a scale factor of 80% draws the tetrahedra at 80% of their original size. Use the **Scale factor** slider to increase or decrease the size of the tetrahedra.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transparency	The degree of transparency for the tets. This is useful for viewing objects or plots behind the current plot. Use the **Transparency** slider to increase or decrease plot transparency.
Mesh Type	Whether to display the tets as wire frame or shaded, and whether to Add Grid.
Mesh Color for Line and Fill	The color for the tet edge lines and fill. Clicking the button for each displays a color selection dialog box. You must select **Specify** to enable fill color.
Surface Only	Whether to plot the surface only, or all tets inside selected objects.
Plot Quality	The resolution quality of the plot. Normal by default. Use the drop-down menu to select Coarse, Normal, Fine, or Very Fine. The higher the resolution, the more memory used.
Real Time	Whether to show changes to a mesh immediately in the view window. If this option is cleared, click **Apply** when you want to see the changes.

For designs that include Stress feedback from Ansys Workbench Integration, an additional **Deformation Scale** option is available. See: Interface Changes for Stress Feedback from Mechanical to HFSS.

For **Plot Quality**, higher quality selections permit visualization of curvilinear mesh elements. With the **Coarse** setting, facets do not snap to the midpoint edge:



The **Normal** setting displays snapping to the midpoint edge:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Fine** setting displays more of the curvilinear tets:

The **Very fine** setting gives the most accurate display, but uses the most memory:

3. Click **Save as default** button if you want the tab's settings to apply to mesh plots created after this point.
4. Click **Close**.

User Defined Outputs (UDOs)

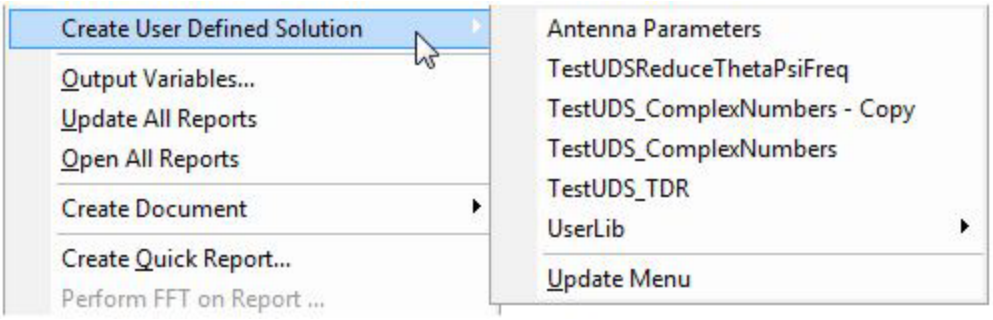
User defined outputs (UDOs) allow users to define calculations through IronPython scripts or any .NET language (and used by the IronPython script). The UDO scripts need to be in the **UserDefinedOutputs** directory under either **syslib**, **userlib**, or **Personallib** with any directory structure needed for organization. (The **Lib** directory name is special and its purpose will be explained later on in the document.)

The UDO scripts that are placed in syslib/UserDefinedOutputs, userlib/UserDefinedOutputs, or Personallib/UserDefinedOutputs become available to the user to create "User Defined Solutions" through the **Results** > **Create User Defined Solution** menu.

Use **Results** > **Create User Defined Solution** > **Update Menu** to refresh the menu to include the new UDO scripts that might have been copied to syslib, userlib, or Personallib; or exclude them if they have been deleted, after the launch of desktop. Once the user-defined-solution is

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

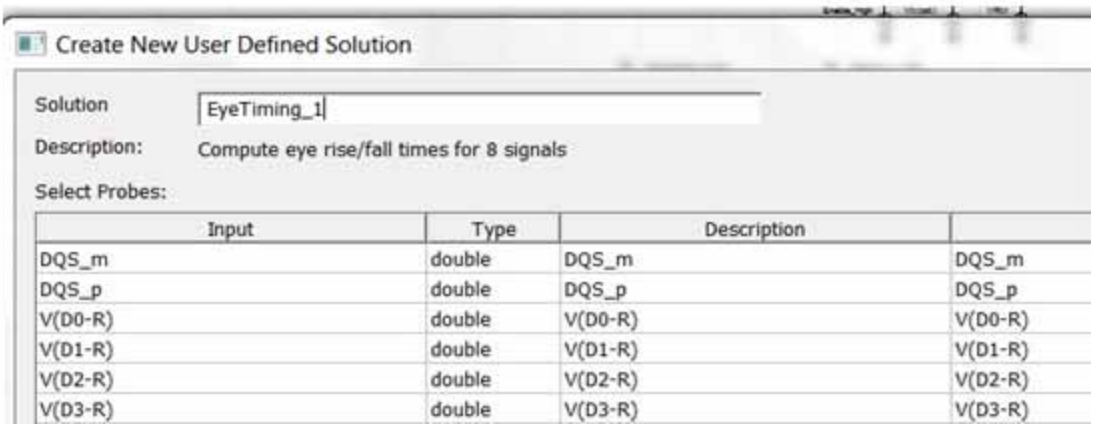
created, the solution and the calculations defined by UDO become available in Reporter as any other quantities in a new "User Defined" report type.



Named Probes and Properties in User Defined Outputs

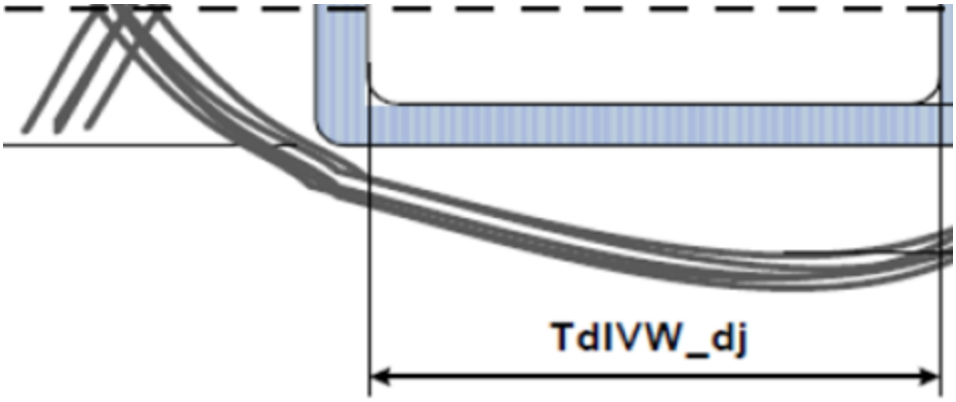
UDOs allow processing data across traces, solutions, and report types. A UDO specifies the named probes and properties for which user selects or enters the values at the time of creation of user-defined solution. Probes are very similar to traces except that the user selects the values of only intrinsic variables for probes. The values of design or project variables are selected when a trace is created based upon the user defined solution in reporter.

For example, you could create a user-defined solution called EyeTiming_1.



You can then access this solution in the Reporter.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In another example, you could create a user-defined solution called **UDS Distance Trace Arithmetic Result1**:

Then access this solution in the Create User Defined Solution menu:

Computation of Traces Based UDO Calculations

When traces that are based upon UDO outputs are computed, the data for probes is computed and passed to the UDO script for each design variation. Along with the probe data, the values of properties entered by user are also passed. The information about the UDO calculations that need to be computed is also made available. The UDO then performs the computation and passes the results to reporter. Note that UDOs can compute and pass back more calculations than have been requested at that point of time. This allows UDOs to compute a set of calculations that take almost same amount of computational resources as any one calculation in that set and cache that with reporter. *When those calculations are subsequently plotted by user, reporter will use the cached results instead of invoking the computation on UDO*.

Dimensions Reduction by UDO Calculations

The probes in a UDO can have heterogeneous dimensions of data, for example, one probe in a UDO can have data that is a function of *n* intrinsic variable, while another probe in same UDO can have data that is function of *m* intrinsic variables, with *n* and *m* are potentially different. UDOs allow you to reduce any number of these intrinsic variables. In the above example, UDO calculations can be a function of any number of intrinsic variables including not being function of any intrinsic variable at all. UDO calculations can also be a function of an intrinsic variable that none of the probes is function of. *The only restriction is that Freq cannot be reduced if any of the probes are on a Fields report type.*

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dynamic Probes

In addition to named probes and properties, UDOs can specify named dynamic probes. The difference between probes and dynamic probes is that while the end user of UDO specifies the complete trace definition for probe, the expression for dynamic probe is specified by UDO code itself and not by end user. This allows UDOs to access the data for probes without requiring the end user to enter each individual probe. For example, a UDO can access data for a huge S matrix for 100 port design without having the end user enter the probe information for each of those 10,000 quantities. Each dynamic probe is associated with a named probe that is entered by user. Information about solution, context, and intrinsic variables is used from selected probes; however multiple dynamic probes can be associated with the same user selected probe. The dynamic probes are acquired from UDOs at the time of trace computation and not at the time of creation of user defined solution.

This means that you select solution, context, and values of intrinsic variables just once, and the same information is used (in this case) for all clock and data signals. The expression for those signals comes from the UDO code.

User Defined Outputs: Python Script API

A User Defined Output (UDO) extension is implemented as an IronPython script that defines a class with a specific name: **UDOExtension**, which derives from a specific base class **IUDOPluginExtension** and implements its abstract methods.

- UDO Extension Implementation
- Data Types Used in UDO Python Scripts
- Working With Properties for UDO
- Other Application Specific Classes Used in Python Scripts
- User Defined Outputs: Messaging Methods
- Using .NET Collection Classes and Interfaces in Python Scripts

UDO Extension Implementation

The purpose, argument list, and expected return types for each of the IUDOPluginExtension abstract methods, which the UDO author is expected to implement are described below.

- Import Statements
- UDOExtension Class
- IUDOPluginExtension Abstract Class

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Import Statements

The base class to use and the types it uses in turn are contained in .NET assemblies. The use of these requires that the assemblies be imported into the UDO script. The following import statements should be added to the top of the Python script:

```
from Ansys.Ansoft.ModulePluginDotNet.Common.API import *
 from Ansys.Ansoft.ModulePluginDotNet.Common.API.Interfaces import *
 from Ansys.Ansoft.ModulePluginDotNet.UDO.API.Interfaces import *
 from Ansys.Ansoft.ModulePluginDotNet.UDO.API.Data import *
```

UDOExtension Class

The UDO itself should be implemented as an IronPython class called **UDOExtension** which *must* derive from the **IUDOPluginExtension** abstract base class (from the **Ansys.Ansoft.ModulePluginDotNet.UDO.API.Interfaces** namespace).

Note that power users could derive a class hierarchy tuned toward a specific type of UDOs and that they can derive from their own base classes. The only requirement is that directly or indirectly, the UDO class must derive from **IUDOPluginExtension**.

The UDOExtension abstract class declares the optional Validate abstract method that may be implemented in the UDOExtension class or one of its base classes.

Example:

```
def BaseClassUDO ((IUDOPluginExtension):
 #base class implementation
 …
 def UDOExtension ((BaseClassUDO):
 #UDO class implementation …
```

Validate

This optional method is used to validate the user choices. The values of the properties entered, the probes etc. can be checked for suitability. This function, while a part of the IUDOExtensionabstract class, has a meaningful default implementation and is therefore optional. However, it can be overridden to take advantage of advanced functionality.

UI Access	NA		
Parameters	Name <errorStringList>	Type List<string>	Description [out] C# list of Python strings. Should be set only if validation failed;

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

			ignored if validation is successful. One error string should be set per each validation error.
	<udsProbParams>	List<UDSProbeParams>	[in] C# list of UDSProbeParams objects.
	<propList>	IPropertyList object	[in] list of properties
	< userSelectionForDynamicProbes >	List<UDSProbeParams>	[in] C# list of UDSProbeParams objects.
Return Value	Boolean. True on validation success and false on failure. The default implementation always returns true.		

Python Syntax	Validate (<*errorList*>, <*probeList*>, <*propertiesList*>, <*dynamicProbes*>)

Python Example	`def Validate(self, errorStringList, probeList, propList, dynamicProbes):`
	`  if probeList == None or probeList.Count == 0:`
	`  errorStringList.Add("Empty probe list")`
	`  return False`
	`  return True`

IUDOPluginExtension Abstract Class

The implementation of the IUDOPluginExtension class will be described in this section using a simple UDO example that expects a single probe and reduces its dimension returning as its outputs, the max, min, and average of its input probe data. The script in its entirety will also be listed later on.

Required functions:

The IUDOPluginExtension abstract class declares the following abstract methods that must be implemented in the UDOExtension class or one of its base classes. Not implementing any of these methods will result in a run-time error and a non-functioning UDO. The UDS refers to User Defined Solution parameters.

- GetUDSName
- GetUDSDescription

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- GetUDSSweepNames
- GetCategoryNames
- GetQuantityNames
- GetQuantityInfo
- GetInputUDSParams
- GetDynamicProbes
- Compute

GetUDSName

Return a string that is used as a prefix for all solution instances created using this UDO.

UI Access	NA
Parameters	None
Return Value	String

Python Syntax	GetUDSName

Python Example	`def GetUDSName(self):`
	`  return "MinMaxAvg"`
	`udsName = UDOPluginEntention.GetUDSName`

GetUDSDescription

Returns a description for the UDO, its purpose, etc. This is used in multiple UDO related dialogs in the application to describe the UDO.

UI Access	NA
Parameters	None
Return Value	String

Python Syntax	GetUDSDescription
Python Example	`def GetUDSDescription(self):` `  return "Sample UDO for dimension reducing quantities`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GetUDSSweepNames

Returns a list of sweep names to be used for the solution generated by the UDO. These will appear in the sweeps list displayed in the standard reporter dialog when used to create reports from the solution generated by the UDO.

UI Access	NA
Parameters	None
Return Value	List of strings. If the UDO outputs have no sweeps, return the empty list [].

Python Syntax	GetUDSSweepNames

Python Example	`# Returns list of sweeps names`
	`# We have no sweeps as we reduce them.`
	`def GetUDSSweepNames(self):`
	`    return []`

GetCategoryNames

The outputs that the UDO solution provides or generates can be classified into multiple categories, like how the application is displayed in the report creation dialog box.

UI Access	The output will be listed in the categories box in the dialog when creating reports from the UDO generated solution data.
Parameters	None
Return Value	List of strings

Python Syntax	GetCategoryNames
Python Example	`def GetCategoryNames(self):` `    return ["UDOOutputs"]`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GetQuantityNames

For each of the category names returned from the GetCategoryNames method, this function is called to return a list of quantities to be organized under that category name.

> **Note:**
>
> The quantity names must be unique across the categories; that is, no two categories can have quantities with the same name.

UI Access	NA		
Parameters	Name <*categoryName*>	Type String	Description [in] Name of category.
Return Value	List of strings		

Python Syntax	GetQuantityNames(<*categoryName*>)

Python Example	`# returns a list of quantity names for the supplied category name`
	`def GetQuantityNames(self, catName):`
	`  if catName == "UDOOutputs":`
	`  return ["min_val", "max_val", "avg_val"]`
	`  else:`
	`  return []`

GetQuantityInfo

For each quantity that the UDO creates, it must also describe the quantity (unit and other details). This method is called for each quantity name (across all categories) as returned from an earlier call of the GetQuantityNames method.

UI Access	NA		
Parameters	Name <*quantityName*>	Type String	Description Name of quantity
Return Value	Object of type QuantityInfo.		

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Syntax	GetQuantityInfo(<*quantityName*>)

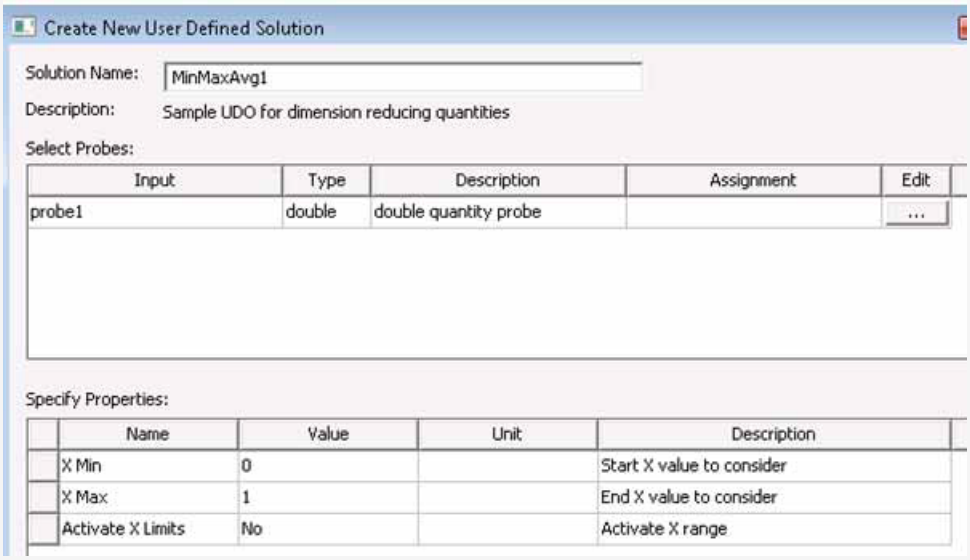
Python Example	`# Returns an instance of QuantityInfo for the qtyName supplied or None if such a`
	`# quantity could not be found`
	`def GetQuantityInfo(self, qtyName):`
	`    # All the quantities we have are simple doubles`
	`    # we can leave them unitless`
	`    return QuantityInfo(Constants.kDoubleParamStr)`

GetInputUDSParams

This is the main definition method of the UDO. The supplied arguments populate details of the parameters to which the UDO user will specify value, specify the probe names and their types as well as the dynamic probe selections.

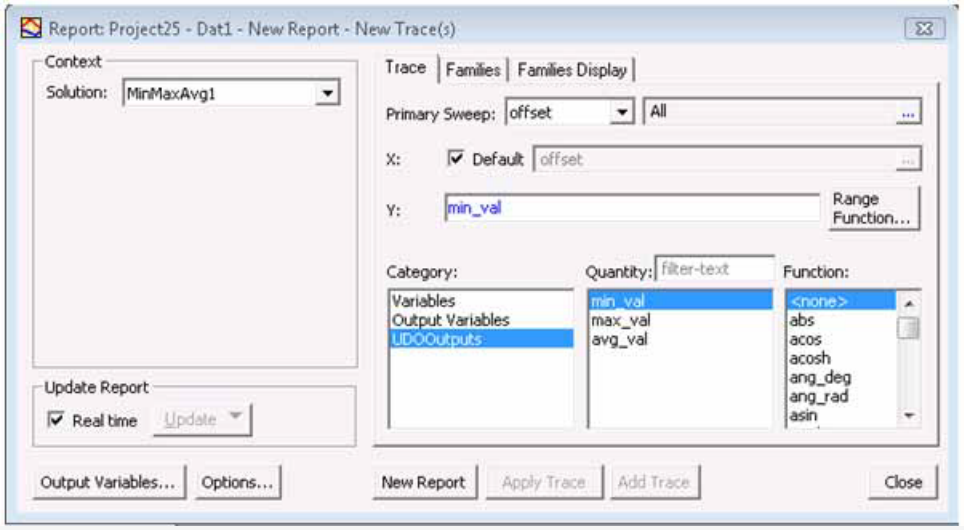
UI Access

The **GetInputUDSParams** function results in the following dialog when you click **Results**>**Create User Defined Solution**. The mapping from the UDSParams and the properties to the GUI elements should be unambiguous. The name and description of the UDS are also displayed in the **Create New User Defined Solution** window.



When a report is created from the UDO dialog box, the category and quantity names specified by the UDO are used (as seen below).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Parameters

Name	Type	Description
<udsParams>	List< UDSProbeParams >	Parameters for the probe. The UDO script must add one instance of UDSProbeParams for each probe definition it will display. When creating the UDO solution, the UDO user must assign a matching quantity to each probe. When user define a UDS in the GUI, the user must select a design-solution-quantity for each UDSProbeParam .
<propList>	IPropertyList	The propList object is used to add properties that should be displayed to the user for data collection. These properties with their user-supplied values are returned to the UDO script by the Compute method.
<	List<	When user defines a

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	userSelectionForDynamicProbes >	UDSProbeParams >	UDS in the GUI, the user ONLY needs to specify the Design-Solution. This quantity will be assigned dynamically by matching the component-expression defined in each UDSProbeParam.
Return Value	Boolean. If true, the event was handled successfully. If false, it was not.		

Python Syntax	GetInputUDSParams(<*udsParams*>, <*propList*>, <*userSelectionForDynamicProbes*>)

Python Example	`# Returns list of UDSParams and list of dynamic properties`
	`# Adds setup time properties to the propList`
	`def GetInputUDSParams(self, udsParams, propList, userSelectedDynamicProbes):`
	
	`  # Add the probes. We need only one double quantity`
	`  param1 = UDSProbeParams("probe1", "double quantity probe", Constants.kDoubleParamStr, "", "")`
	`  udsParams.Add(param1)`
	
	`  # Add the properties we want the user to supply`
	`  # In this case, we will ask for a start/end range for`
	`  # X parameters. Since we cannot reasonably provide defaults`
	`  # as we have no idea what the sweep limits will be, we will`
	`  # also ask if the limits are to be activated.`
	`  prop = propList.AddNumberProperty("X Min", "0")`
	`  prop.Description = "Start X value to consider"`
	
	`  prop = propList.AddNumberProperty("X Max", "1")`
	`  prop.Description = "End X value to consider"`
	
	`  # For menus, the first option is the default.`
	`  prop = propList.AddMenuProperty("Activate X Limits", ["No", "Yes"])`
	`  prop.Description = "Activate X range"`
	
	`  return True`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GetDynamicProbes

This is the primary mechanism by which the UDO script obtains the probe data (as double precision values) for its compute process.

UI Access	NA		
Parameters	Name <*dynamicProbes*>	Type List< UDSDynamicProbes >	Description [out] List of dynamic probes
Return Value	Boolean. If true, the method was successful. If false, it was not.		

Python Syntax	GetDynamicProbes(<*dynamicProbes*>)

Python Example	`# Returns list of UDSParams and list of dynamic properties`
	`# output UDSDynamicProbeCollection probes`
	`  def GetDynamicProbes(self, probes):`
	`  pass`

Compute

This is the main computation method which generates the data for the quantities that make up the UDO solution.

UI Access	The data is received from the UI using IUDSInputData. It is processed and the result data is sent to the UI using IUDSOutputData.		
Parameters	Name <*inData*> <*outData*> <*propList*> <*progressMonitor*>	Type IUDSInputData IUDSOutputData IPropertyList IProgressMonitor	Description Used to get the input probe data. Used to set the UDO solution quantity and sweep data. Used to get the user entered values for each of the properties defined during the GetInputUDSParams call. This can be used to set progress for long running calculations, check for user initiated abort etc.
Return Value	Boolean. If true, the method was successful. If false, it was not.		

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Syntax	Compute (<*inData*>, <*outData*>, <*propList*>, <*progressMonitor*>)

Python Example	`# IUserDefinedSolutionHandle API implementation.`
	`# Calculates output values and sets them using IUDSInputData/IUDSOutputData API.`
	`def Compute(self, inData, outData, propList, progMon):`
	
	`  # Get the sweeps associated with the probe and validate`
	`  # use the probe name that we had defined earlier`
	`  sweeps = inData.GetSweepNamesForProbe("probe1")`
	`  if (sweeps == None or sweeps.Count > 1):`
	`  AddErrorMessage(self.GetName() + "Unexpected sweep count 0 or > 1 in Compute")`
	`  return False`
	
	`  # Get the data associated with our probe`
	`  probeData = inData.GetDoubleProbeData("probe1")`
	`  sweepData = inData.GetSweepsDataForProbe("probe1", sweeps[0])`
	
	`  # Get the user specified properties.`
	`  # Note that ideally, these "X Min" etc names should be`
	`  # written as constant members and referred to in both`
	`  # the GetInputUDSParams and in Compute to reduce the`
	`  # change of typos.`
	`  useXRangeProp = propList.GetMenuProperty("Activate X Limits").SelectedMenuChoice`
	`  xRangeStart = propList.GetNumberProperty("X Min").ValueSI`
	`  xRangeEnd = propList.GetNumberProperty("X Max").ValueSI`
	
	`  # At this stage, one can look at the`
	`  # RequestedQuantities and create a dictionary to later`
	`  # check against. However, I am simply computing all`
	`  # the quantities.`
	`  minVal = 0`
	`  maxVal = 0`
	`  avgVal = 0`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
# Check if we need to perform range computation
if useXRangeProp == "Yes":
  seenAny = False
  avgSum = 0
  count = 0

# zip is used since we also need to pull in sweep data
# an index and the array notation could also have been
# used
for probeVal, sweepVal in zip(probeData, sweepData):
  if sweepVal < xRangeStart or sweepVal > xRangeEnd:
    pass

# Note that in a better written script, this code
# would be refactored into its own function to
# avoid code duplication
if not seenAny:
  minVal = probeVal
  maxVal = probeVal
  avgSum = probeVal
  seenAny = True
  count = 1
else:
  if probeVal < minVal:
    minVal = probeVal

  if probeVal > maxVal:
    maxVal = probeVal

  avgSum += probeVal
  count += 1

if seenAny:
  avgVal = avgSum/count

else:
  seenAny = False
  avgSum = 0
  for probeVal in probeData:
    if not seenAny:
      minVal = probeVal
      maxVal = probeVal
      avgSum = probeVal
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Data Types Used in UDO Python Scripts

There are several types that you must use while authoring a Python script. Some of them are used to pass data from the UI to a Python script and to provide interface for working with this data. Some are used to pass data from a Python script to the UI.

To pass data from a Python script to the UI, the objects must be created in the Python script. Then they can be set as a function's return values or set to the output parameters using their API.

Constants

- **kTraceTypeStr** : string constant used to specify an input of trace type
- **kSolutionTypeStr** : string constant used to specify an input of solution type
- **kNumberTypeStr** : string constant used to specify an input of number type
- **kTextTypeStr** : string constant used to specify an input of text type
- **kBoolTypeStr** : string constant used to specify an input of boolean type
- **kStandardReportStr** : string constant to specify a standard report
- **kEyeDiagramReportStr** : string constant to specify an eye diagram report
- **kUserDefinedReportStr** : string constant to specify a user defined report
- **kSweepDomainStr** : string constant to specify the sweep domain
- **kTimeDomainStr** : string constant to specify the time domain

Abstract Classes

- IProgressMonitor
- IUDSInput
- IUDSOutput

IUDSInputData

The purpose of this class is to get data (probe and sweep) from Desktop. The IUDSInputData abstract class declares the following abstract methods:

- GetDoubleProbeData
- GetSweepsDataForProbe
- GetComplexProbeData
- GetSweepNamesForProbe
- GetRequiredQuantities

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- GetVariableValues
- GetInterpolationOrdersData

Examples in this section are just to show proper syntax of the function calls. For actual usage of the class, see the Compute function's example.

GetDoubleProbeData

This is the primary mechanism by which the UDO script obtains the probe data (as double precision values) for its compute process.

UI Access	NA		
Parameters	Name *<probeName>*	Type String	Description Name of the probe for which data is requested. This is one of the many probes supplied during a call to the UDO's GetInputUDSParams method. Each probeName must be unique within an UDO.
Return Value	Double array of data for the specified probe if the probe exists or null if the probe is unknown.		

Python Syntax	GetDoubleProbeData (*<probeName>*)
Python Example	`# doubleData is a list of floats` `doubleData = inData.GetDoubleProbeData("probe1")`

GetSweepsDataForProbe

All probe data that is supplied is associated with one or more sweep (an intrinsic quantity like Time, Frequency, Theta, Phi, etc. that is swept) quantities.

UI Access	NA		
Parameters	Name *<probeName>* *<sweepName>*	Type String String	Description Name of the probe for which data is requested. This is one of the many probes supplied during a call to the UDO's GetInputUDSParams method. Each probeName must be unique within an UDO. Name of the sweep.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Return Value	Double array of data for the specified probe and sweep.

Python Syntax	GetSweepsDataForProbe(<*probeName*>, <*sweepName*>)
Python Example	`# sweepData is C# Array of doubles (floats in python)` `sweepData = inData.GetSweepsDataForProbe ("FarFieldsProbe","Freq"])`

GetComplexProbeData

The primary mechanism by which the UDO retrieves data for its input probes (if it expects complex data for the probe).

UI Access	NA		
Parameters	Name <*probeName*>	Type String	Description Name of the probe for which data is requested. This is one of the many probes supplied during a call to the UDO's GetInputUDSParams method. Each probeName must be unique within an UDO.
Return Value	Double array (float in Python) of data for the specified probe. Each pair of floats represents one complex number. The first value is for real part and the second value it the imaginary part. For instance, array [10.0, 0, 5.1, 2.1] represents 2 complex numbers: (10.0, 0) and (5.1, 2.1).		

Python Syntax	GetComplexProbeData(<*probeName*>)

Python Example	`# complexDataAsDouble is C# Array of doubles (floats in Python)`
	`# each pair of floats represents one complex number`
	`complexDataAsDouble = inData.GetComplexProbeData ("FarFieldsProbe")`
	`# creating a list of complex numbers from complexDataAsDouble array`
	`complexData = []`
	`if complexDataAsDouble != None:`
	`  for i in xrange(0,complexDataAsDouble.Count, 2):`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	`complexData.append(complex(complexDataAsDouble [i],complexDataAsDouble[i+1]))`

GetSweepNamesForProbe

Retrieves a list of sweep quantity names associated with a given probe. This also indicates the dimensionality of the data. One name implies that the probe-data is 2D (probe-quantity vs Sweep Quantity) and two names implies 3D data (probe-quantity vs Sweep 1 X Sweep 2).

UI Access	NA		
Parameters	Name <*probeName*>	Type String	Description Name of the probe for which data is requested. This is one of the many probes supplied during a call to the UDO's GetInputUDSParams method. Each probeName must be unique within an UDO.
Return Value	IList<string> - list of sweep names for the current probe name.		

Python Syntax	GetSweepNamesForProbe(<*probeName*>)
Python Example	`# sweepNames is C# Array of strings` `sweepNames = inData.GetSweepNamesForProbe("FarFieldsProbe")`

GetRequiredQuantities

A given UDO can specify that it provides one or more computed quantities. The user might choose to create a report from only a few among the various available UDO outputs. This function returns the list of UDO output quantities that the user requested. Only these need be computed in the UDO's Compute method.

UI Access	NA
Parameters	None
Return Value	IList<string>

Python Syntax	GetRequiredQuantities()

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Example	`# quantities is C# Array of strings` `quantities = inData.GetRequiredQuantities()`

GetVariableValues

This allows the UDO to obtain the names and values of all the design variables for which the UDO quantities are being requested.

UI Access	NA
Parameters	None
Return Value	IDictionary<string,string> of key-value pairs for variables. Both key and value are strings.

Python Syntax	GetVariableValues()

Python Example	`# theDict is C# Dictionary<string, string>`
	`theDict = inData.GetVariableValues()`
	`if theDict != None:`
	`  #varPair is of .Net KeyValuePair type`
	`  for varPair in theDict:`
	`  varName = varPair.Key #string`
	`  varValue = varPair.Value #string`

GetInterpolationOrdersData

Returns the interpolation orders that are associated with the probe-data. The probe data is specified at each value of the various sweeps. Any value in between the sweep data points, can use the interpolation data to get a possibly more accurate (compared to linear interpolation) inter-sweep value.

UI Access	NA		
Parameters	Name *<probeName>*	Type String	Description Name of the probe for which data is requested. This is one of the many probes supplied during a call to the UDO's GetInputUDSParams method. Each probeName must be unique within an UDO.
Return Value	Byte array of interpolation order for the specified probe. These are to be		

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	treated as 8-bit signed integers, that is, their values range from 0-127.

Python Syntax	GetInterpolationOrdersData(<*probeName*>)

Python Example	`# interData is C# Array of bytes (integers in Python)`
	`interData = inData.GetInterpolationOrdersData(kProbeNames[0])`
	`for interValue in theDict:`
	`  # interValue and order are integers`
	`  order = interValue`

IUDSOutputData

This type is a twin of the IUDSInputData in that it is used to store the values computed by the UDO's Compute method. The IUDSOutputData abstract class declares the following abstract methods:

- SetSweepsData
- SetDoubleQuantityData
- SetComplexQuantityData

Examples in this section are just to show proper syntax is function calls. For actual usage of the class, see the Compute function example.

SetSweepsData

Each quantity that is computed by the UDO can be associated with a sweep. If it is, the values that make up the sweep's data points must be specified using this call.

UI Access	NA		
Parameters	Name	Type	Description
	<*sweepName* >	String	Name of the sweep
	<*sweepData*>	List<floats>	Sweep data for the specified sweep.
Return Value	Boolean. If true, the event was handled successfully. If false, it was not.		

Python Syntax	SetSweepsData (<*sweepName*>,<*sweepData*>)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Example	`sweepList = [12.3, 14.5, 16.7]` `outData.SetSweepsData("Freq", sweepList)`

SetDoubleQuantityData

This method is used to record the computed quantity data for each output that is computed. Please note that unless all the sweeps are reduced, this should be used in conjunction with SetSweepsData.

UI Access	NA		
Parameters	Name <*qtyName*> <*qtyData*>	Type String List<Floats>	Description Name of the quantity. Quantity data for the specified quantity.
Return Value	Boolean. If true, the event was handled successfully. If false, it was not.		

Python Syntax	SetDoubleQuantityData (<*qtyName*>,<*qtyData*>)
Python Example	`doubleList = [12.3, 14.5, 16.7]` `outData.SetDoubleQuantityData("V1PlusV2", doubleList)`

SetComplexQuantityData

If the quantity computed is a complex quantity, use this method to set the quantity values. Any sweep values must be set separately via the SetSweepsData method.

UI Access	NA		
Parameters	Name <*qtyName* > <*qtyData* >	Type String List<floats>	Description Name of the quantity. Quantity data for the specified quantity. Complex numbers are passed as pairs of floats in a list.
Return Value	Boolean. If true, the method was successful. If false, it was not.		

Python Syntax	SetComplexQuantityData (<*qtyName*>,<*qtyData*>)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Example	doubleFromComplexList=[]
	complexList = [(1+1j), (2+4j), (9.1+3.2j)]
	for aComplex in complexList:
	    doubleFromComplexList.append(aComplex.imag)
	    doubleFromComplexList.append(aComplex.real)
	    outData. SetComplexQuantityData ("V1PlusV2", doubleFromComplexList)

Working With Properties for UDO

A property is the unit for collecting and using input from the user that is used to influence the UDO's Compute. These are initially set up when the UDOs **GetInputUDSParams** method is called and are retrieved in the UDO's Compute method.

There are three supported property types that could be used in the UDO script:

- INumberProperty to specify number properties (with unit support).
- IMenuProperty to allow the user to select from a list of options.
- ITextProperty to allow the user to enter text.

The IPropertyList type implements a collection for these properties:

- IPropertyList Abstract class
- IProperty Abstract class
- INumberProperty Abstract class
- ITextProperty Abstract class
- IMenuProperty Abstract class

IPropertyList Abstract Class

Attributes:

- AllProperties (IEnumerable<IProperty> - see IProperty)
- NumProperties (int)

Functions:

- GetProperty(string propName): Returns a named property as an IProperty.
- GetMenuProperty (string propName): Returns the named property as an IMenuProperty.
- GetTextProperty (string propName): Returns the named property as an ITextProperty.
- GetNumberProperty (string propName): Returns the named property as an INumberProperty.
- DeleteProperty (string propName): Deletes an already added named property.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- AddNumberProperty(string name, string numberWithUnits): Adds a new number property. If a property with the same name already exists, it is overwritten.
- AddTextProperty(string name, string textValue): Adds a new named text property with the supplied value. Any existing property with the same name is overwritten.
- AddMenuProperty(string name, IList<string> menuChoices): Creates a new named menu property with the supplied list of choices. The default selection is set to item 0 (the first item). Any property with the same name is overwritten.

IProperty Abstract Class

Attributes:

- Name (string)
- Description (string)
- PropType (read-only EPropType - see Constants)

Constructor:

- IProperty(string name, EPropType type)

The class is used as base class for INumberProperty, IMenuProperty, and ITextProperty.

INumberProperty Abstract Class

Base class:

- abstract class IProperty

Attributes:

- ValueSI (read-only double)
- ValueInUnits (read-only double)
- Units (read-only string)
- HasUnits (read-only bool)

Constructor:

- INumberProperty(string name)

Functions:

- Set(string numberWithUnits)
- SetDouble(double number, string unitString)

ITextProperty Abstract Class

Base class:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- abstract class IProperty

Attributes:

- Text (string)

Constructor:

- ITextProperty(string name)

IMenuProperty Abstract Class

Base class:

- abstract class IProperty

Attributes:

- MenuSelection (int): This represents the index into the MenuChoices list.
- SelectedMenuChoice (string): This is the item in the MenuChoices list corresponding to the MenuSelection index.
- MenuChoices (IList<string>)

Constructor:

- IMenuProperty (string name)

Example:

```
# adding data to IPropertyList propList; used in Compute function
prop = propList.AddNumberProperty('Offset 1', '0')
prop.Description = 'Trace 1 Offset'
prop = propList.AddNumberProperty("TRATE", "800 MHz")
prop.Description = "Frequency"
prop = propList.AddTextProperty("Text", "The Text")
prop.Description = "Text Property"
prop = propList.AddMenuProperty('Operation', ['Add', 'Subtract',
'Max' ,'Min','Mean'])
prop.Description = 'Operation menu'

# reading data from IPropertyList propList; used in Validate function
numOfNumberProperties = 0
if propList != None and propList.AllProperties != None:
        for prop in propList.AllProperties:
                if prop.PropType == Constants.EPropType.PT_NUMBER:
                        numOfNumberProperties ++
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Other Application-Specific Classes Used in Python Scripts

This section describes other classes used in Python scripts:

Constants Class

UDSProbeParams Class

UDSDynamicProbes Class

QuantityInfo Class

IProgressMonitor Abstract Class

Constants Class

The constants used in a Python script are defined in the Constants class.

Attributes:

- kDoubleParamStr : string constant used to specify *double* as the type of a quantity
- kComplexParamStr: string constant used to specify *complex* as the type of a quantity
- Enum EPropType: (used to set property type)
  - EPropType.PT_NUMBER
  - EPropType.PT_TEXT
  - EPropType.PT_MENU

Example:

```
paramType = Constants.kDoubleParamStr
propType = Constants.EPropType.PT_NUMBER
```

UDSProbeParams Class

This class defines which data quantity that a UDO will be pulled from the design to compute the UDO output (results). The objects of this class must be created in a Python script with the GetInputUDSParams function. They are supplied to the Validate function if implemented.

Attributes:

- ProbeName (read-only string)
- ProbeDescription (read-only string)
- ParamType (read-only string)
- ReportTypeName (read-only string)
- ComponentExpression (read-only string)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Constructor: UDSProbeParams(string probeName, string probeDescription, string paramType, string reportTypeName, string componentExpression);

- probeName - required.
- probeDescription - optional (can be empty string).
- paramType - required; can be one of the Constants:
  - kDoubleParamStr
  - kComplexParamStr
- reportTypeName - optional (can be empty string)
- ComponentExpression - optional (can be empty string)

Example:

```
udsProbParam = UDSProbeParams("probe1","", Constants.kDoubleParamStr,
"", "",)
```

UDSDynamicProbes Class

Attributes:

- UDSParam (read-only UDSProbeParams)
- UserSelectedProbeName (read-only string)

Constructor: UDSDynamicProbes (UDSProbeParams udsParam, string userSelectedProbeName)

- udsParam - required
- userSelectedProbeName - required

Example:

```
udsProbParam = UDSProbeParams("probe1","", Constants.kDoubleParamStr,
"", "",)
selectedName = "probe1"
udsDynamicProbParam = UDSDynamicProbes(udsProbParam, selectedName )
```

QuantityInfo Class

Attributes:

- ParamType (read-only string)
- FullUnitType (read-only string)

Constructors:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- QuantityInfo(string paramType)
- QuantityInfo(string paramType, string fullUnitType)

Parameters:

- paramType can be one of the Constants:
  - kDoubleParamStr
  - kComplexParamStr
- fullUnitType is a case insensitive string representing full unit type. It is not defined in Constants. Instead you can use any of the units in string representation - for example, "mm" or"ghz".

Example:

```
quantityInfo1 = QuantityInfo(Constants.kDoubleParamStr)
quantityInfo2 = QuantityInfo(Constants.kDoubleParamStr,"ghz")
```

IProgressMonitor Abstract Class

The object of this class is a progress monitor. It displays a task's calculated progress in the UI and checks if the user has requested to abort the computation. Use this object for computations that take a long time to complete or if the UI might freeze during the computation. When displayed in the application, each progress message has four items:

- A task name
- A sub-task name
- The progress amount
- A button to abort the task in progress

This class provides the following functionality and abort interaction:

- SetTaskName (string taskName)
- SetSubTaskName (string subTaskName)
- BeginTask (string name)
- SetTaskProgressPercentage(int progressPercent)
- CheckForAbort()
- EndTask (bool passFail)

Example:

```
progMon.BeginTask("Process DQS")
progMon.SetSubTaskName("Compute UI segments")
progMon.SetTaskProgressPercentage(33)
progMon.SetSubTaskName("Compute the rest")
progMon.SetTaskProgressPercentage(100)
progMon.EndTask(True)
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> There should be the same number of calls to BeginTask as to EndTask.

SetTaskName

Sets the name of the task whose progress is being monitored.

UI Access	NA		
Parameters	Name <*TaskName*>	Type String	Description Name of the task
Return Value	None		

Python Syntax	SetTaskName (<*TaskName*>)
Python Example	`progMon.SetTaskName("Compute UI")`

SetSubTaskName

Sets the name of the subtask whose progress is being monitored.

UI Access	NA		
Parameters	Name <*TaskName*>	Type String	Description Name of the task
Return Value	None		

Python Syntax	SetSubTaskName (<*TaskName*>)
Python Example	`progMon.SetSubTaskName("Compute UI segments")`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BeginTask

Sets the name of the task whose progress will be monitored. Call this method only once for each task and subtask.

UI Access	NA		
Parameters	Name <*Name*>	Type String	Description Name of the task
Return Value	None		

Python Syntax	BeginTask (<*Name*>)
Python Example	`progMon.BeginTask("Process DQS")`

SetTaskProgressPercentage

Sets the progress percentage of a task to a specific value.

UI Access	NA		
Parameters	Name <Percent>	Type Integer	Description Percent of progress
Return Value	None		

Python Syntax	SetTaskProgressPercentage (<*Percent*>)
Python Example	`progMon.SetTaskProgressPercentage(33)`

CheckForAbort

Checks if a user has aborted the task. Call this method if the quantities are computationally expensive. This method can be called multiple times.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UI Access	Abort button on the progress dialog.
Parameters	None
Return Value	Boolean flag that if true indicated that the user requested an abort and you should call EndTask. If false, allow the task to continue.

Python Syntax	CheckForAbort
Python Example	`should_abort = progMon.CheckForAbort`

EndTask

Stops a task from running. Call this method only once for each task and subtask. In a script, there should be the same number of BeginTask calls as **EndTask** calls.

UI Access	NA		
Parameters	Name <PassFail>	Type Boolean	Description Flag that indicates whether to continue the main task or not. If true, the task will continue to process the next subtask. If false, the main task stops and an error icon appears in the UI.
Return Value	None		

Python Syntax	EndTask (<*PassFail*>)
Python Example	`progMon.EndTask(True)`

Using .NET Collection Classes and Interfaces in Python Scripts

Some of the API functions specified above use .Net collection classes and interfaces, that is, Array class, IList interface, IEnumerable interface, and IDictionary interface. The following section describes how to work with the .Net collection objects in Python scripts.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

.NET Array, IEnumerable, and IList objects can be indexed and iterated over as if they were Python lists. You can also check for membership using 'in'. To get .Net Array and IList sizes you can use Python's 'len' or .Net 'Count'.

Example:

Getting size:

```
arraySize = doubleDataArray.Count
   arraySize = len(doubleDataArray)
listSize = sweepsNamesList.Count
 listSize = len(sweepsNamesList)
```

Iterating:

```
for sweep in sweepsNamesList:
   print sweep
for in in xrange(listSize)
      print sweepsNamesList[i]
```

Checking for membership:

```
if 'Time' in sweepsNamesList:
   doThis()
  else:
   doThat()
```

For .NET IDictionary, the same as for Array and IList, you can get size with 'len' or 'Count' and check for membership of the keys using 'in'. Getting values for the keys also works the same way as in Python 'dict'.

Example:

Getting size:

```
varValuesSize = varValues.Count
 varValuesSize = len(varValues)
```

Checking for membership:

```
if 'offset' in varValues:
   print varValues['offset']
```

Getting value:

```
if 'offset' in varValues:
   offsetValue = varValues['offset']
```

As for iteration .NET Dictionary is different from Python dict. While iterating, python dict will return keys, .Net Dictionary will return .Net KeyValuePair.

Example:

Iterating:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for .NET IDictionary:

```
for varPair in varValues: #varPair is of .Net KeyValuePair type
     varName = varPair.Key
    varValue = varPair.Value
```

for Python dict:

```
for varName in varValues:
   varValue = varValues[varName]
```

You can use Python types instead of .NET types if you prefer. For this, you need to cast .NET array and .Net iList to Python list type and .NET Dictionary to Python dict type.

Casting should not be used for data arrays - it can be extremely costly for the memory usage as well as time consuming.

Example:

```
aPythonList = list(dotNetArray)
  aPythonList = list(dotNetList)
   aPythonDict = dict(dotNetDictionary)
```

User Defined Outputs: Messaging Methods

Messaging methods are provided to convey additional information to the user from any of the UDOs methods. The Compute function is the one typically location where such use is anticipated. Any message sent via these functions are displayed in the application's message window using the appropriate icon.

These functions can also be used for debugging purposes.

- AddErrorMessage
- AddInfoMessage
- AddWarningMessage

Example script:

```
############################################################
# Imports
############################################################
from Ansys.Ansoft.ModulePluginDotNet.Common.API import *
from Ansys.Ansoft.ModulePluginDotNet.Common.API.Interfaces import *
from Ansys.Ansoft.ModulePluginDotNet.UDO.API.Interfaces import *
from Ansys.Ansoft.ModulePluginDotNet.UDO.API.Data import *
class UDOExtension(IUDOPluginExtension):
  def __init__(self):
    pass

  #--- IDA IUDOPluginExtension -----------------------
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
def GetUDSName(self):
  return "MinMaxAvg"

#--- ISA IUDOPluginExtension ------------------------
def GetUDSDescription(self):
  return "Sample UDO for dimension reducing quantities"

#--- ISA IUDOPluginExtension ------------------------
# Returns list of category names
 def GetCategoryNames(self):
  return ["UDOOutputs"]

#--- ISA IUDOPluginExtension ------------------------
# returns a list of quantity names for the supplied category name
def GetQuantityNames(self, catName):
  if catName == "UDOOutputs":
    return ["min_val", "max_val", "avg_val"]
  else:
    return []

#--- ISA IUDOPluginExtension ------------------------
# Returns an instance of QuantityInfo for the qtyName supplied or
# None if such a quantity could not be found
def GetQuantityInfo(self, qtyName):
  # All the quantities we have are simple doubles
  # we can leave them unitless
  return QuantityInfo(Constants.kDoubleParamStr)

#--- ISA IUDOPluginExtension ------------------------
# Returns list of UDSParams and list of dynamic properties
# Adds setup time properties to the propList
def GetInputUDSParams(self, udsParams, propList,
userSelectedDynamicProbes):
  # Add the probes. We need only one double quantity
  param1 = UDSProbeParams("probe1",
    "double quantity probe",
    Constants.kDoubleParamStr,
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
      "", "")
    udsParams.Add(param1)


    # Add the properties we want the user to supply
    # In this case, we will ask for a start/end range for
    # X parameters. Since we cannot reasonably provide defaults
    # as we have no idea what the sweep limits will be, we will
    # also ask if the limits are to be activated.
    prop = propList.AddNumberProperty("X Min", "0")
    prop.Description = "Start X value to consider"

    prop = propList.AddNumberProperty("X Max", "1")
    prop.Description = "End X value to consider"

    # For menus, the first option is the default.
    prop = propList.AddMenuProperty("Activate X Limits", ["No",
    "Yes"])
    prop.Description = "Activate X range"

    return True

#--- ISA IUDOPluginExtension ------------------------
# Returns list of UDSParams and list of dynamic properties
# output UDSDynamicProbeCollection probes
def GetDynamicProbes(self, probes):
    pass

#--- ISA IUDOPluginExtension ------------------------
# Returns list of sweeps names
# We have no sweeps as we reduce them.
def GetUDSSweepNames(self):
    return []

#-----------------------------------------------------------------
# IUserDefinedSolutionHandle API implementation.
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
# Calculates output values and sets them using
IUDSInputData/IUDSOutputData API.
def Compute(self, inData, outData, propList, progMon):

  # Get the sweeps associated with the probe and validate
  # use the probe name that we had defined earlier
  sweeps = inData.GetSweepNamesForProbe("probe1")
  if( sweeps == None or sweeps.Count > 1):
    AddErrorMessage(self.GetName() + "Unexpected sweep count 0 or >
    1 in Compute")
    return False

  # Get the data associated with our probe
  probeData = inData.GetDoubleProbeData("probe1")
  sweepData = inData.GetSweepsDataForProbe("probe1", sweeps[0])

  # Get the user specified properties.
  # Note that ideally, these "X Min" etc names should be
  # written as constant members and referred to in both
  # the GetInputUDSParams and in Compute to reduce the change
  # of typos.
  useXRangeProp = propList.GetMenuProperty("Activate X
  Limits").SelectedMenuChoice
  xRangeStart = propList.GetNumberProperty("X Min").ValueSI
  xRangeEnd = propList.GetNumberProperty("X Max").ValueSI

  # At this stage, one can look at the RequestedQuantities and
  # create a dictionary to later check against. However, I am
  # simply computing all the quantities.
  minVal = 0
  maxVal = 0
  avgVal = 0

  # Check if we need to perform range computation
  if useXRangeProp == "Yes":
    seenAny = False
    avgSum = 0
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    count = 0


    # zip is used since we also need to pull in sweep data
    # an index and the array notation could also have been used
    for probeVal, sweepVal in zip(probeData, sweepData):
                if sweepVal < xRangeStart or sweepVal > xRangeEnd:
        pass


                # Note that in a better written script, this code
      would be
                # refactored into its own function to avoid code
                # duplication
                if not seenAny:
        minVal = probeVal
        maxVal = probeVal
        avgSum = probeVal
        seenAny = True
        count = 1
                else:
        if probeVal < minVal:
          minVal = probeVal
        if probeVal > maxVal:
          maxVal = probeVal
        avgSum += probeVal
        count += 1
    if seenAny:
                avgVal = avgSum/count
  else:
    seenAny = False
    avgSum = 0
    for probeVal in probeData:
                if not seenAny:
        minVal = probeVal
        maxVal = probeVal
        avgSum = probeVal
        seenAny = True
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
            else:
        if probeVal < minVal:
          minVal = probeVal
        if probeVal > maxVal:
          maxVal = probeVal
        avgSum += probeVal
    if seenAny:
            avgVal = avgSum/probeData.Count


# Finally set the output values. Note that these are always set
# as lists even if we have just one item.
outData.SetDoubleQuantityData("min_val", [minVal])
outData.SetDoubleQuantityData("max_val", [maxVal])
outData.SetDoubleQuantityData("avg_val", [avgVal])


# And we are done.
return True
```

AddErrorMessage

Call this method to convey an error condition to the user.

UI Access	NA		
Parameters	Name <message>	Type String	Description Error message
Return Value	None		

Python Syntax	AddErrorMessage (<*message*>)
Python Example	`AddErrorMessage(self.GetName() + "Unexpected sweep count 0 or > 1 in Compute"))`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AddInfoMessage

Call this method to convey an informational message to the user. This is the call to use when outputting messages for debugging purposes.

UI Access	NA		
Parameters	Name <message>	Type String	Description Warning message
Return Value	None		

Python Syntax	AddInfoMessage (<*message*>)
Python Example	`AddInfoMessage("Enter starting X value.")`

AddWarningMessage

Call this method to convey a warning message, typically used for conditions that are not ideal but can be tolerated by the script.

UI Access	NA		
Parameters	Name <message>	Type String	Description Warning message
Return Value	None		

Python Syntax	AddWarningMessage (<*message*>)
Python Example	`AddWarningMessage("Unexpected sweep count in Compute")`

User Defined Outputs: Script Organization

As described in the Introduction section, the UDO scripts should all reside under the **UserDefinedOutputs** folder under either of the three library locations (system, user, or personal).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using Script Libraries

If you decide that you need base classes, additional data files, and etc., to organize your UDOs better, you can do so. This type of library organization allows code reuse between similar UDOs and can be very helpful. There is special support provided for this type of script-library organization:

- **All script-library and other support files need to be in a *Lib* sub-directory under the *UserDefinedOutputs* directory**. Any *.py* files found in such **Lib** directories are ignored and not displayed in the GUI as a valid UDO choice.
- For a UDO script at any given directory depth, all Lib directories in its parent directories will be automatically added to the system include path (and so, any support script files from any Lib directory through the top level *UserDefinedOutputs* directory can be imported).

Using Additional .NET Assemblies

Because the UDO functionality uses IronPython, we have access to the full .NET eco system. If needed, any subset of the UDO functionality can be implemented in any .NET language and used by the UDO script. There are simple rules to follow to achieve this.

1. Build your .NET assembly for .NET 2.0 runtime.
2. Drop the built assembly in any **Lib** directory upstream of the UDO script location: that is, if you have your UDO script in *C:\Users\x\PersonalLib\UserDefinedOutputs\a\b\c\myudo.py* and have a .NET assembly called *com.Acme.UDOLib* You can keep the .NET assembly under:
    - UserDefinedOutputs\Lib
    - UserDefinedOutputs \a\Lib
    - UserDefinedOutputs \a\b\Lib
    - UserDefinedOutputs\a\b\c\Lib
3. Add the following lines to your Python script:
    - `Import clr`
    - `clr.AddReference("com.Acme.UDOLib")`
    - `import com.Acme.UDOLib` -or- `from com.Acme.UDOLib import * etc`

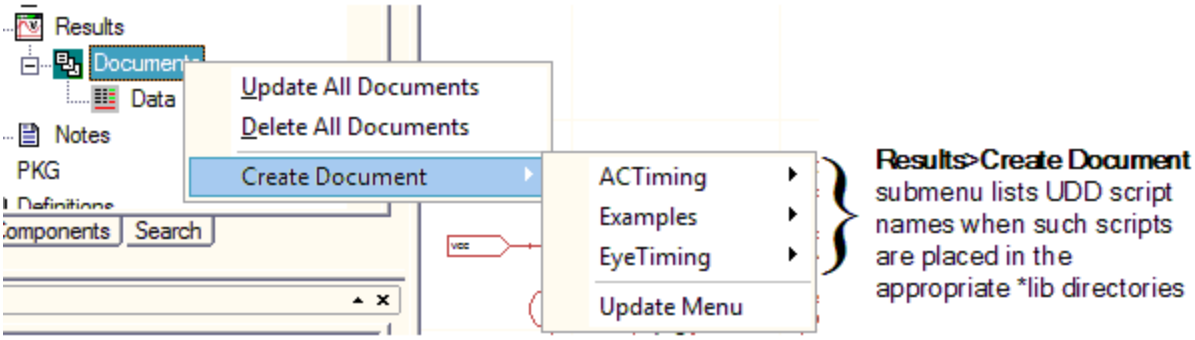
If for some reason you cannot place the .NET assemblies into a Lib directory under *UserDefinedOutputs*, you need to do a couple more steps before step 3 listed above.

```
Import sys
sys.path.append("full path to your .NET assembly location")
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User-Defined Documents (UDDs)

User-defined documents (UDDs) are custom reports that you define through IronPython scripts. Once placed in a Lib directory, you can access the scripts via the **Create Document** command. The scripts describe a *Create User Defined Document* dialog box that lets you specify trace and solution inputs. After you confirm your input selections, XML, HTML, and PDF documents are generated. A web browser window opens to display the generated HTML file. The created document appears in the Project Manager, under Results in the Documents folder.



The general UDD process flow is as follows:

The UDD Python scripts must be placed in the **UserDefinedDocuments** directory under either of **syslib**, **userlib**, or **Personallib** with any subdirectory structure needed. The Lib directory can contain Python scripts that have common code that other scripts can use.

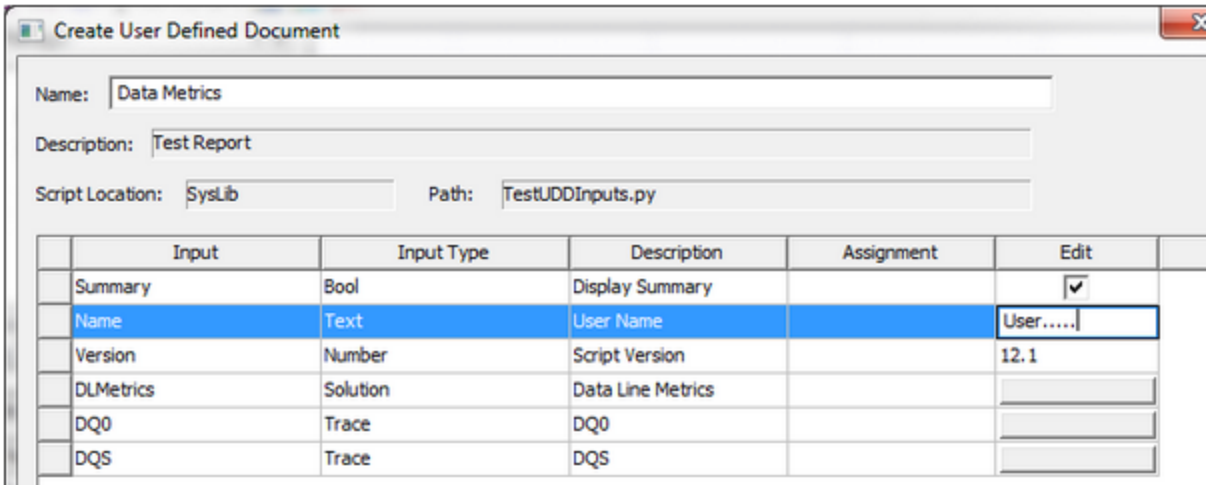
Use **HFSS** > **Results** > **Create Document**> **Update Menu** to refresh the menu to include the new UDD scripts that have been copied to syslib, userlib, or Personallib, or to exclude them if they have been deleted after the launch of desktop.

The UDD scripts that are in syslib/UserDefinedDocuments, userlib/UserDefinedDocuments, or Personallib/UserDefinedDocuments are available through the **HFSS** > **Results** > **Create Document** menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create User-Defined Document Inputs

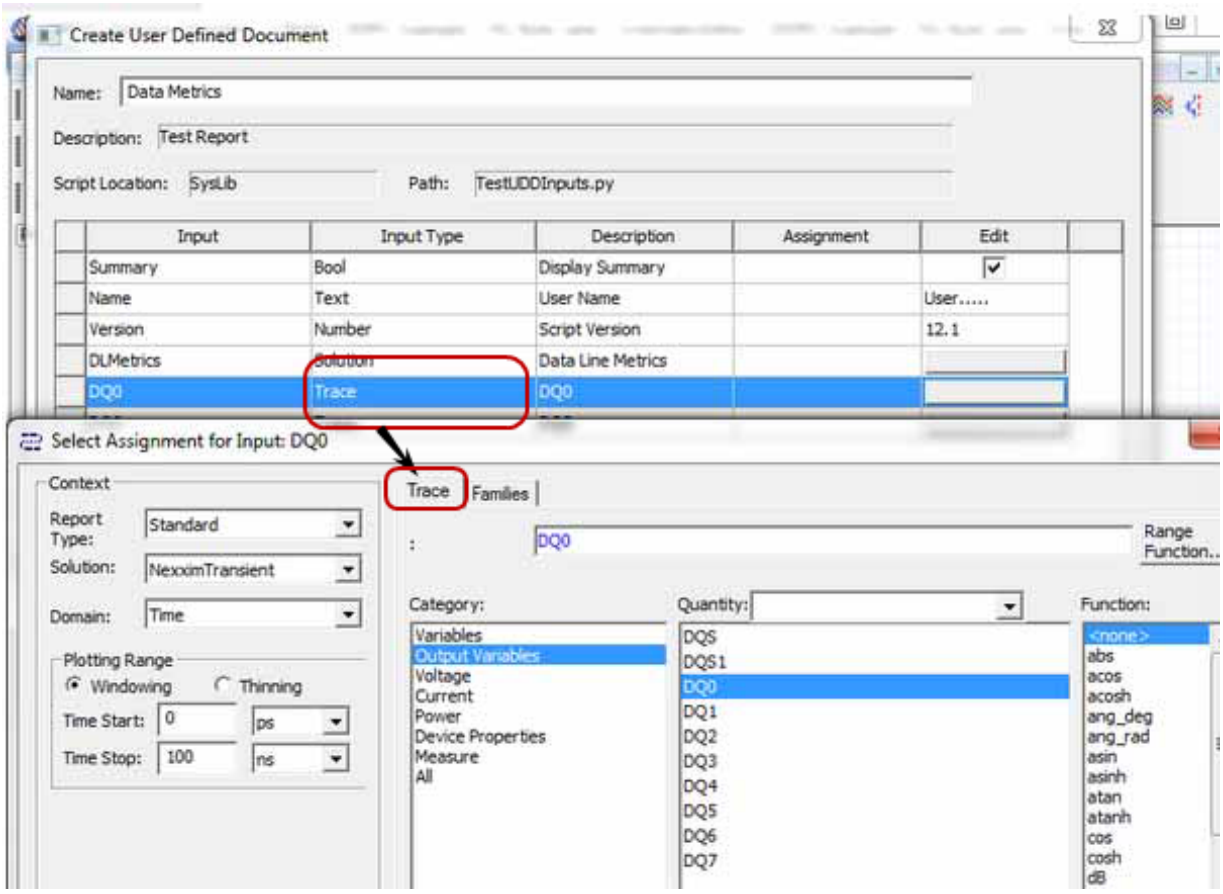
User-defined documents allow data from traces, solutions, and report types as inputs. A UDD can specify the named inputs, which you select or enter the values of in the *Create User Defined Document* dialog box that displays when you run **HFSS** > **Results** > **Create Document**> <*scriptName*>.



Input Types can be of Boolean, number, text, trace, or solution type. The boolean, number, and text type can be given a default value that you can interactively override when the document is created or modified. For example, you can select a trace when you create or modify a UDD document. The trace data is available to the user and can be accessed from the Python script.

At the time of selection, you can choose from the *Reporter* dialog box the report type (Standard, Eye Diagram, User Defined), solution name, context, and the quantity for which you want the trace data.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

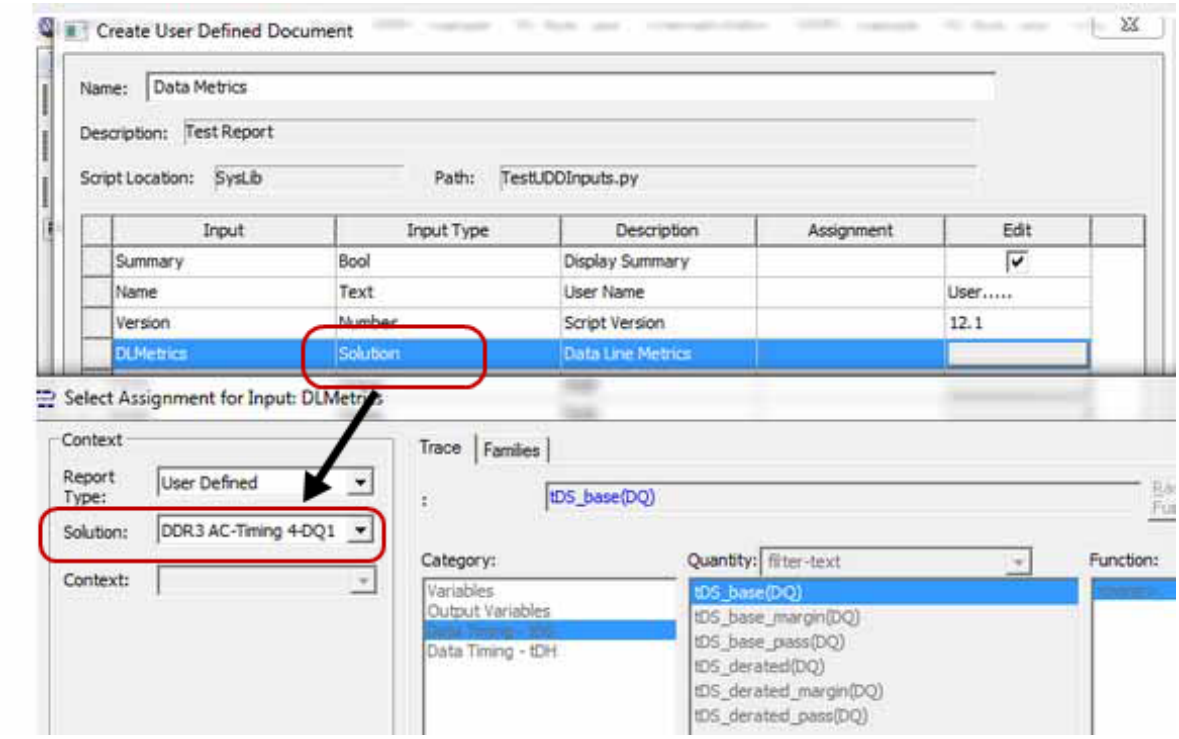


Input Type can also be Solution. You can select an entire solution when the document is created or modified. The solution data in its entirety is now available to the user and can be accessed from the Python script.

At the time of selection, you can choose from the reporter dialog the report type (Standard, Eye Diagram, User Defined), solution name, and context. A specific quantity cannot be selected since data for all quantities in the solution are available.

> **Note:** The Category/Quantity/Function portion of the dialog is disabled for user input.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

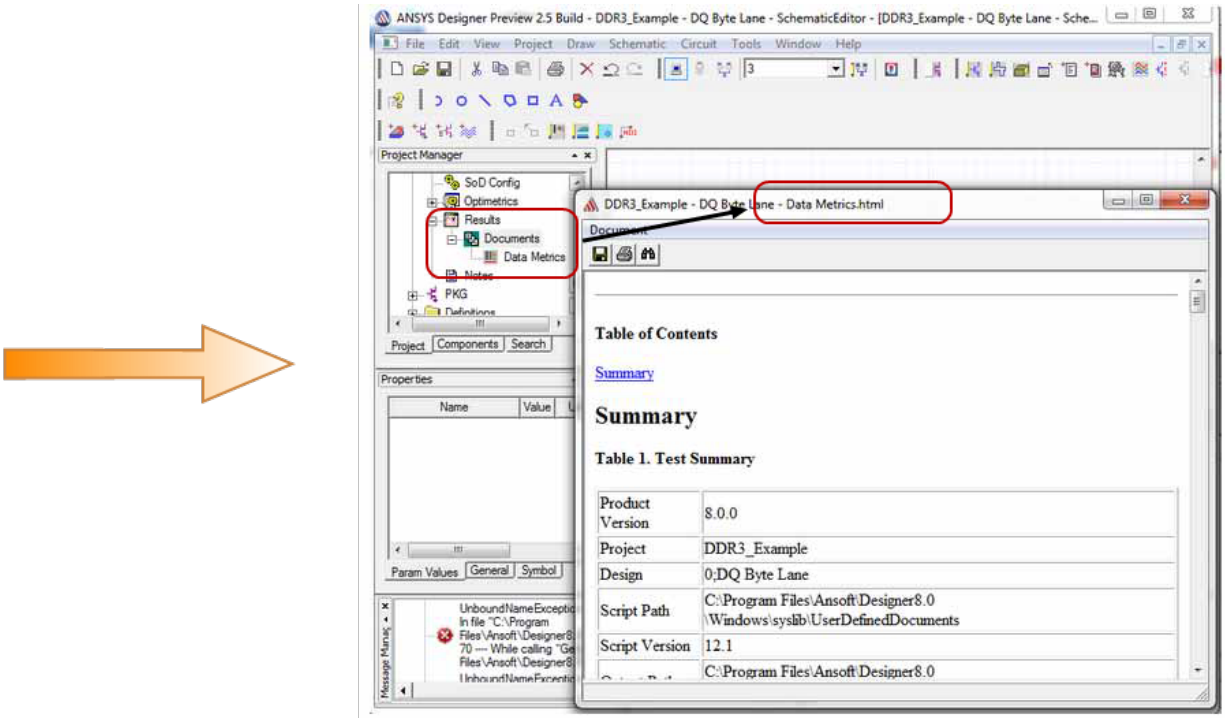


UDD Document Creation and Display

After all the input selections for a UDD are confirmed, based on the script, XML, HTML, and PDF documents are generated based on the inputs provided by the user. (The XML, HTML, and PDF generation is based on specific calls in the Python script, which are explained in a following section.) A web browser window also opens to display the generated HTMLfile.

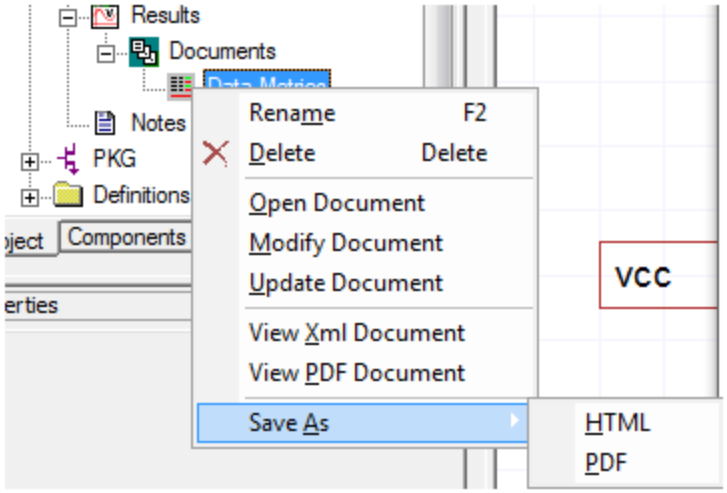
The created document will be placed under a new folder named **Documents** under the **Results** folder. All documents that are created by the user for the design will be placed under this folder.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Managing Documents Listed in the Project Window Under Results

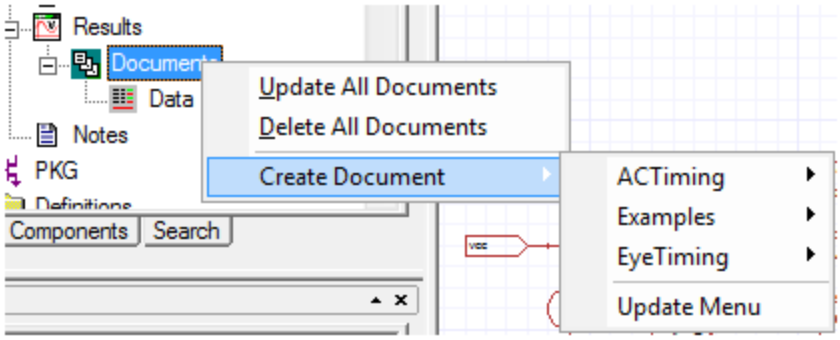
Right-click a user-defined document displayed in the Project Manager tree to bring up a menu where you can rename, delete the document. **Open document** opens the web browser with the html document. **Modify document** opens the setup dialog box, where you can change the selections for the input. To view the XML and the PDF document simply choose the appropriate menu items. There is also a menu item to save the document in a different location.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Documents Folder's Context Menu

Right-click the documents folder has the menu options to **Update All Documents** or **Delete All Documents**. It also provides the option of creating a document from here.

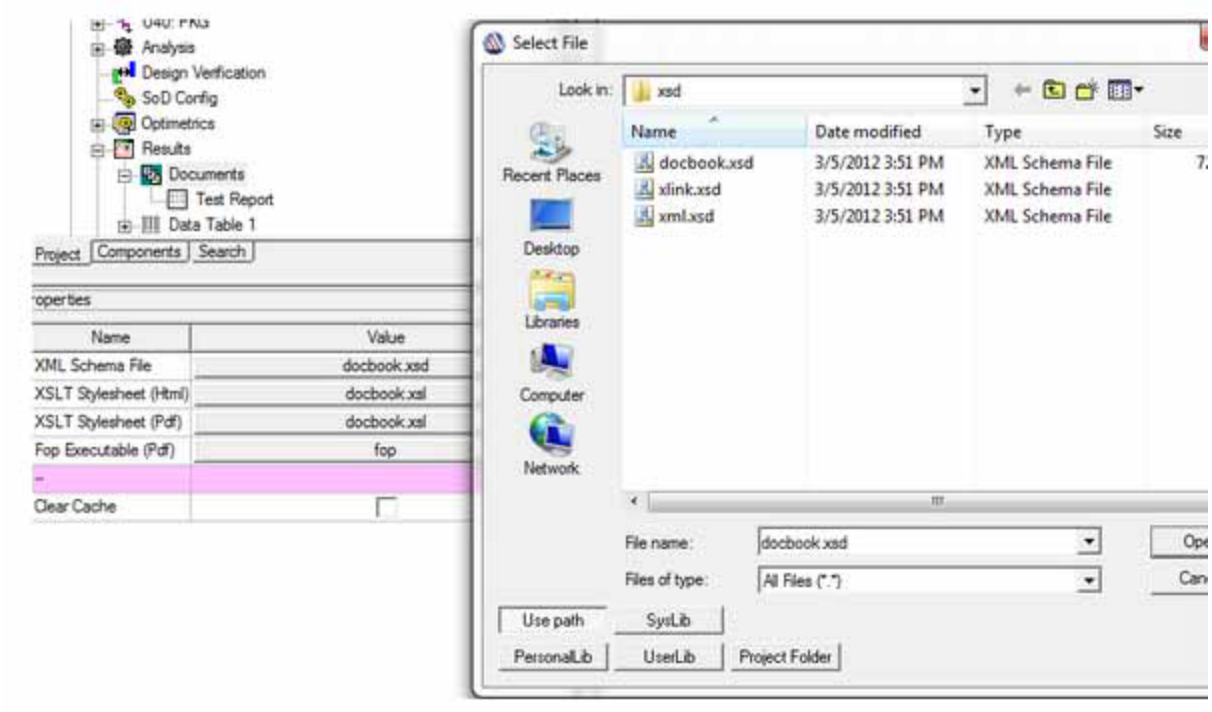


Document Folder's Property Window

When the documents folder is selected, the *Property* window shows the following properties:

- **XML Schema File** - File path to the XML schema file.
- **XSLT StyleSheet (HTML)** - File path to the XSLT stylesheet file used for HTML generation.
- **XSLT StyleSheet (Fo)** - File path to the XSLT stylesheet file used for PDF generation.
- **Fop Executable (PDF)** - File path the Fop executable used for PDF generation.
- **Clear Cache** - Clears the cached XSL transform object and forces creation of a new one. (The caching is done to save time during document generation, so subsequent generation or update of the document can use the cached transform object. But sometimes you may want to force a recompile of the document if you change the stylesheet).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The XML, HTML, and PDF generation require the XML schema file and XSLT stylesheets to generate proper output. In addition, the PDF generation requires a FOP executable. You can use the defaults provided in the installation or provide the file paths of your own preferred stylesheets and fop executable installed in his machine.

Viewing UDDs with an HTML Web Browser

The XML and HTML documents can be viewed in a web browser with some basic functionality like printing the document, searching the document for a phrase or sentence, and saving the document.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

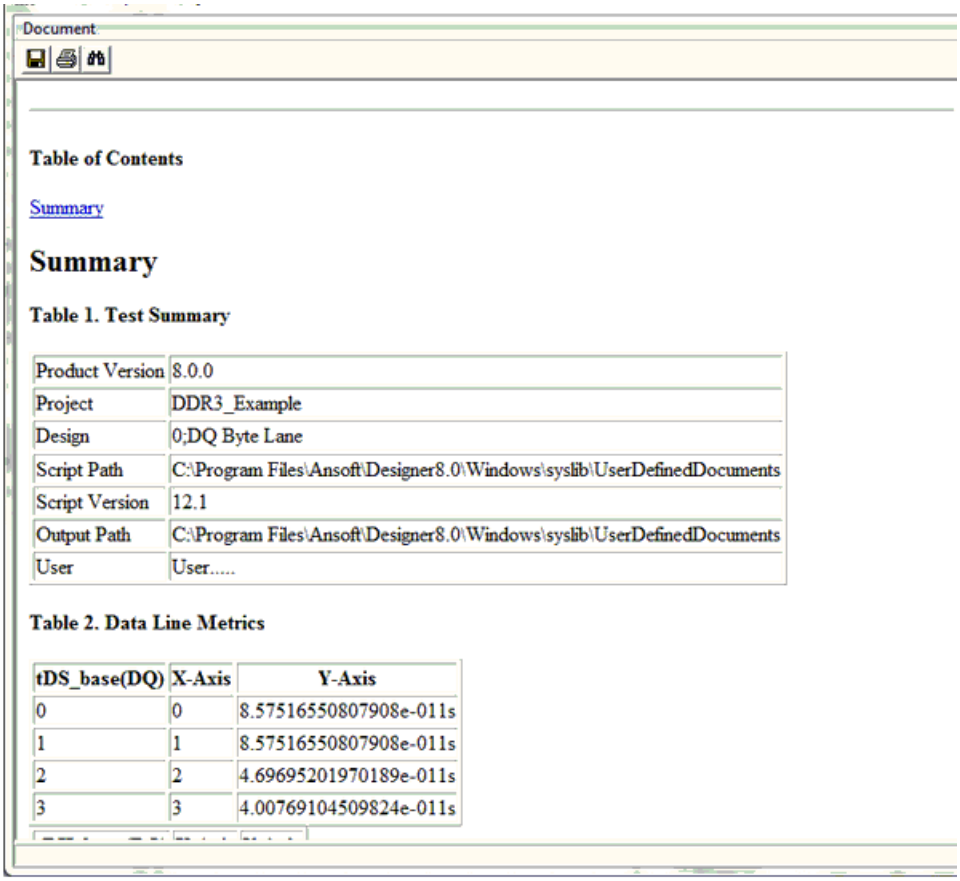

Document

Table of Contents

Summary

Summary

Table 1. Test Summary

Product Version	8.0.0
Project	DDR3_Example
Design	0;DQ Byte Lane
Script Path	C:\Program Files\Ansoft\Designer8.0\Windows\syslib\UserDefinedDocuments
Script Version	12.1
Output Path	C:\Program Files\Ansoft\Designer8.0\Windows\syslib\UserDefinedDocuments
User	User.....

Table 2. Data Line Metrics

tDS_base(DQ)	X-Axis	Y-Axis
0	0	8.57516550807908e-011s
1	1	8.57516550807908e-011s
2	2	4.69695201970189e-011s
3	3	4.00769104509824e-011s

UDD Script Libraries

Base classes and data files shared between similar UDDs can be organized to reuse the code in a better way. All script-library and other support files need to be in a Lib sub-directory under the UserDefinedDefinitions directory. Any .py files found in such Lib directories are ignored and not displayed in the GUI as a valid UDD choice. For a UDD script at any given directory depth, all Lib directories in its parent directories will be automatically added to the system include path (and so, any support script files from any Lib directory through the top level UserDefinedDefintions directory can be imported).

The UDD functionality uses IronPython so we have access to all the .NET assembles. If needed, any subset of the UDD functionality can be implemented in any .NET language and used by the UDD script. There are simple rules to follow to achieve this.

1. Build your .NET assembly for .NET 2.0 runtime.
2. Drop the built assembly in any Lib directory upstream of the UDD script location: that is, if you have your UDD script in C:\Users\x\PersonalLib\UserDefinedDefintions\a\b\c\myudd.py and have a .NET

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

assembly called com.Acme.UDDLib You can keep the .NET assembly under:

- UserDefinedDefintions\Lib
- UserDefinedDefintions\a\Lib
- UserDefinedDefintions\a\b\Lib
- UserDefinedDefintions\a\b\c\Lib

3. Add the following lines to your Python script:

```
Import clr

clr.AddReference("com.Acme.UDDLib")

import com.Acme.UDDLib -or- from com.Acme.UDDLib import * etc
```

If for some reason you cannot place the .NET assemblies into a Lib directory under UserDefinedDefintions, you need to do a couple more steps before step 3 listed above:

```
Import sys

sys.path.append("full path to your .NET assembly location")
```

User-Defined Documents: Python Script API

A User-Defined Documents (UDD) extension is implemented as an IronPython script that defines a class with a specific name: UDDExtension which derives from a specific base class IUDDPluginExtension and implements its abstract methods.

This API supports multiple data types in the forms of constants and classes. It also has several input interfaces.

User-Defined Document scripting commands are provided in this product's scripting guide. An complete example with a line by line explanation of how to use these methods is available.

Import Statements

The base class to be used and the types it uses in turn are contained in .NET assemblies. The use of these requires that the assemblies be imported into the UDD script: the following import statements should be added to the top of the python script:

```
from Ansys.Ansoft.DocGeneratorPluginDotNet.DocGenerator.API.Data
import *
from
Ansys.Ansoft.DocGeneratorPluginDotNet.DocGenerator.API.Interfaces
import *
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Data Types Used in UDD Python Scripts

There are several types that you must use while authoring a Python script. Some of them are used to pass data from the UI to a Python script and to provide interfaces for working with this data. Some are used to pass data from a Python script to the UI.

To pass data from a Python script to the UI, the objects of the C# class must be created in the Python script using their C# constructors. Then they can be set as a function's return values or set to the output parameters using their API.

Constants

- **kTraceTypeStr** : string constant used to specify an input of trace type
- **kSolutionTypeStr** : string constant used to specify an input of solution type
- **kNumberTypeStr** : string constant used to specify an input of number type
- **kTextTypeStr** : string constant used to specify an input of text type
- **kBoolTypeStr** : string constant used to specify an input of boolean type
- **kStandardReportStr** : string constant to specify a standard report
- **kEyeDiagramReportStr** : string constant to specify an eye diagram report
- **kUserDefinedReportStr** : string constant to specify a user defined report
- **kSweepDomainStr** : string constant to specify the sweep domain
- **kTimeDomainStr** : string constant to specify the time domain

Abstract Classes

- IProgressMonitor Abstract Class
- UDDExtension Class
- UDDInputData class
- UDDInputParams class

IProgressMonitor Abstract Class

The object of this class is a progress monitor. It displays a task's calculated progress in the UI and checks if the user has requested to abort the computation. Use this object for computations that take a long time to complete or if the UI might freeze during the computation. When displayed in the application, each progress message has four items:

- A task name
- A sub-task name
- The progress amount
- A button to abort the task in progress

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This class provides the following functionality and abort interaction:

- SetTaskName (string taskName)
- SetSubTaskName (string subTaskName)
- BeginTask (string name)
- SetTaskProgressPercentage(int progressPercent)
- CheckForAbort()
- EndTask (bool passFail)

Example:

```
progMon.BeginTask("Process DQS")
progMon.SetSubTaskName("Compute UI segments")
progMon.SetTaskProgressPercentage(33)
progMon.SetSubTaskName("Compute the rest")
progMon.SetTaskProgressPercentage(100)
progMon.EndTask(True)
```

> **Note:**
>
> There should be the same number of calls to BeginTask as to EndTask.

SetTaskName

Sets the name of the task whose progress is being monitored.

UI Access	NA		
Parameters	Name <*TaskName*>	Type String	Description Name of the task
Return Value	None		

Python Syntax	SetTaskName (<*TaskName*>)
Python Example	`progMon.SetTaskName("Compute UI")`

SetSubTaskName

Sets the name of the subtask whose progress is being monitored.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UI Access	NA		
Parameters	Name <*TaskName*>	Type String	Description Name of the task
Return Value	None		

Python Syntax	SetSubTaskName (<*TaskName*>)
Python Example	`progMon.SetSubTaskName("Compute UI segments")`

BeginTask

Sets the name of the task whose progress will be monitored. Call this method only once for each task and subtask.

UI Access	NA		
Parameters	Name <*Name*>	Type String	Description Name of the task
Return Value	None		

Python Syntax	BeginTask (<*Name*>)
Python Example	`progMon.BeginTask("Process DQS")`

SetTaskProgressPercentage

Sets the progress percentage of a task to a specific value.

UI Access	NA		
Parameters	Name <Percent>	Type Integer	Description Percent of progress
Return Value	None		

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Syntax	SetTaskProgressPercentage (<*Percent*>)
Python Example	`progMon.SetTaskProgressPercentage(33)`

CheckForAbort

Checks if a user has aborted the task. Call this method if the quantities are computationally expensive. This method can be called multiple times.

UI Access	Abort button on the progress dialog.
Parameters	None
Return Value	Boolean flag that if true indicated that the user requested an abort and you should call EndTask. If false, allow the task to continue.

Python Syntax	CheckForAbort
Python Example	`should_abort = progMon.CheckForAbort`

EndTask

Stops a task from running. Call this method only once for each task and subtask. In a script, there should be the same number of BeginTask calls as **EndTask** calls.

UI Access	NA		
Parameters	Name <PassFail>	Type Boolean	Description Flag that indicates whether to continue the main task or not. If true, the task will continue to process the next subtask. If false, the main task stops and an error icon appears in the UI.
Return Value	None		

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Syntax	EndTask (<*PassFail*>)
Python Example	`progMon.EndTask(True)`

IUDDPluginExtension Abstract Class

The **IUDDPluginExtension** abstract class declares the following abstract methods which must be implemented in the UDDExtension class or one of its base classes. If any of these methods are not implemented, a run-time error will occur and the UDD will not function.

- GetUDDName
- GetUDDDescription
- ShowDefaultSetupDialog
- GetUDDInputParams
- Generate

This class also provides for the following optional methods:

- SetupUDDInputParams
- HandleUDDEvents
- GetUDDSchema
- GetUDDStyleSheetForHtml
- GetUDDStyleSheetForPdf
- GetFopExecutable
- GetUDDAppContext
- GetUDDDesignContext

GetUDDName

Retrieves a prefix to use for all solution instances created using this UDD.

UI Access	NA
Parameters	None
Return Value	String that is the prefix

Python Syntax	GetUDDName()

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Python Example	`def GetUDDName(self):` `  return "MinMaxAvg"`

GetUDDDescription

Retrieves a description for the UDD, its purpose, etc.

UI Access	NA
Parameters	None
Return Value	String that is the description

Python Syntax	GetUDDDescription()
Python Example	`def GetUDDDescription(self):` `  return "Sample UDD"`

ShowDefaultSetupDialog

Retrieves whether to show the default setup dialog box. If not, the user might want to implement or show a customized setup dialog.

UI Access	NA
Parameters	None
Return Value	Boolean. If true, the default dialog should be shown. If false, the user does not want to see the default dialog.

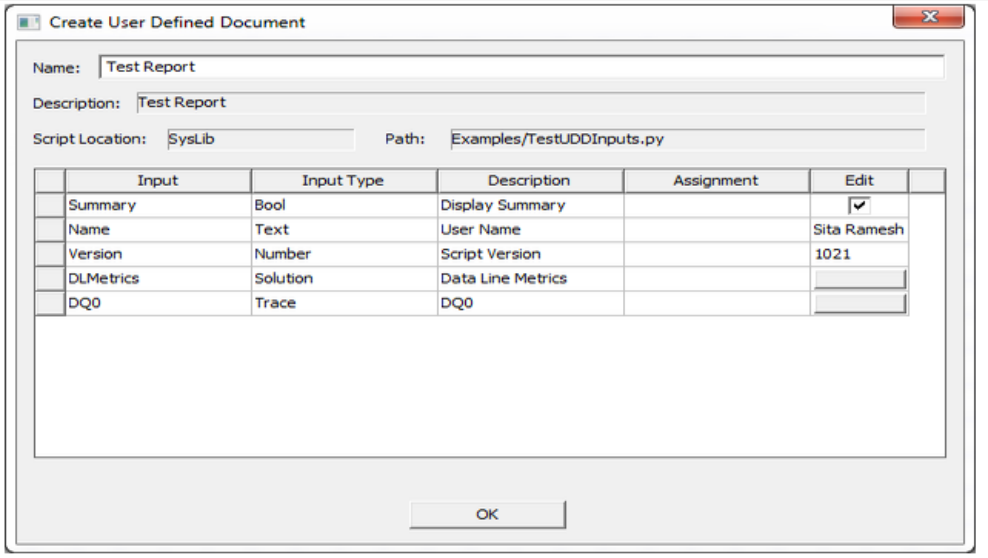
Python Syntax	ShowDefaultSetupDialog()
Python Example	`def ShowDefaultSetupDialog(self):` `  return True`

GetUDDInputParams

Retrieves inputs parameters for the User Defined Document. These parameters are the input's name, description, and data type. The UDD user specifies the value of the inputs through a

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dialog or within the script.

UI Access	Depending on the input parameters, the following dialog may be displayed when you click **HFSS**> **Results**> **Create Document**. The name and description of the UDD are also displayed in this dialog. 
Parameters	Name: *<uddInputs>* Type: List< UDDInputParams >* Description: The UDD script is expected to add one instance of UDDInputParams for each input definition that it will to display. When creating the UDD, the user will assign a matching value to each such input.
Return Value	Boolean. If true, the method was successful. If false, it was not.

Python Syntax	GetUDDInputParams(*<probeName>*)

Python Example	`def GetUDDInputParams(self, uddInputs)`
	`    # Boolean input`
	`    param1 = UDDInputParams("Summary","Display Summary", Constants.kBoolTypeStr, True)`
	`    uddInputs.Add(param1)`

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    # Text input
    param2 = UDDInputParams("Name","User Name",
Constants.kTextTypeStr, "Sita Ramesh")
    uddInputs.Add(param2)
    # Number input
    param3 = UDDInputParams("Version","Script Version",
Constants.kNumberTypeStr, 1021)
    uddInputs.Add(param3)
    # Solution input
    param5 = UDDInputParams("DLMetrics","Data Line
Metrics",Constants.kSolutionTypeStr)
    uddInputs.Add(param4)
    # Trace input
    param5 = UDDInputParams
("DQ0","DQ0",Constants.kTraceTypeStr)
    uddInputs.Add(param5)
    return True
```

Generate

This is the main method which accesses the data from the uddInputs and generates the document.

UI Access	NA		
Parameters	Name	Type	Description
	<uddInputs>	List< UDDInputData>	The list of inputs that the user setup in the dialog box. They are now available to query for data.
	<generator>	IUDDGenerator	This is the document generator object which used to create different elements of the document like titles, sections, tables, images and write the data too. See the Document Generator Interfaces.
	<progressMonitor>	IProgressMonitor	This can be used to set progress for long running calculations and check for any user-initiated aborts.
Return Value	Boolean. If true, the method was successful. If false, it was not.		

Python	Generate(*<uddInputs>*, *<generator>*, *<progressMonitor>*)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Syntax	

Python Example	def Generate(self, input, docgen, progMon):
	
	    # Gather data from inputs
	    boolinput = input[0].Data()
	    textinput = input[1].Data()
	    dblinput = input[2].Data()
	
	    # Get document root
	    docroot = docgen.GetDocumentRoot()
	
	    # Add Section
	    section1 = docroot.AddSection("Summary", "Overall Results ")
	
	    # Add a table
	    table1 = section1.AddTable("Test Summary")
	
	    #Add a table group with 2 columns
	    tgroup1 = table1.AddTableGroup(2)
	
	    # get desktop application
	    oApp = self.GetUDDAppContext()
	    if oApp != None:
	        oDesktop = oApp.GetAppDesktop()
	    if oDesktop != None:
	        # version number
	        version = oDesktop.GetVersion()
	        text1 = tgroup1.AddContent()
	        text1 .Add(0, "Product Version")
	        text1 .Add(1, version)
	
	    oProject = oDesktop.GetActiveProject()
	    if oProject != None:
	        projectname= oProject.GetName()
	        text1 = tgroup1.AddContent()
	        text1 .Add(0, "Project")
	        text1 1.Add(1, projectname)
	
	    oDesign = self.GetUDDDesignContext()
	    if oDesign != None:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        designname = oDesign.GetName()
        text1 = tgroup1.AddContent()
        text1 .Add(0, "Design")
        text1 .Add(1, designname)

    # Provides a script path
    scriptpath = docgen.GetScriptPath()
    text1 = tgroup1.AddContent()
    text1 .Add(0, "Script Path")
    text1 .Add(1, scriptpath)

    #Provides the script version
    text1 = tgroup1.AddContent()
    text1 .Add(0, "Script Version")
    text1 .Add(1, str(dblinput))

    #Provides the output xml path
    outputpath = docgen.GetOutputFilePath()
    text1 = tgroup1.AddContent()
    text1.Add(0, "Output Path")
    text1.Add(1, outputpath)

    #Provides the user information
    text1 = tgroup1.AddContent()
    text1 .Add(0, "User")
    text1 .Add(1, textinput)

    # Generate Xml output
    docgen.Write(False)

    # Generate Html output
    docgen.WriteHTML()

    # Generate PDF output
    docgen.WritePDF()

    return True
```

SetupUDDInputParams

This optional method displays a customized dialog and returns the user choices for the input params.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UI Access	NA		
Parameters	Name <*uddInputs*>	Type List<UDDInputParams>	Description .NET list of UDDInputParams objects with values for each of them. These can be the user choice for each input obtained through a custom dialog or some other non-graphical assignment. This method cannot process trace and solution types of input with a custom dialog because there is no way of assigning solution data to the input without the invocation of the reporter dialog.
Return Value	Boolean. If true, the event was handled successfully. If false, it was not.		

Python Syntax	SetupUDDInputParams(<*uddInputs*>)

Python Example	`def SetupUDDInputParams(self, uddInputs)`
	`    udddialog = BaseExampleUDDDialog()`
	`    if udddialog.ShowDialog() == Forms.DialogResult.OK:`
	`        # Boolean input`
	`        param1 = udddialog.GetInput("Summary")`
	`        uddInputs.Add(param1)`
	
	`        # Text input`
	`        param2 = udddialog.GetInput("Name")`
	`        uddInputs.Add(param2)`
	
	`        # Number input`
	`        param3 = udddialog.GetInput("Version")`
	`        uddInputs.Add(param3)`

HandleUDDEvents

This optional method is the event handler for all link events set by the SetEventLink() method on a IUDDText. Refer to the definition of the IUDDText object in the Document Generator Interface. The tags associated with the event are received by plugin using this abstract class.

UI Access	NA

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameters	Name Type Description <*eventTags*> List<string> Event tags.
Return Value	Boolean. If true, the event was handled successfully. If false, it was not.

Python Syntax	HandleUDDEvents(<*eventTags*>)

Python Example	`def HandleUDDEvents(self, uddLinks):`
	`  if uddLinks[0] == "Open Report":`
	`  # Get Design Name`
	`  oDesign = self.GetUDDDesignContext()`
	`  if oDesign != None:`
	`  oDesign.OpenReport(uddLinks[1])`
	`  return True`

GetUDDSchema

This optional method retrieves the file path of the schema to validate the XML. This will override the default schema used.

UI Access	NA
Parameters	None
Return Value	String containing the full file path of the schema.

Python Syntax	GetUDDSchema()

Python Example	`def GetUDDSchema(self):`
	`  return "C:\\Program Files\\ANSYS Inc\\v251\\AnsysEM\\common\\docbook\\schema\\xsd\\docbook.xsd"`
	`schemaPath = pluginExt.GetUDDSchema()`

GetUDDStyleSheetForHtml

This optional method retrieves the file path of the style sheet used to generate the HTML document. This will override the default stylesheet for HTML.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UI Access	NA
Parameters	None
Return Value	String containing the full file path of the style sheet.

Python Syntax	GetUDDStyleSheetForHtml()

Python Example	`def GetUDDStyleSheetForHtml(self):`
	`  return "C:\\Program Files\\ANSYS Inc\\v251\\AnsysEM\\common\\docbook\\"`
	`styleSheet = pluginExt.GetUDDStyleSheetForHtml()`

GetUDDStyleSheetForPdf

This optional method retrieves the file path of the style sheet used to generate the PDF document. This will override the default stylesheet for PDF.

UI Access	NA
Parameters	None
Return Value	String containing the full file path of the style sheet.

Python Syntax	GetUDDStyleSheetForPdf()

Python Example	`def GetUDDStyleSheetForPdf(self):`
	`  return "C:\\Program Files\\ANSYS Inc\\v251\\AnsysEM\\common\\docbook\\xsl\\fo\\docbook.xsl"`
	`styleSheet = pluginExt.GetUDDStyleSheetForPdf()`

GetFopExecutable

This optional method retrieves the file path of the fop executable used to generate the PDF document.

UI Access	NA

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameters	None
Return Value	String containing the full file path of the fop executable.

Python Syntax	GetFopExecutable()

Python Example	`def GetFopExecutable(self):`
	`  return "C:\\Program Files\\ANSYS Inc\\v251\\AnsysEM\\common\\ApacheFOP\\fop-1.0\\fop"`
	`fopExe = pluginExt.GetFopExecutable()`

GetUDDAppContext

This optional method retrieves the UDD Owner if it is set. This is a Dispatch wrapper that is essentially a COM IDispatch implementation and corresponds to the IDispatch pointing to the desktop app.

UI Access	NA
Parameters	None
Return Value	the IDispatch for the desktop object

Python Syntax	GetUDDAppContext()
Python Example	`oDesktop = self.GetUDDAppContext()`

GetUDDDesignContext

This optional method retrieves the UDD Owner if it is set. This is a Dispatch wrapper that is essentially a COM IDispatch implementation and corresponds to the IDispatch pointing to the design.

UI Access	NA
Parameters	None

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Return Value	IDispatch for the specified design object

Python Syntax	GetUDDDesignContext()
Python Example	`oDesign = self.GetUDDDesignContext()`

UDDExtension Class

The UDD itself should be implemented as an IronPython class called **UDDExtension** which must derive from the IUDDPluginExtension abstract base class (from the **Ansys.Ansoft.DocGeneratorPluginDotNet.DocGenerator.API.Interfaces** namespace).

Note that power users could derive a class hierarchy tuned toward a specific type of UDDs and that they can derive from their own base classes. The only requirement is that directly or indirectly, the UDD class must derive from **IUDDPluginExtension**.

Example:

```
def BaseClassUDD ((IUDDPluginExtension):
#base class implementation
…
def UDDExtension ((BaseClassUDD):
#UDD class implementation
…
```

> **Note:**
>
> This class is modeled after the UDO class UDOExtension, therefore the usage is similar.

UDDInputData Class

This class contains user input data and can be used in the generated document. UDDInputData has the constructor UDDInputData(string name) and the following properties accessors:

Property accessor	Purpose	Form
Name	Get/Set the name of an input.	string Name();
Type	Get/Set the type of an input	string Type();

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The data can be any of the following types, based on the UDDInputParams type specified in the script.

- UDDInputBool
- UDDInputDouble
- UDDInputSolution
- UDDInputText
- UDDInputTrace

UDDInputBool

Method purpose	Return Value	Data type of the returned value in C#
Retrieves boolean value of the input data.	a boolean value of input data	string Data();

UDDInputDouble

Method purpose	Return Value	Data type of the returned value in C#
Retrieves double data.	a double value of input data	double Data();

UDDInputSolution

Method purpose	Return Value	Data type of the returned value in C#
Retrieves x and y double data for a probe	Data collection	IDictionary<double, double> DoubleData (string name);
Retrieves x and y double data for a probe for a particular variation	Data collection	IDictionary<double, double> DoubleData (string name, IDictionary<string, string> variation);
Retrieves x and y complex data for a probe	Data collection	IDictionary<double, double[]> ComplexData (string name);
Retrieves x and y complex data for a probe for a particular variation	Data collection	IDictionary<double, double[]> ComplexData (string name, IDictionary<string, string> variation);
Retrieves x and y text data for a probe	Data collection	IDictionary<string, string> TextData(string name, IDictionary<string, string> variation);

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Retrieves x and y text data for a probe for a particular variation	Data collection	IDictionary<string, string> TextData(string name);
Retrieves a list of category names	A string collection	IList<string> CategoryNames();
Retrieves a list of quantity names given a category name	A string collection	IList<string> QuantityNames(string category);
Retrieves a list of variable values	A string collection	IList<Dictionary<string, string>> VariableValues();

UDDInputText

Method purpose	Return Value	Data type of the returned value in C#
Retrieves string data	a string value of input data	string Data();

UDDInputTrace

Method purpose	Return Value	Data type of the returned value in C#
Retrieves x and y double data	Double	IDictionary<double, double> DoubleData();
Retrieves x and y double data for a particular variation	Double	IDictionary<double, double> DoubleData (IDictionary<string, string> variation);
Retrieves x and y complex data	Double	IDictionary<double, double[]> ComplexData();
Retrieves x and y complex data for a particular variation	Double	IDictionary<double, double[]> ComplexData (IDictionary<string, string> variation);
Retrieves x and y text data	Text	IDictionary<string, string> TextData();
Retrieves x and y text data for a particular variation	Text	IDictionary<string, string> TextData (IDictionary<string, string> variation);
Retrieves a list of variable values	A string collection	IList<Dictionary<string, string>> VariableValues();
Retrieves an image of the trace data in a plot	An image	string Image();

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDDInputParams Class

The objects of this class must be created in Python script in the GetUDDInputParams and SetUDDInputParams functions.

Attributes :

- Input Name (string)
- Input Description (string)
- Input Type (string) (This can be boolean, number, text, trace, or solution.)
- BoolData (boolean)
- DoubleData (double)
- TextData (string)
- ReportType (string)
- SolutionName (string)
- DomainName (string)

Constructors:

- UDDInputParams(string name, string description, string type)
- UDDInputParams(string name, string description, string type, bool data)
- UDDInputParams(string name, string description, string type, double data)
- UDDInputParams(string name, string description, string type, string data)
- UDDInputParams(string name, string description, string type, string reportType, string solutionName, string domainName)

Property Accessors :

- Name : Get/Set the name of an input
- Description : Get/Set the description of an input
- Type : Get/Set the type of an input
- BoolData : Get/Set the data of a boolean input
- DoubleData : Get/Set the data of a number input
- TextData : Get/Set the data of a text input
- ReportType : Get/Set the report type
- SolutionName : Get/Set the name of the solution
- DomainName : Get/Set the name of the domain

UDD Input Interfaces

The Generate function takes in a list of inputs which allows the user to access data from the design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IUDDInputBool : This interface exposes 3 methods.

- Name() : Gets the inputs name.
- Type() : Gets the input type.
- Data() : Gets the boolean data, set by the user in the setup dialog.

IUDDInputDouble : This interface exposes 3 methods.

- Name() : Gets the inputs name.
- Type() : Gets the input type.
- Data() : Gets the double data, set by the user in the setup dialog.

IUDDInputText : This interface exposes 3 methods

- Name() : Gets the inputs name.
- Type() : Gets the input type.
- Data() : Gets the text data, set by the user in the setup dialog.

IUDDInputTrace : This interface exposes 9 methods

- Name() : Gets the inputs name.
- Type() : Gets the input type.
- DoubleData() : Method used to return x and y double data as a IDictionary<double, double>
- DoubleData(IDictionary<string, string> variation) : Method used to return x and y double data as a IDictionary<double, double>, given a variation.
- ComplexData() : Method used to return x data and y complex data as a IDictionary<double, double[]>
- ComplexData(IDictionary<string, string> variation) : Method used to return x data and y complex data as a IDictionary<double, double[]>, given a variation.
- TextData() : Method used to return x data and y data as a IDictionary<string, string>
- TextData(IDictionary<string, string> variation) : Method used to return x data and y data as a IDictionary<string, string>, given a variation.
- VariableValues() : Method used to get a list of variations as a IList<Dictionary<string, string>>

IUDDInputSolution : This interface exposes 11 methods

- Name() : Gets the inputs name.
- Type() : Gets the input type.
- DoubleData(string name): Method used to return x and y double data as a IDictionary<double, double>, given a quantity name.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- DoubleData(string name, IDictionary<string, string> variation) : Method used to return x and y double data as a IDictionary<double, double>, given a quantity name and a variation.
- ComplexData(string name) : Method used to return x data and y complex data as a IDictionary<double, double[]>, given a quantity name.
- ComplexData(string name, IDictionary<string, string> variation) : Method used to return x data and y complex data as a IDictionary<double, double[]>, given a quantity name and a variation.
- TextData(string name) : Method used to return x data and y data as a IDictionary<string, string> given a quantity name.
- TextData(string name, IDictionary<string, string> variation) : Method used to return x data and y data as a IDictionary<string, string>, given a quantity name and a variation.
- CategoryNames() : Method to return a list of category names in the solution as an IList<string>
- QuantityNames(string category) : Method to return a list of quantity names in the solution as an IList<string>, given a category.
- VariableValues() : Method used to get a list of variations as a IList<Dictionary<string, string>>

Examples

```
def Generate(self, input, docgen, progMon):
# Getting the boolean data set by the user
boolinput = input[0].Data()
# Getting the double data set by the user
dblinput = input[1].Data()
# Getting the text data set by the user
textinput = input[2].Data()
# Getting the category names in a solution
categories = input[3].CategoryNames()
# Getting the quantity names based on a category
quantities = input[3].QuantityNames(categories[0])
# Getting the XY data from the trace
xydata = input[4].DoubleData()
```

Explication of a Sample UDD Script

This VB script defines a document. It is a portion of one of the UDD example VB scripts.

```
Array("NAME:Test Report",              ' Name of the document
"Test Report",                         ' Description of the document
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
"SysLib",                                   ' Location of the python script: Syslib, Userlib, PersonalLib, etc.
"TestUDDReport",                            ' Relative path of the script in the UserDefinedDocuments folder

' This array is the start of the input definition.
Array("NAME:Inputs",                        ' Document Inputs keyword

' This array contains the Solution input.
Array("NAME:DLMetrics",                     ' Input name
"Solution",                                 ' Solution Input Type
"Data Line Metrics",                        ' Input Description
-1,                                         ' Solution ID
-1),                                        ' Report ID

' This array contains the trace input.
Array("NAME:DQ0",                           ' Input name
"Trace",                                    ' Trace Input Type
"DQ0",                                      ' Input Description
-1,                                         ' Solution ID
-1),                                        ' Report ID

' This array contains the text input.
Array("NAME:Name",                          ' Input name
"Text",                                     ' Text Input Type
"User Name",                                ' Input Description
Array("Sita Ramesh")),                      ' Default Value

' This array contains the Bool input.
Array("NAME:Summary",                       ' Input name
"Bool",                                     ' Boolean Input Type
"Display Summary",                          ' Input Description
Array(true)),                               ' Default Value

' This array contains the number input.
Array("NAME:Version",                       ' Input name
"Number",                                   ' Number Input Type
"Script Version",                           ' Input Description
Array(1021)))),                             ' Default Value
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
' This array contains trace selection for the solution and trace inputs.
Array("NAME:DocTraces",                    ' Document traces keyword

' This array has input for "DLMetrics".
Array("NAME:DLMetrics",                    ' Input name

' This array defines a trace similar to the UDO. This trace definition is a User defined solution
Array("User Defined", "", "DDR3 AC-Timing 4-DQ1", Array("Context:=",
"", Array("Index:=", Array("All"), "Trise:=", Array("Nominal"),
"Tfall:=", Array("Nominal"), "Pulse_Width:=", Array("Nominal"),
"Data_Rate:=", Array("Nominal"), "Length:=", Array("Nominal")), Array
("Probe Component:=", Array("")), Array())),

' This array is for input "DQ0".
Array("NAME:DQ0",

' This array defines a trace similar to the UDO. This trace definition is a Standard solution.
Array("Standard", "DQ0", "NexximTransient", Array("NAME:Context",
"SimValueContext:=", Array(1, 0, 2, 0, false, false, -1, 1, 0, 1, 1,
"", 0, 0, "DE", false, "0", "DP", _
false, "20000000", "DT", false, "0.001", "WE", false, "100ns", "WM",
false, _
"100ns", "WN", false, "0ps", "WS", false, "0ps")), Array("Time:=",
Array("All"), "Trise:=", Array( _
"Nominal"), "Tfall:=", Array("Nominal"), "Pulse_Width:=", Array
("Nominal"), "Data_Rate:=", Array("Nominal"), "Length:=", Array
("Nominal")), Array("Probe Component:=", Array( _
"DQ0")), Array())))
```

Document Generator Interfaces

This document briefly describes the API interfaces available in the document generator plugin.

(Ansys.Ansoft.DocGeneratorPluginDotNet.dll)

Scripting objects available in the script for the Generate function:

- oApp = self.GetUDDAppContext()

  Gets the application context. Use this interface to get the active project ad the version of the product.

  ```
  oDesktop = oApp.GetAppDesktop()
  ```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
if oDesktop != None:
vr = oDesktop.GetVersion()
oProject = oDesktop.GetActiveProject()
```

- oDesign = self.GetUDDDesignContext()

  Gets the design context. Use this interface to get the design name.

```
oDesign = self.GetUDDDesignContext()
  if oDesign != None:
    nm = oDesign.GetName()
```

- IUDDGenerator interface

  This interface available in the Generate method of the UDDPluginExtension. This interface can be used to:

  1. Set the document output file path.
     ```
     docgen.SetOutput("C:\\Examples\\DocumentOutput.xml")
     ```
  2. Get the document root.
     ```
     docroot = docgen.GetDocumentRoot()
     ```
  3. Write out to the output file.
     ```
     docgen.Write()
     ```
  4. Write HTML document
     ```
     void WriteHTML();
     ```
  5. Write PDF document
     ```
     void WritePDF();
     ```
  6. Load the HTML transform object
     ```
     void LoadHTMLTransform();
     ```
  7. Load the cached PDF transform object
     ```
     void LoadPDFTransform();
     ```
  8. Get script path
     ```
     string GetScriptPath();
     ```
  9. Get output file path
     ```
     string GetOutputFilePath();
     ```

- IUDDRoot interface

  Calling GetDocumentRoot() on the IUDDGenerator interface provides you with the this interface. This interface can be used to:

  1. Add a new section to the document. Provide a section title.
     ```
     section1 = docroot.AddSection("Section title")
     ```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Add a new section to the document. Provide a section title and subtitle.
   ```
   section1 = docroot.AddSection("Section title", "Section subtitle")
   ```
3. Add a new title
   ```
   section1 = docroot.AddTitle("Title")
   ```
4. Add a new subtitle
   ```
   section1 = docroot.AddSubtitle("Subtitle")
   ```

- IUDDSection interface

  Calling AddSection() on the IUDDRoot interface provides you with this interface. This interface can be used to:

  1. Set an ID for the section for internal links.
     ```
     section1.SetID("id")
     ```
  2. Add a new table to the document. Provide a table title.
     ```
     table1 = section1.AddTable("Table title")
     ```
  3. Add a new image to the document. Provide an image title and a file path to the image file.
     ```
     image1 = section1.AddImage("Image title")
     ```
  4. Add text to the document.
     ```
     text1 = section1.AddText("Random text........")
     ```

- IUDDImage interface

  Calling AddImage() on the IUDDSection interface provides you with this interface. In this interface, you can call the following methods:

  1. Set an ID for the image for internal links.
     ```
     image1.SetID("id")
     ```
  2. Set alignment information. Can be "center", "left", and "right".
     ```
     image1.SetAligment("center")
     ```
  3. Set the file path of the image file. Not necessary if image file path is set through the AddImage() method.
     ```
     image1.SetFileRef("Image path")
     ```
  4. Set the format of the image file. Can be "BMP", "PNG", "JPEG", "JPG", "DVI", etc.
     ```
     image1.SetFormat("format")
     ```

- IUDDText interface

  Calling AddText() on the IUDDSection interface provides you with the this interface. In this interface, you can call the following methods:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set an ID for the text for internal links.
   ```
   text1.SetID("id")
   ```
2. Set the emphasis attribute on the text.
   ```
   text1.SetEmphasis()
   ```
3. Set the quotes attribute on the text.
   ```
   Text1.SetQuotes()
   ```
4. Set the block quotes attribute on the text.
   ```
   text1.SetBlockquotes()
   ```
5. Set quotes on the text.
   ```
   Text1.SetQuotes()
   ```
6. Set the wordsize attribute on the text
   ```
   text1.SetSize(size as an integer)
   ```
7. Set a link to an ID of any element to provide internal links
   ```
   text1.SetLink("linkname")
   ```
8. Set an event link to handle an event. The HandleUDDEvents method should be implemented in the script to handle the event.
   ```
   text1.SetEventLink("linkname")
   ```

- IUDDTable interface

  Calling AddTable() on the IUDDSection interface provides you with the this interface. In this interface, you can call the following methods:

  1. Set an ID for the table for internal links.
     ```
     table1.SetID("id")
     ```
  2. Set alignment information. Can be "center", "left", and "right".
     ```
     table1.SetAlignment("center")
     ```
  3. Set the background color of the table
     ```
     table1.SetBgColor(string bgcolor)
     ```
  4. Set the frame type. Can be "all", "bottom", "top", "sides", and "topbot".
     ```
     table1.SetFrame(string frame)
     ```
  5. Add a table group and specify the number of columns. A table can have multiple table groups.
     ```
     IUDDTableGroup table1.SetTableGroup(int columns)
     ```

- IUDDTableGroup interface

  Calling AddTableGroup() on the IUDDTable interface provides you with the this interface. In this interface, you can call the following methods:

  1. Set an ID for the table group for internal links.
     ```
     tgroup1.SetID("id")
     ```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Set alignment information. Can be "center", "left", and "right".
   ```
   tgroup1.SetAlignment("center")
   ```
3. Set the column width of a column given the index of the column and the required width. Width can be set in 2 ways.
   - Width can be set relative to 1. E.g Setting it to "2*" makes the column width double the width of the others.
   - If the entire table width is considered to be 99.99 units. Width can be a number relative to this.

   ```
   tgroup1.SetColumnWidth(int index, string width)
   ```
4. Add a header to the table group
   ```
   IUDDTableRow tgroup1.AddHeader()
   ```
5. Add a header with multiple rows to the table group. Takes number of sub rows.
   ```
   IUDDTableRow tgroup1.AddHeader(int rows)
   ```
6. Add a row of content to the table group
   ```
   IUDDTableRow tgroup1.AddContent()
   ```
7. Add content with multiple rows to the table group. Takes number of sub rows.
   ```
   IUDDTableRow tgroup1.AddContent(int rows)
   ```

- IUDDTableRow interface

  Calling AddHeader() & AddContent() on the IUDDTableGroup interface provides you with the this interface. In this interface, you can call the following methods:

1. Set an ID for the table row for internal links.
   ```
   trow1.SetID("id")
   ```
2. Set alignment information. Can be "center", "left", and "right".
   ```
   trow1.SetAlignment("center")
   ```
3. Set cell text. Can be cell content or header text. Takes a column index and a text string. It is added to the first row.
   ```
   IUDDTextElement trow1.Add(int column, string text)
   ```
4. Set cell text. Can be cell content or header text. Takes a column index, row index and a text string. Takes in a row number because a table row can have multiple sub rows.
   ```
   IUDDTextElement trow1.Add(int column, int subrow, string
   text)
   ```
5. Set cell content. Takes a column index and an int value. It is added to the first row.
   ```
   IUDDTextElement trow1.Add(int column, int value)
   ```
6. Set cell content. Takes a column index, row index and a int value.
   ```
   IUDDTextElement trow1.Add(int column, int subrow, string
   text)
   ```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Set cell text. Takes a column index and a double value. It is added to the first row.
   ```
   IUDDTextElement trow1.Add(int column, double value)
   ```
8. Set cell text. Takes a column index, row index and a double value.
   ```
   IUDDTextElement trow1.Add(int column, int subrow, double value)
   ```
9. Set cell text spanning 2 columns. Can be cell content or header text. Takes a sub row index , starting column index., ending column index and a text string.
   ```
   IUDDTextElement trow1.AddSpanningcolumnst(int subrow, int columnstart, int columnend, string text)
   ```
10. Set cell text. Can be cell content or header text. Takes a column index, starting sub row index, ending sub row index and a text string.
    ```
    IUDDTextElement trow1.AddpanningRows(int column, int subrowstart, int subrowend, string text)
    ```

- IUDDTableRow interface

  Calling Add() on the IUDDTableGroup interface provides you with the this interface. In this interface, you can call the following methods :

  1. Set an ID for the table row for internal links.
     ```
     trow1.SetID("id")
     ```
  2. Set alignment information . can be "center", "left", and "right".
     ```
     trow1.SetAlignment("center")
     ```

Includes all the methods exposed by the IUDDText interface.

Toolkit

The Toolkit command in the solver menu provides access to a design type specific IronPython script, such as the Hearing Aid Compliance Test, MIMO calculation or a module specific task, such as for Boundary, Excitation, or Solve Setup. The Hearing Aid Compliance Test toolkit example involves multiple modules. The MIMO example includes a Toolkit script and a User Defined Solution Script. The Cable modeling toolkits serve automotive cable applications and oil and gas applications.

An **MI Materials Gateway** toolkit that can interact with the Granta Gateway to bring in new materials at a project level, and so, it appears on the **Tools>Toolkit** menu, rather than the **<solver>Toolkit** menu. It works in conjuction with scripting commands.

The python script provides default but customizable UI and canned automations. The UDD format can be adjusted/extended as needed. The default UI is a modal dialog.

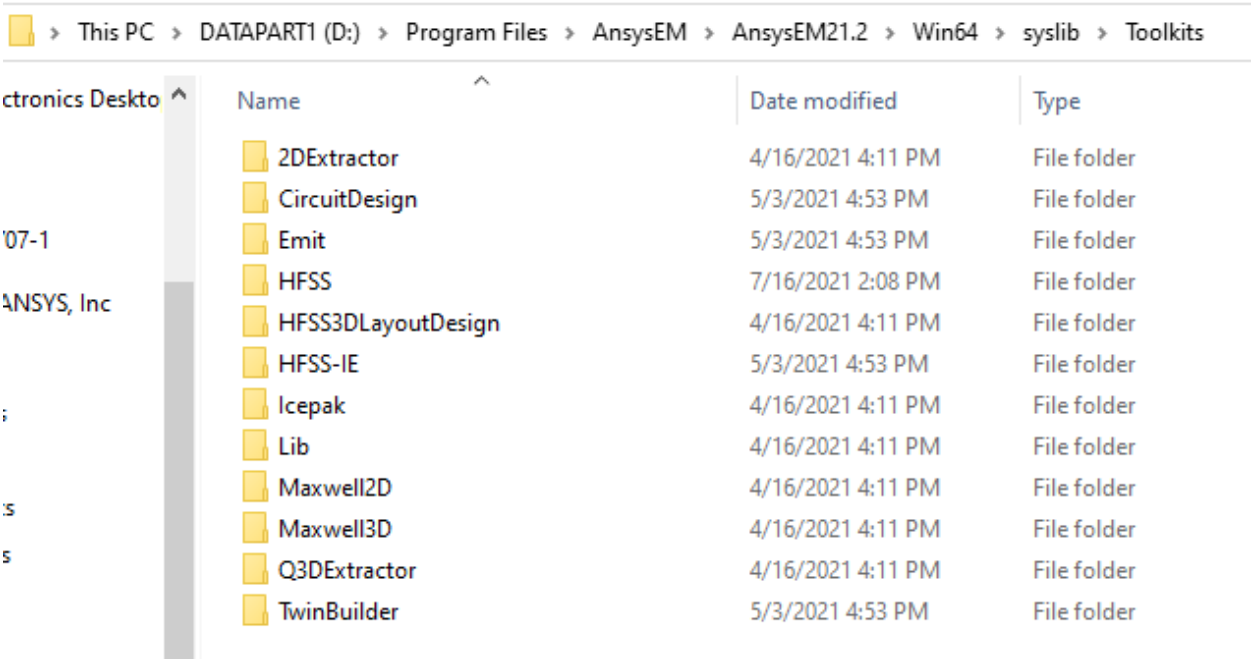
This allows us to support changing standards, or multiple standards, without necessarily being tied to the product release cycle. Similarly, you can create new UDD to customize the test report.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDD are saved in project result directory. This type of reports can be created prior to simulation, but will only be populated when there is solution.

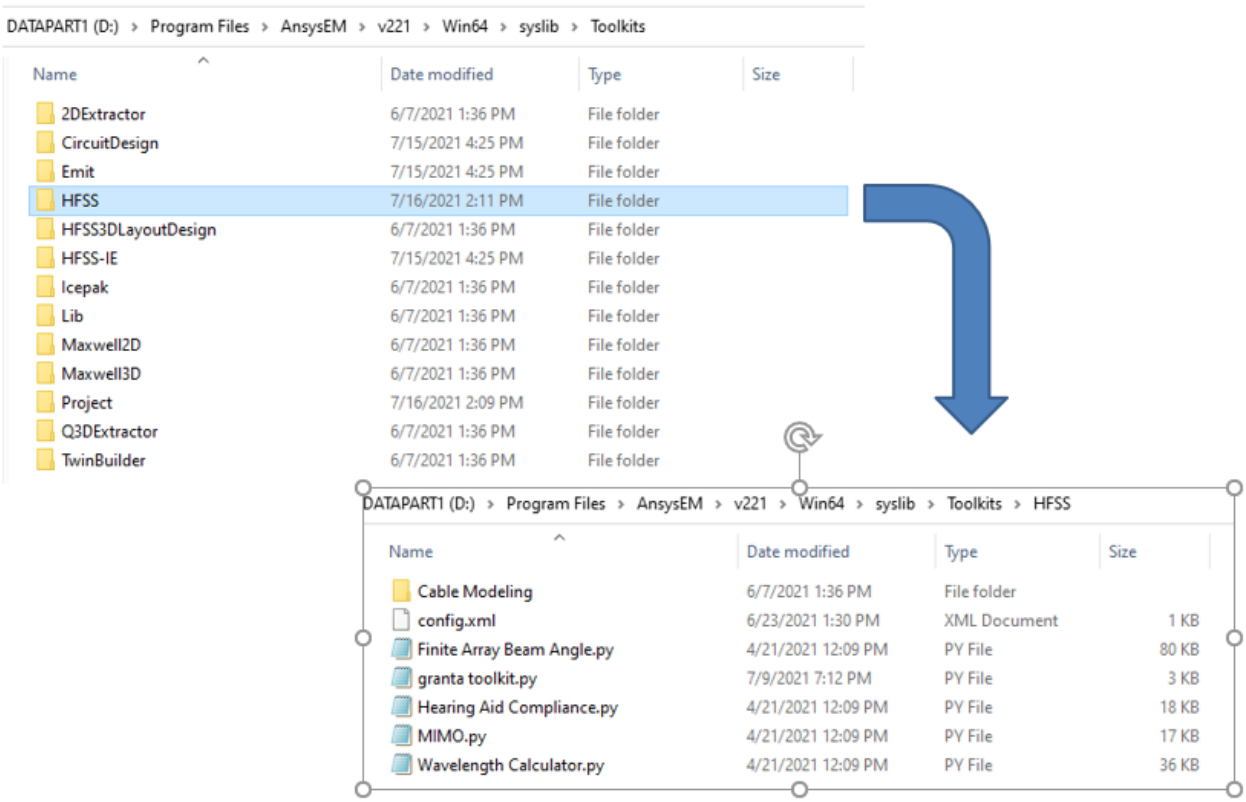
Undo/Redo/Scripting is supported. If a script includes multiple commands, it takes multiple undo (s) to revert all the changes made by the python script. Also, the script playback might not work if the baseline of the design has changed.

A directory called "Toolkits" appears in the syslib, userlib and personallib of HFSS installation. Menu items for files found in <installation>/syslib will be inserted at the first level of the menu.



There are product specific directories in "Toolkits" and each directory contains toolkits (scripts) that are specific for that product. The following snapshot illustrates the file structure on disk that provides access to the "HearingAidCompliance" toolkit at the design level menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Chip Model Analyzer (CMA)

Ansys Chip Model Analyzer (CMA) is an advanced Chip Power Model (CPM) creation tool for performance checking and failure diagnosis of the Power Delivery Network (PDN). CMA can use either HSpice or Nexxim simulators. HSpice and external Nexxim require licenses, but the Nexxim Internal simulator does not.

CMA does not currently interact with Electronics Desktop. However, you can launch CMA from within Electronics Desktop by clicking **Tools** > **Chip Model Analyzer (CMA)**.

From there, you can import CPM files for analysis or create a pseudo CPM.

For more information about using CMA, consult the CMA training documentation by launching CMA and selecting **Help** > **Content**.

PinToPinUtility

The PinToPin utility can quickly extract HFSS or SIwave models for specified nets of package and PCB geometries. It establishes a repeatable process for robust geometry extraction, port configuration, and passive model assignment. Process execution can be done via the user interface (Windows only) or in non-graphical mode (Windows or Linux).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information, consult the SIwave and HFSS 3D Layout help.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

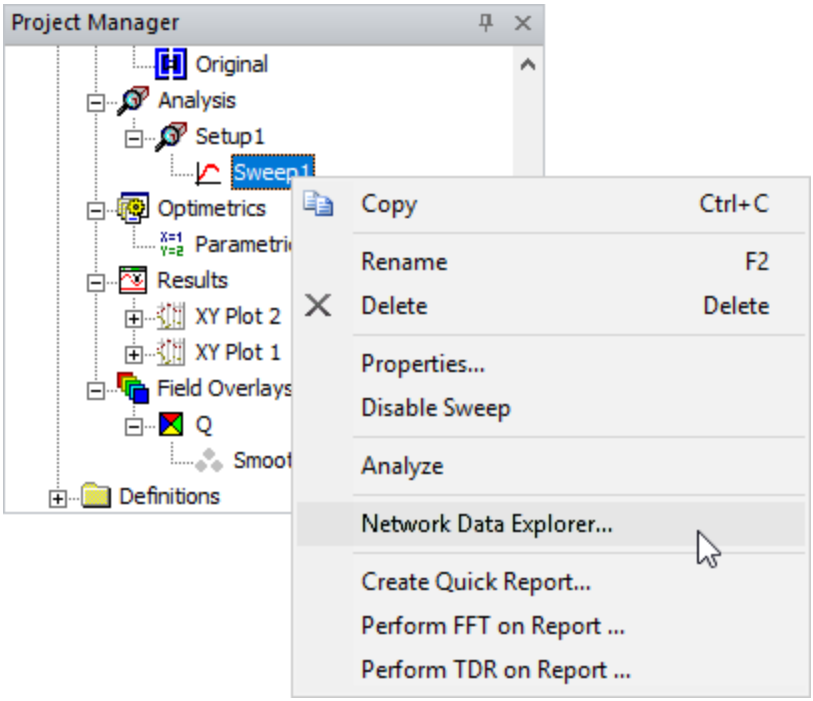
24 - Network Data Explorer

The Network Data Explorer provides visualization, analysis, and manipulation tools for network data.

To access Network Data Explorer, click **Tools** > **Network Data Explorer**.

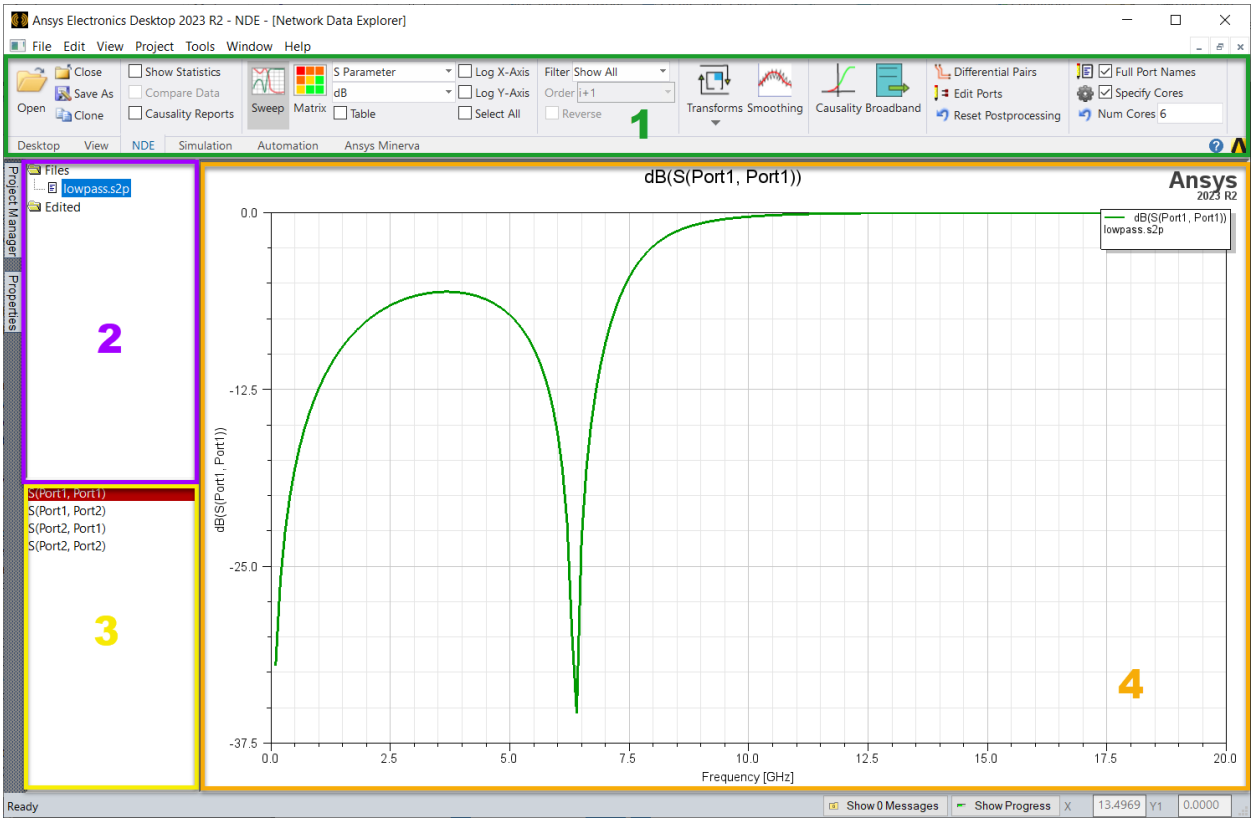
If you have added a Frequency Sweep, you can also access Network Data Explorer through the Sweep shortcut menu in the Project Manager.

> **Note:**
>
> Accessing Network Data Explorer from this menu automatically loads sweep data.



The **Network Data Explorer** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



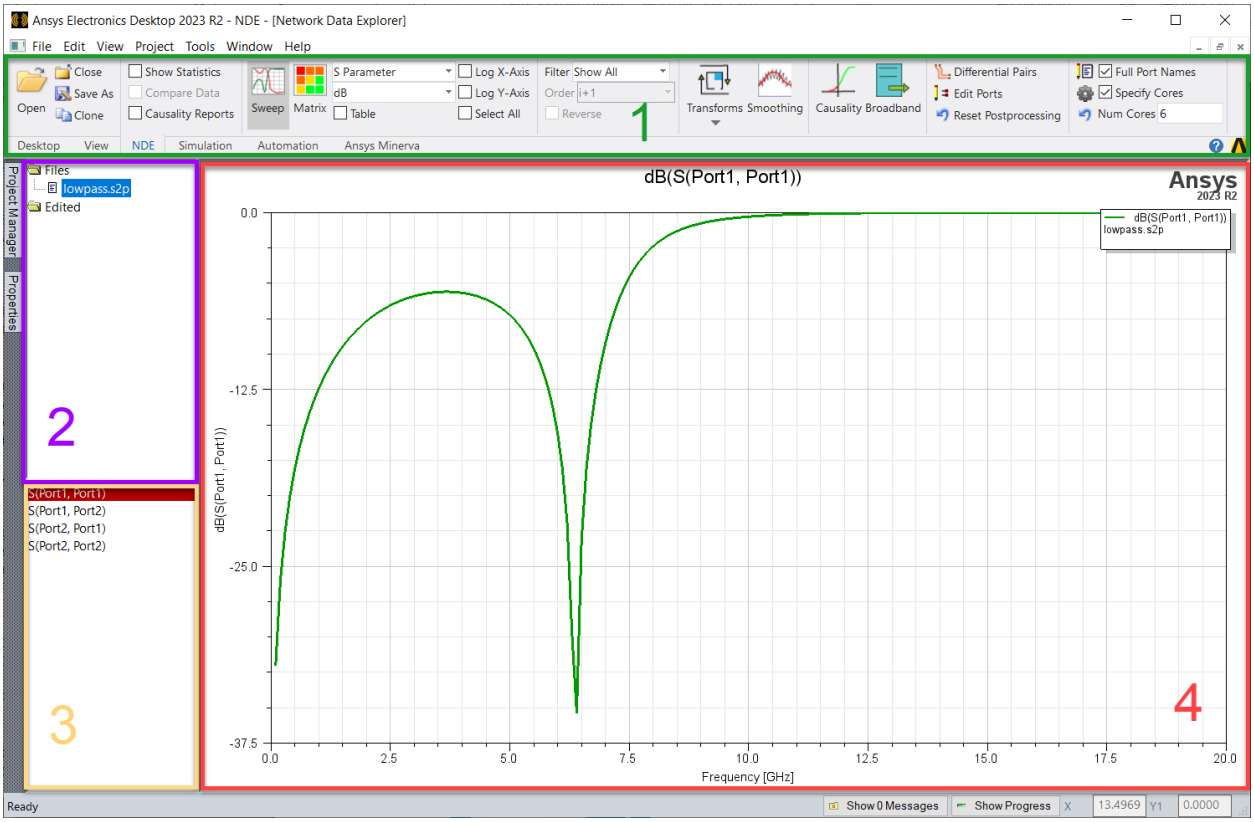
Network Data Explorer Overview

The Network Data Explorer window is divided into the following panes:

1. **NDE ribbon** – allows you to perform many functions of the Network Data Explorer.
2. **Network Data Selection pane** – allows you to select a network data file.
3. **Cell and Frequency Selection pane** – allows you to narrow your selection.
4. **Data View pane** – displays data in table or plot format.

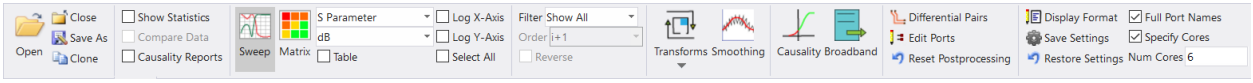
The panes are shown in the following figure. Additional information about each pane follows.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



NDE Ribbon

The ribbon provides access to many of the Network Data Explorer's functions and display options.



On this ribbon, you can control:

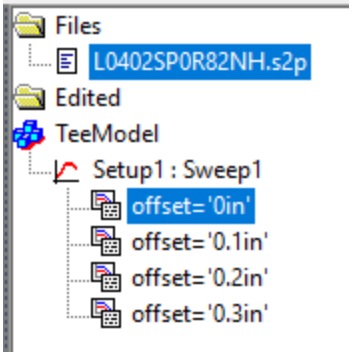
- Plotting - allows you to determine how the data is displayed.
- Quantity – allows you to select the type of quantity to display (parameter values, matrix statistics, or causality plots).
- Parameter Type – allows you to choose the parameter for display (S, Y, or Z parameters, Port Impedance, or Gamma).
- Format – allows you to decide the display function to apply to the data (e.g., magnitude, phase, dB, real, imaginary).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Export – allows you to export either SYZ data (*.s1p, *.ts, *.nmf, *.tab, *.m, *.cit) or Broadband data (*.sp).
- Check – allows you to check causality.
- Cores – allows you to enable or disable multithreading.
- Post-Process Selection – allows you to choose between Terminal Data and Differential Pairs, if your design includes Differential Pairs

Network Data Selection Pane

This pane allows you to view and compare various data sets. Original data sets appear under **Files**. Click one of these to see the data set as it was when it was opened. Altered data sets are listed under **Edited**. These data sets appear here when they have been smoothed, transformed, or changed in some way.



This pane also lists available variations for AEDT design solution data. Variations are listed under a setup name icon (for example “Setup1 : Sweep1”) that is listed under a design icon (“TeeModel”). Variations can be selected and displayed just like other data sets.

Use the **Shift** and **Ctrl** keys to select and display data sets from multiple files simultaneously. Multiple data sets can be selected to display multiple traces in sweep plots, but not in tables. Click the **Files** icon to select all data sets under **Files**. Click the **Edited** icon or a design/setup icon to do the same.

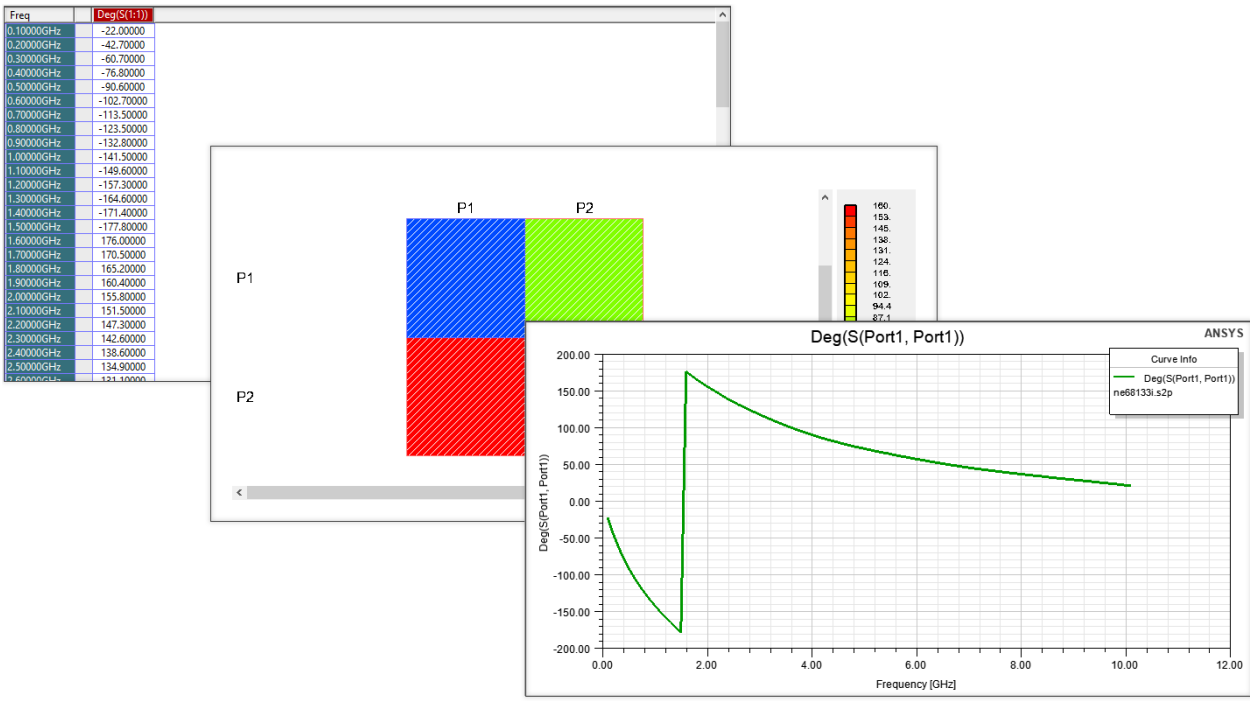
Cell and Frequency Selection Pane

Select the **Sweep** button on the NDE ribbon to display cell entries, for example S(1,1). Select the **Matrix** button to display frequencies. When displayed by frequency, the entire matrix is presented in the Data View pane for each selected frequency. When displayed by matrix cell, the data for the individually chosen cells is shown across all frequencies. Use the **Select All** check box to select all frequencies or cells.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Data View Pane

The Data View Pane displays data for **Sweep** or **Matrix** in either plots or a table, depending on your selection.



Loading Data into Network Data Explorer

You can launch **Network Data Explorer** from within several Ansys products. In the Project Manager, open <project> > **Analysis** > **Setup***n* > **Sweep***n*. Right click on the sweep and click **Network Data Explorer** in the shortcut menu. The current solution data is automatically loaded and ready for viewing. Otherwise, you must load a data file into Network Data Explorer.

> **Note:** When solution data that is loaded into NDE is modified and resimulated in another Ansys product, the NDE data automatically updates.

You can import the following file types:

- Touchstone Format (*.s*p)
- Touchstone 2 Format (*.ts, *.sp)
- Citifile (*.cit)
- Neutral Format (*.nmf)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- State Space File (*.sss)

> **Note:**
>
> When this type of file is loaded, Network Data Explorer regenerates s-parameter data based on the file.
>
> You can compare the regenerated s-parameters to the original data.

To import a file into Network Data Explorer, either drag and drop an analysis from the Project Manager into Network Data Explorer or

1. On the **NDE** ribbon, click **Open**. An **Open** window opens.
2. Navigate to and select a file.
3. Click **Open**. The file appears in the **Files** tree.

The file browser allows you to open multiple files at a time. However, the displayed data always corresponds to the data set indicated in the Network Data Selection pane. Click the file you want in that pane to switch between data sets.

Exporting Data from Network Data Explorer

Network Data Explorer allows you to export data to a variety of different file formats.

In this section, you will learn about:

Exporting SYZ Data

Exporting Macro Model

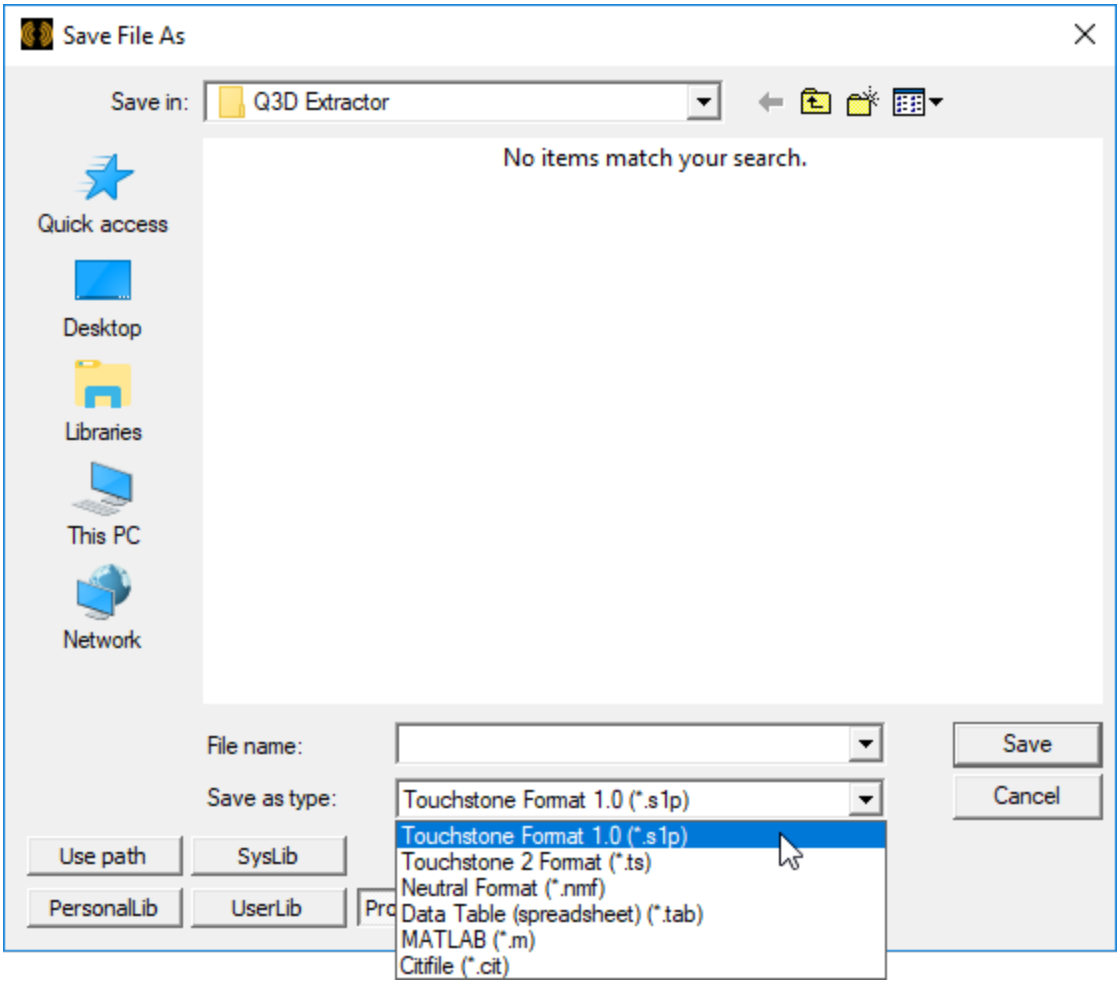
Creating an NPort Model

Scripting for Network Data Explorer

Exporting SYZ Data

To export SYZ data from within Network Data Explorer, click the **Save As** icon on the **NDE** ribbon. The **Save File As** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

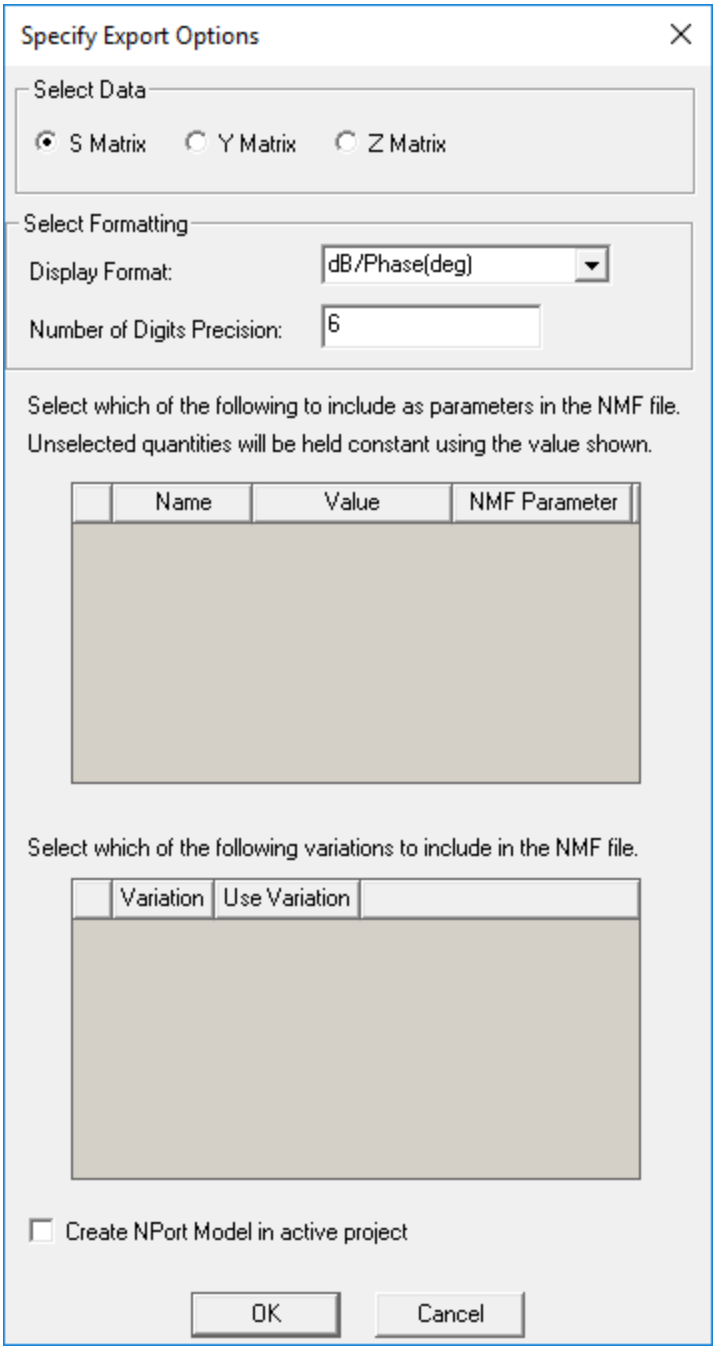


You can export data in any of six file types:

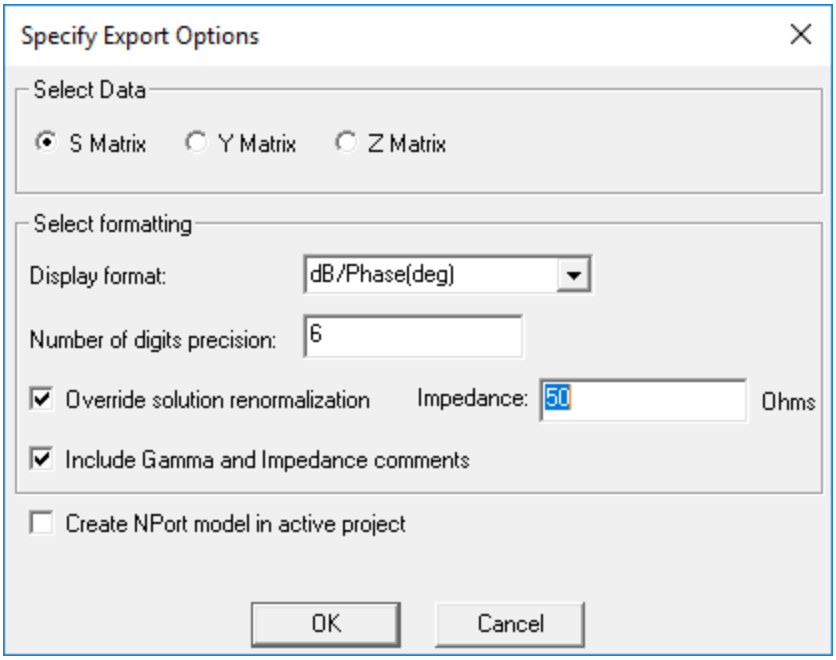
- Touchstone Format 1.0 (*.s*p)
- Touchstone 2 Format (*.ts)
- Neutral Format (*.nmf)
- Data Table Spreadsheet (*.tab)
- MATLAB (*.m)
- Citifile (*.cit)

Select a file type and name for export. A **Specify Export Options** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Depending on the type of export file, different options appear. However, all file types allow you to:

- Select from **S Matrix**, **Y Matrix**, and **Z Matrix** data.
- Select the **Display Format**.

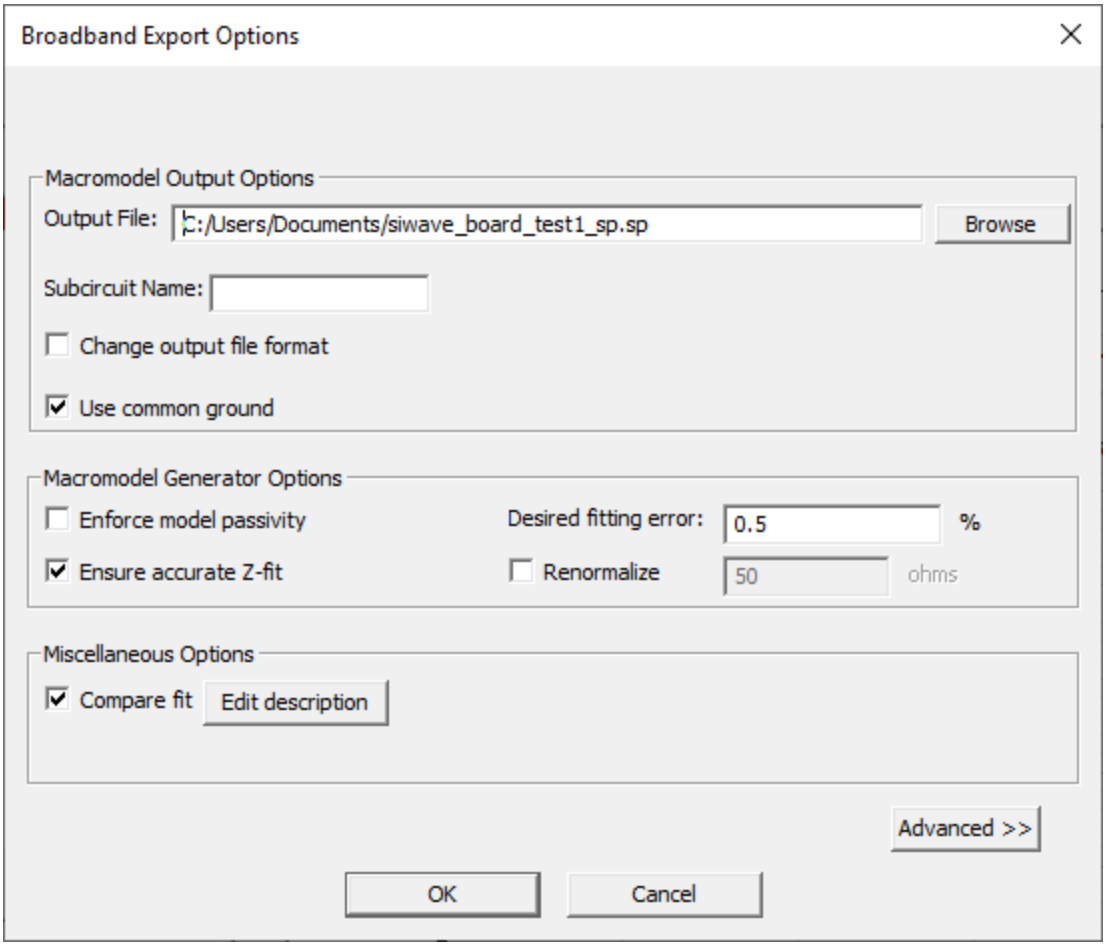
Some types allow you to create an NPort Model in the active project.

Exporting Macro Model

Network Data Explorer lets you export macro model data. To export data, click the **Broadband** icon on the **NDE** ribbon.

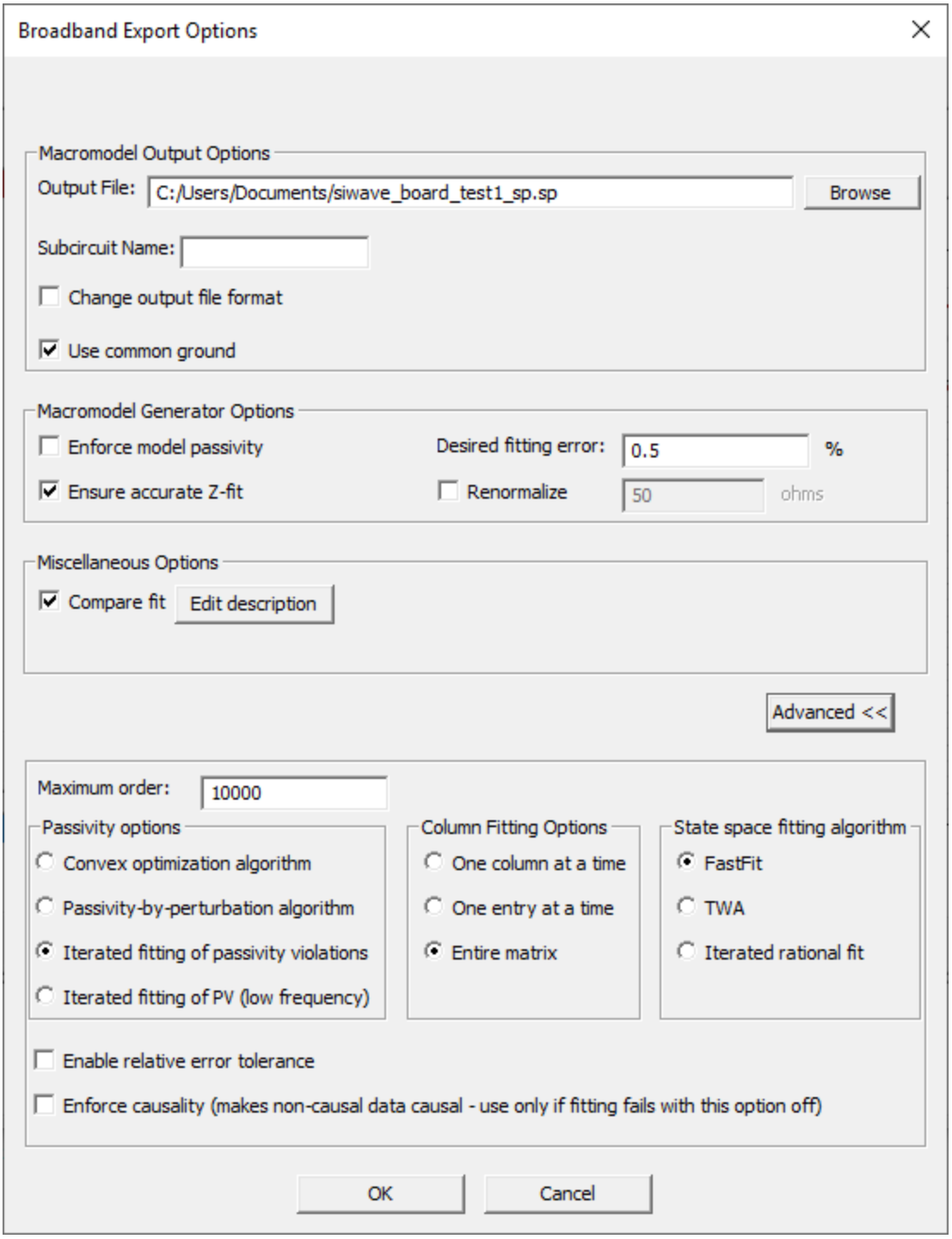
The **Broadband Export Options** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Click **Advanced >>** to view all options.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Macromodel Output Options include:

- **Output File** – Allows you to choose the name and location of the file.
- **Subcircuit Name** – Use this field to name the subcircuit.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Change Output File Format** – Check this box to open a submenu allowing you to select a new output format.
- **Use Common Ground** – Check this box to use common ground. When this option is on, ports are referenced to ideal ground (node 0). When this option is off, extra ports are generated to provide the reference levels. Common grounding is best when the pins are physically near to each other and ideal ground is suitable. For distant connections and circuits with non-ideal reference levels such as differential pairs, common grounding is not used.

> **Note:**
>
> - R and L values may be quite sensitive to the values of the S-parameters. This is an issue if the actual impedance value is much greater than or much less than the reference impedance of the S-parameters.
> - Since resistances of power cables is typically in the milliohms range at DC, using a reference impedance of 50 ohms is 5000 times higher. This causes any fitting errors in the state space model to get multiplied by 5000 times when the R and L values are computed.
> - As a general rule, for high power applications a reference impedance of 1 ohm is probably a better choice than 50 ohms.

Macromodel Generator Options include:

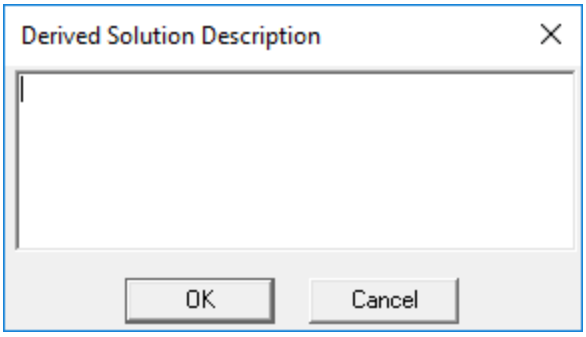
- **Enforce Model Passivity** – Check this box to enforce passivity.
- **Ensure Accurate Z-fit** – Check this box when state-space fitting of Y-parameters or Z-parameters does not produce an accurate fit.
- **Desired Fitting Error** – Allows you to select the value at which rational fitting fails (if the fitting error exceeds this value).
- **Renormalize** – Check this box to renormalize using the specified impedance item. 50 ohms is the default setting, but you can type a different value.

> **Note:**
>
> - R and L values may be sensitive to S-parameter values. This presents an issue if the actual impedance value is much greater than or much less than the reference impedance of the S-parameters.
> - Since resistances of power cables are typically in the milliohms range at DC, using a reference impedance of 50 ohms is 5000 times higher. This causes any fitting errors in the state space model to be multiplied by 5000 when the R and L values are computed.
> - For high-power applications, a reference impedance of 1 ohm is generally a better choice than 50 ohms.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Miscellaneous Options include:

- **Compare Fit** – When this box is checked, the original and derived solution will be available for comparison. You can click **Edit Description** to open the **Derived Solution Description** window and add a text description to better identify the export.



Advanced Options include:

- **Maximum Order** – Allows you to specify the number of poles. See Note below.
- **Passivity Options** – If you enabled **Enforce Model Passivity**, this area allows you to select the passivity enforcement method.
  - **Convex optimization algorithm** – guarantees a passive state-space realization, but is very slow and memory-intensive. Not practical for numbers of ports beyond ten.
  - **Passivity-by-perturbation algorithm** – designed for systems with a large number of ports. Less accurate than the Convex optimization method.
  - **Iterated fitting of passivity violations (IFPV)** (default) – less accurate than other algorithms but more suitable for larger numbers of ports.
  - **Iterated fitting of PV (low frequency)** – similar to IFPV while improving the fit to "Z" at DC and low frequencies. A better choice of passivity enforcement when the fit to corresponding "Z-data" is important such as power delivery, EMI/EMC applications.

    > **Note:**
    >
    > For a more detailed explanation on any of the passivity options, see the technical note section of the Circuit help.

- **Column Fitting Options** – This area allows you to choose how poles are matched to columns:
  - **One Column at a Time** – The set of poles will be shared across all entries of a single column.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **One Entry at a Time** – Each entry will be fitted using a separate set of poles.
- **Entire Matrix** – The set of poles will be shared across all entries of the matrix being fitted.

> **Note:**
>
> - Typically, using the same set for all entries is adequate, and yields the most compact models. However, if all the entries of the matrix have completely unrelated transfer functions, it may be better to fit them using separate pole sets.
> - The options **One column at a time** and **One entry at a time** do not work when either **Ensure accurate *Z*-fit** or FastFit is used.

- **State Space Fitting Algorithm** – Allows you to select FastFit, TWA, Iterated rational fitting.
  - **FastFit** (default) – FastFit is the Ansys-proprietary method for state-space fitting. Network Data Explorer uses FastFit for calculating the state-space matrices from the network data. The FastFit algorithm for state-space fitting is an alternative to the Tsuk-White algorithm (TWA) and Iterated Rational Fitting (IRF) methods. FastFit is generally as accurate as TWA, but is significantly faster than both TWA and IRF. It also aims to fit the lower frequencies with higher fidelity.
  - **TWA** – The Tsuk-White Algorithm is an Ansys-proprietary method for fitting a state space model to extracted s-parameter data. It uses techniques based on Singular Value Decomposition (SVD) to quickly determine required number of poles for fitting a model.
  - **Iterated Rational Function** – The IRF fitting approach takes a matrix of S-parameter data and, for each matrix entry, tries a succession of different pole-zero approximations (increasing the number of poles used at each iteration) until it can find an acceptable fit to the data. For broad frequency sweeps and large numbers of excitations, this process can be time consuming because of all the iterations and is not guaranteed to produce a good fit to the data. It is retained as a fallback if the TWA algorithm fails.
- **Enable Relative Error Tolerance** – Allows you to enable relative error tolerance, which works best with TWA fitting.

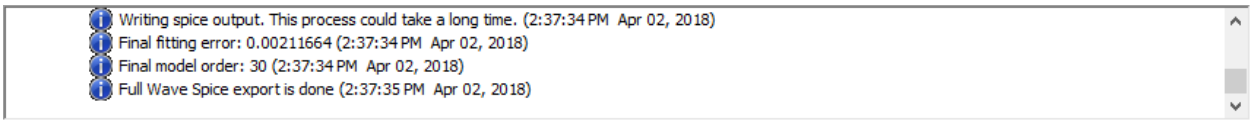
> **Note:**
>
> The **Enable Relative Error Tolerance** option works best with the TWA fitting algorithm, is not recommended for use with iterated rational fitting, and is disabled when either FastFit or **Ensure accurate Z-fit** is used.

- **Enforce Causality** – Allows you to make non-causal data causal. Use this option only if fitting fails without it.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:**
>
> Broadband models are built from a rational-function approximation of the data. The fidelity of this approximation can be controlled by setting the Maximum order (number of poles).

Click **OK** to begin the export. The **Messages** pane details the export process.



Comparing Original S-Parameters with Exported S-Parameters

If **Compare Fit** was checked during export, the **Data Selection** pane updates to list both the original and exported solution *and* the **Compare** checkbox is checked.

Creating an NPort Model

Touchstone, Neutral Format, and Citifile exports allow you to select an option to **Create NPort Model in active project**.

Creating an NPort Model exports the active data set back to the design, either as a static n-port model or as a parametric n-port model. If there are multiple variations, a parametric n-port model is automatically created containing all variations. When only a single variation exists, the user can create a static n-port model that either links to a file (the data is exported first) or stores the data itself. Linking to a file reduces the size of the ADSN file. The newly created model can be placed as a component in a circuit. This is particularly useful after reducing the number of ports via termination.

The import of the exported solution is done by reading the exported Touchstone file.

Scripting for Network Data Explorer

Scripting is available for each Network Data Explorer export method, which means a script can be recorded to duplicate the export process.

Network Data Explorer can be invoked in the following contexts, and scripting is available from the Project Context and the Design Instance Context (simulation setup):

- **Project Context** – In the context of a project, you can open a touchstone file and then export.
- **Design Instance Context** – In the context of a solution (RCM from the simulation setup in a design), you can export the corresponding network data solution.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

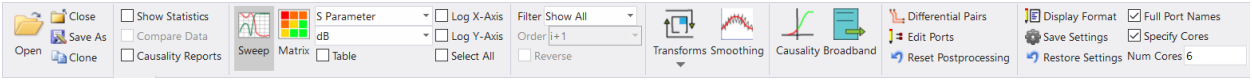
When there is no design available, export functionality and scripting are not available.

When there is no design available, export functionality and scripting are not available.

For more information, see Network Data Explorer Script Commands in the *Ansys Electronics Desktop Scripting Guide*.

Network Data Explorer Ribbon

The **NDE** ribbon provides access to many of the Network Data Explorer's functions.



> **Note:**
>
> The **Define Differential Pairs** command is disabled in HFSS because it handles differential pairs data differently from Network Data Explorer. If you have defined differential pairs in HFSS, you can see them by using the Post Process Selection menu.

In this section, you will learn about:

Data Sources

Setting Display Format

Displaying Full Port Names

Saving or Resetting Default Settings

Smoothing All Frequencies

Cell Filtering

Editing Port Properties

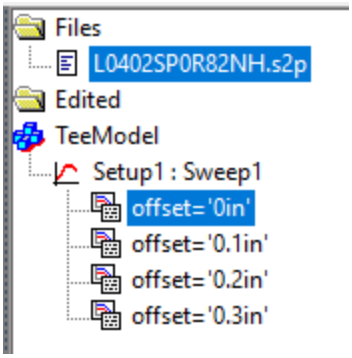
Defining Differential Pairs and Displaying Mixed Mode Parameters

Resetting All Port Properties

Network Data Explorer Data Sources

Network Data Explorer allows you to easily view all data sources in the **Network Data Selection** pane.

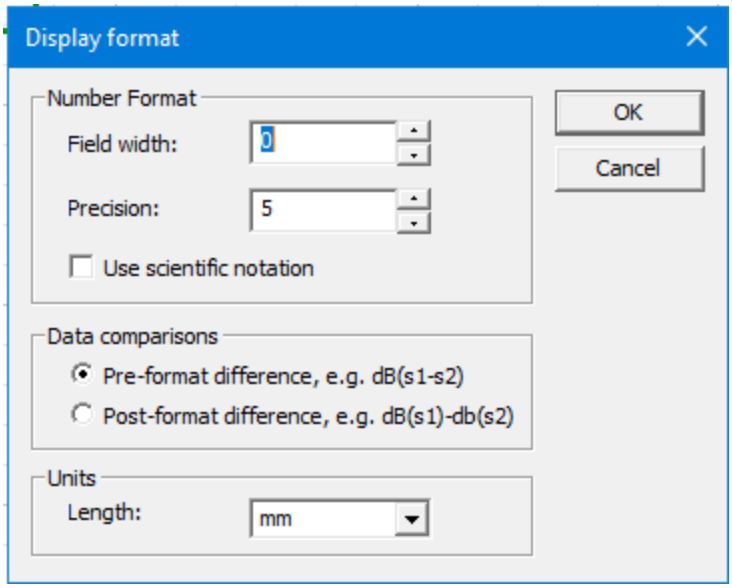
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Network Data Explorer Display Format

The **Display Format** window affords additional control over the display of values in Network Data Explorer.On the **NDE** ribbon, click the **Display Format** icon.

The **Display Format** window appears.



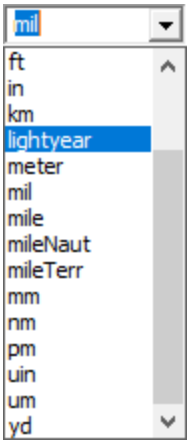
The **Number Format** options allow you to specify **Field Width** (the minimum number of characters used to display a number) and **Precision** (the number of decimals to display). You can also check the **Use scientific notation** check box, if desired.

The **Data comparisons** options allow you to choose Post-format difference or Pre-format difference.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Pre-format difference**- when comparing data sets, subtract values before applying the formatting function (e.g. dB, magnitude); the values displayed will be the magnitude, dB, etc., of the complex difference.
- **Post-format difference** – when comparing data sets, subtract values after applying the formatting function (e.g. dB, magnitude); the values displayed will be the difference between the magnitude, dB, and so on.

The **Units** option allows you to specify the **Length** unit (the unit used to display and interpret length values). The default is mm.

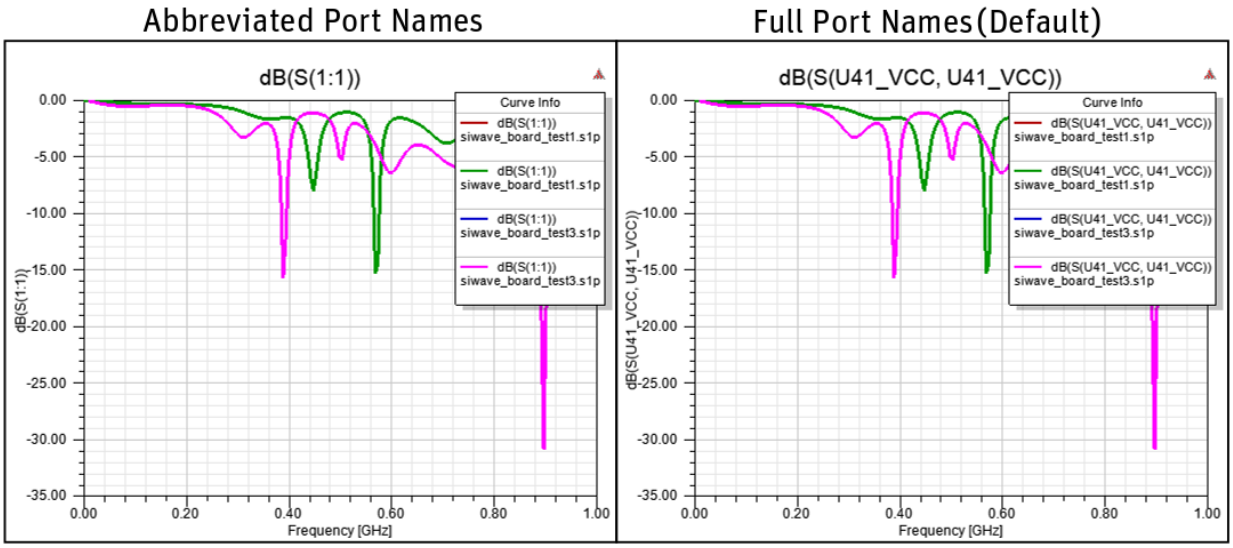


Network Data Explorer Display Full Port Names

By default, full port names are displayed. This applies to both the Data Selection pane and the Data View pane. To change this so that port names in Network Data Explorer are displayed in an abbreviated form (P1, P2, etc.), click the **Full Port Names** check box on the **NDE** ribbon.

The following figure shows the difference in display for a plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

Tool-tips always display the full port name.

Network Data Explorer Save or Reset Default Settings

To save field settings as the default, click the **Save Settings** icon on the **NDE** ribbon. The next time Network Data Explorer is opened, the chosen settings will be selected by default.

To restore the default settings (i.e. the settings previously saved using the **Save Settings** button), click the **Restore Settings** icon on the **NDE** ribbon. The next time Network Data Explorer is opened, the original settings will be selected by default.

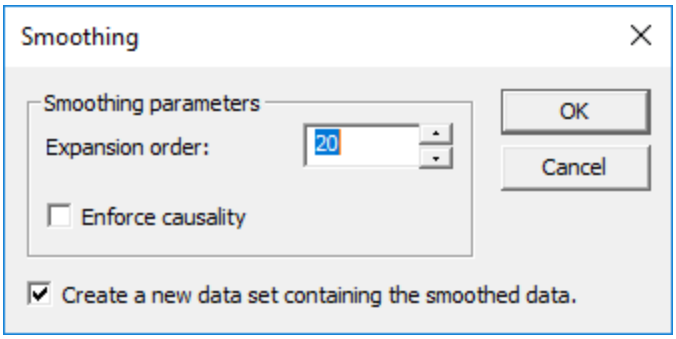
Smoothing

To access data smoothing options:

1. Click the **Smoothing** icon on the **NDE** ribbon.

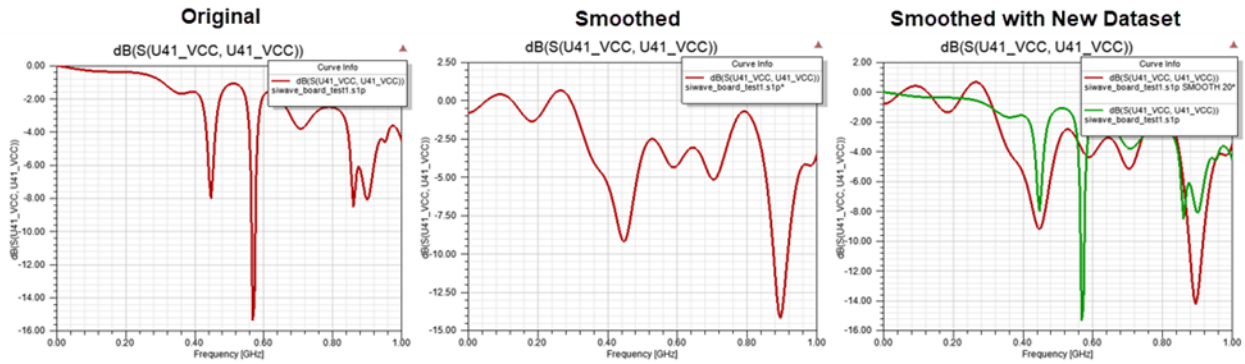
   The **Smoothing** window appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. The **Smoothing Parameters** area allows you to choose the **Expansion Order**. This can be any discrete value between 1 and 150.
3. If desired, check the **Enforce Causality** check box.
4. If desired, check the **Create a new data set containing the smoothed data** check box. If selected, the smoothed data appears alongside the original data.
5. Click **OK**.

The Data View Pane updates. The following image shows a plotted result with **Create a new data set containing the smoothed data** unchecked and checked.

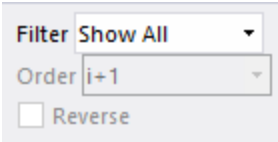


A least-squares polynomial fit of the specified order is used to interpolate new data points for the magnitude and phase components of the S-parameters.

Cell Filtering

The cells available in the data selection pane may be restricted using cell filtering. The **Cell Filtering** control are located on the **NDE** ribbon.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Cell filtering is modeless, and filters are immediately applied to the cell list. Filtering remains in effect when the window has been closed.

For an n-port model with a total of 2n pins in the standard arrangement, the choices are:

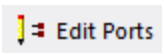
- **Show all** – display all available cells. There are n-squared choices.
- **Return loss** – show S(i, i). There are n choices.
- **Insertion loss** – show S(i, i+1). There are n choices.
- **Lower triangle** – show S(i, j) for all j < i. There are n(n-1)/2 choices.

Three pin arrangements are recognized:

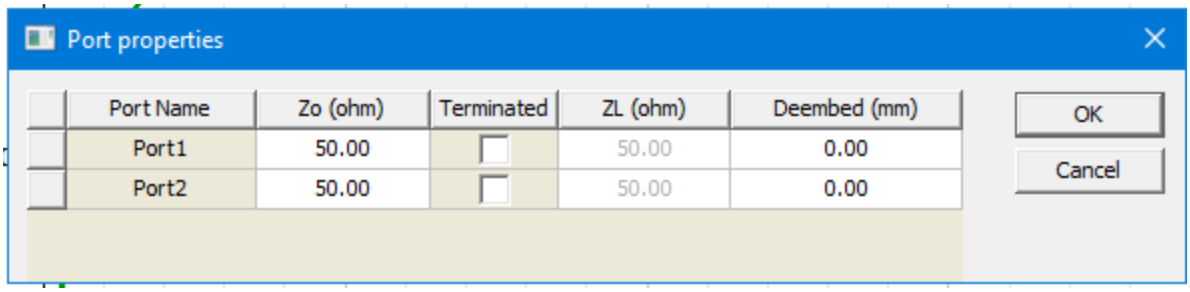
- S(i, **i+1**) and its **Reverse order**, S(i, i-1)
- S(i, **i+n**) and its **Reverse order**, S(i, i-n)
- S(i, **2*n-i+1**) and its **Reverse order**, S(i, 2*n-i-1)

Changing Port Properties and Reducing Matrix Size

The normalization impedance, termination, port order, gamma values, and de-embedding distance may all be edited from the **Port Properties** window.

To access these options, click **Edit Ports** 

on the **NDE** tab. The **Ports Properties** window opens.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

> **Note:** For HFSS Driven Terminal designs, selecting the solution data will disable this ribbon button. The user must make port/differential pair changes directly in the HFSS Design and re-import the solution.

Ports appear in a table. Click a column heading to sort by that column. Click within a cell to edit the port property:

- **Zo (ohm)** and **ZL (ohm)** – specify Impedance values. Accepted syntaxes are:
  - real (e.g., 50)
  - real + imag i (e.g., 50+5i)
  - imag i (e.g., 5i).
- **Terminated** – use the check box to terminate a port. Terminated ports are eliminated from the matrix, reducing the matrix size. Existing data sets with mismatching port numbers will no longer be available for data comparisons.
- **De-Embedding** – this column appears only if gamma values are available. Default units can be changed from the Set Display Format window.

To reorder ports, click and drag a row to a new location.

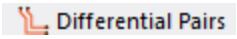
To save changes, click **OK**.

Displaying Mixed-Mode Parameters using Differential Pairs

Network Data Explorer displays mixed-mode parameters when differential pairs are both defined and activated.

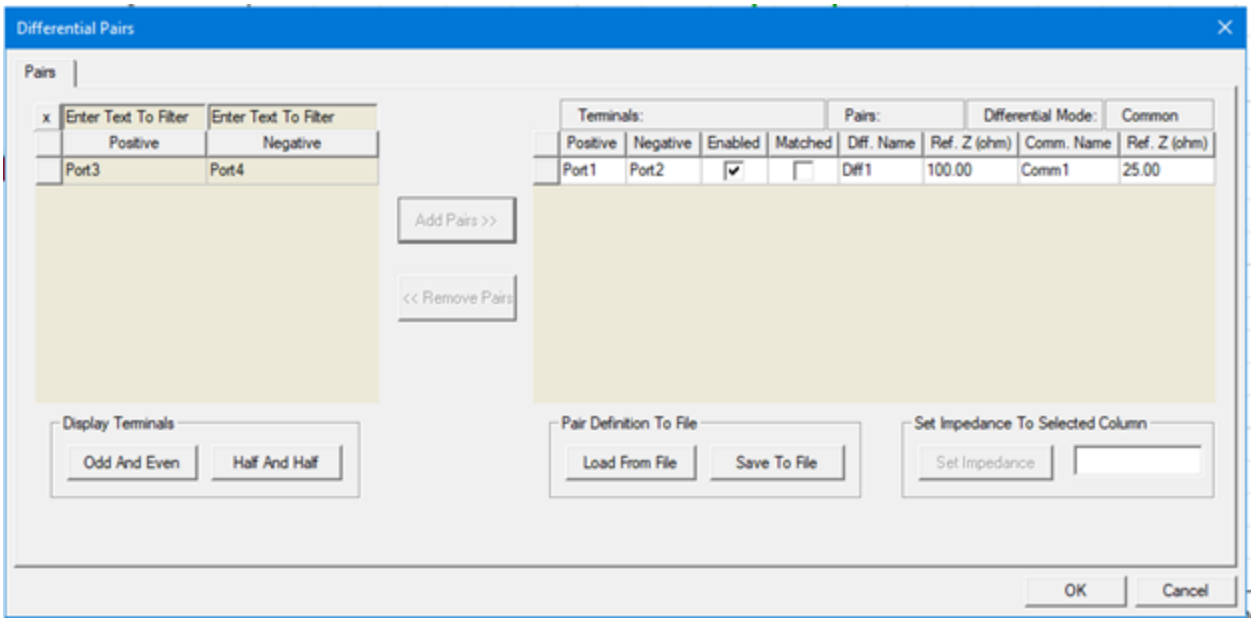
To define differential pairs:

1. Select existing ports.
2. Open Network Data Explorer (**Tools**>**Network Data Explorer**).
3. On the **NDE** ribbon, select **Differential Pairs**.



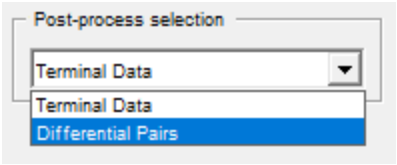
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Differential Pairs** window appears.



4. Select a pair from the list on the left and click **Add Pairs**.
5. Click **OK**.
6. To disable all differential pairs, click the **Enabled** column header in the **Differential Pairs** dialog to deselect all pairs.

For HFSS Driven Terminal designs, selecting the solution data will disable this ribbon button. The user must make port/differential pair changes directly in the HFSS Design and re-import the solution. When the HFSS Driven Terminal Design has differential pairs defined, NDE shows this drop down menu and you can change between showing reports for Terminals or Differential Pairs.Select **Differential Pairs** from the **Post-process selection** field to view mixed-mode parameters.



Note:

The Network Data Explorer **Edit** menu option **Reset All Port Properties** deactivates all pairs, but it does not clear the differential pair settings. And since **Reset All Port Properties** also clears reference impedances and terminations, it should not be used when the user simply wishes to disable all differential pairs.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reset All Port Properties

Reset All Ports resets all changes in the **Edit Ports** window. It also deactivates all differential pairs defined in the **Differential Pairs** window but does not remove the definitions.

> **Note:** For HFSS Driven Terminal designs, selecting the solution data will disable this ribbon button. The user must make port/differential pair changes directly in the HFSS Design and re-import the solution.

Data View Pane Context Menus

The Data View pane presents different right-click menu options, depending on the context. Some commands are the same as those on the NDE ribbon. Others appear only in the context menus.

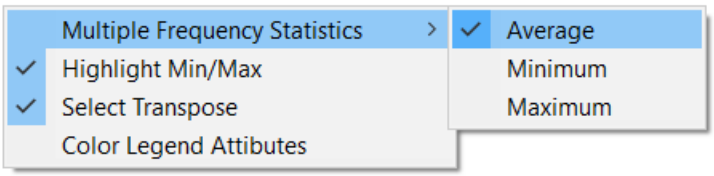
Network Data Explorer commands unique to the context menus are:

- Multiple Frequency Statistics
- Highlight Min/Max
- Select Transpose
- Color Legend Attributes
- Matrix Entries Plot Menu

Multiple Frequency Statistics

The **Multiple Frequency Statistics** menu option determines the statistical composite to display when multiple frequencies have been selected for the matrix display. The statistical data is always the first matrix displayed, followed by matrices for each individual frequency. The **Multiple Frequency Statistics** option also indicates the data used in the colored matrix plot when multiple frequencies have been selected.

This right click menu option appears in the **Matrix** plot, regardless of whether you are in Table or Plot view.



The menu options are:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

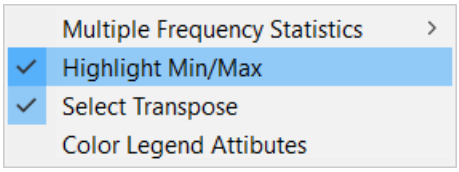
- **Average** – display the average of the matrix values across selected frequencies.
- **Minimum** – display the minimum matrix values across selected frequencies.
- **Maximum** – display the maximum matrix values across selected frequencies.

The selected information also appears in a tool-tip when you hover the cursor over a cell.

Highlight Min/Max

The **Highlight Min/Max** menu option determines whether the minimum and maximum matrix entries should be highlighted in the matrix table and color plot view.

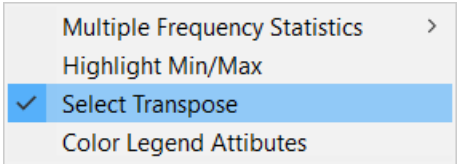
This right click menu option appears in the **Matrix** plot, regardless of whether you are in Table or Plot view.



Select Transpose

The **Select Transpose** menu option determines whether transpose cells are highlighted along with selected cells.

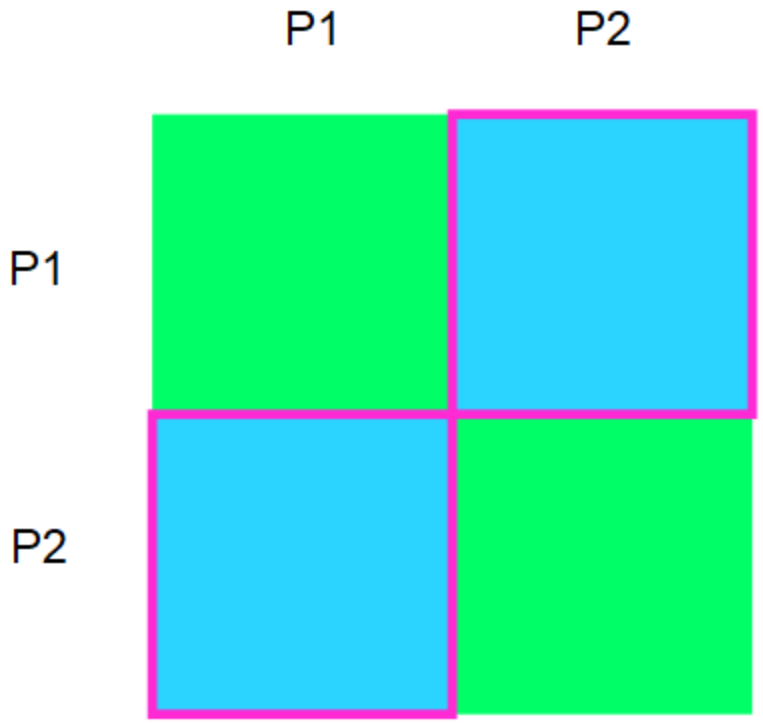
This right click menu option appears in the **Matrix** plot, regardless of whether you are in Table or Plot view.



Transpose cells are highlighted in Table or Plot view, as shown below.

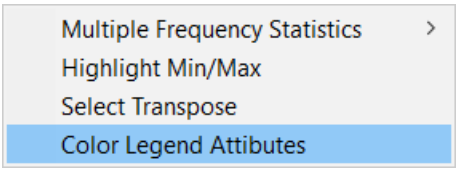
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0.01000GHz	P1	-57.10340, -81.97841	-57.72581, 85.70836	-0.00213, -0.18916
	P2	-57.72582, 85.70825	-57.09556, -81.98749	-66.27113, 99.16916
	P3	-0.00213, -0.18916	-66.27110, 99.16941	-57.10246, -81.95892
	P4	-66.27112, 99.16942	-0.00213, -0.18929	-57.72556, 85.71465



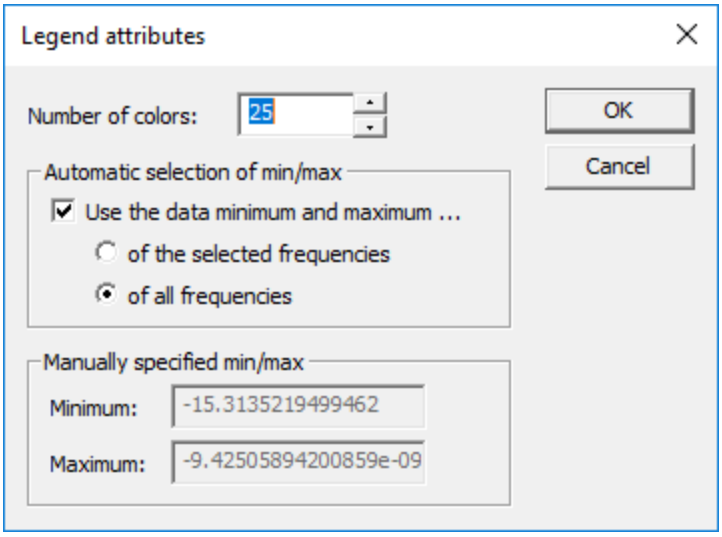
Color Legend Attributes

The **Color Legend Attributes** menu option allows you to change the granularity of the color scheme and the value range for plots. This right click menu option appears in the **Matrix** plot.



Alternatively, double-click the matrix plot's legend to open the **Legend Attributes** window.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



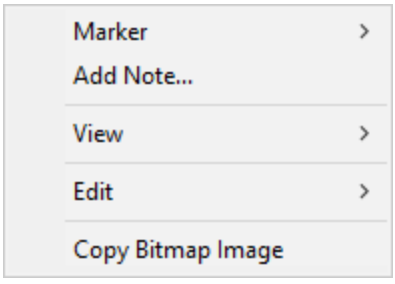
Options include:

- **Number of colors** – allows you to select number of color entries in the legend (the number of divisions between the start/end of the data range). This can be set to any discrete number between 1 and 50.
- **Automatic selection of min/max** – check the **Use the data minimum and maximum** check box to automatically select the data range using the minimum and maximum values from either selected frequencies or all frequencies in the data set.
- **Manually specified min/max** – when the when the range is not automatically determined, these fields permit the user to manually enter hard values. For example, for S parameter data magnitude data, you could enter a minimum of 0 and a maximum of 1.

Using a standard range across all frequencies permits you to quantitatively compare plots, and ndExplorer remembers legend settings for each data-type and display-format pair.

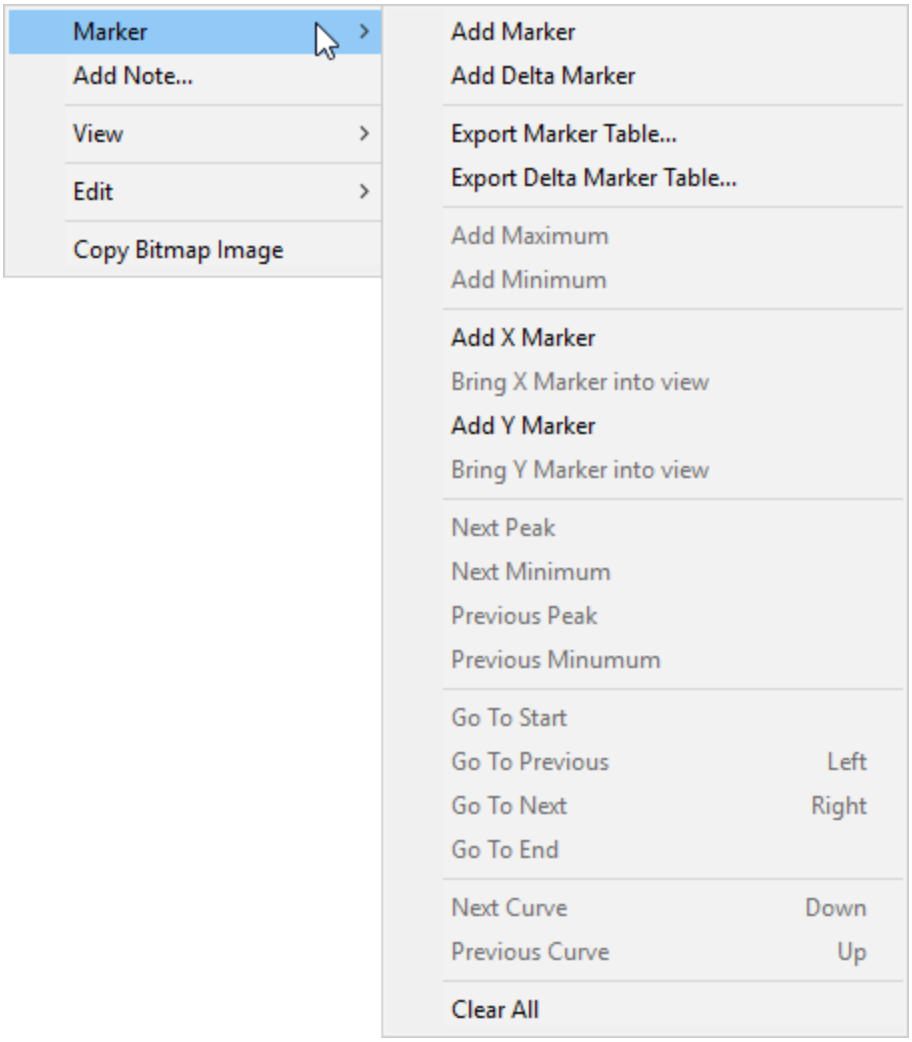
Matrix Entries Plot Menu

On the **Matrix** plot, several new right-click menu options appear.

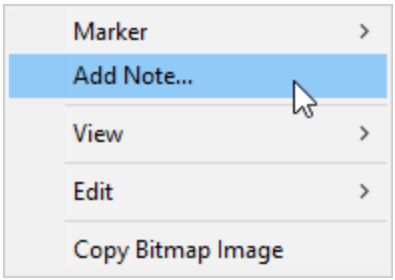


The **Marker** sub-menu provides commands for adding markers to plots.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

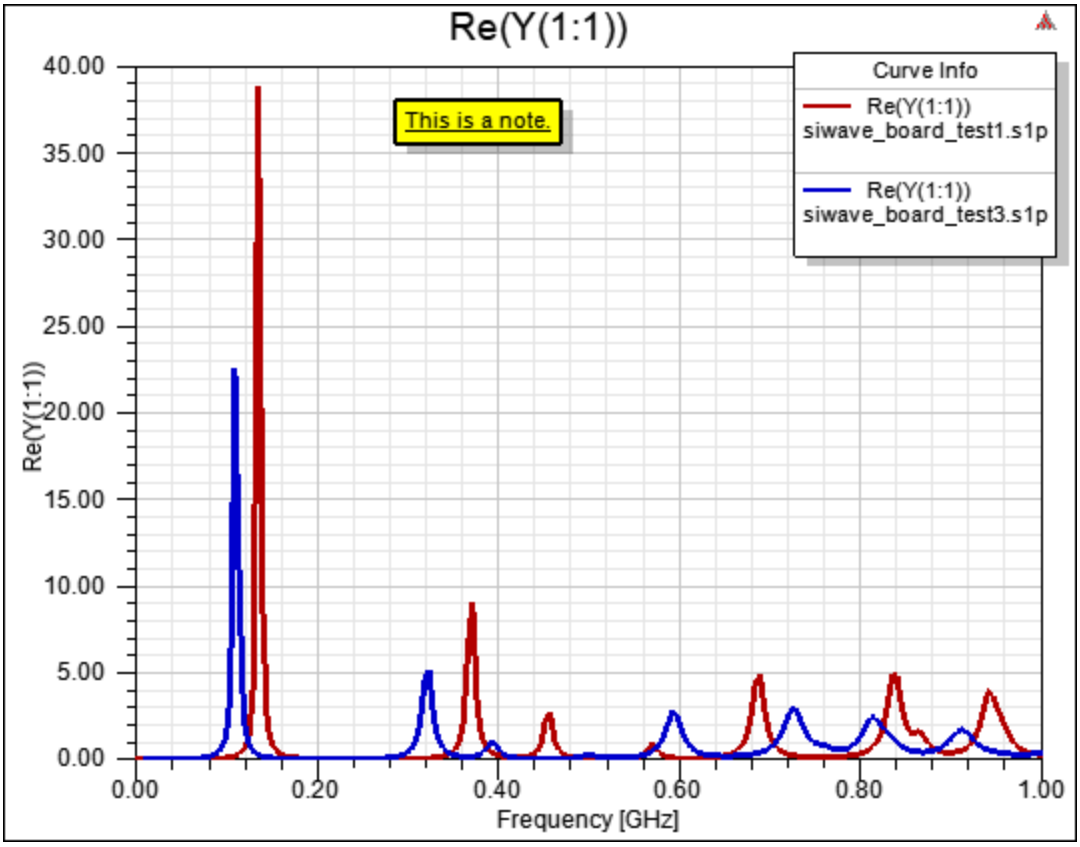


The **Add Note** menu option allows you to add a note to the plot.



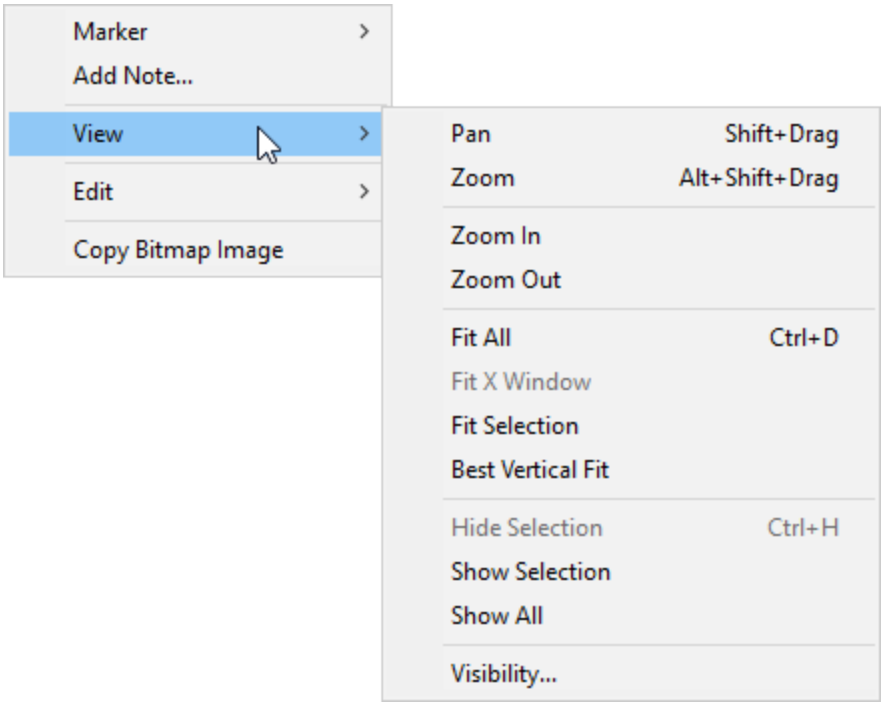
The note appears at the location you right-clicked. You can click and drag the note to a new location, or double-click the note to change its color and font.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



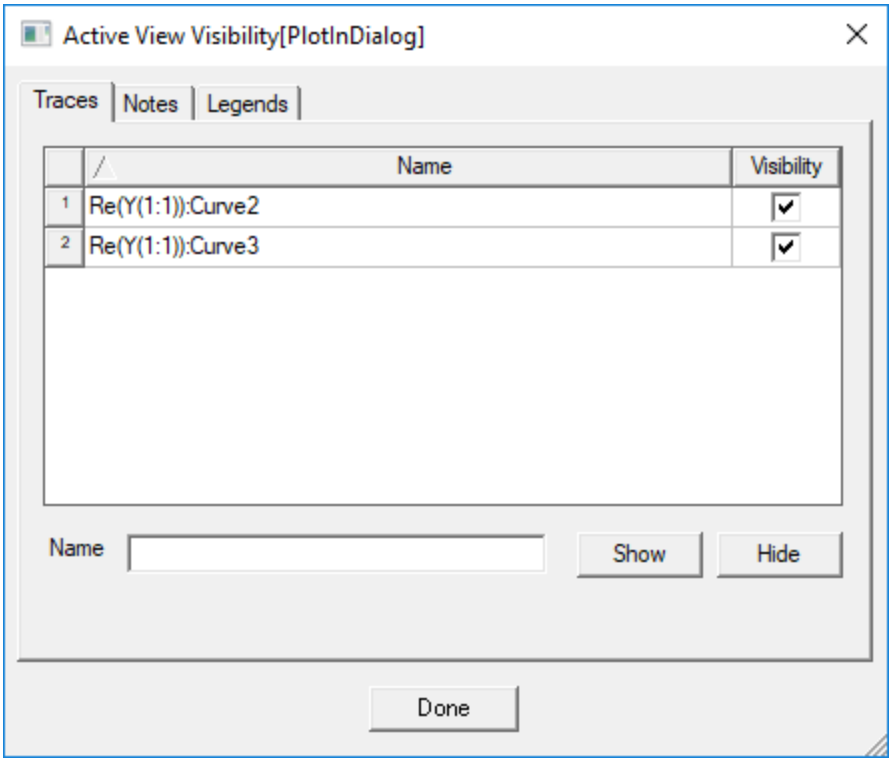
The **View** sub-menu provides commands for viewing, panning, zooming, and fitting elements to your plot.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

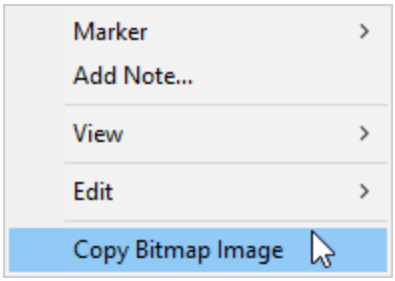


Click **Visibility** to open the **Active View Visibility** window, where you can select the visibility of traces, notes and legends.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Copy Bitmap Image** menu option copies the plot to your clipboard. You can then paste it into a graphics editor.



Exploring Network Data and Modifying the Display

Network Data Explorer allows you to view data and modify various aspects of the display, including color plots, color coding, viewing across frequencies, and displaying individual statistics. This section provides examples of Network Data Explorer capabilities:

- Viewing the S, Y, or Z Matrix for a Frequency
- Viewing a Color-coded Matrix Plot
- Displaying a Cell Graph Across All Frequencies

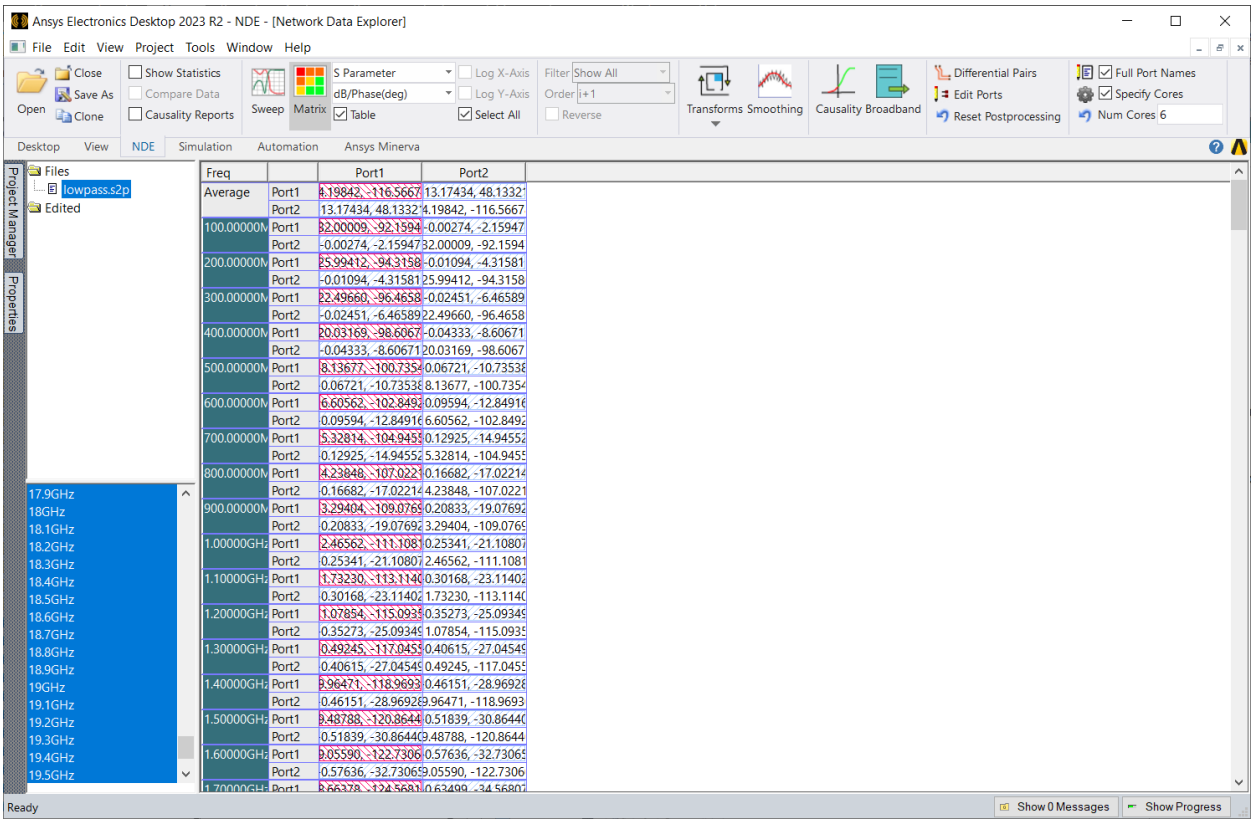
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Displaying Matrix Statistics by Frequency
- Displaying Individual Statistics for All Frequencies
- Creating a Statistics Plot
- Comparing Variations

Viewing a Matrix Table

To view the S, Y or Z matrix:

1. On the **NDE** ribbon, click **Matrix**.
2. Click the **Table** check box.
3. Use the **Parameter** type drop-down menu to select **S parameter**, **Y parameter**, **Z parameter**, or another choice.
4. Choose a format from the drop-down menu, for example dB or Mag.
5. In the Cell and Frequency Selection pane, select frequencies to display or click the **Select All** check box on the **NDE** ribbon to select all frequencies. The S, Y, or Z matrix displays.



6. Hover over a cell to see more information in a tool tip.
7. Right click to see a right click menu with other commands.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum values are highlighted in red stripes. If Select Transpose is enabled, transposes are highlighted in red stripes as well.

Minimum values are highlighted in blue stripes.

Selected cells appear in solid blue. If Select Transpose is enabled, the transpose is highlighted in blue as well.

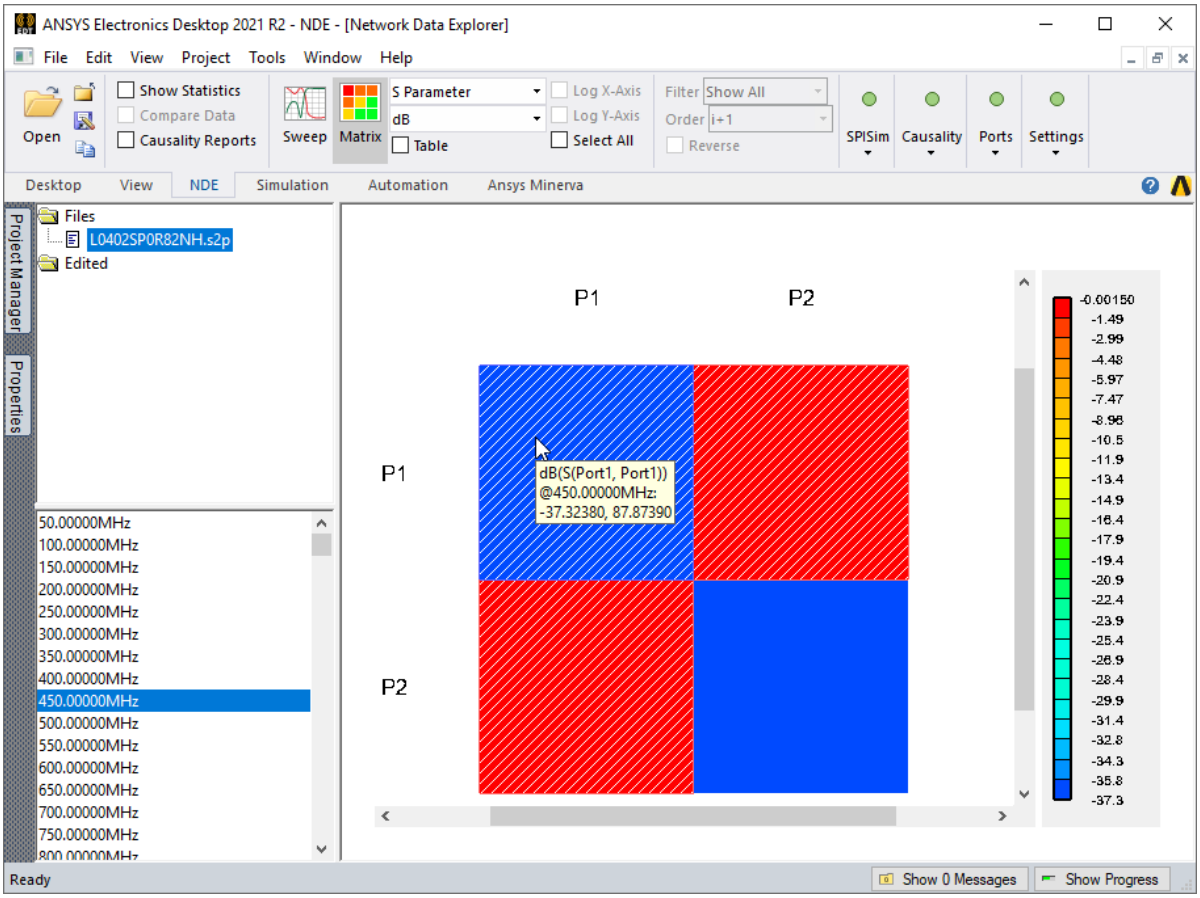
Double-clicking a cell switches to a matrix cell view, in which values for all frequencies for that cell are displayed. The double-clicked frequency is highlighted with solid red shading

Complex values are compared using their modulus. When multiple frequencies are selected, the data display depends on the Multiple Frequency Statistics setting.

Viewing a Color-Coded Matrix Plot

1. On the **NDE** ribbon, click **Matrix**.
2. Use the **Parameter** type drop-down menu to select **S parameter**, **Y parameter**, **Z parameter**, or another choice.
3. In the Cell and Frequency Selection pane, select the frequencies you want to plot or click the **Select All** check box on the **NDE** ribbon to select all frequencies.
4. Hover over a cell to see more information in a tool tip.
5. Right click to see a right click menu with other commands.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Matrix values display in a color-coded grid. If the selected **Format** is a complex value, only the real component is used to determine the display color. When multiple frequencies or variations are selected, the data display depends on the Multiple Frequency Statistics setting. Maximum values are highlighted in red. Minimum values are highlighted in dark blue.

Hover the cursor over any cell to view information about it.

Click any cell to select it. Selected cells appear with a pink outline. If Select Transpose is enabled, the transpose is selected in pink as well.

Double-click any cell to view a matrix cell plot in which all frequency values for that matrix cell are displayed as a graph.

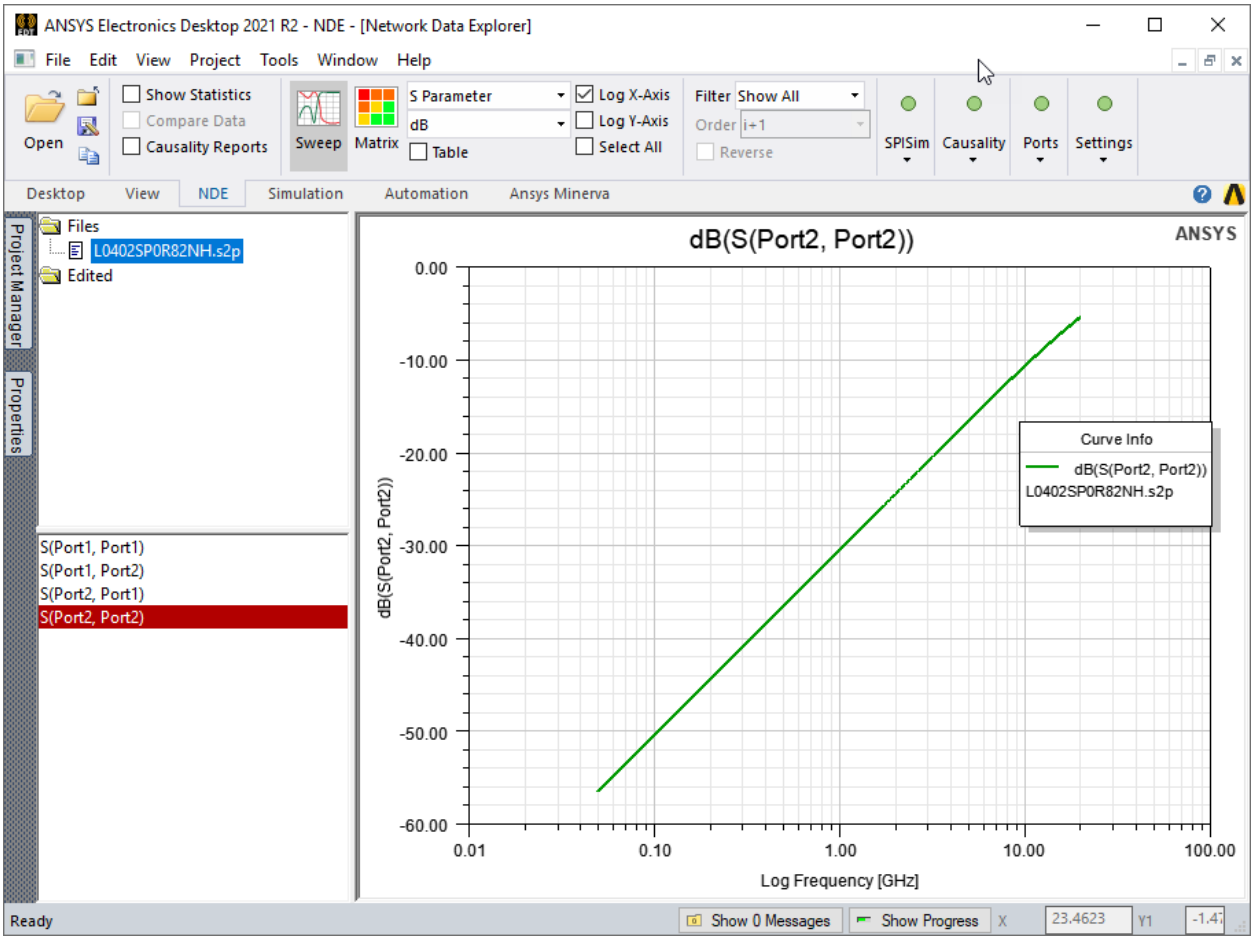
Displaying a Cell Graph Across All Frequencies

Network Data Explorer can plot a cell across all frequencies.

1. On the **NDE** ribbon, click **Sweep**. The Data View Pane updates if necessary.
2. Use the **Parameter** type drop-down menu to select **S parameter**, **Y parameter**, or **Z parameter**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the Cell and Frequency Selection pane, select cells to display or click the **Select All** check box on the **NDE** ribbon to select all cells.
4. To add a log scale to the X-axis, click **Log X-Axis** on the **NDE** ribbon.



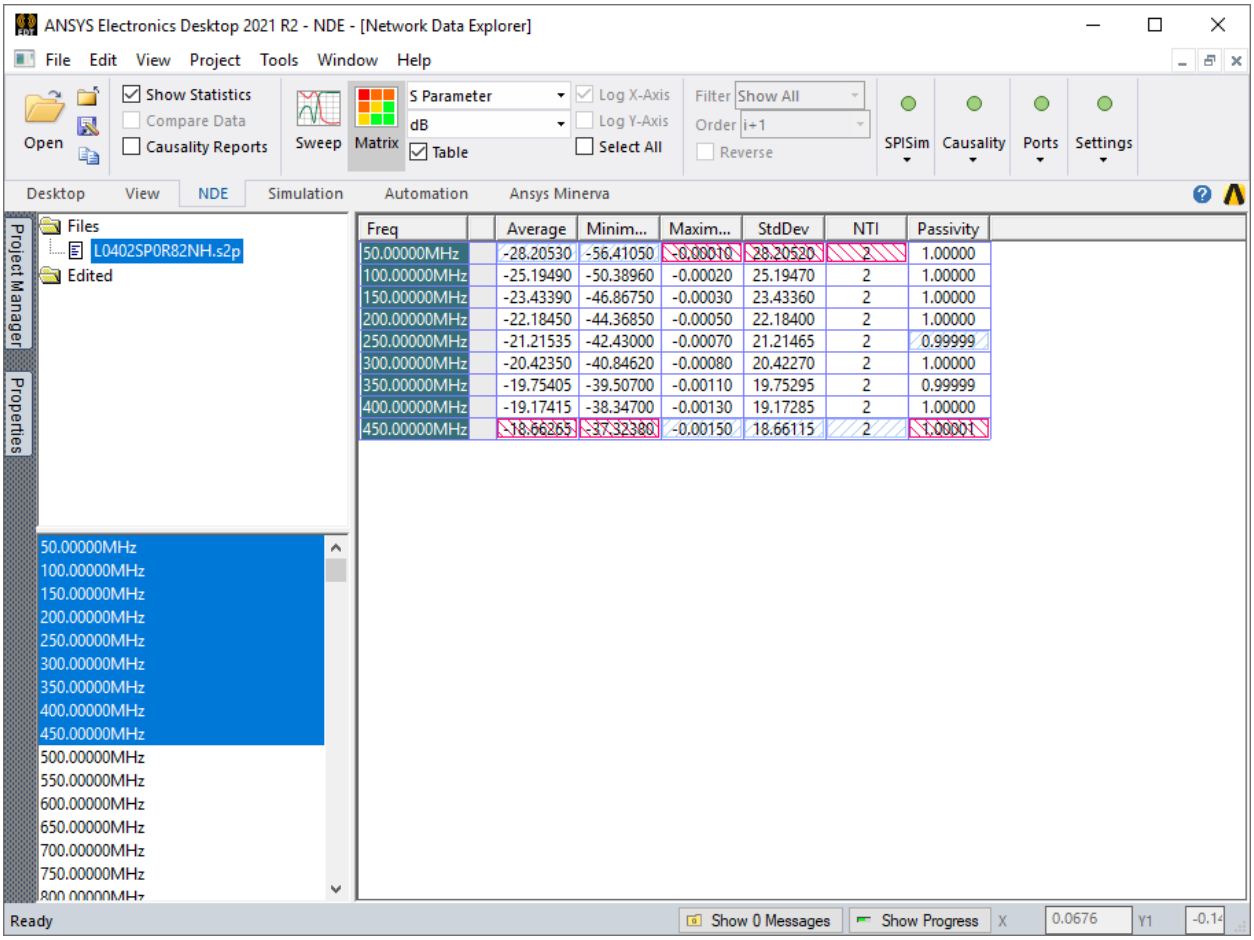
5. To add a log scale to the Y-axis, click **Log Y-Axis** on the **NDE** ribbon.
6. Right click to see a right click menu with other commands.

Displaying Matrix Statistics by Frequency

Network Data Explorer can display various statistical measurements.

1. On the **NDE** ribbon:
   a. Click **Matrix**.
   b. Click **Table**.
   c. Click **Show Statistics**.
2. In the Cell and Frequency Selection pane, select frequencies to display.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Click a column header to sort data by that column.

Hover the cursor over a cell to view information about it.

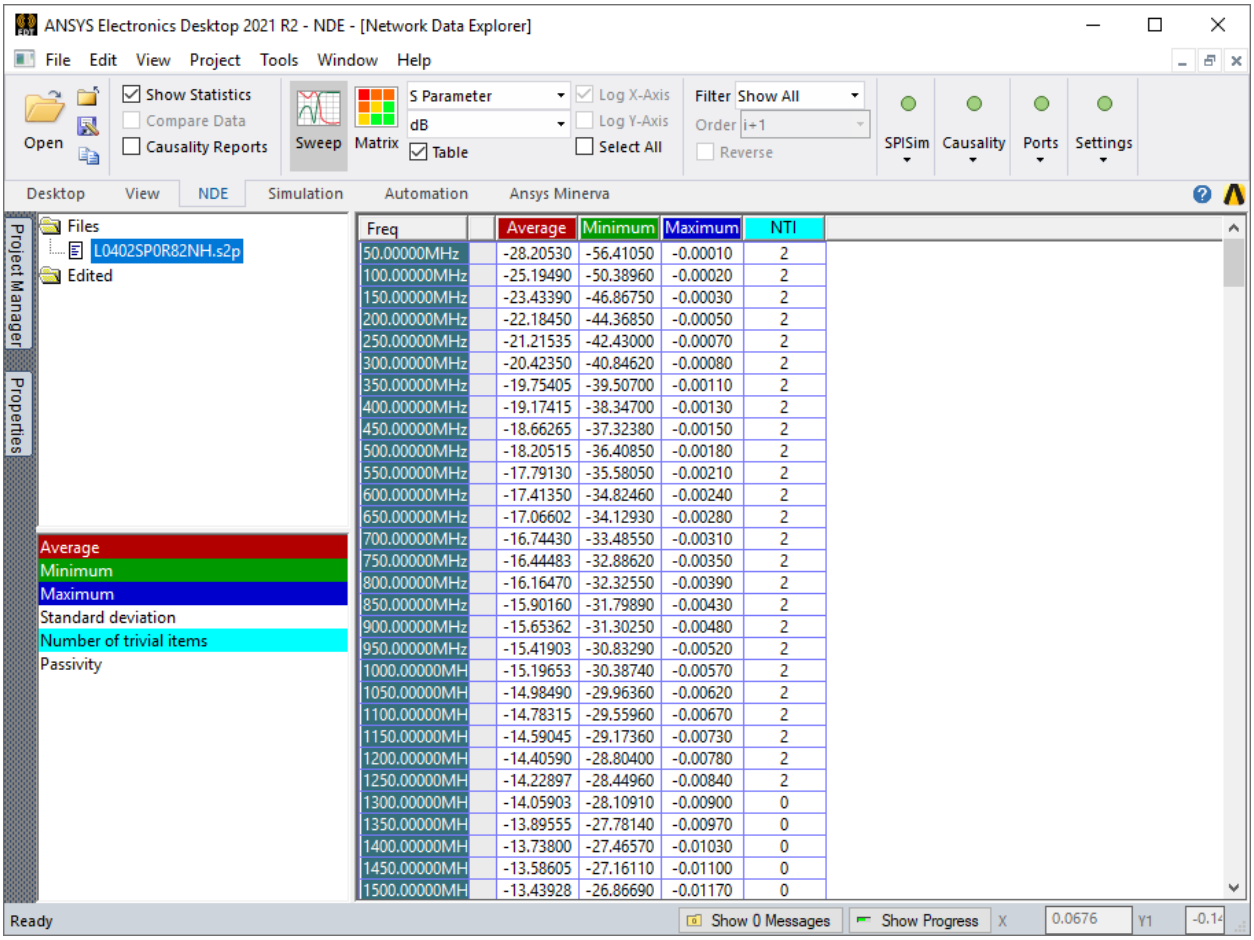
Only real (not complex) data formats are offered for statistical analysis. **Passivity** is only available for S-parameter data (comparisons inactive). **NTI** refers to the number of trivial items; for S-parameters, this includes all zeros and ones; for all other data (and data comparisons), only zeros are counted as trivial. The minimum value for each column is highlighted in blue; the maximum is highlighted in red.

Displaying Individual Statistics for All Frequencies

1. On the **NDE** ribbon:
    a. Click **Sweep**.
    b. Click **Table**.
    c. Click **Show Statistics**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the Cell and Frequency Selection pane, select statistics to display. The information displays in a table in the Data View pane.



Selected statistics are displayed for all frequencies.

Passivity is only available for S-parameters (comparisons inactive).

For S-parameters, the **Number of Trivial Items (NTI)** includes all values of 0 and 1. For other data and data comparisons, only values of 0 are counted as trivial.

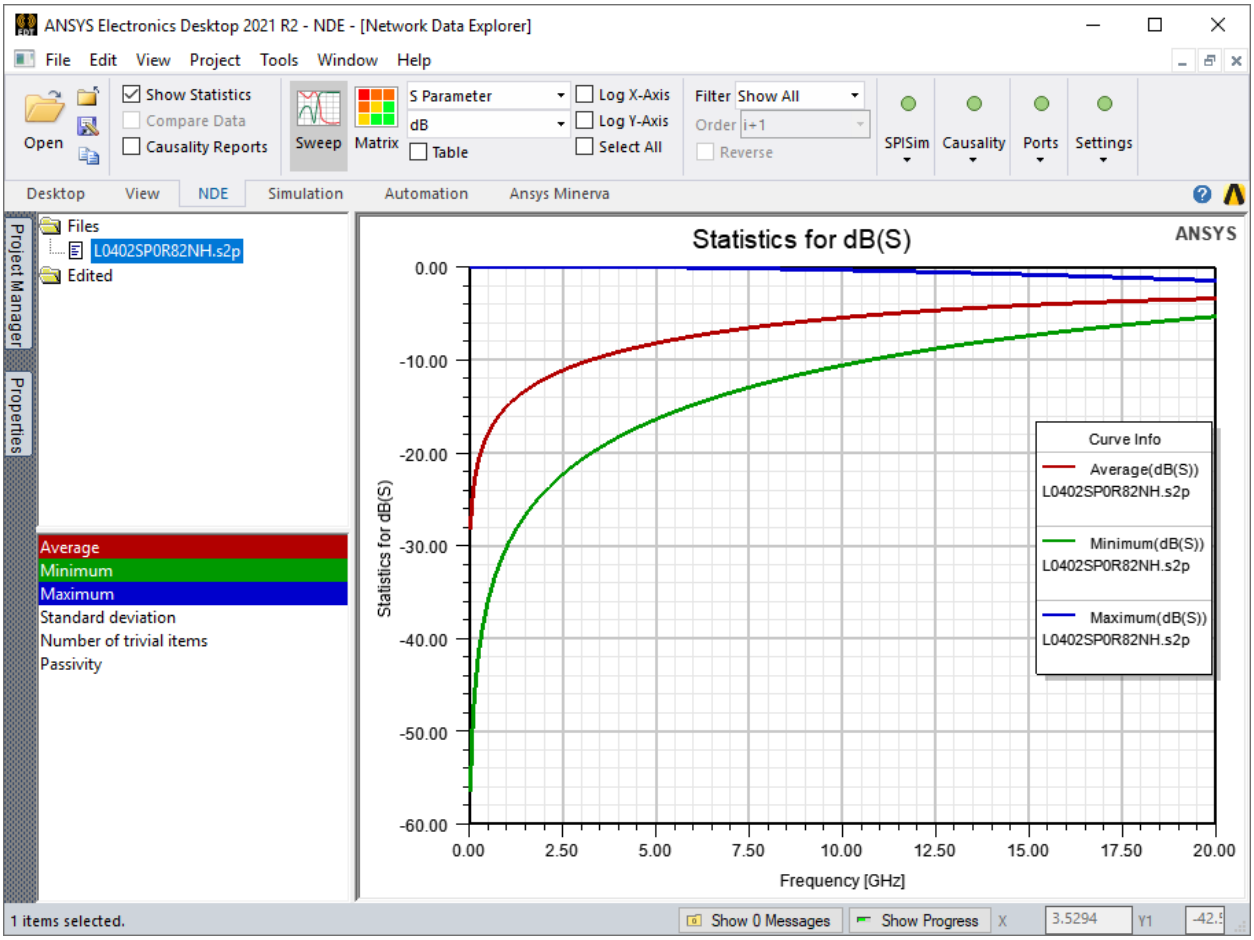
Creating a Statistics Plot

Network Data Explorer can display a graph of selected statistical measures across all frequencies.

1. On the **NDE** ribbon:
    a. Click **Sweep**.
    b. Click **Show Statistics**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the Cell and Frequency Selection pane, select statistics to display. The selected statistics are plotted.



Hover the cursor over a statistic to view more information about it.

Passivity is only available for S-parameters (comparisons inactive).

For S-parameters, the **Number of Trivial Items (NTI)** includes all values of 0 and 1. For other data and data comparisons, only values of 0 are counted as trivial.

Comparing Network Data

Network Data Explorer can compare variations for two network data sets that are the same size.

1. On the **NDE** ribbon, click **Sweep**.
2. In the Network Data Selection pane, select exactly two data sets.
3. On the **NDE** ribbon, click **Compare Data**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. In the Cell and Frequency Selection pane, select which cells you want to compare . For each value along the X-axis, the Y-axis values are subtracted, one from the other, to create the comparison plot. The second selected data is subtracted from the first selected data.
5. Optionally, check **Show Statistics** to show values applied to all cells and all frequencies

It is not possible to compare a data set against itself *unless the data set has been cloned.* Then, you can compare the original data set to the clone.

Traces for a given cell or statistical measure are displayed for all data sets; you can use tool tips to distinguish between them.

If a single cell or statistical measure is displayed, different colors are used for each data trace. If multiple cells or statistical measures are selected, a single color is used for all data traces for each cell or statistical measure.

In a data comparison, traces are shown for all selected data sets. This is true for both cell and statistical traces.

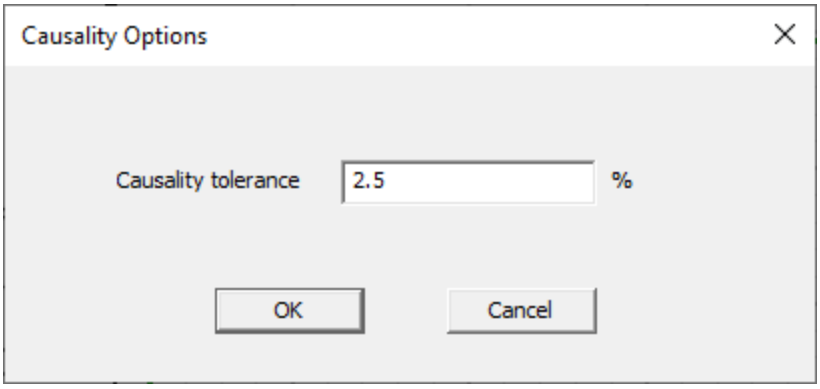
If **dB** format is shown, you can either subtract values before or after applying the dB function. See Display Format to make this choice.

Causality Checking and Plots

Network Data Explorer can perform a causality check on S-parameter data from any source (solution or file), and provide plots of the results in various formats.

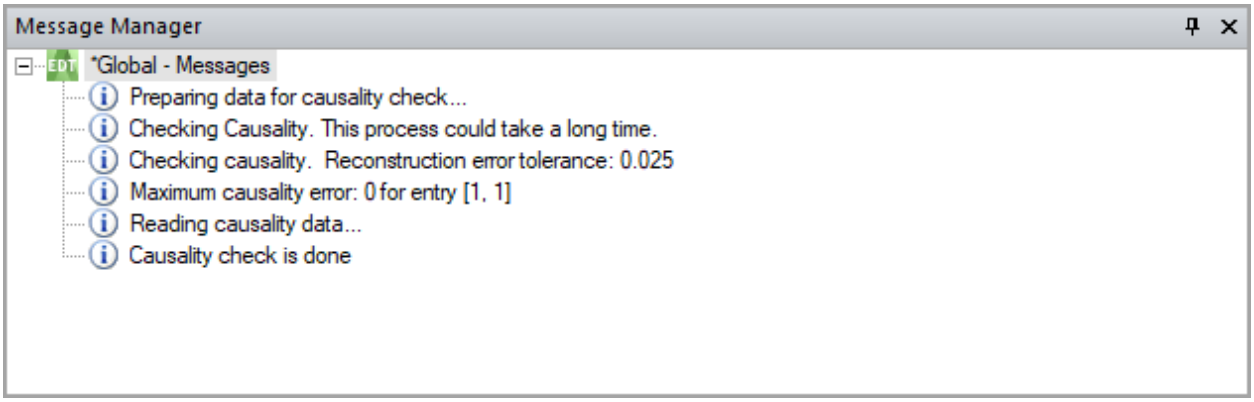
When S-parameter data is loaded into Network Data Explorer, the **Causality** button is enabled.

1. On the **NDE** ribbon, click **Causality**. The **Causality Options** dialog opens.



2. Enter a **Causality tolerance** and click **OK** to start the causality check. Depending on the size of the S-parameter data, the causality check may take several minutes to complete. The check's status displays in the Message Manager.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

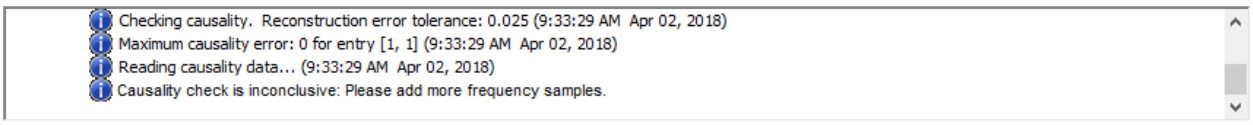


When the check completes, the Message Manager updates to display a summary of results.

Reconstruction Error Tolerance – Causality of a frequency response is determined by calculating the generalized Hilbert transform of the data at all frequencies. A causal frequency response is equal to its generalized Hilbert transform. The reconstruction error is the difference between the tabulated data and its transform at a given frequency. The message shows the maximum reconstruction error tolerance for a causal frequency response. The default tolerance of 0.01 is equal to the state-space fitting tolerance.

Maximum Causality Error – The maximum causality error for all port pairs and all frequencies, along with the matrix indices (port numbers) where the maximum noncausality occurs. A noncausal response is one where all matrix entries can be conclusively analyzed, and at least one entry exceeds the causality tolerance. The maximum reconstruction appears first, followed by port numbers in brackets (e.g., [port number, port number]). When all results are conclusive but no matrix reconstruction error exceeds the tolerance, the maximum causality error is reported as zero, and no matrix entry is listed.

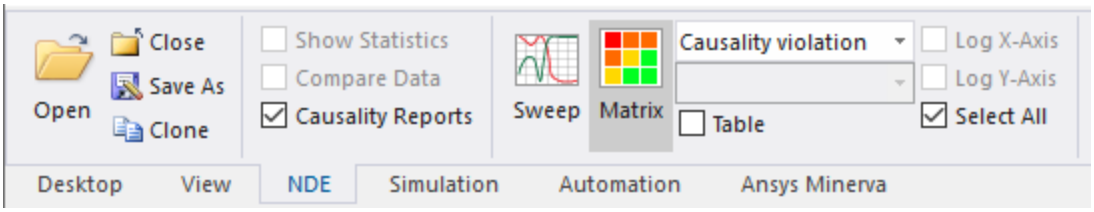
If the data does not contain enough frequency points to determine whether the data is or is not causal, the Messages Pane will note an inconclusive result. Network Data Explorer will also report the data set as inconclusive if any cells are inconclusive, even if other entries exhibit causality violations.



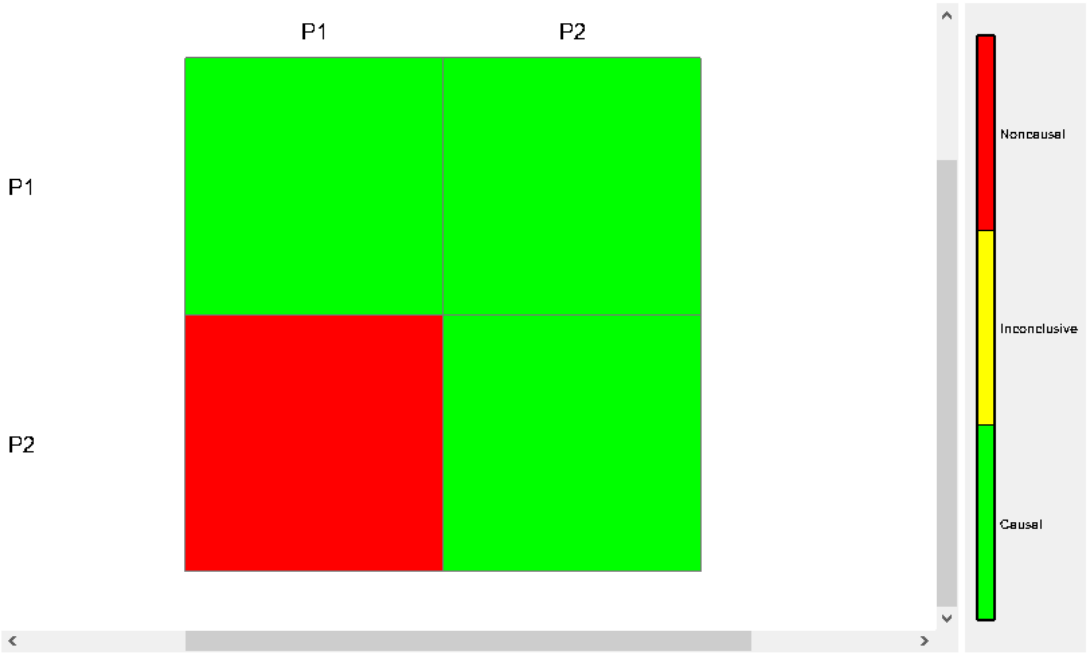
To see the plotted results of the causality check:

1. On the **NDE** ribbon, click **Causality Reports**.
2. Either click the **Matrix** icon or select **Causality violation** in the **Parameter** type drop-down menu.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A rectangular plot displays, with dimension N x N and color-coding to indicate the causality status of each port pairing.



In this plot, the cells go from (Port 1, Port 1) at the upper-left area to (Port N, Port N) at the lower-right corner. The result shows the causality over all frequencies in the data. In this example, the matrix is asymmetric, so that S12 is noncausal, while S11, S21, and S22 are causal.

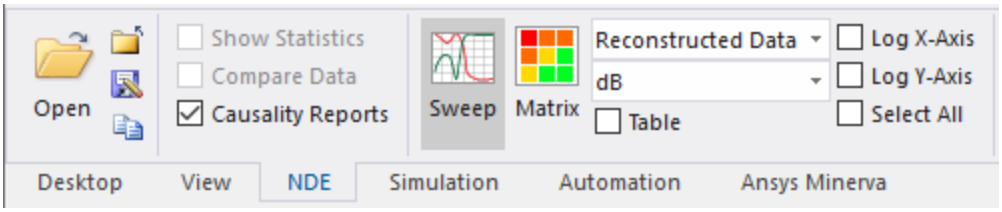
To see the details for each frequency, click the **Table** check button on the **NDE** ribbon.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Freq		Port1	Port2	Port1	Port2
0.10000GHz	Port1	0.01715, 0.00000	0.00050, 0.00000	0.01493, 0.00000	0.00098, 0.00000
	Port2	0.53111, 0.00000	0.00579, 0.00000	0.11471, 0.00000	0.00832, 0.00000
0.20000GHz	Port1	0.01367, 0.00000	0.00059, 0.00000	0.00736, 0.00000	0.00052, 0.00000
	Port2	0.27237, 0.00000	0.00394, 0.00000	0.05933, 0.00000	0.00423, 0.00000
0.30000GHz	Port1	0.01362, 0.00000	0.00105, 0.00000	0.00344, 0.00000	0.00028, 0.00000
	Port2	0.24509, 0.00000	0.00239, 0.00000	0.01921, 0.00000	0.00195, 0.00000
0.40000GHz	Port1	0.00907, 0.00000	0.00103, 0.00000	0.00171, 0.00000	0.00006, 0.00000
	Port2	0.16665, 0.00000	0.00163, 0.00000	0.01530, 0.00000	0.00084, 0.00000
0.50000GHz	Port1	0.00693, 0.00000	0.00062, 0.00000	0.00075, 0.00000	0.00006, 0.00000
	Port2	0.11877, 0.00000	0.00106, 0.00000	0.00542, 0.00000	0.00040, 0.00000
0.60000GHz	Port1	0.00295, 0.00000	0.00044, 0.00000	0.00047, 0.00000	0.00005, 0.00000
	Port2	0.05535, 0.00000	0.00042, 0.00000	0.00462, 0.00000	0.00015, 0.00000
0.70000GHz	Port1	0.00015, 0.00000	0.00009, 0.00000	0.00029, 0.00000	0.00003, 0.00000
	Port2	0.00510, 0.00000	0.00012, 0.00000	0.00114, 0.00000	0.00008, 0.00000
0.80000GHz	Port1	0.00058, 0.00000	0.00068, 0.00000	0.00047, 0.00000	0.00004, 0.00000
	Port2	0.02772, 0.00000	0.00088, 0.00000	0.00115, 0.00000	0.00010, 0.00000
0.90000GHz	Port1	0.00210, 0.00000	0.00081, 0.00000	0.00040, 0.00000	0.00013, 0.00000
	Port2	0.05492, 0.00000	0.00133, 0.00000	0.00173, 0.00000	0.00010, 0.00000
1.00000GHz	Port1	0.00221, 0.00000	0.00064, 0.00000	0.00033, 0.00000	0.00010, 0.00000
	Port2	0.07703, 0.00000	0.00142, 0.00000	0.00111, 0.00000	0.00009, 0.00000
1.10000GHz	Port1	0.00410, 0.00000	0.00045, 0.00000	0.00037, 0.00000	0.00008, 0.00000
	Port2	0.08851, 0.00000	0.00151, 0.00000	0.00137, 0.00000	0.00011, 0.00000
1.20000GHz	Port1	0.00433, 0.00000	0.00078, 0.00000	0.00017, 0.00000	0.00013, 0.00000
	Port2	0.09013, 0.00000	0.00100, 0.00000	0.00168, 0.00000	0.00017, 0.00000
1.30000GHz	Port1	0.00377, 0.00000	0.00072, 0.00000	0.00020, 0.00000	0.00018, 0.00000
	Port2	0.09487, 0.00000	0.00151, 0.00000	0.00081, 0.00000	0.00012, 0.00000
1.40000GHz	Port1	0.00380, 0.00000	0.00035, 0.00000	0.00020, 0.00000	0.00014, 0.00000
	Port2	0.08918, 0.00000	0.00136, 0.00000	0.00139, 0.00000	0.00009, 0.00000
1.50000GHz	Port1	0.00417, 0.00000	0.00005, 0.00000	0.00012, 0.00000	0.00011, 0.00000
	Port2	0.08091, 0.00000	0.00174, 0.00000	0.00094, 0.00000	0.00021, 0.00000
1.60000GHz	Port1	0.00315, 0.00000	0.00038, 0.00000	0.00006, 0.00000	0.00015, 0.00000
	Port2	0.07512, 0.00000	0.00208, 0.00000	0.00065, 0.00000	0.00011, 0.00000
1.70000GHz	Port1	0.00275, 0.00000	0.00034, 0.00000	0.00006, 0.00000	0.00021, 0.00000
	Port2	0.06069, 0.00000	0.00152, 0.00000	0.00101, 0.00000	0.00021, 0.00000

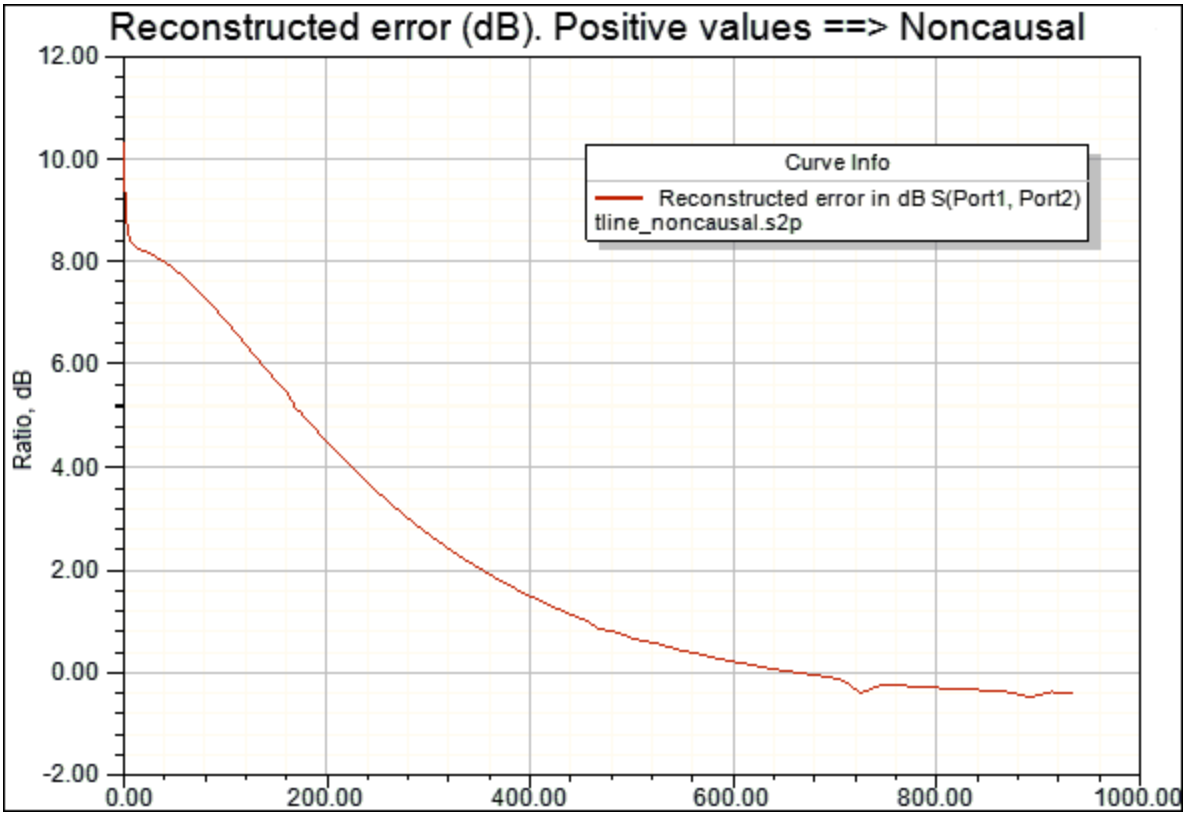
To plot the reconstructed frequency response generated by the causality checker:

1. On the **NDE** ribbon, click **Causality Reports**.
2. Either click the **Sweep** icon or select **Reconstructed Data** in the **Parameter** type drop-down menu.
3. In the **Format** drop-down menu, select the desired format. **dB** is selected by default.



The plot appears, showing the reconstruction error at each frequency divided by the tolerance.

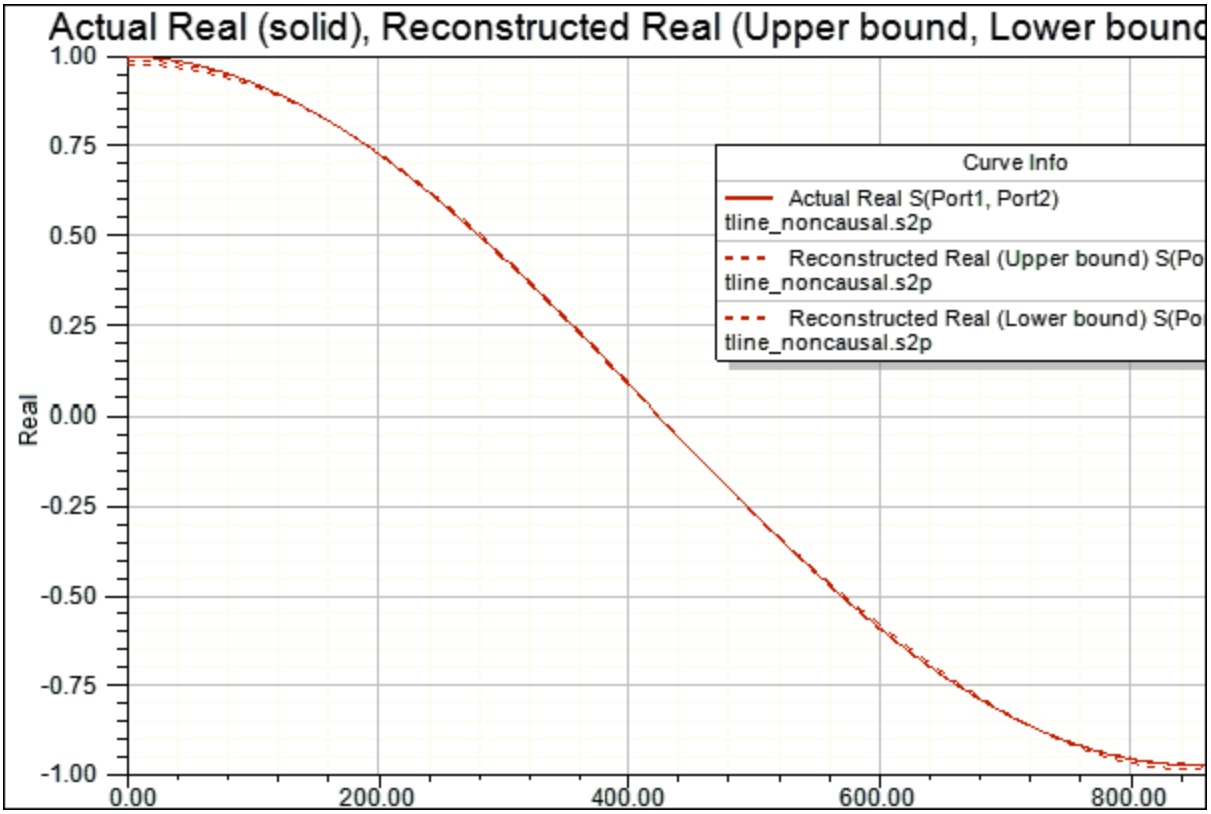
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The reconstruction error ratio for parameter S12 is positive for frequencies less than about 680MHz, indicating a broad range of noncausal behavior.

4. To compare the real part of the reconstructed data to the real part of the actual data, set the **Format** to **Real**.

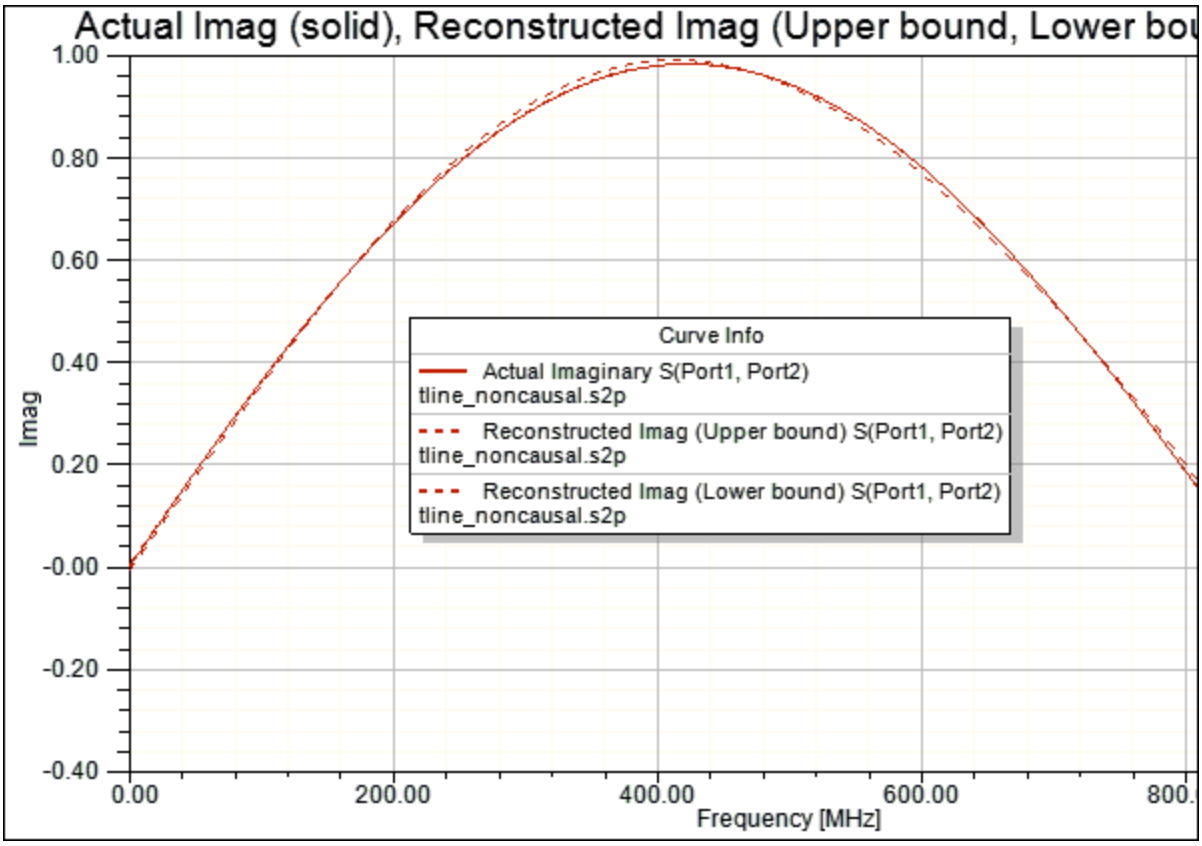
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For a causal frequency response, the actual data (solid line) will be within the upper and lower bounds of the reconstruction (dotted lines) at all frequencies.

5. To compare the imaginary part of the reconstructed data to the imaginary part of the actual data, set the **Format** to **Imaginary**.

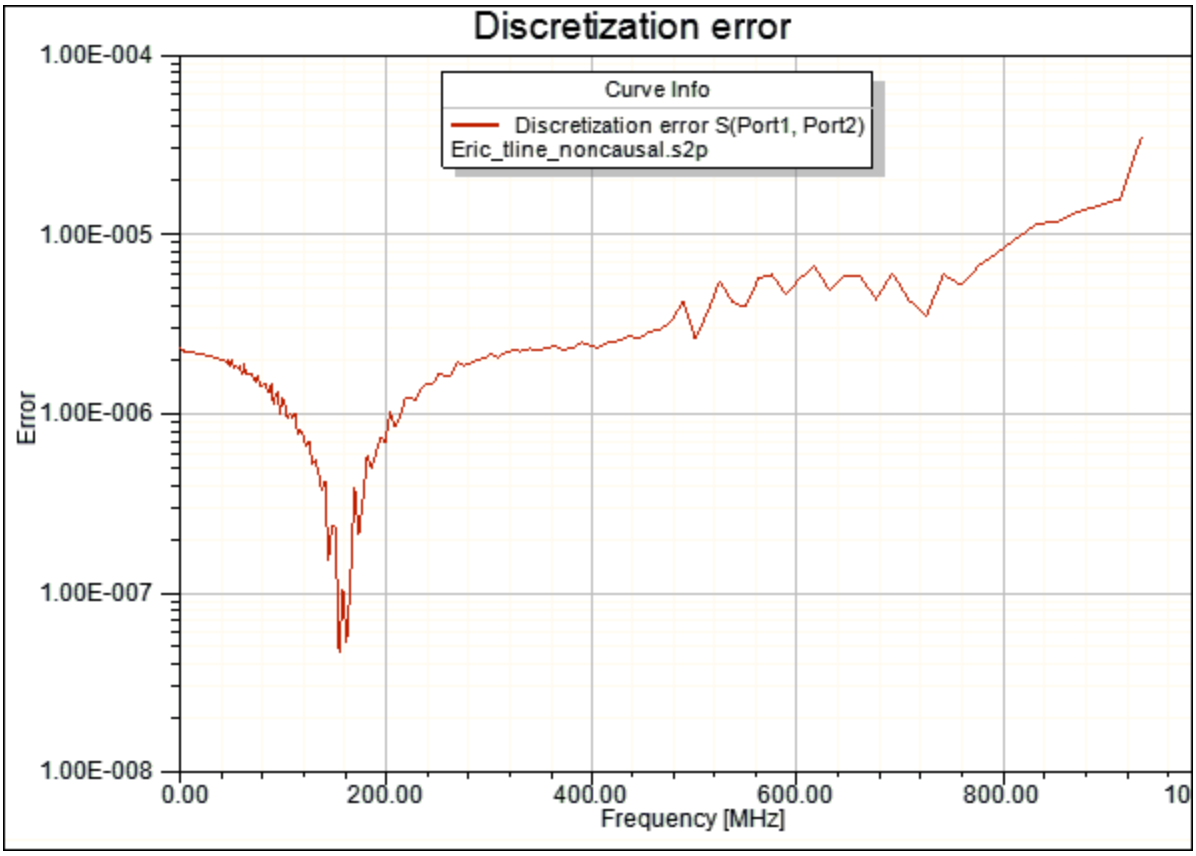
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For a causal frequency response, the actual data (solid line) will be within the upper and lower bounds of the reconstruction (dotted lines) at all frequencies.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. To view the frequency-dependent discretization error, set the **Format** to **Discretization**.



The discretization error is the error that is due to the fact that the data are available only at discrete frequencies rather than for a continuous spectrum. A discretization error near or greater than the causality tolerance renders the causality check inconclusive. Data at more frequencies could reduce the discretization error and render the analysis conclusive. This set of data exhibits low discretization errors (<<0.01) at all frequencies, and the causality check is conclusive (conclusively noncausal in this example).

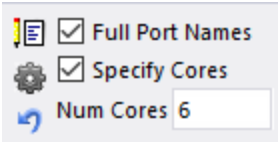
Multithreading

By default, multithreading (execution on multiple cores) is enabled for Causality Checking and Macro Model export. Multithreading saves significant time in the Causality Check calculation, and improves the time for other state-space fitting operations. See Technical Notes.

The **Cores** field in the Network Data Explorer Control Pane defaults to half the number of cores detected on your computer.

For best performance, disable hyper-threading on your computer. When hyper-threading is enabled, the number of cores includes the physical cores and an equal number of logical cores. With hyper-threading disabled, the display shows only half the number of physical cores.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To disable multithreading, uncheck the check box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25 - Scripting

Information about creating, editing, and using IronPython scripts and VB scripts is in the product's scripting guide in the help. You can also access PDF versions of the scripting guide from the help system.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26 - Ansys Workbench Integration Overview

Ansys Workbench combines the strength of its core product solvers with the project management tools necessary to manage project workflow. In Ansys Workbench, analyses are built as *systems*, which can then be combined into a *project*. The project is driven by a schematic workflow that manages the connections between the systems.

From the schematic, you can interact with applications (called workspaces) that are native to Ansys Workbench and that display within the Ansys Workbench interface. Native workspaces include: Project Schematic, Engineering Data, and Design Exploration (Parameters and Design Points).

You can also launch applications that are data-integrated with Ansys Workbench, meaning the application's interface remains separate, but the data from the application communicates with the native Ansys Workbench data. Thus, data can be passed back and forth between any Ansys Electromagnetics product on a Workbench Project Schematic and any supported Ansys or Ansys Electromagnetics desktop product. Depending on the application, data integration can include basic actions such as saving projects, as well as more complex actions such as the coupling of Ansys Electromagnetics product variables to Workbench Design Exploration parameters.

Data-integrated applications include the following Ansys Electromagnetics products:

Circuit, HFSS, Icepak, Maxwell/RMxprt, Mechanical, Q3D Extractor, and Twin Builder.

> **Note:**
>
> For detailed information on working with Ansys Workbench, please refer to the Workbench documentation.

Integrating Ansys EM Suite with Ansys Workbench

You can integrate the Ansys Electromagnetics Suite with the Ansys Workbench application after both have been installed on your system. For successful integration, both preinstalled applications must be of the same version. A single operation completes the integration process for all supported design types.

The same procedure is used to integrate the applications or to decouple them (if they had previously been integrated). The procedure is as follows:

1. From the Windows Start Menu, select **All Programs** > **Ansys EM Suite 20*yy* R*n*** > **Modify Integration with Ansys 20*yy* R*n***.

   A *Modify Integration with Ansys 20yy Rn* command window appears.

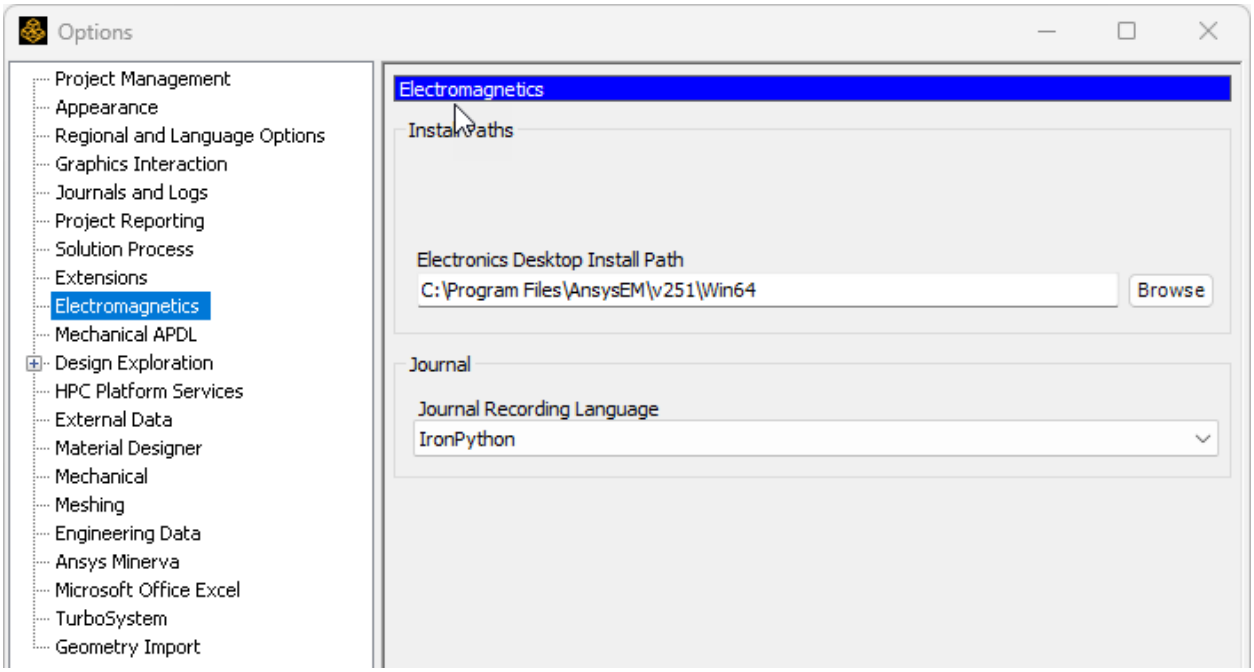
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If both applications are present on the system, their installation paths will be reported, and you will be asked if you wish to integrate them.
- Alternatively, if integration has already been preformed, you will be asked if you wish to decouple them.

2. Type **yes** to perform the prompted action or **no** to abort the operation without making any changes. Then press **Enter**
   - If you enter *"yes,"* a confirmation line appears in the command window indicating that integration (or decoupling) has been completed, and the window remains open.

     Press **Enter** again to close the command window.
   - If you enter *"no,"* the operation aborts and the command window closes immediately.

> **Note:**
>
> You can confirm that the Ansys 20*yy* R*n* application is “aware” of the integrated application via the *Options* dialog box in Workbench. The path to the integrated Electronics Desktop application is listed in the *Electromagnetics* group of settings.



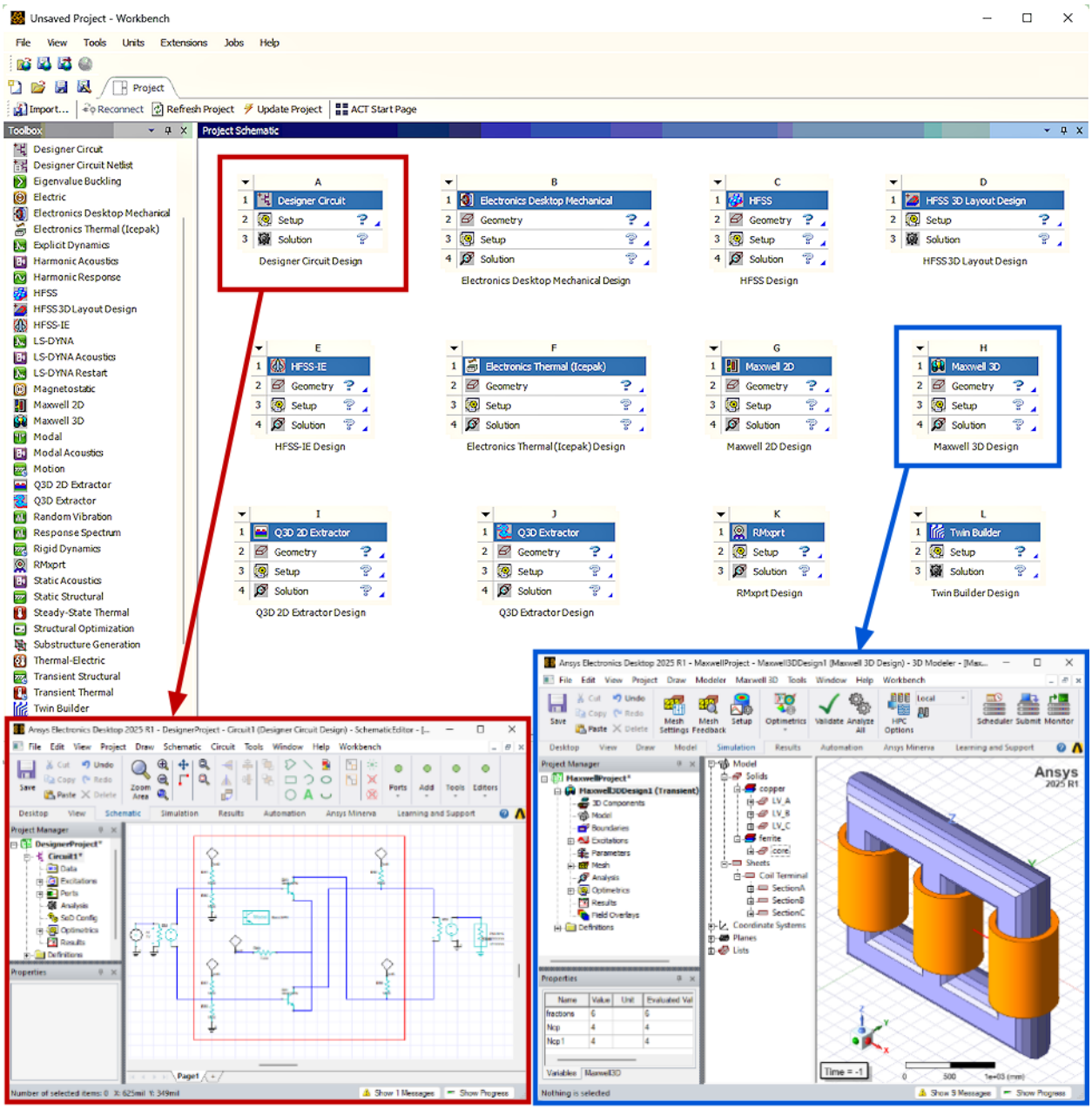
In this group of options, you can also specify Workbench's default **Journal Recording Language** for use when interoperating with the Ansys Electromagnetics products. **Iron Python** is the default language, and the alternative choice is **Visual Basic** (VB). Changes applied here

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

affect any new Workbench or Ansys Electronics Desktop session launched after the change. Any open instance will continue recording in the language that was in effect when the program was launched.

Workbench Data Integration Overview

Ansys EM data-integrated applications can reside on a Workbench **Project Schematic**, as shown.

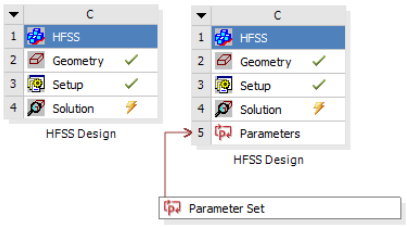


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Objects placed on a Workbench Project Schematic (such as instances of Ansys Electromagnetics projects) are referred to as *systems*. The number of cells in the system depend on the desigh type, as follows:

- **RMxprt, Circuit, HFSS 3D Layout,** and **Twin Builder:** Appear on the Workbench Project Schematic as systems with three cells: **<*System_Type_Name*>, Setup,** and **Solution**.
- **HFSS, Maxwell, Q3D Extractor, Icepak,** and **Mechanical:** Include an additional **Geometry** cell along with the **<*System_Type_Name*>, Setup,** and **Solution** cells, for a total of four cells.

If you invoke Ansys **DesignXplorer** to use variables for refining a design, a **Parameters** cell is added, with a link to the associated Workbench **Parameter Set**. Refer to the Ansys Workbench help for details on working with systems, cells, and parameter sets.



All supported Ansys Electromagnetics Suite products integrate with Workbench commands, services, and DesignXplorer in a similar manner. Some capabilities include:

- Adding New Analysis Systems
- Importing Existing Ansys EM Projects into Workbench
- Editing Ansys EM Models in Workbench
- Ansys EM to Workbench Geometry Transfer
- Ansys EM to Workbench Material Data Transfer
- Analyzing Ansys EM Models in Workbench
- Performing Parameter Studies in Workbench
- Scripting in Workbench

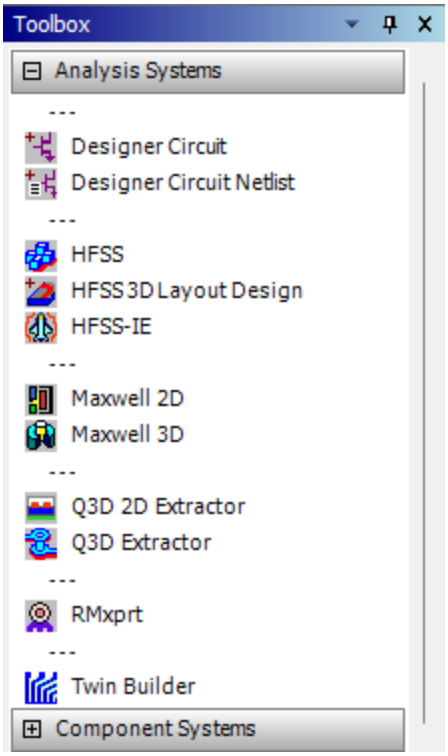
In addition to these major features, Workbench can archive, save, back up, duplicate, and delete Ansys Electromagnetics projects used in a Workbench project. Progress information and messages from integrated Ansys EM projects are also displayed in Workbench.

> **Note:**
>
> Look in the *Ansys Workbench Help* to find detailed information about performing the above-listed operations.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

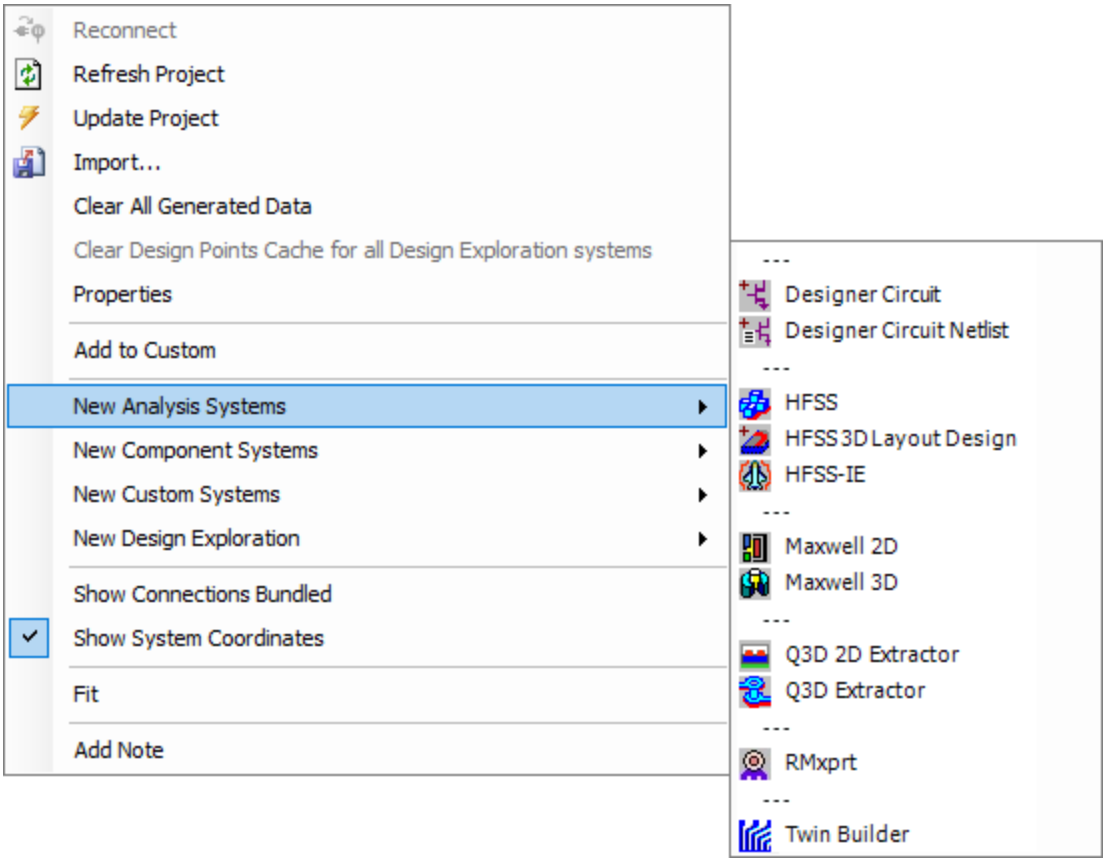
Adding New Analysis Systems

A new Ansys Electromagnetics Analysis System can be added to a Workbench Project Schematic by dragging and dropping it from the Toolbox:



Alternately, you can add a system by right-clicking in the Workbench Project Schematic area and selecting **New Analysis Systems** > **[System Type]**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

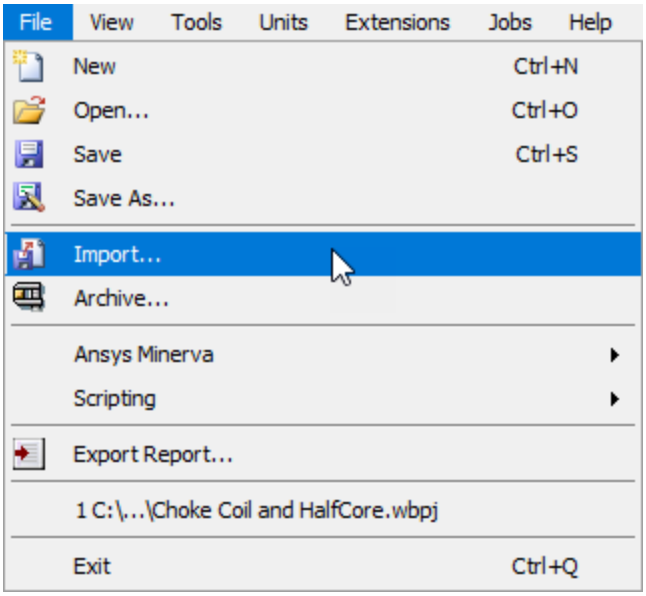


Importing Existing Ansys EM Projects into Workbench

You can import existing Ansys Electromagnetics Suite projects into a Workbench project schematic using **File** > **Import**. When you do, a copy of the Ansys Electromagnetics project is placed in the Workbench project folder. The original Ansys Electromagnetics project remains intact.

> **Note:**
>
> Object, material, and parameter names with non-ASCII characters are not allowed for data transfer. Such transfers fail and produce an error message.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

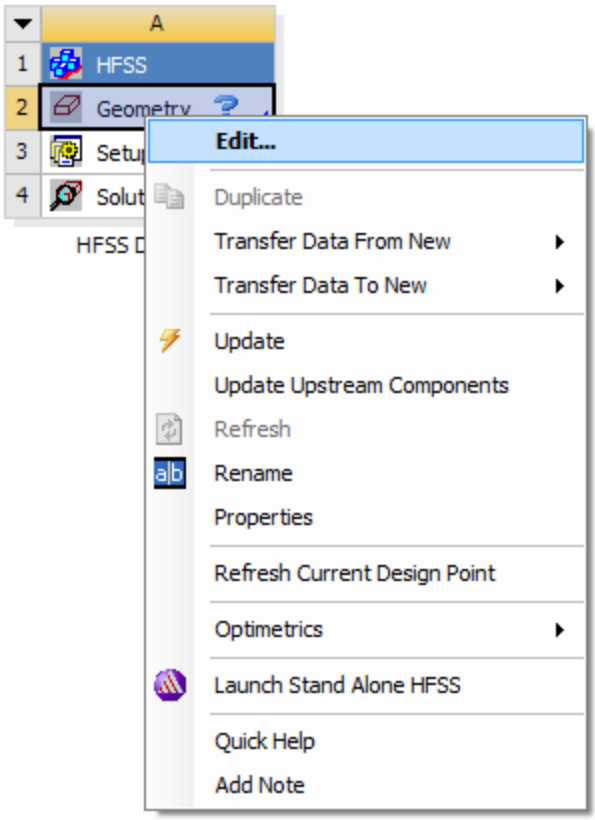


Editing Ansys EM Models in Workbench

You can edit various properties and parameters of the Ansys Electromagnetics project (geometry, setup, solution, etc.) either by right-clicking the project in Workbench and selecting **Edit**, or by double-clicking the project.

Doing so launches the Ansys Electromagnetics desktop application and loads the project so that you can set up your project in a familiar desktop environment. Changes made to the Ansys Electromagnetics project are saved to the project instance in the Workbench project folder.

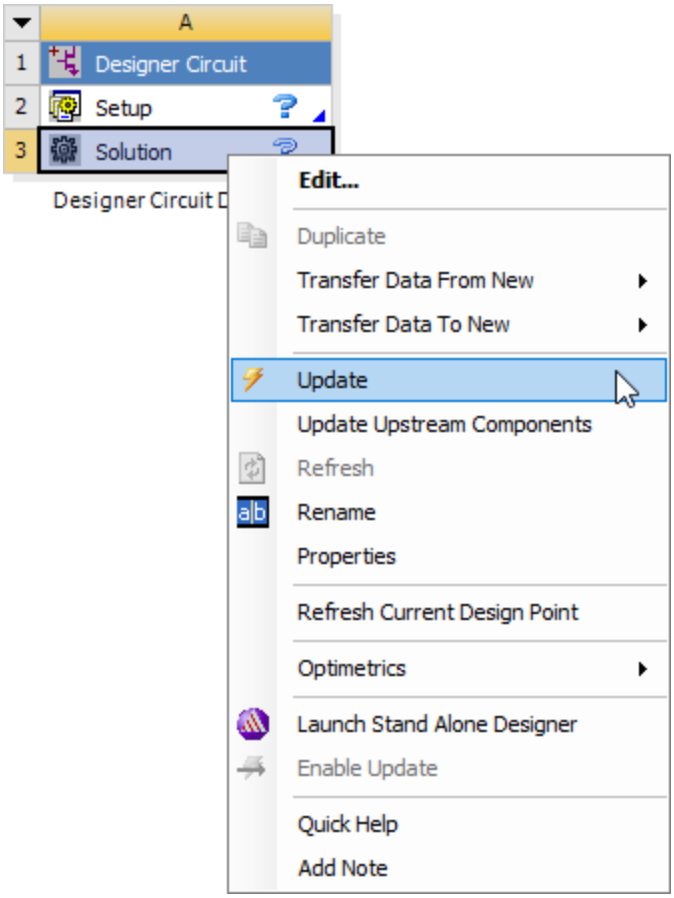
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



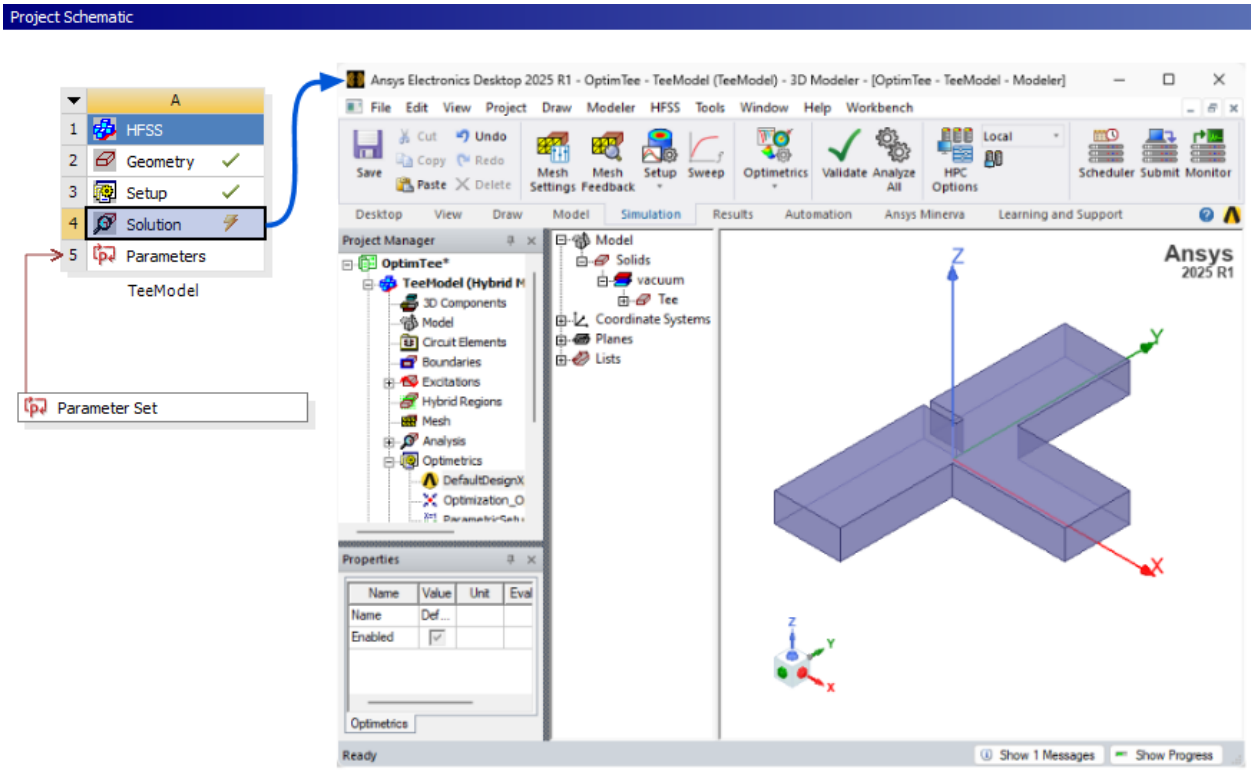
Analyzing Ansys EM Models in Workbench

You can use Workbench's **Update** command to run analyses in the integrated Ansys Electromagnetics project. Progress information is also shown in Workbench.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



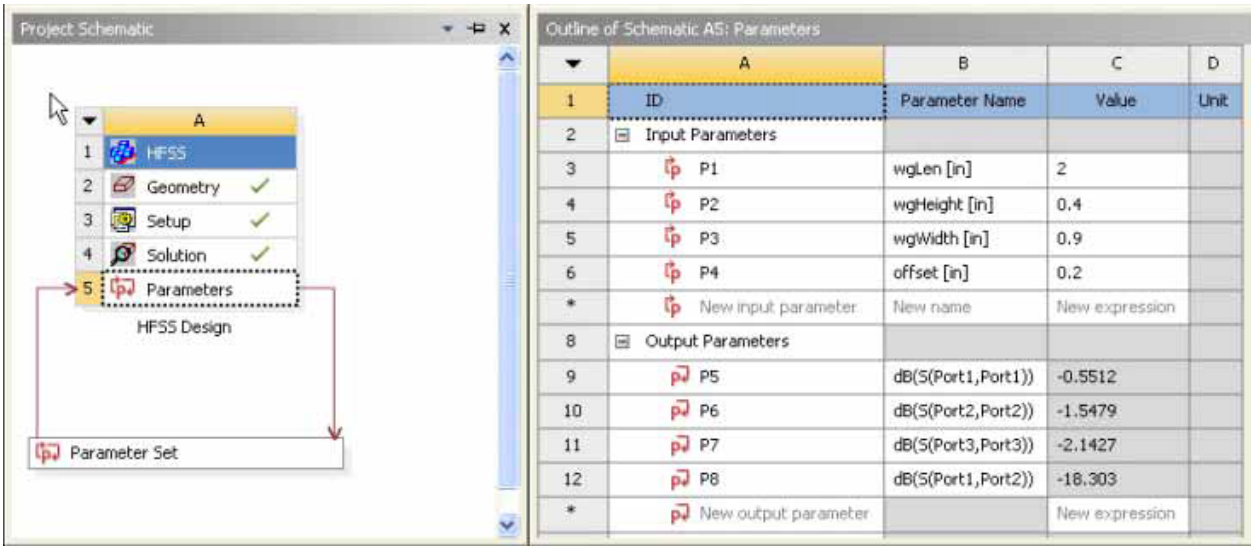
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Performing Parameter Studies in Workbench

Workbench **Parameter Sets** allow you to change parameter values and units of measure, or add new parameters. Parameter data is passed back to the Ansys Electromagnetics application for updated analyses. After analysis, the Workbench Parameter table should be updated correctly for all design points.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

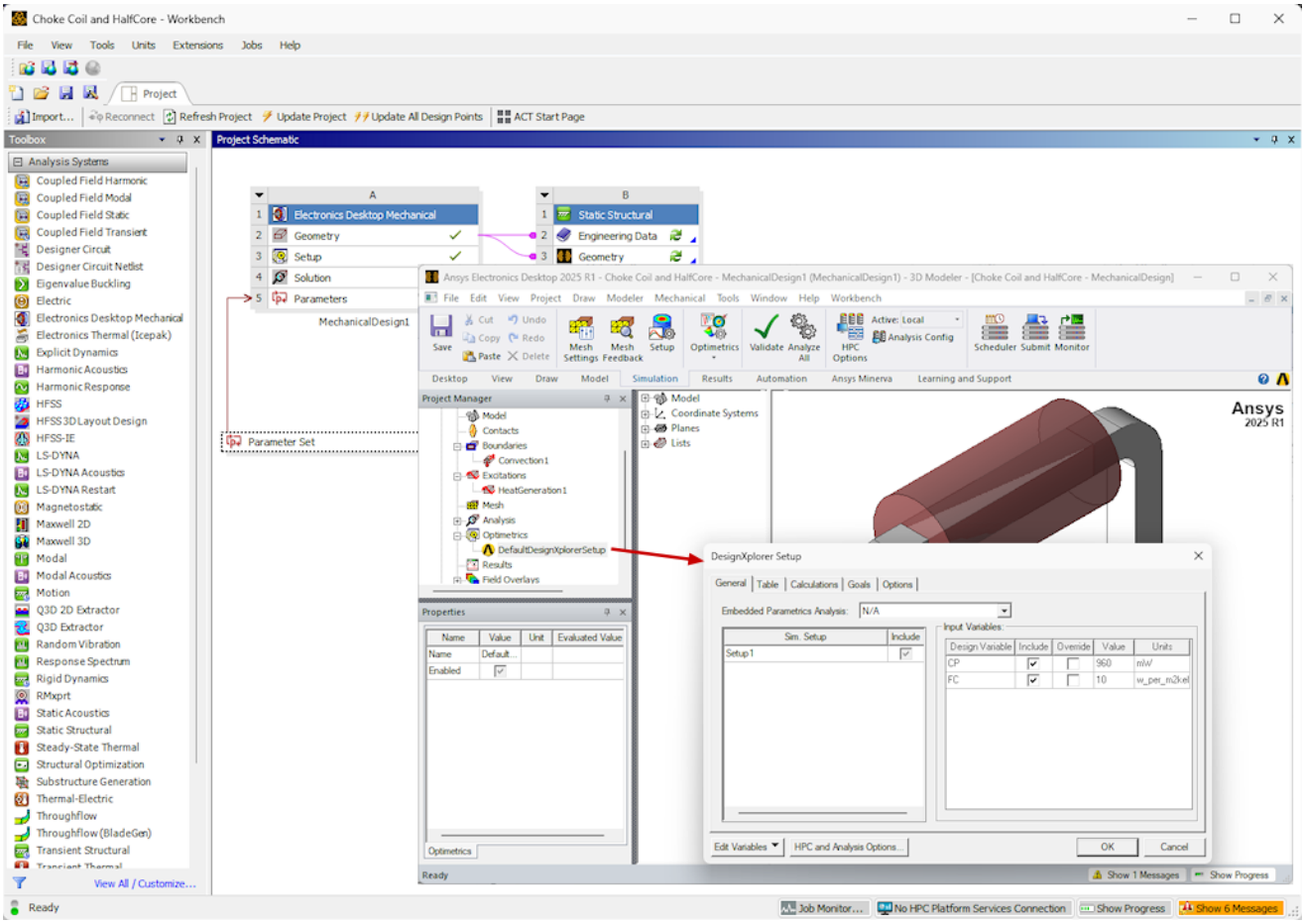


Note:

Because Workbench and Ansys Electromagnetics Desktop have different unit systems, units defined in Desktop are Dimensionless in the Workbench Parameters table and thus cannot be changed. Refer to the *Ansys Design Modeler User's Guide* > *Project Schematic Operations* > *Parameters in Project Schematic* > *Parameter Units* topic for additional information.

Parameters from the Ansys Electromagnetics project are exposed to Workbench through the **DesignXplorer** setup. The Ansys Electromagnetics system's cell status on the Workbench project is updated as changes are made in the Ansys Electromagnetics application. The following example is a Mechanical, Steady-State Thermal design in which two variables have been defined. One variable (CP) is the coil power (960 mW heat generation) and the other (FC) is the film coefficient of the assigned convection boundary:

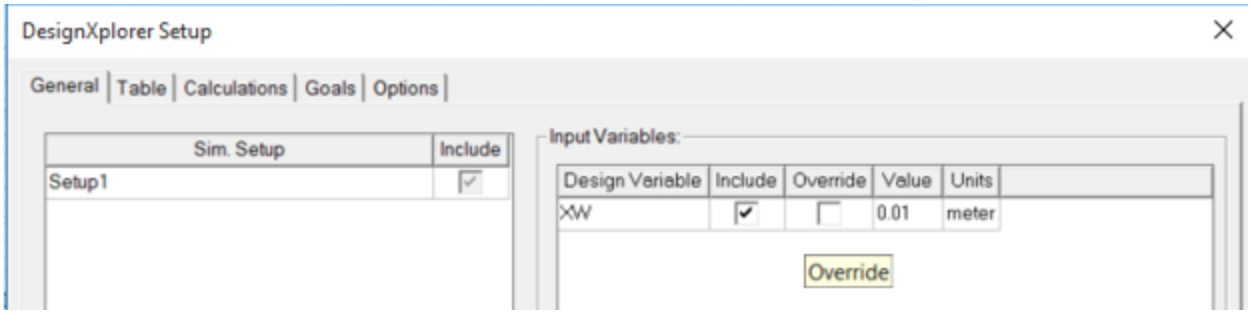
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



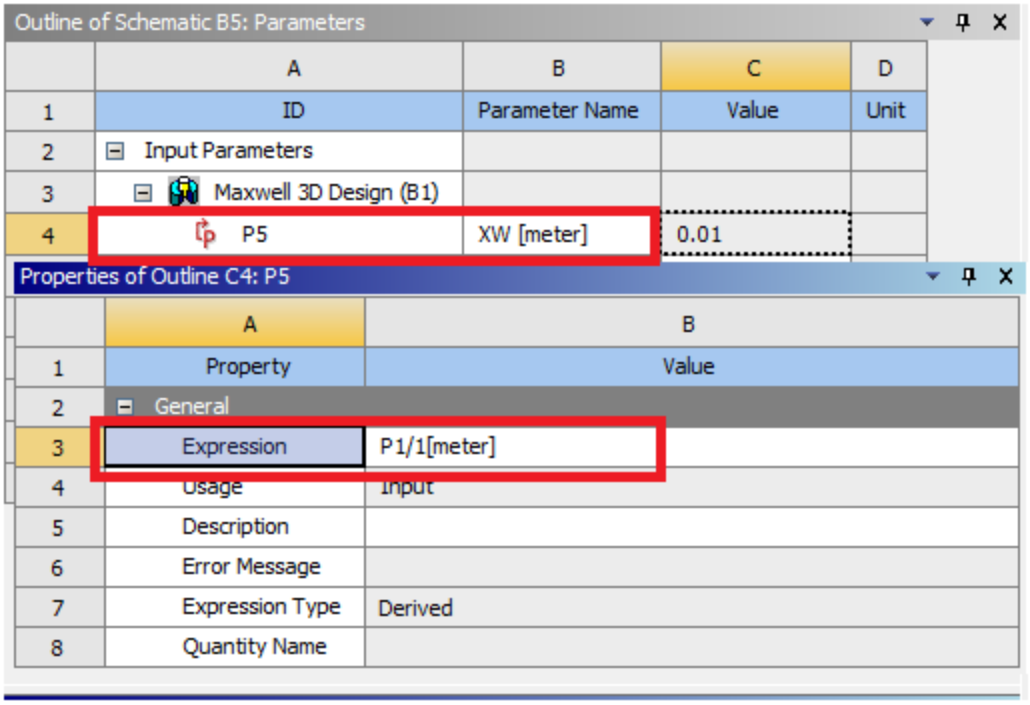
The workflow for using Electronics Desktop systems with the Workbench to take advantage of Distributed Analysis is as follows:

1. In the Ansys Electronics Desktop, specify a variable an as input parameter. The following figure shows a DesignXplorer Setup that includes a geometry variable called XW This variable is mapped to a variable in the corresponding Workbench Design, in this example, P5.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

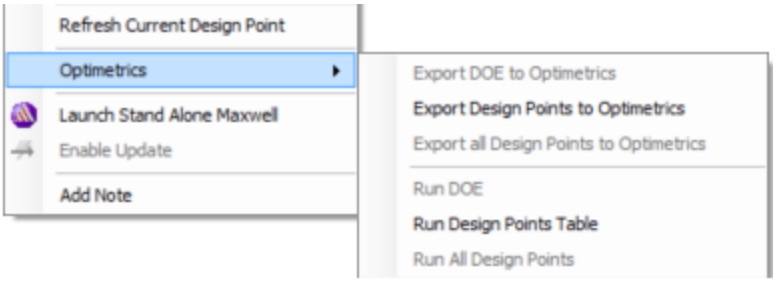


2. In the Workbench Design, change the P5 expression to a new value (in this example figure, the P5 parameter is assigned an expression of P1/1[meter] to tie it to the geometry system's variable.



In Workbench, the **Export Design Points to Optimetrics** and **Run Design Points Table** commands are enabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

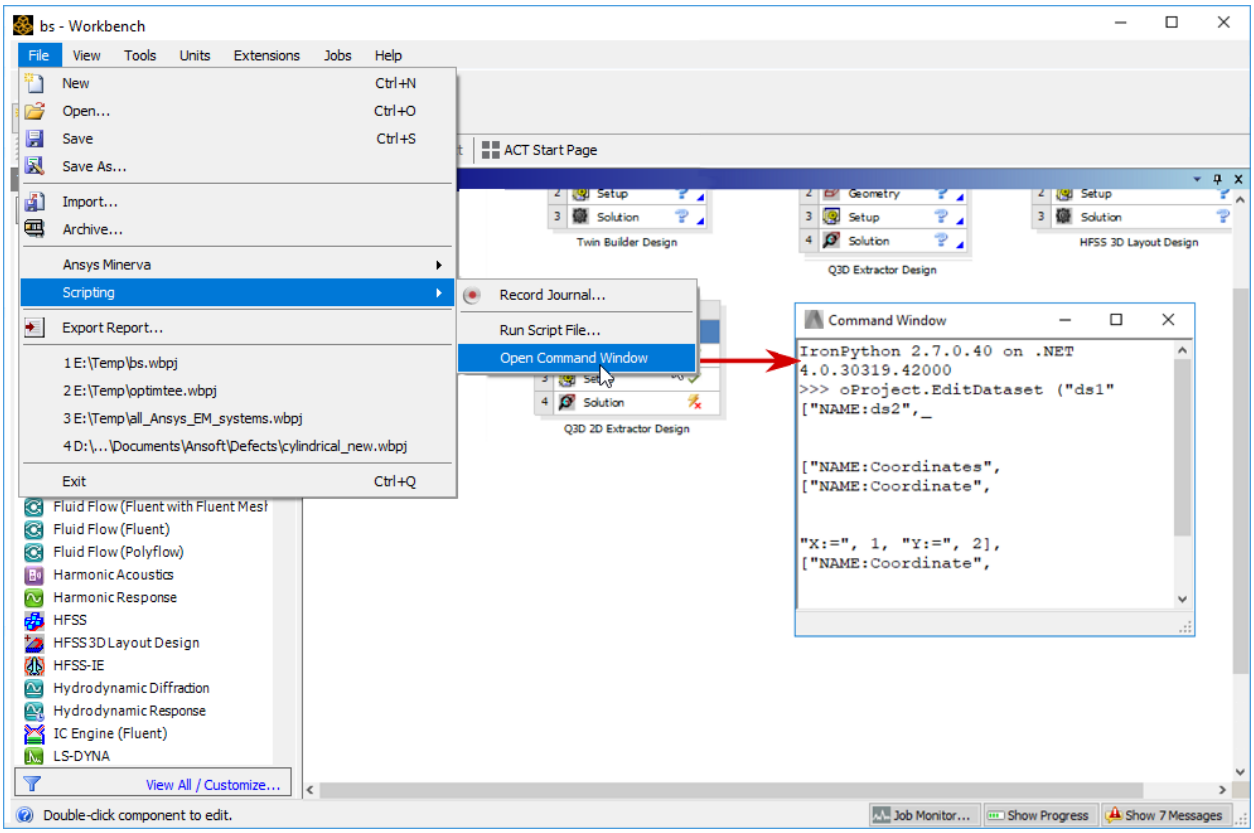


3. In Electronics Desktop, you can click on the default DesignXplorer Setup, and choose **Generate Variation Data**.

If you have configured your Analysis Configuration for multiple machines, the Project is ready for Distributed Analysis. After analysis, the Workbench Parameter table should be updated correctly for all design points.

Scripting in Workbench

Scripts that include Ansys Electromagnetics projects can be recorded and run in Workbench.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When a Workbench journal is run from the command line, Electronics Desktop allows the Ansys EDT portion to also run in batch mode (that is, when using RunWB2 -b).

Example: From a Windows Command prompt, enter the following command to replay the journal named "myReplayJournal.wbjn":

```
RunWB2 -b -r myReplayJournal.wbjn
```

Ansys EM - Ansys Multiphysics Coupling

Data integration provides an improved multiphysics workflow between Ansys Electromagnetics designs and Ansys applications such as Mechanical and Thermal. Coupling is provided through project schematic links. Heat losses and force data are automatically transferred to Ansys Mechanical, and there is no need to export or import transfer XML files. Edits made in Ansys Electromagnetics applications are automatically transferred to the Ansys application through the Workbench **Refresh** command. Workbench commands also enable easier automation of iterative coupling of thermal feedback.

The following sections provide some examples of multiphysics coupling.

Multiphysics Coupling on Workbench with Ansys Thermal

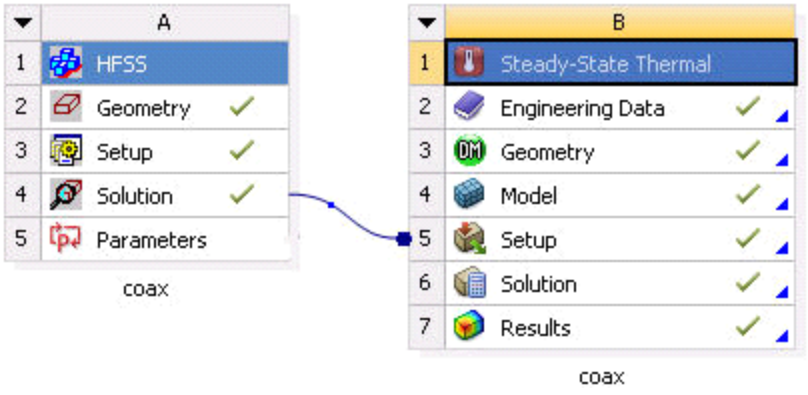
Multiphysics Coupling on Workbench with Ansys Structural

Multiphysics Coupling between Ansys EM Field Systems on Workbench

Multiphysics Coupling on Workbench with Ansys Thermal

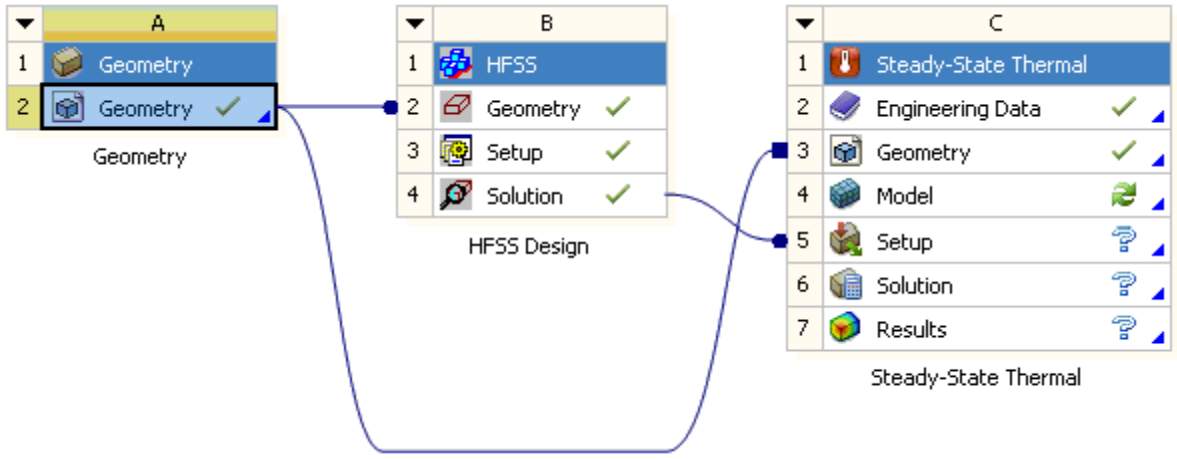
Using data integration, HFSS, Maxwell, and Q3D Extractor provide heat losses (heat generation and heat flux) to Ansys Thermal. In HFSS, both Driven and Eigenmode projects can be coupled for multiphysics. You first need to enable feedback as described in Setting the Temperature of Objects.

Note how the HFSS design is linked visually to Ansys Thermal on the Workbench project schematic.

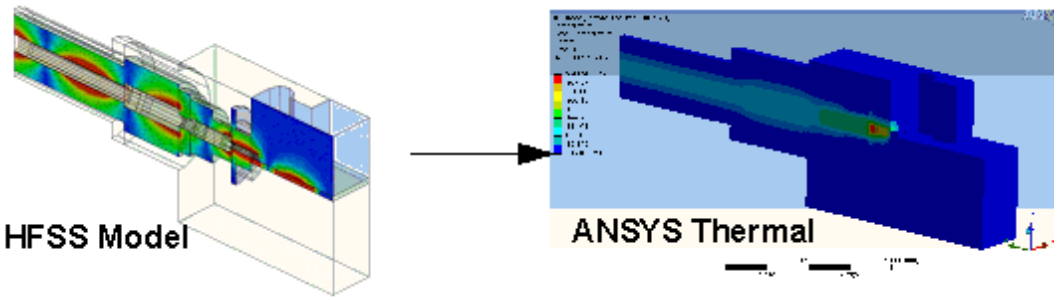


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometry sharing is possible, provided you activate Beta Options in Workbench. A geometry from the Component Systems in the Toolbox in Workbench Schematic can be shared as shown in the image below.



In this example, HFSS coax model Solution provides heat loss data as a thermal load to the Ansys Thermal Setup. The resulting analysis shows a thermal "hotspot", providing the user with the information needed to adjust the design's material to fix the problem.

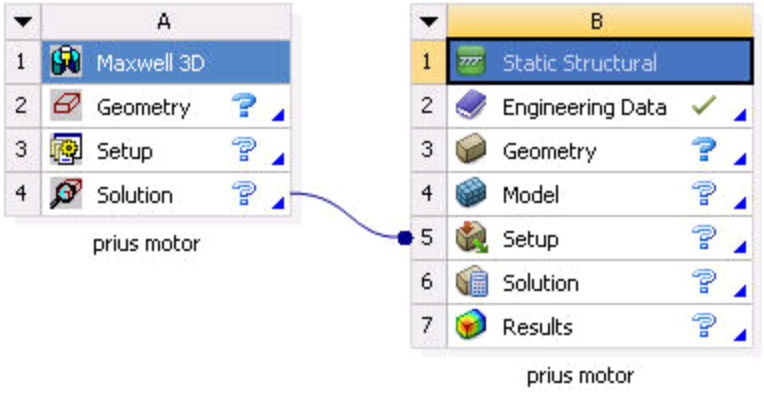


You enable this feature by checking Enable Feedback in the dialog for Setting the Temperature of Objects.

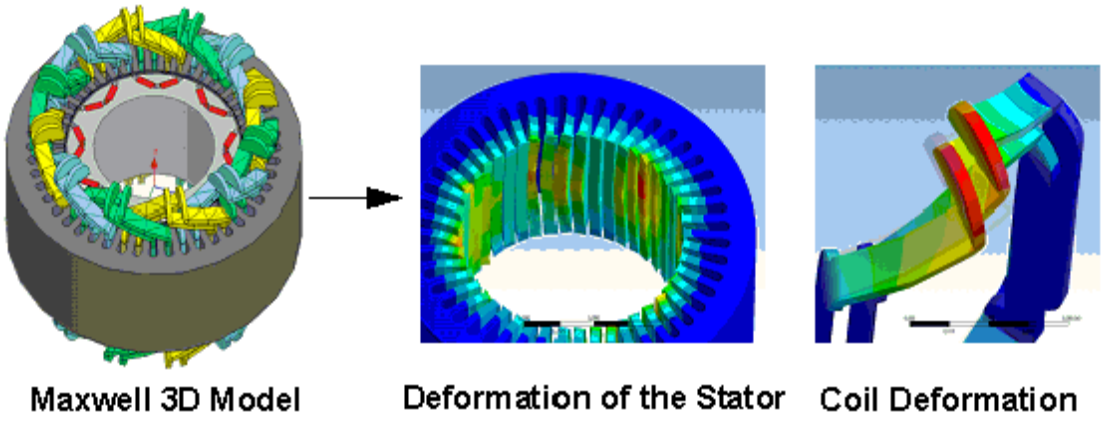
Multiphysics Coupling on Workbench with Ansys Structural

Using data integration, Maxwell 2D and Maxwell 3Dcan provide forces to Ansys Structural.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In this example, the Maxwell 3D electromagnetic force density **Solution** is used as the load in Ansys Structural to determine how these forces deform the motor's stator and coils.

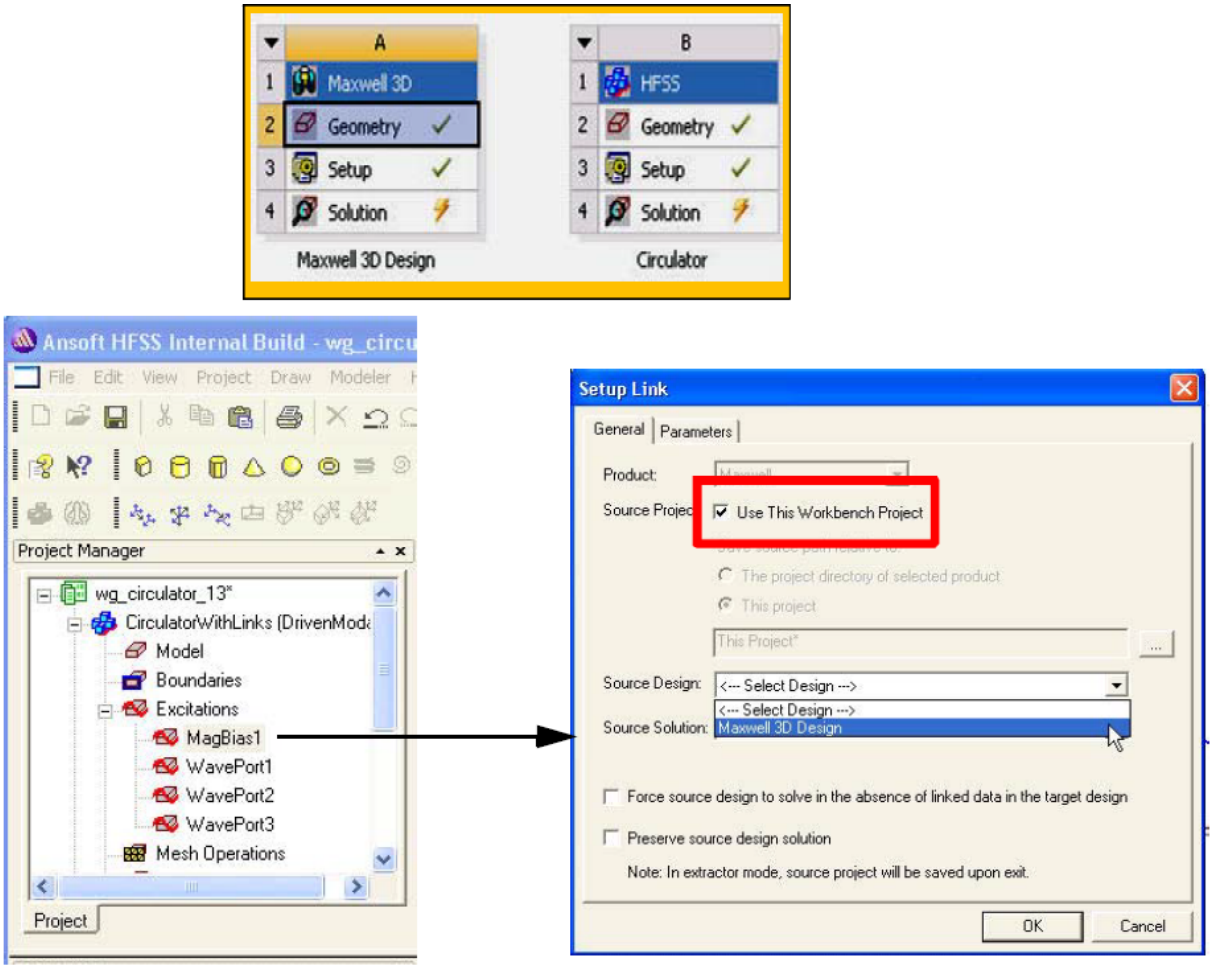


Multiphysics Coupling between Ansys EM Field Systems on Workbench

You can set up links between Ansys Electromagnetics field systems that reside on a Workbench project schematic. Linking is set up in the Ansys Electromagnetics application as shown in the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

following example.

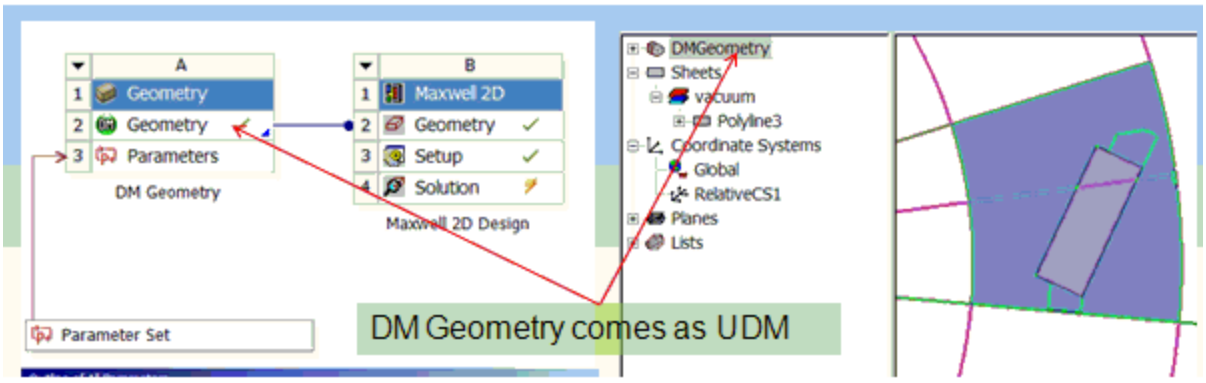


Ansys EM CAD Integration through Workbench

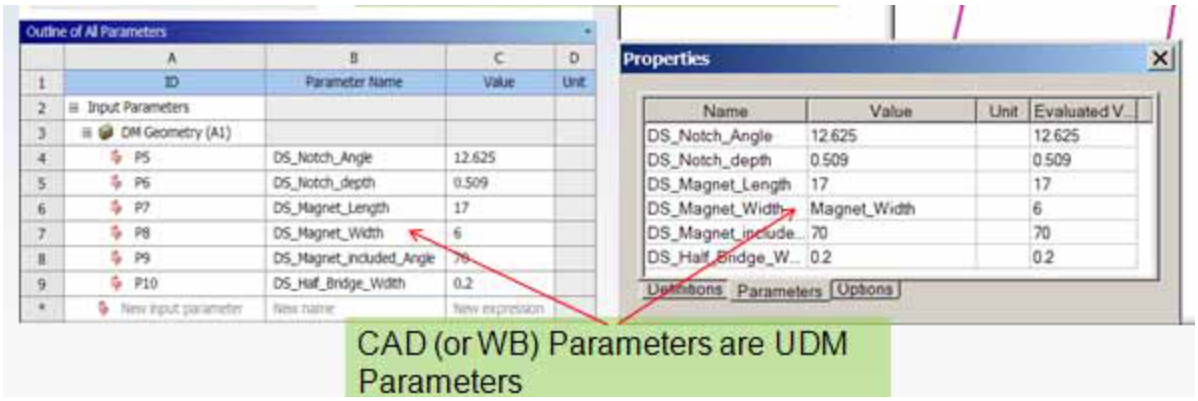
Ansys Electromagnetics CAD integration is a Workbench feature available for Ansys Electromagnetics 3D Products - HFSS, Maxwell, Q3D, Icepak, and Mechanical, as the Ansys Framework. The feature is available only through Workbench, and is not available via standalone Ansys Electromagnetics products.

Ansys CAD integration provides a bi-directional dynamic link through Workbench. This makes it possible to get updated geometry from CAD, modify CAD parameters in Ansys Electromagnetics products, and return an updated geometry. The feature is non-associative due to a need to reassign boundaries if a modified CAD model is used. The process creates a User Defined Model (UDM) for each geometry source.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



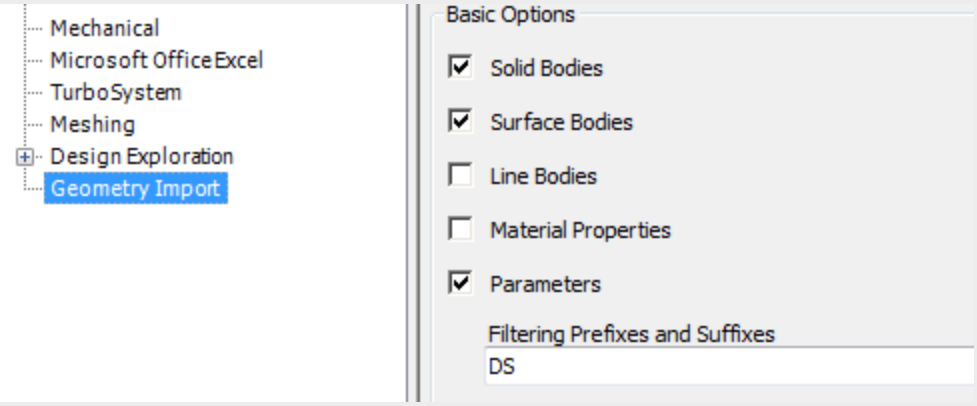
The UDM format makes it possible to exchange parameters. Ansys online help contains further description of the UDM feature and function.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

The parameters shown in the previous example all have a DS prefix. This is the default for the Workbench **Tools>Options** for **Geometry Import**. To import parameters with different prefixes or names, assign an appropriate prefix or clear the **Filtering Prefixes and Suffixes** field.



Ansys Electromagnetics CAD integration makes it possible to consume geometry from multiple upstream sources (any CAD or Ansys Electromagnetics product).

This feature supports direct interfaces with major CAD systems, including

- Creo Parametric
- UG NX
- CATIA V5
- SOLIDWORKS
- Autodesk Inventor
- Ansys Design Modeler (DM)
- Discovery UDM model

Note:

The CAD software must be installed on the user machine (not required for solve nodes).

The following sections contain additional information:

CAD Integration Functionality

CAD Integration and Geometry Sharing

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bi-Directional CAD Integration

CAD Integration Model Edits

Multiple Geometry Links for CAD Integration

Creating Dynamic Links to EDT Designs

Adding a Design Variable for Dynamic Links

CAD Integration Functionality

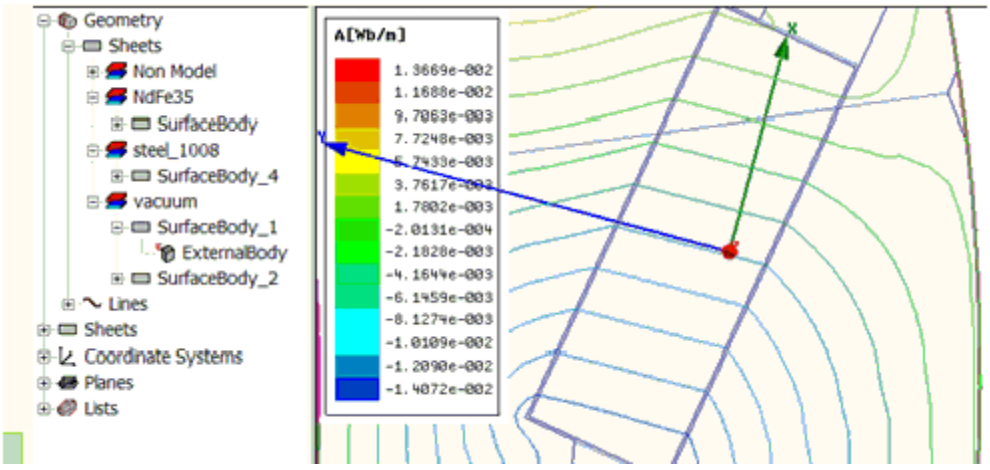
Healing with CAD Integration

Important Geometry Options for CAD Integration

CAD Integration Functionality

CAD Integration includes the following functionality:

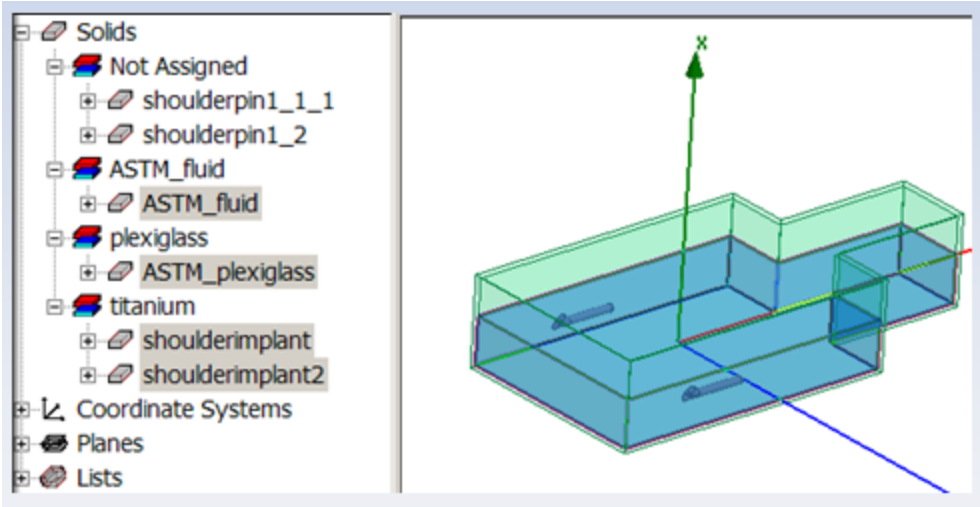
- DX analysis
- WB Update Project
- WB Update All Design Points
    - Parametric analysis with DSO
    - Animation
    - Geometry
    - Field plots



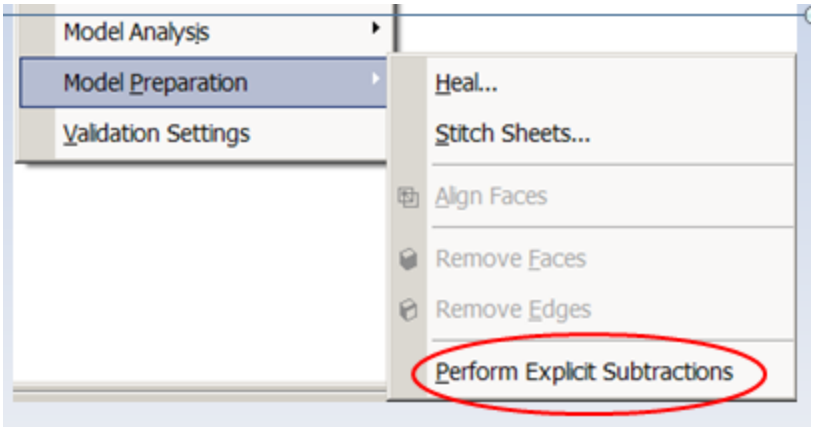
Ansys Mechanical does not do implicit subtraction. Ansys Electromagnetics products can do explicit subtractions before sending geometry to Ansys using **Modeler > Model Preparation > Perform Explicit Subtraction**.

For example, consider the following model.

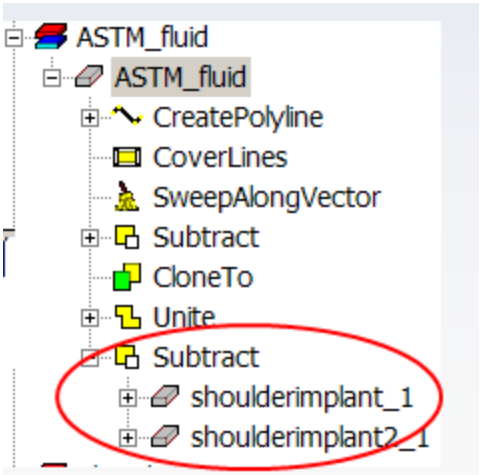
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Perform Explicit Subtractions can be performed.

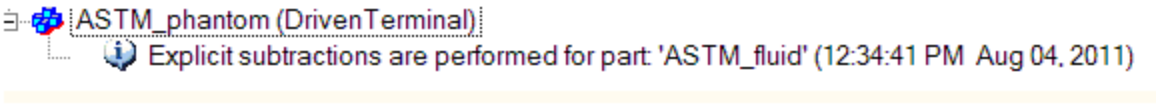


Results appear in the History tree, as shown:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Messages** window reports this action.

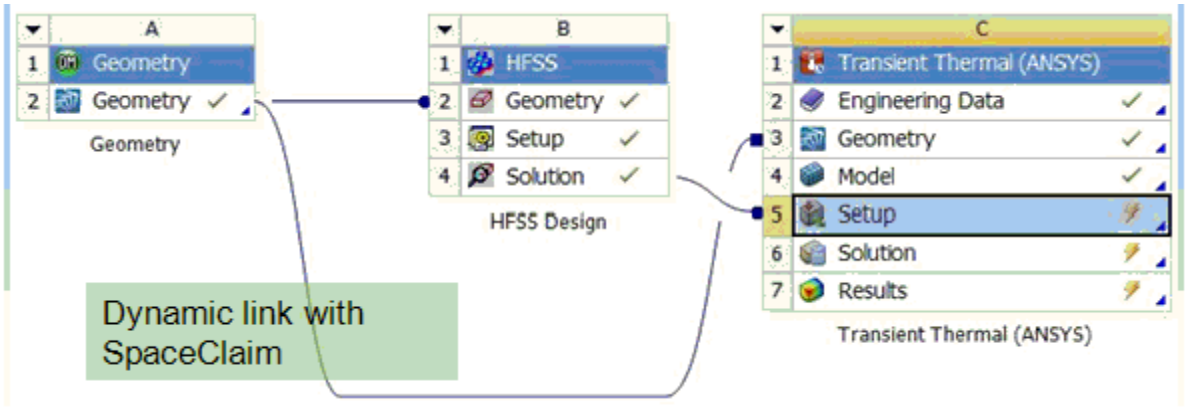


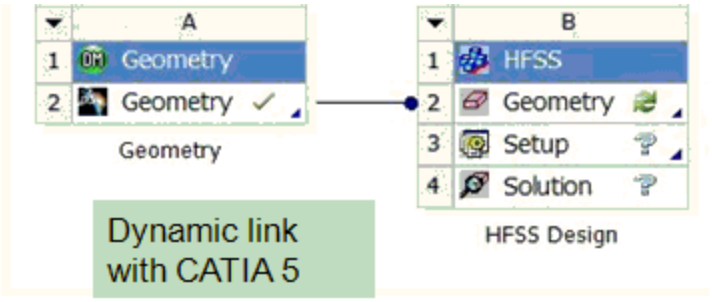
CAD Integration and Geometry Sharing

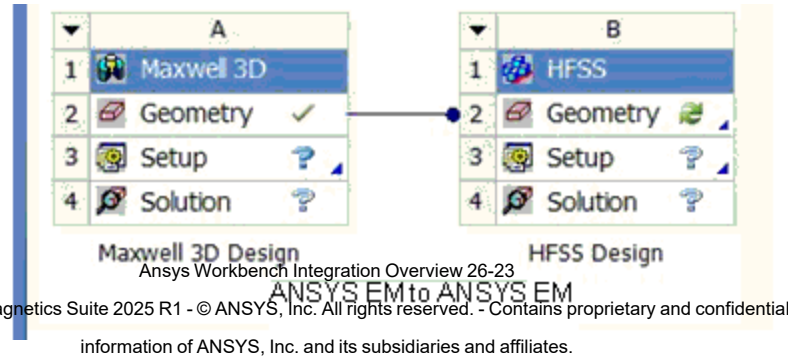
A CAD model comes into Ansys Electromagnetics as a User Defined Model (UDM).

The inputs to Ansys Electromagnetics from CAD are:

- Geometry/Topology with persistent IDs
- CAD parameters
- Material assignment
- Attributes, such as name and color

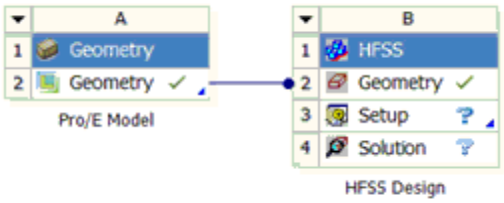




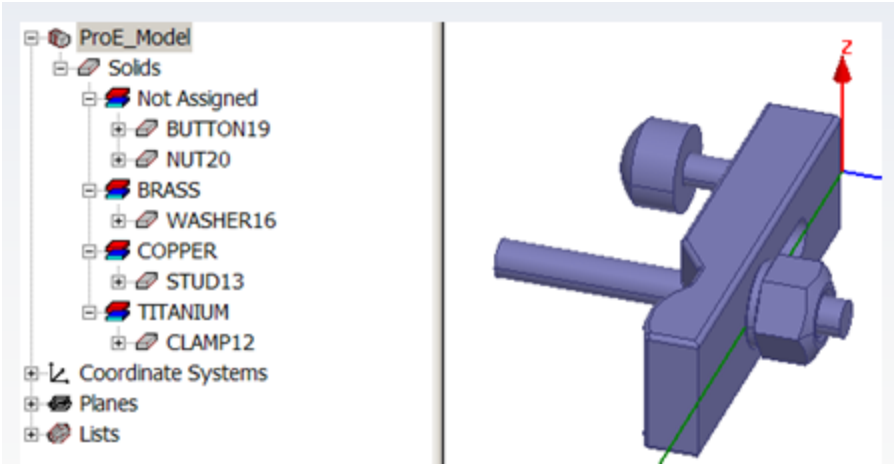


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, in Workbench, a Pro/E Model can be linked to Ansys.



The geometry can then be viewed in HFSS as a UDM.



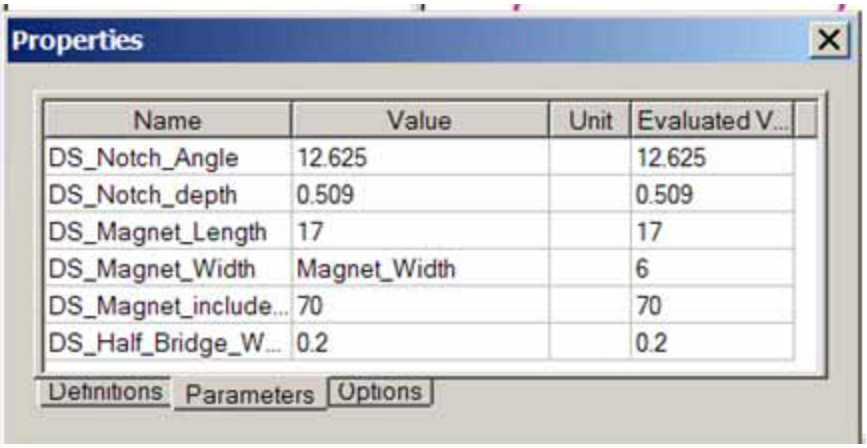
The CAD or WB model parameters appear in the Workbench:

Outline of All Parameters

	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	DM Geometry (A1)			
4	P5	DS_Notch_Angle	12.625	
5	P6	DS_Notch_depth	0.509	
6	P7	DS_Magnet_Length	17	
7	P8	DS_Magnet_Width	6	
8	P9	DS_Magnet_included_Angle	70	
9	P10	DS_Half_Bridge_Width	0.2	
*	New input parameter	New name	New expression	

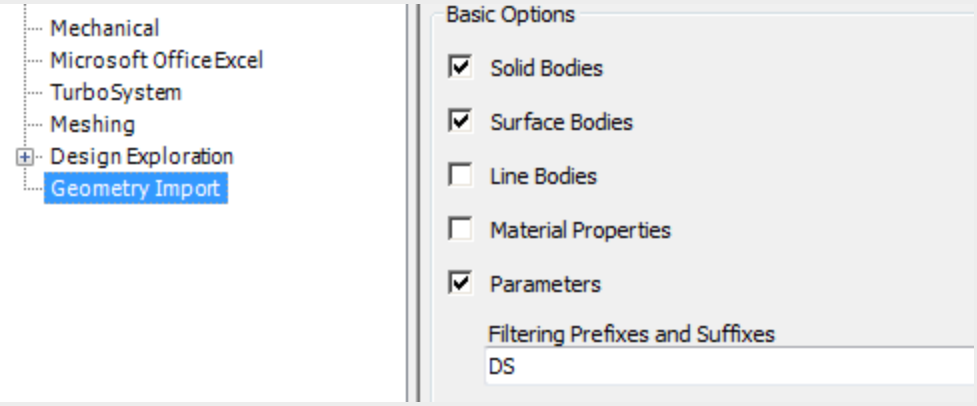
Through the Ansys Electromagnetics CAD integration, the linked UDM includes the same parameters.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



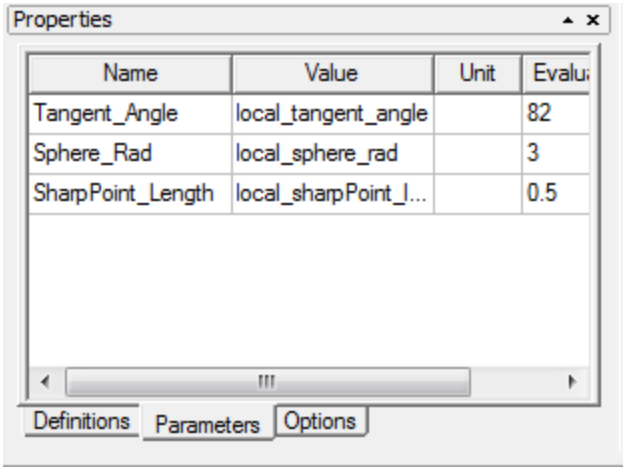
Note:

The parameters shown in these examples all have a DS prefix. This is the default for the Workbench **Tools>Options** for **Geometry Import**. To import parameters with different prefixes or names, assign an appropriate prefix or clear the **Filtering Prefixes and Suffixes** field.



Once you import a geometry with parameters to an Electromagnetics application in the desktop, you need to map to local variables. In the target Electromagnetics application, select the geometry associated with the parameters. Update the geometry and view the **Parameters** tab in the **Properties** window. The **Value** column shows the values of the imported parameters. Type names for local variables in the **Name** column.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once you have local variables for the imported Design Modeler parameters, you are ready to use Design Xplorer for multiple design parameters.

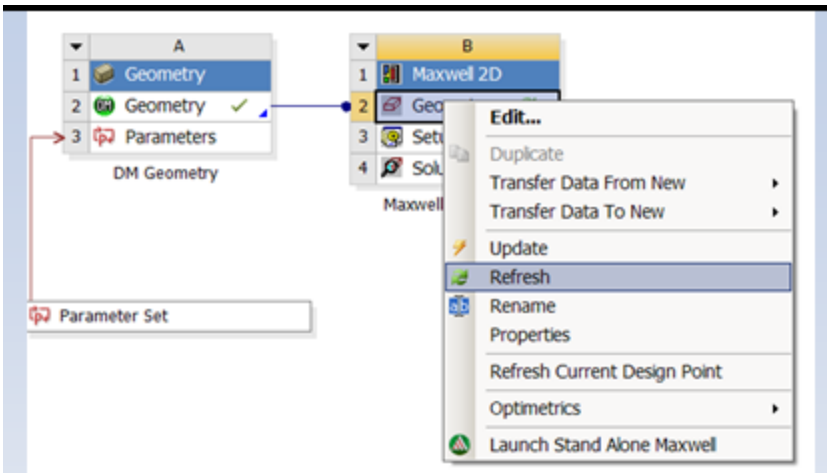
Table of Schematic D2: Design of Experiments (Custom)

	A	B	C
1	Name	P3 - Tangent_Angle (degree)	P4 - Sphere_Rad (in)
2	1	80	3
3	2	75	3
4	3	85	3
5	4	80	2.7

Bi-Directional CAD Integration

Ansys Electromagnetics CAD Integrations uses **Refresh** or **Generate** to pass updates.

You make an edit in a CAD application and run **Refresh** on an Ansys Electromagnetics Geometry Cell.

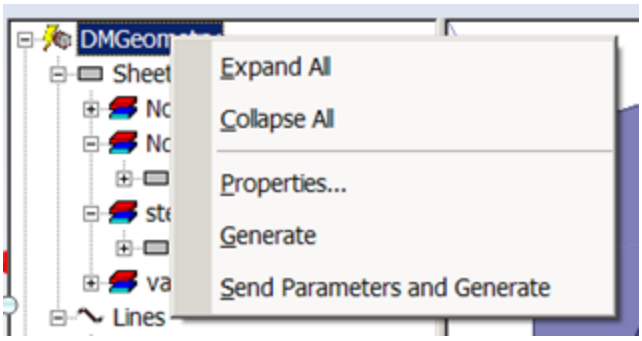


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refresh pulls the current state of CAD model (geometry, parameters, materials etc) and updates the corresponding data in the Ansys Electromagnetics application.

Alternatively, you can run **Generate** on the UDM in the Ansys Electromagnetics window.

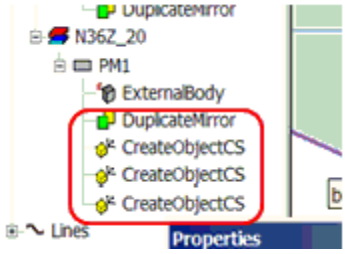
If you edit UDM (CAD) parameters in the Ansys Electromagnetics modeler window you can run the **Send Parameters and Generate** command.



The command passes the edited parameters to the linked CAD application and then pulls corresponding CAD geometry.

CAD Integration Model Edits

Several modeling operations are allowed on a CAD model in the Ansys Electromagnetics Modeler window. Operations are included in the History tree and retained during model **Refresh**.

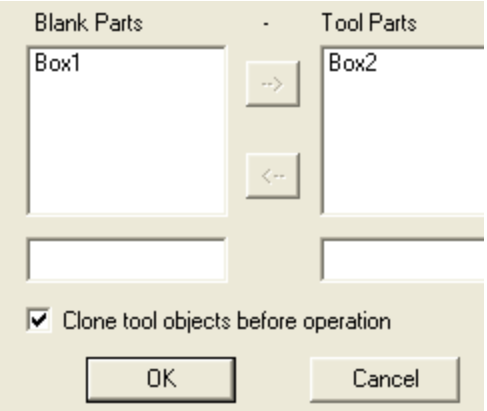


The following operations are not allowed:

- Non-history tree operations like Heal or defeature.
- Operations that use UDM parts as tools, such as Sweep

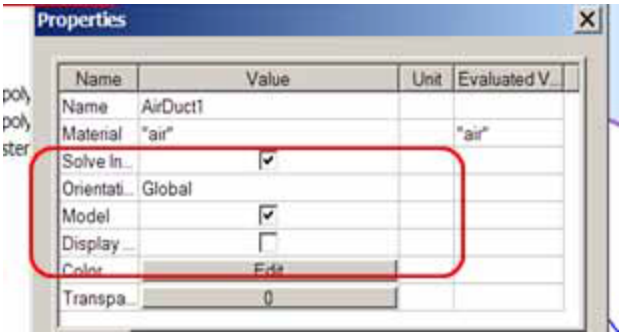
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Boolean operations like Split or Unite (allowed when you select the clone tool option)



The following part attributes can be modified for UDM parts:

- Model/Non Model flag
- Solve Inside flag
- Part orientation
- Color
- Display Wireframe



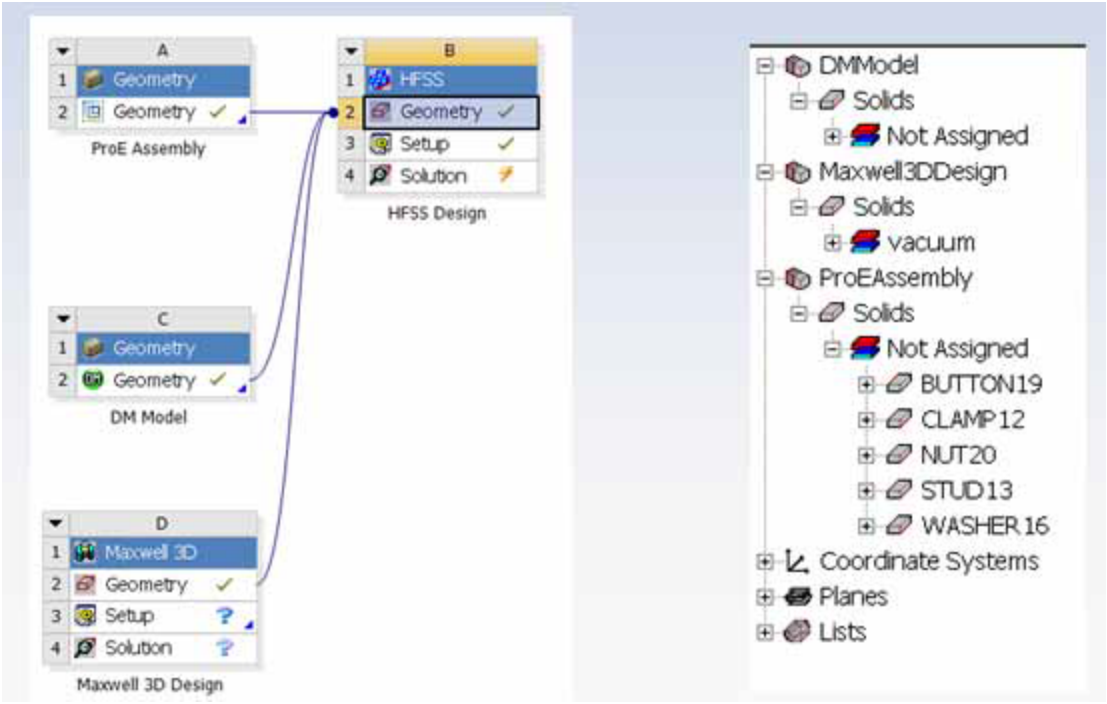
It is not possible to delete individual parts of UDM.

Multiple Geometry Links for CAD Integration

With CAD Integration you can consume geometry from multiple upstream sources. The source can be any of supported CAD or Ansys Electromagnetics products. This creates a UDM for each geometry source.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

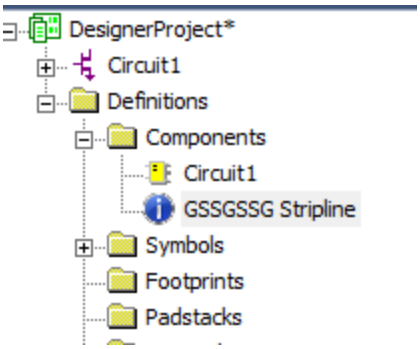
For example, the following figure shows a DM Model, a Maxwell model, and ProE model linked to HFSS in Workbench, and displayed in the HFSS History tree as three UDMs.



Creating Dynamic Links to EDT Designs

If you have a Circuit Design and one or more HFSS, 2D Extractor, Q3D, Icepak, or Mechanical designs open in a Workbench project, you can create dynamic links to those designs.

Update the Circuit Design to open the Electronics Desktop. In the menu bar, select **Workbench > Update Dynamic Link Components**. This command will create components in the project tree for each design in the Workbench project.

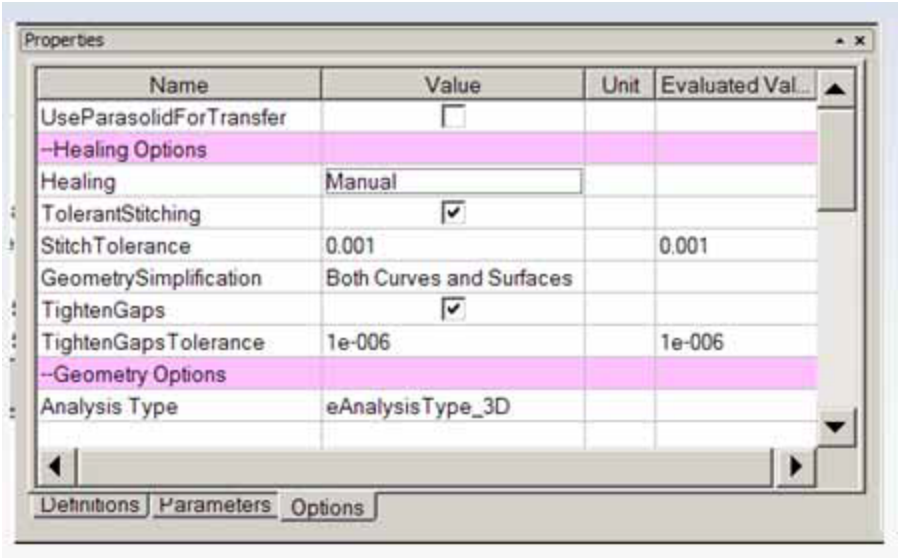


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that in this example, a GSSGSSG Stripline component was created corresponding to the GSSGSSG Stripline design in WorkBench. Once created, you can drag the component into a schematic to place it.

Healing with CAD Integration

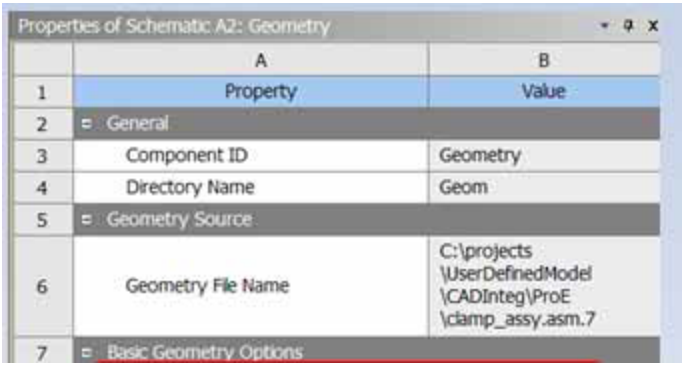
It is not possible to use the Heal command in Ansys Electromagnetics Modeler. Instead similar healing options are available under UDM properties option tab.



Healing options are None, Auto and Manual. By default, healing is off (None) and should be turned on only if required.

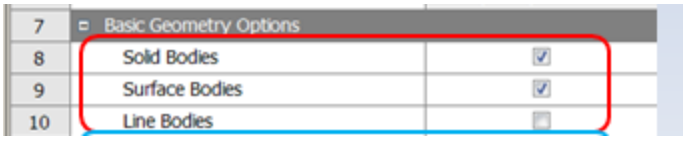
Important Geometry Options for CAD Integration

Select a Geometry Cell in Workbench to see options in the Properties window.



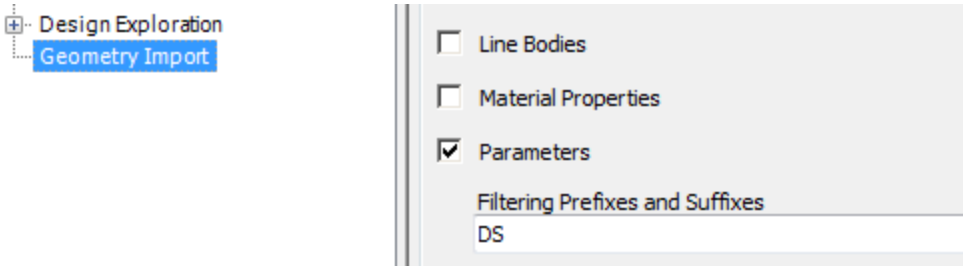
Control dimension of bodies coming from CAD

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Make sure **Parameters** is checked and that the parameter key (filter) is appropriate for CAD parameters.

The default parameter key filtering means that only parameters whose names start with DS will come through. This is the default for the Workbench **Tools>Options** for **Geometry Import**. To import parameters with different prefixes or names, assign an appropriate prefix or clear the **Filtering Prefixes and Suffixes** field.



The **Attribute Key** should be empty or "Color" to bring in CAD colors.

Material Properties must be checked to bring in the material assignment.

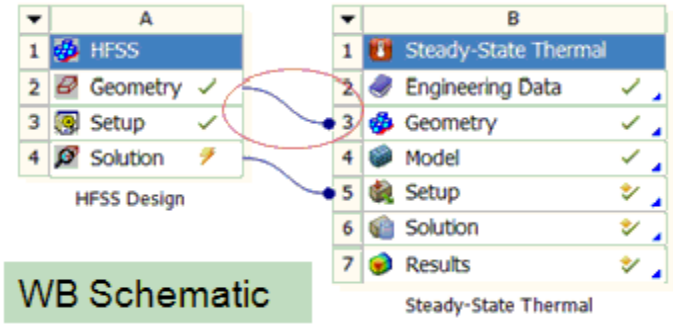
The **Mixed Import Resolution** option is used to resolve parts with mixed dimension (typically from Pro/E)

Refer to Ansys Help for details.

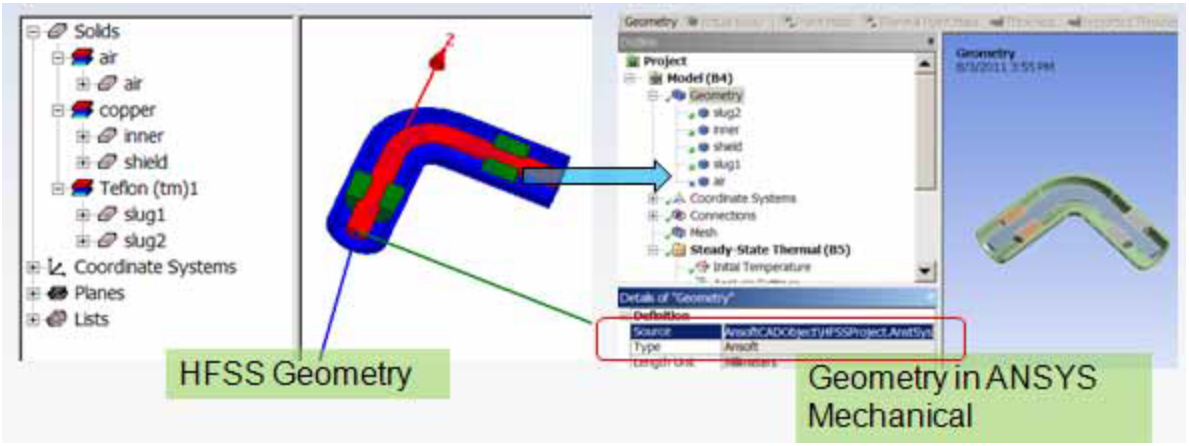
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys EM to Ansys Geometry Transfer

Ansys Electromagnetics CAD Integration transfers model information based on Workbench links.

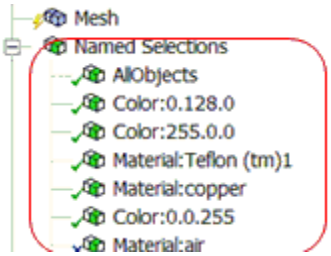


The following figure shows how the information is transferred between simulators.



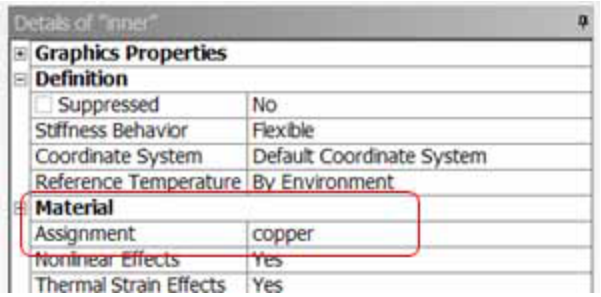
Information transferred includes:

- Geometry
- Ansys Electromagnetics lists and material assignment as Named Selection



- Material assignment

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The CAD Integration geometry link is dynamic (you can get updated geometry from Ansys Electromagnetics) and associative (IDs persist between Ansys Electromagnetics model and Ansys model during model refresh).

Boundary conditions in Ansys are preserved. Object, material, and parameter names with non-ASCII characters are not allowed for data transfer. Such transfers fail and produce an error message.

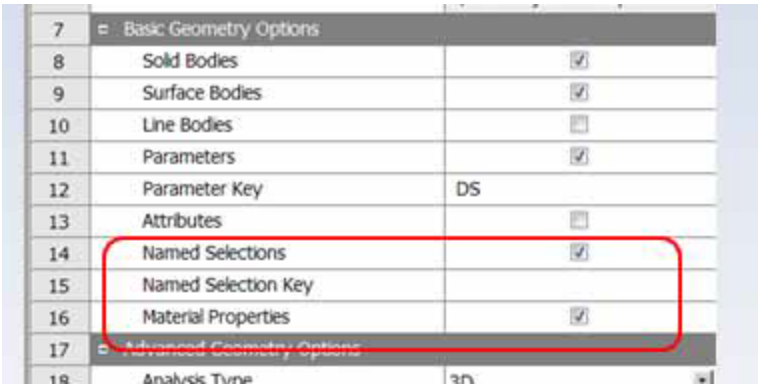
Material Assignment Transfer

There are two prerequisites for material assignment transfer:

- Engineering Data should have materials used in Ansys Electromagnetics model and material names should match (including case)
- Material Properties in Geometry Options must be checked

Before transferring as a named selection, **Named Selections** must be checked in **Basic Geometry Options**.

The Named Selection Key should either be empty or contain "Material."

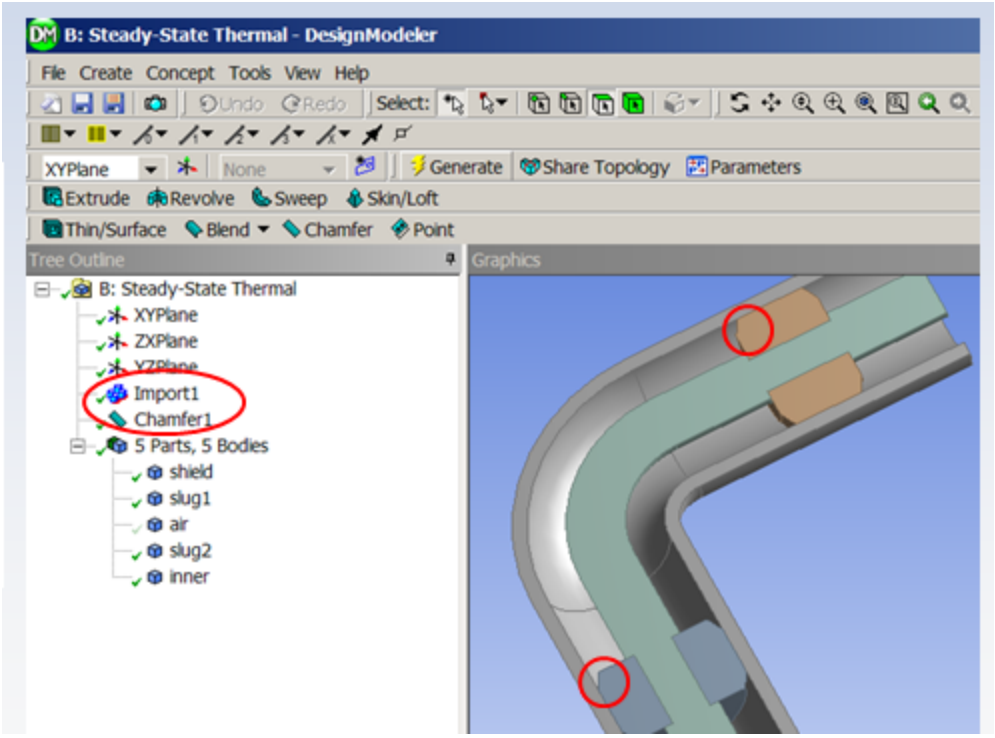


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometry Transfer through Ansys DesignModeler (DM)

It is possible to edit geometry in Ansys DesignModeler (DM) before consuming it in Mechanical. This is useful when geometry requires pre-processing for Ansys simulation.

Consider an HFSS model linked to DM through the Workbench. In the following example, a chamfer operation on geometries is being imported.

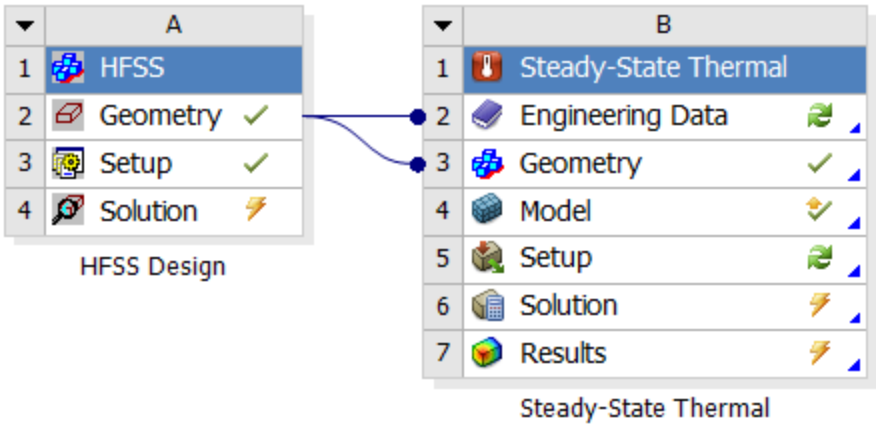


Workbench Material Data Transfer

A link can be created from an Ansys Electromagnetics Desktop system (e.g., HFSS) geometry cell to a downstream Engineering Data (ED) cell. When the downstream ED cell is refreshed, it updates with materials (names and properties) used by the upstream project. This ensures that the geometry-to-geometry link connecting the same two systems, which specifies the material assignments for various parts, refers to the correct materials and their properties.

Detailed Behavior

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The geometry-to-ED link can be created by dragging the Ansys Electromagnetics Desktop system geometry cell to the ED cell of a downstream system. When the downstream system's ED cell refreshes, the upstream system generates a MatML format XML file, which is consumed by Workbench to update the downstream system. This file is stored in the workbench project files according to design point. Note that Workbench ED cell material names are case-sensitive, as are material names within Ansys Electromagnetics Desktop products.

For material assignments to work with the ED in the downstream model, the downstream geometry cell must have the **Material Properties** check box selected. A convenient way to do this is to have this property checked by default, by setting this in **Tools > Options > Geometry Import**. Systems created after this default option is set will have **Material Properties** automatically checked for the geometry cell.

When the downstream model cell is refreshed or updated, material properties from the ED in the same system are used in the model's materials, based on material assignments set during the geometry cell refresh. Note that if material assignment fails, the Workbench default material (Structural Steel) is used.

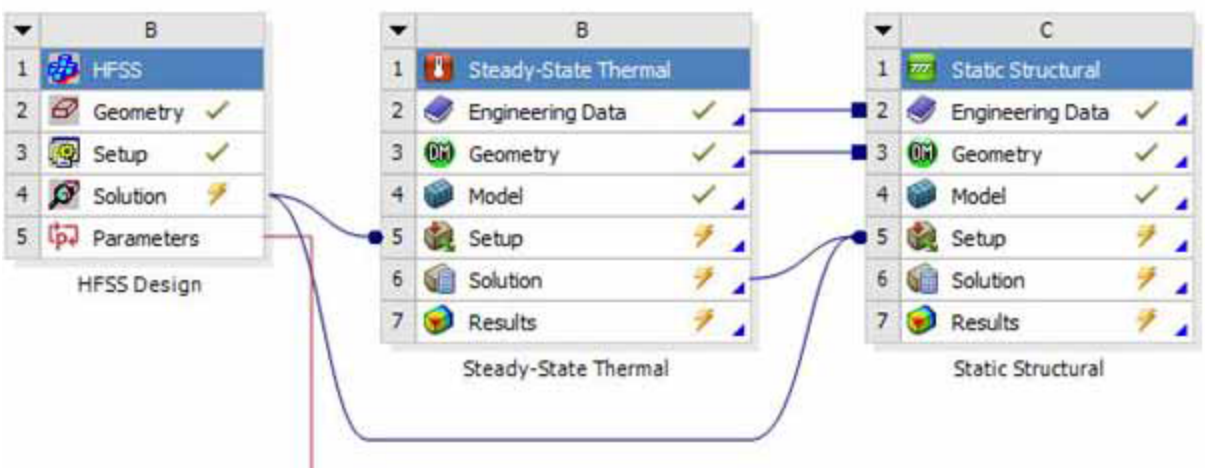
Workbench resolves name conflicts by changing the name of any materials being transferred over from the upstream system. For example, if the upstream Ansys Electromagnetics Desktop system uses a material named "Structural Steel", this will conflict with the existing downstream default material of the same name. The update will result in the Ansys Electromagnetics Desktop material being named "Structural Steel 2" in the downstream ED, but the downstream model will still use the properties supplied by Ansys Electromagnetics Desktop for the material.

Note that some materials may not satisfy all requirements for downstream system physics. For example, vacuum has no thermal conductivity, but downstream steady state thermal systems require a non-zero value for this property. Material properties in the workbench ED cell of the downstream system can be modified after refresh to ensure that validity criteria can be met. Any property added to a material in the downstream ED is preserved in subsequent ED cell updates. However, any existing property that is edited in the downstream ED will be overwritten by a subsequent ED cell update.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stress Feedback to HFSS using Workbench

Including stress feedback to HFSS involves coupling an HFSS system with both thermal and stress systems. The thermal system serves as an upstream system to the stress system and simulates the effect of heat induced displacement. And through the Workbench link between the Solution cell of HFSS and the setup cell of Mechanical, the displacement can feedback into HFSS. The HFSS project can be Driven or Eigenmode.



You use a dialog to select one or more objects (geometry) in HFSS upon which displacement data from Mechanical applies. This feature lets you avoid applying displacement to the whole model where only certain parts are sensitive to stress. You must select the same objects in the thermal and stress system to apply the respective imported loads.

The results of the process include deformation feedback applied to selected objects. The Solution Profile includes a report of Solve with thermal/displacement feedback, including information Maximum Delta T and Maximum Delta Displacement. You can create field plots with based on Displacement and Mag_Displacement, and you can scale the deformation displayed. The Fields Calculator also includes these input quantities.

Interface Changes for Stress Feedback from Mechanical to HFSS

Process Flow for Stress Feedback to HFSS

Interface Changes for Stress Feedback from Ansys Mechanical to HFSS

When used through the Workbench for stress feedback projects, the HFSS interface contains some commands and information that do not appear in standalone HFSS.

Deformation of Objects dialog

Solution Profile information for Solve with Displacement Feedback

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot Fields Menu with Displacement and Mag_Displacement

Deformation Scale tab for Modify Plot Attributes

Revert to Zero Displacement command under the **Solve Setup** menus

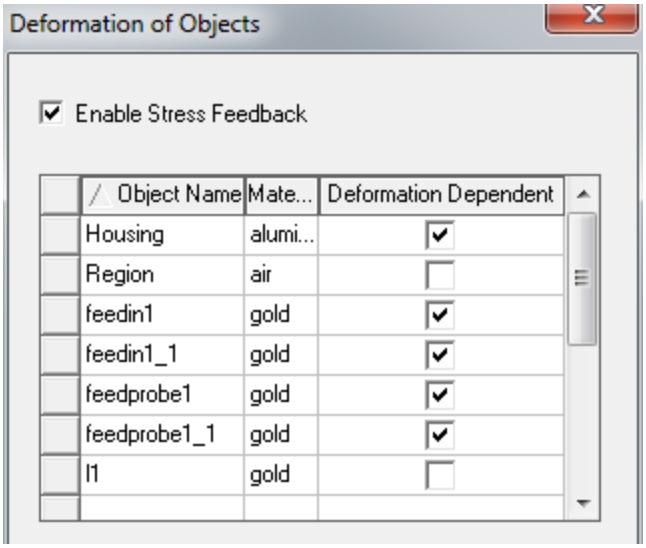
Fields Calculator Field Quantities for Stress Feedback Projects

Plot Fields Context Menu for Stress Feedback Projects

Modify Plot Attributes Dialog for Stress Feedback Projects

Deformation of Objects Dialog

When working with HFSS through Ansys Workbench, the HFSS menu for Driven and Eigenmode projects includes a Deformation of Objects command that opens this dialog for specifying which objects shall be deformation dependent.



The Enable Stress Feedback check box enables object selection. All selections are disabled when this is unchecked. Changing the setting invalidates any existing solutions.

The objects listed do not include objects without materials assigned, non-model objects, or non-3D objects. Objects assigned materials such as air or vacuum are not deformed and don't apply to stress feedback.

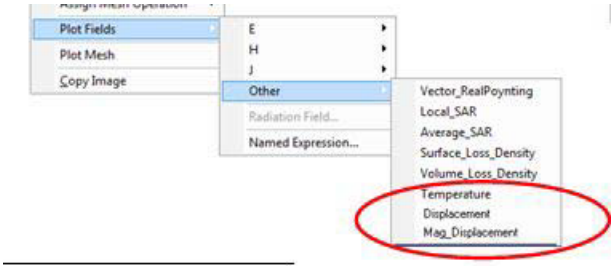
Solution Profile with Displacement Feedback

The Solution Profile includes a report of Solve with thermal/displacement feedback, including information Maximum Delta T and Maximum Delta Displacement.

Plot Fields Menu with Displacement and Mag_Displacement

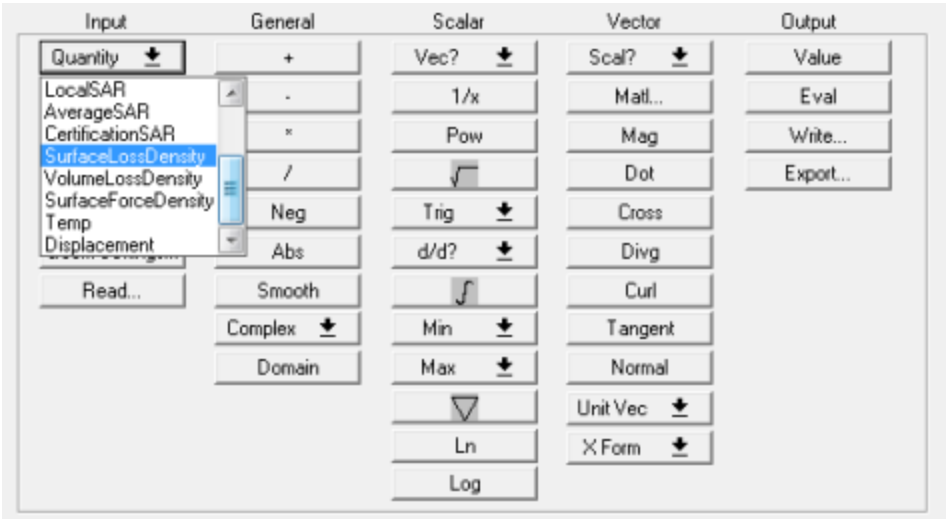
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Plot Fields menu includes entries for displacement and mag_displacement.



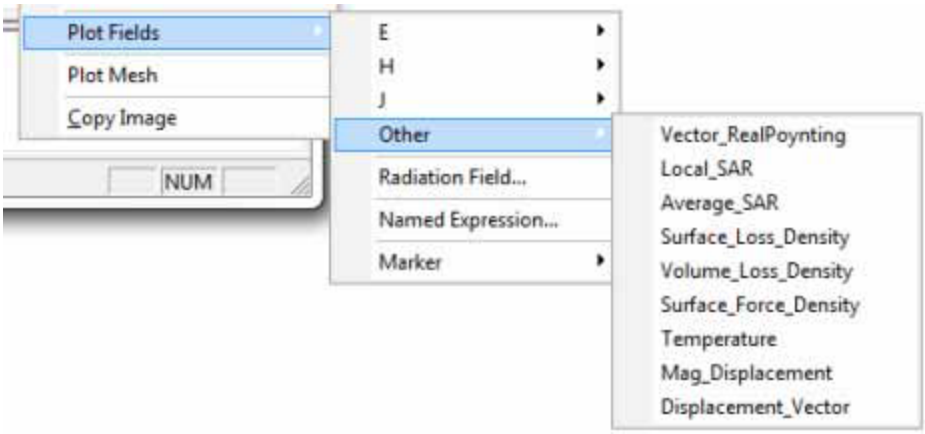
Fields Calculator Field Quantities for Stress Feedback Projects

The Quantity menu for the Fields Calculator lists Displacement for Stress Feedback projects.



Plot Fields Context Menu for Stress Feedback Projects

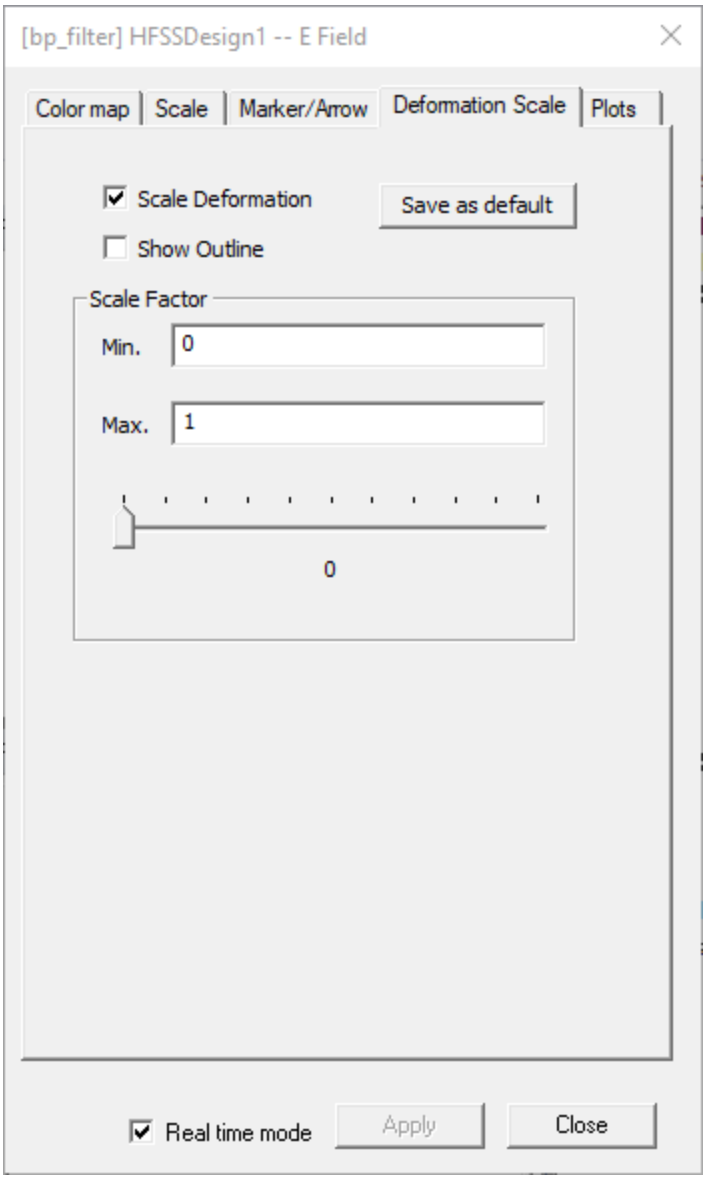
The **Plot Fields>Other** menu includes Displacement_vector and Mag_Displacement, a scalar. A field plot remains empty until the displacement data applies to the HFSS solution.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modify Plot Attributes Dialog for Stress Feedback Projects

When an HFSS plot includes deformation, the **HFSS>Fields>Modify Plot Attributes** dialog includes a tab for scaling the deformation. If you check Scale Deformation, you can also select Show Outline.

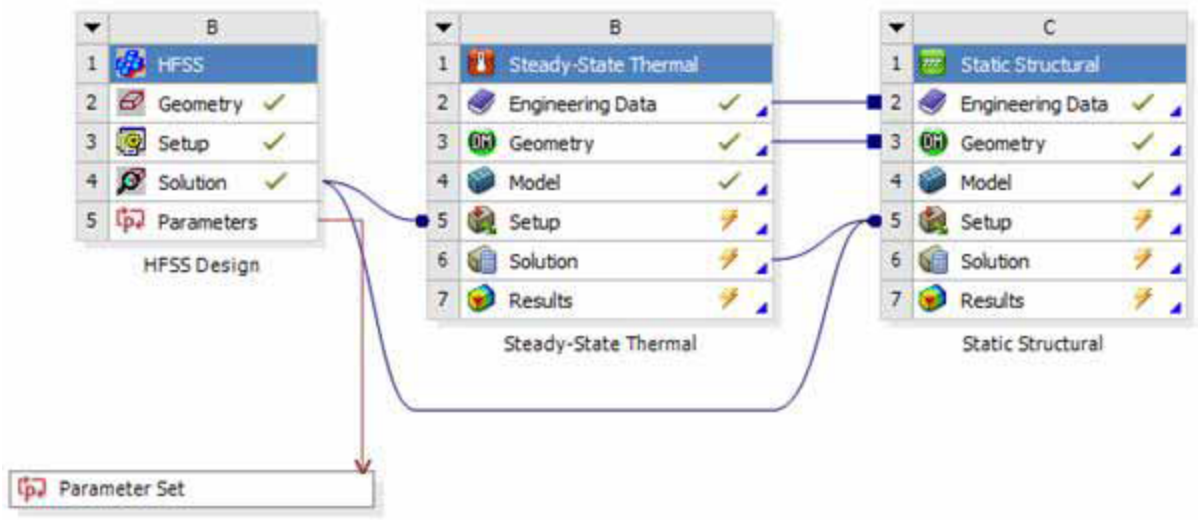


Process Flow for HFSS Workbench for Stress Feedback

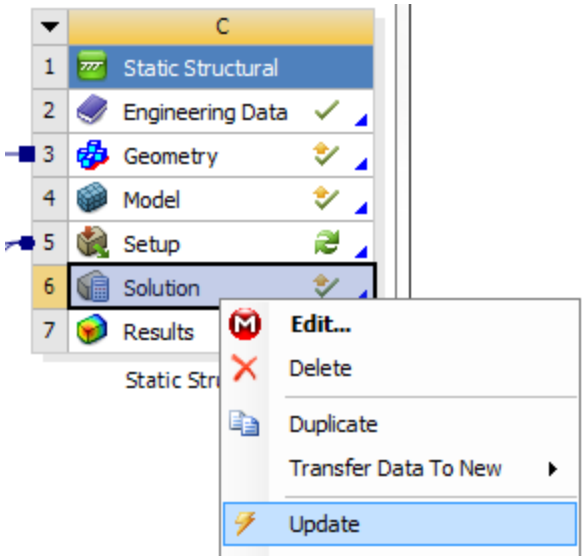
to set up the coupling, drag the "Solution" cell of the HFSS system and drop it at the "Setup" cell of a Thermal and Stress system. You also need to also couple the Thermal and Stress system to capture the effect of thermal force. The following image illustrates a coupling setup where both

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the Thermal and Stress system are ready to be "Updated". The HFSS adaptive solution has converged in 3 passes. The HFSS system has both thermal and stress feedback enabled.



1. Right-click the Solution cell of Stress and select "Update"

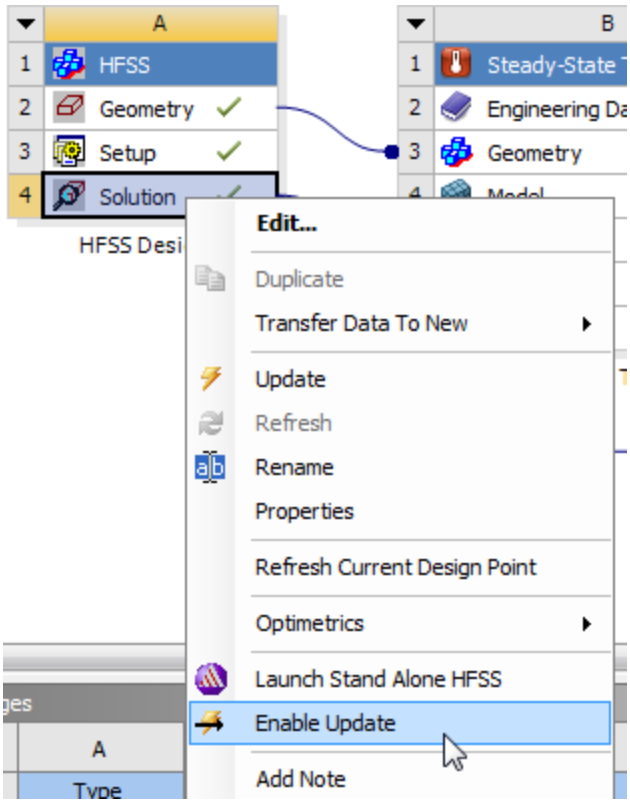


   a. The Setup of Thermal will be "Updated" with em loss from HFSS
   b. The Solution of Thermal will be "Updated" and temperature will be exported to HFSS

   c. The Setup of Stress will be "Updated" with thermal force from Thermal and force density (just zeros) from HFSS
   d. Displacement will be exported to HFSS after Stress finishes simulation

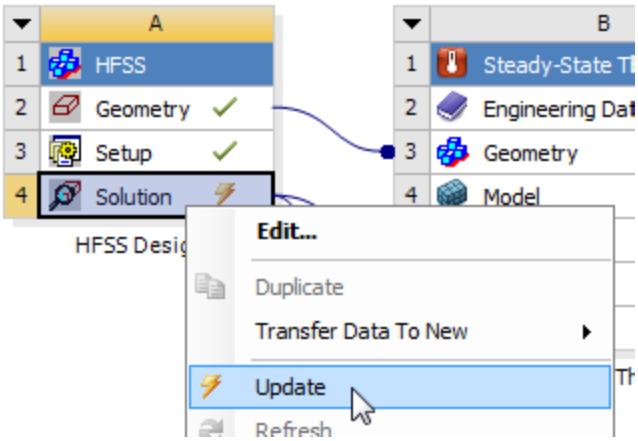
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Right-click the Solution cell of HFSS and select "Enable Update"



The Solution cell icon changes from a green checkmark to the Update lighting icon.

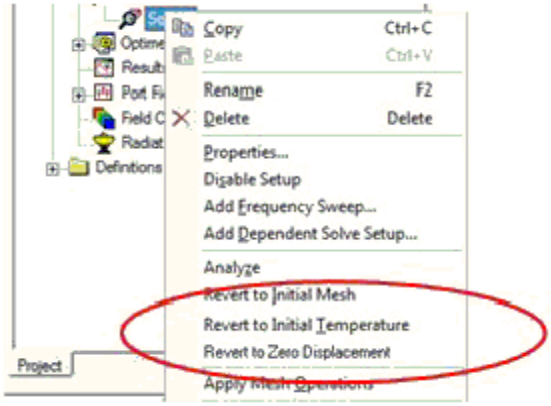
3. Right-click the Solution cell of HFSS and select "Update"



   a. HFSS re-simulate the 3rd pass with its mesh and the temperature/displacement feedback
   b. The profile will show information about the feedback

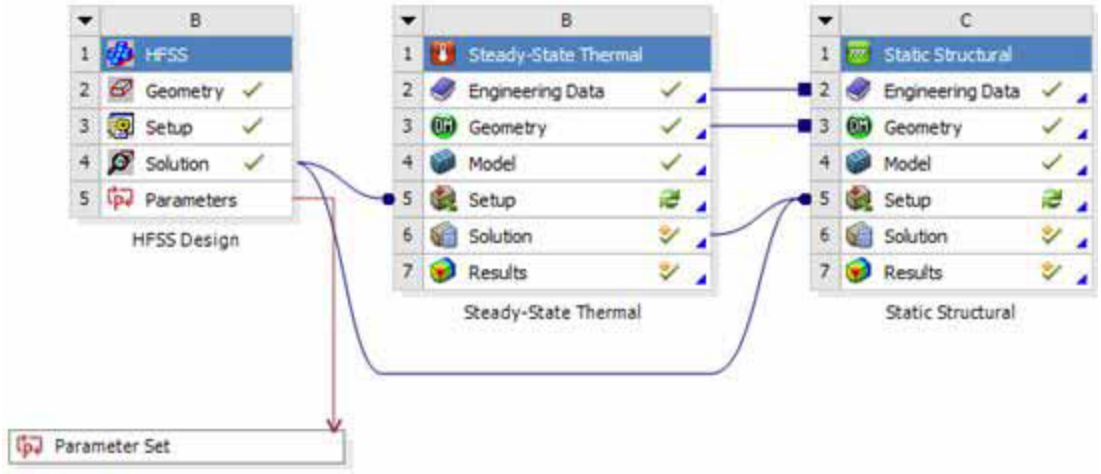
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Both "Revert" menus will be available at the right mouse click menu of the solve setup item.



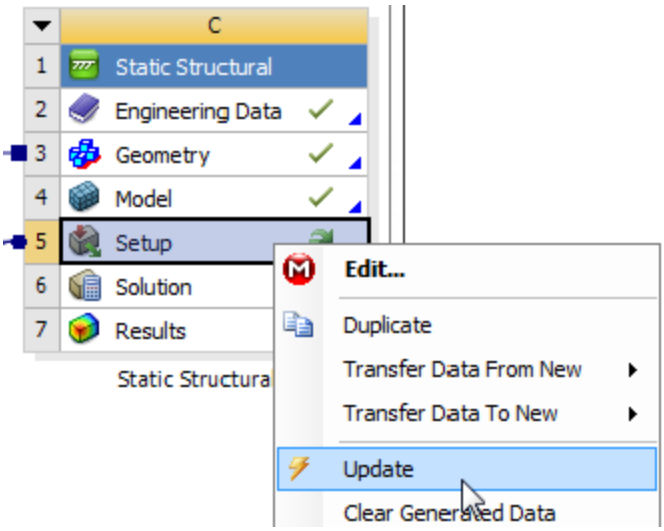
Manual Iteration After First Iteration Completes

The following image illustrates the WB schematic after the previous situation, after the 1st iteration has completed. Notice the changed icons for the status for Setup for Thermal and Structural.

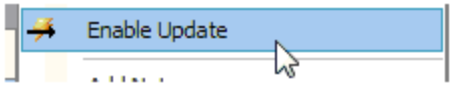


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

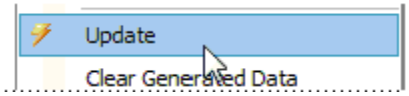
1. Right-click the Solution cell of Stress and select "Update"



   a. The Setup of Thermal will be "Refreshed" and "Updated" with EM loss from HFSS
   b. The Solution of Thermal will be "Updated" and temperature will be exported to HFSS
   c. The Setup of Stress will be "Refreshed" and "Updated" with thermal force from Thermal and force density from HFSS
   d. Displacement will be exported to HFSS after Stress finishes simulation
2. Right-click the Solution cell of HFSS and select "Enable Update"



3. Right-click the Solution cell of HFSS and select "Update"



   a. HFSS re-simulates the 3rd pass with its mesh and NEW temperature/displacement feedback
   b. The profile will show information about the feedback

Note that the delta temperature and displacement is being reported in the profile

Revert the HFSS Solution For Temperature and Displacement

You can revert the temperature and displacement in HFSS solution separately.

1. Select "Revert to Initial Temperature" in HFSS.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A warning message notifies you about the invalidation of solution

2. Right-click HFSS solve setup and select "Analyze"

HFSS re-simulates the 3rd pass with its mesh and displacement that was previously exported from Mechanical, but without temperature.

3. Select "Revert to Zero Displacement" in HFSS, and followed by "Analyze"

HFSS re-simulates the 3rd pass with its mesh without neither temperature nor displacement

Only Stress Feedback

The coupling setup is the same as in Use Case 1, with either one of the following differences

- The HFSS system is not enabled for thermal feedback. Since HFSS is not enabled to support feedback, the "Export Result" properties will be missing in the thermal system's property window.
- The HFSS system is enabled for thermal feedback, but users select not to "Export Result" in the thermal system. This is to disable the "automatic" feedback after Mechanical finishes its simulation.

1. Right-click the Solution cell of Stress and select "Update"
    a. The Setup of Thermal will be "Updated" with em loss from HFSS
    b. The Solution of Thermal will be "Updated"
    c. The Setup of Stress will be "Updated" with thermal force from Thermal and force density from HFSS
    d. Displacement will be exported to HFSS after Stress finishes simulation
2. Right-click the Solution cell of HFSS and select "Enable Update"
3. Right-click the Solution cell of HFSS and select "Update"
    a. HFSS re-simulate the 3rd pass with its mesh and the displacement feedback
    b. The profile will show information about the feedback and "Revert to Zero Displacement" will be enabled at the RMC menu of the solve setup item

Feedback Iterator

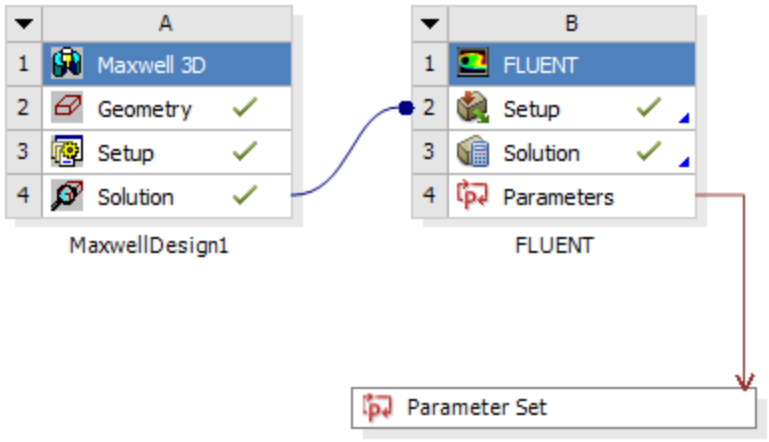
Background Information

Prior to the introduction of the Feedback Iterator, Ansys Workbench supported a two-way loose-coupling protocol with Ansys Electromagnetics products.

SystemCoupling already uses the word coupling to mean low-level solver coupling. Existing Electromagnetics product coupling is loose compared to SystemCoupling and is limited to file transfers at the end of a complete solve in a standalone system/product. No communication occurs during a solve. The coupling is two-way.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The one way portion (Upstream source component to Downstream target component) is handled via existing workbench data/transfer connection mechanism.

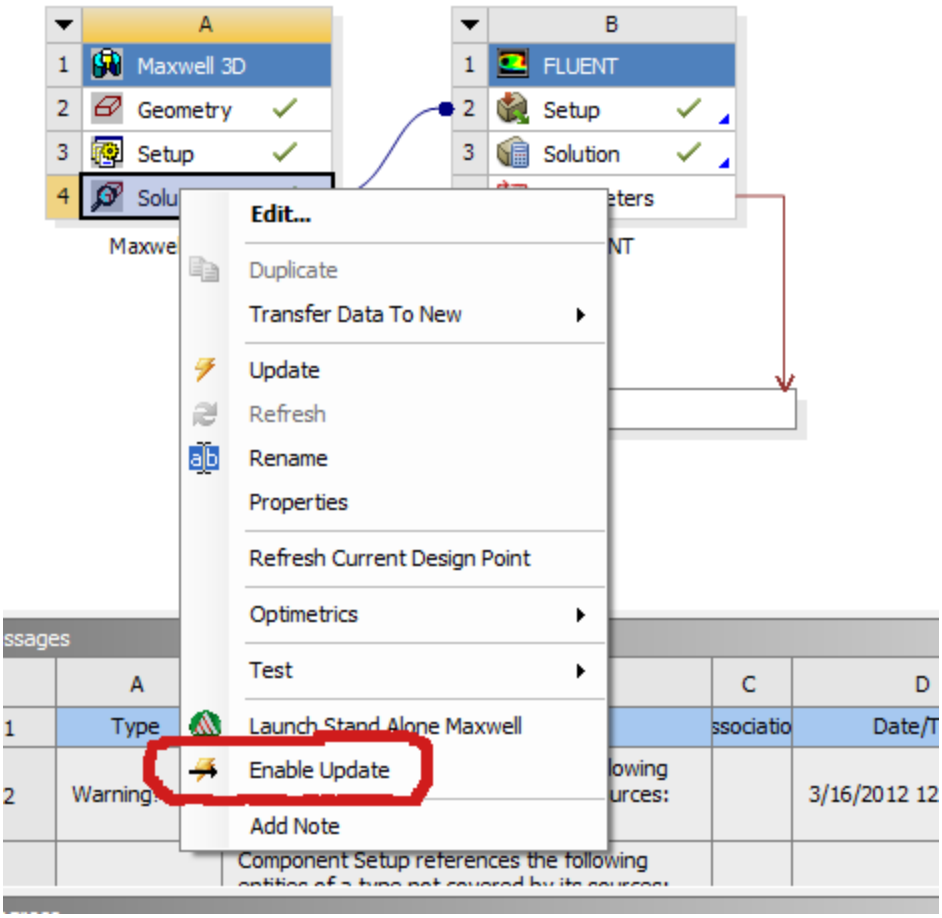


The round trip is handled by a separate protocol, agreed upon by the participating systems, whereby the downstream system exports a set of files to a location specified by the upstream system via its one-way transfer. This exported data is then incorporated by the upstream system in its next update.

To run the next coupled solve iteration, invoke the **Enable Update** operation, as shown below, which updates all systems involved.

These steps (**Enable Update**, **Update Project**) are continued as long as needed.

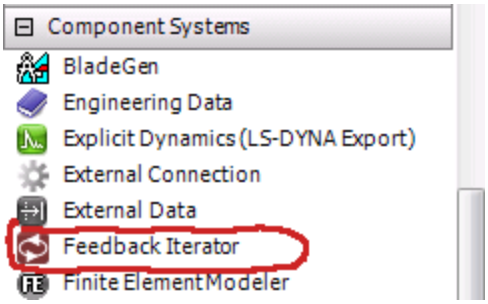
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Feedback Iterator System

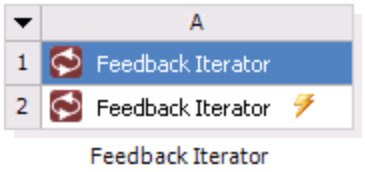
Ansys Electromagnetics Suite provides the Feedback Iterator system for automating the manual steps needed for driving a feedback utilizing system-pair to convergence. In addition to automating feedback incorporation over a user-specified number of iterations, the Feedback Iterator also allows control over the number of iterations (iteration termination criteria), target temperature, and displacement convergence criteria.

The **Feedback Iterator** appears in the Workbench Toolbox user interface under **Component Systems**.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

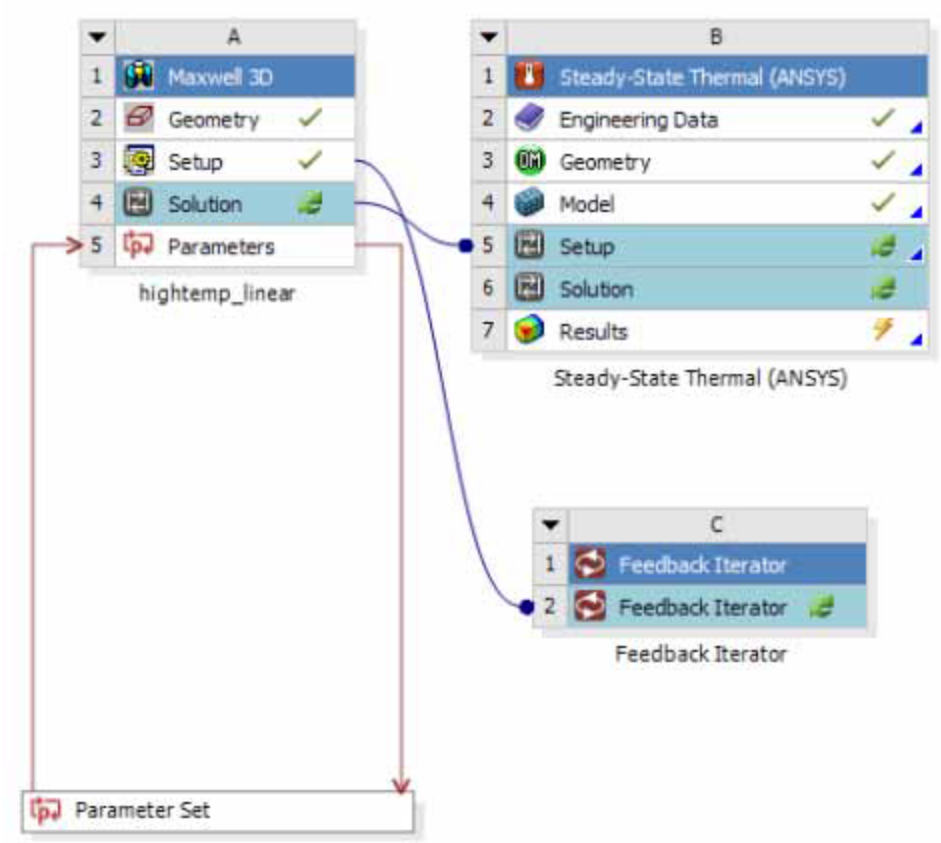
If you drag and drop an instance of the Feedback Iterator into Workbench Schematic, the default system name is Feedback Iterator.



Feedback Iterator in Use

The Feedback Iterator system attaches to the Workbench schematic like any other system.

In the following example, a Maxwell system provides AnsoftHeatLossData to a fluent system. Dropping the Maxwell setup cell onto the Feedback Iterator component results in the links shown.



The Feedback Iterator Addin functions the same way with all systems that participate in Electromagnetics product two-way loose coupling (Maxwell and HFSS for sources and Fluent, Thermal, and Structural for targets).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Feedback Iterator Component Properties

The Feedback Iterator's primary role is to control iterative solves and its properties target this end.

For Maxwell and HFSS, the Feedback Iterator properties are

- **Iterations Completed** – a read-only property that displays the number of iterations completed.
- **Max Iterations** – sets the maximum number of iterative solve loops to perform before terminating iterations in case the *Target Delta Temperature %* or *Target Delta Displacement %* is not achieved. The default value is 100.
- **Target Delta Temperature %** – specifies the maximum *Target Delta Temperature %* that signifies convergence. The default value is 5%, and the value must be 0.01 or greater.

  > **Note:**
  >
  > • The delta temperature error is calculated in Kelvins.
  >
  > • The Maxwell and HFSS design Profile tab displays absolute/relative delta data, while the delta temperate in Workbench means the relative delta (expressed as %).

- **Target Delta Displacement %** – specifies the maximum *Target Delta Displacement %* that signifies convergence. The default value is 5%, and the value must be 0.01 or greater.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

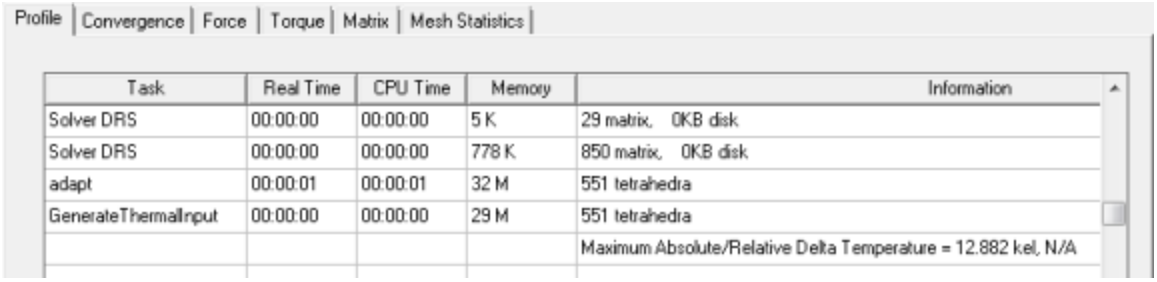
	A	B
1	Property	Value
2	⊟ General	
3	Component ID	FeedbackIterator
4	Directory Name	FeedbackIteratorComponent
5	⊞ Notes	
7	⊞ Used Licenses	
9	⊟ Iterations	
10	Iterations Completed	0
11	Max Iterations	100
12	⊞ Callback	
14	⊟ Temperature Convergence	
15	Target Delta Temperature %	5
16	Latest Delta Temperature %	Not Available
17	⊟ Displacement Convergence	
18	Target Delta Displacement %	5
19	Latest Delta Displacement %	Not Available

The Feedback Iterator properties also display the most recently achieved *Latest Delta Temperature %* and *Latest Delta Displacement %* values, allowing the user to manually abort when satisfied with the achieved deltas.

> **Note:**
>
> Both the Temperature and Displacement Convergence properties sections are always shown irrespective of which feedback types are enabled for the project.

Running a two-way feedback simulation from Workbench while keeping the Maxwell or HFSS Profile tab open shows that the 3D solver keeps track of two feedback related quantities:

- Maximum Absolute/Relative Delta Temperature (if temperature feedback is enabled)
- Maximum Absolute/Relative Delta Displacement (if displacement feedback is enabled)



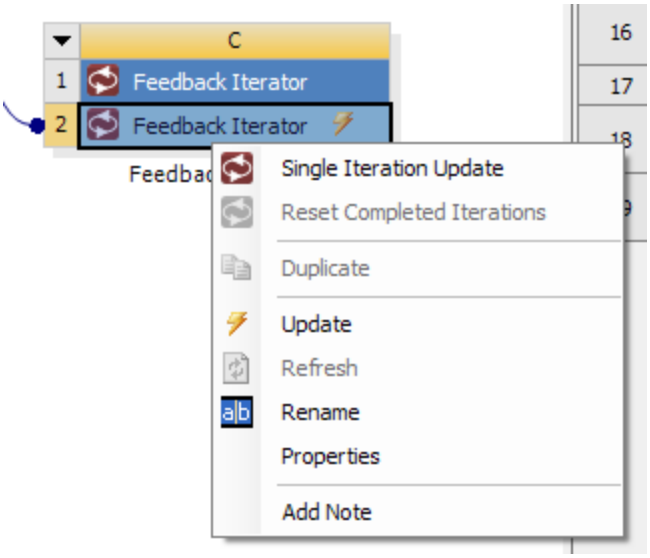
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These signify the maximum difference on the solution mesh of the quantity in question from one feedback iteration to the next (thus the value is only available from the second iteration onward). This data forms the basis for the convergence controls in the FeedbackIterator. These controls allow the user to base the termination of feedback iterations on a target max delta T and/or max delta displacement. The target max delta forms the convergence condition.

Feedback Iterator GUI Operations

The **Single Iteration Update** action allows you to run a single iteration worth of updates, as follows:

1. Updates the Electromagnetics product system, which will incorporate any previous feedback from the downstream system.
2. Updates the downstream system.
3. All coupled systems (Electromagnetics product, Electromagnetics product Downstream, and Iterator) are in the UpToDate state.
4. Iterator's “Iterations Completed” property increments by 1.



Right-clicking the Feedback Iterator component and selecting **Update** allows you to run iterations automatically. Iterations continue to run either until convergence criteria are met, or until the Max Iterations value is reached.

Resetting the Feedback Iterator

Resetting the iterator essentially means starting from iteration-0 conditions, that is:

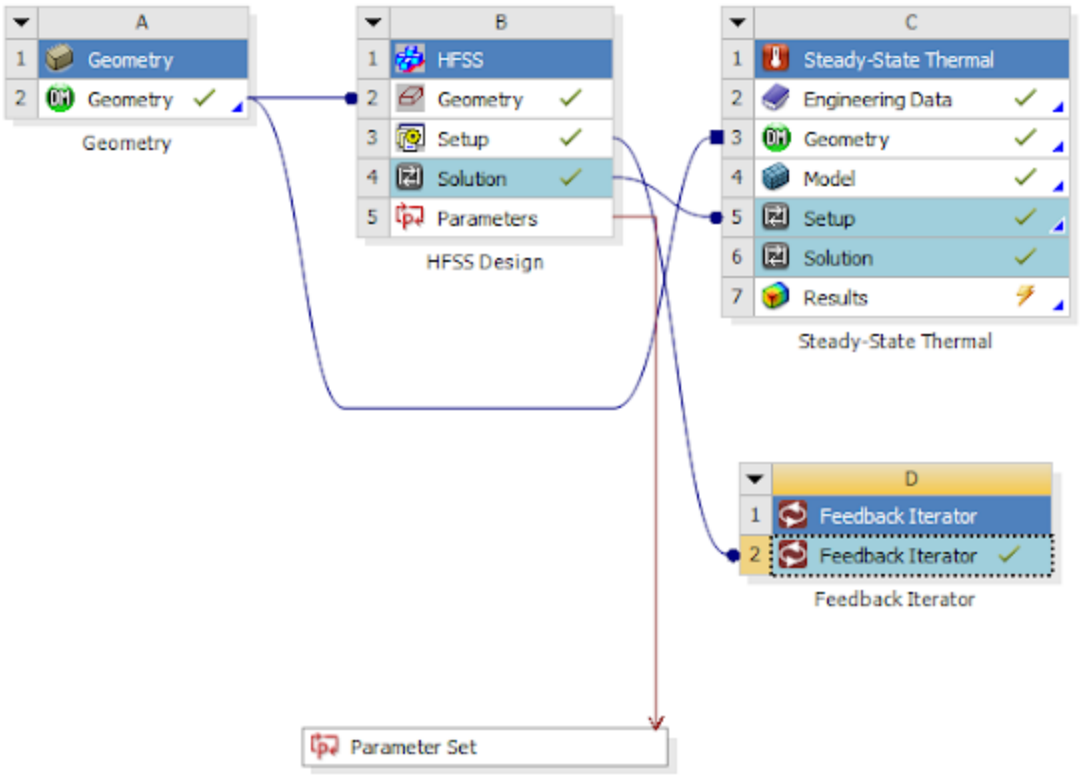
- No Displacement or Temperature feedback from another system is incorporated.
- The first iteration solution will be purely an Ansys EM analysis solution.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Resetting the Feedback Iterator cannot easily be done entirely within Workbench. However, the following two processes can be used:

- Resetting via the Ansys EM user interface
- Resetting via a WB journal script

The following example uses an HFSS/Thermal project. (This same approach applies to the other Ansys EM design types.)



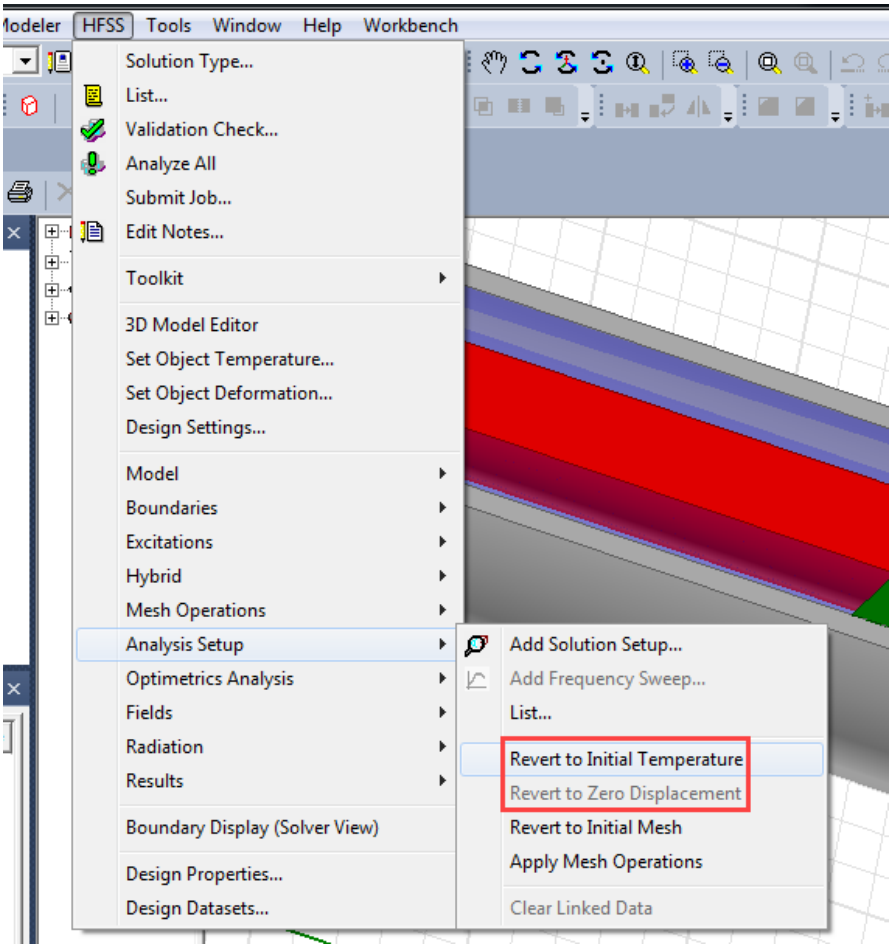
Resetting via the Ansys EM User Interface

Depending on the type(s) of feedback involved, this typically means reverting back to initial conditions for one or both of the following settings:

- **Revert to Initial Temperature**
- **Revert to Zero Displacement**

You can find these under the respective design type menu, such as HFSS shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Resetting via a WB Journal Script

> **Note:** The above Ansys EM user interface actions can be recorded into a WB journal script and played back for the easiest approach.

You also can build a script from scratch using the following as the template – simply update the system name as appropriate, include the appropriate "Revert..." commands, and either save as a script or enter into the WB Command window.

```
oDesign = Ansoft.GetDesign(GetSystem(Name="HFSS"))

oDesign.GetModule("AnalysisSetup").RevertAllToInitialTemperature()

oDesign.GetModule("AnalysisSetup").RevertAllToZeroDisplacement()
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Callback Interface

The callback interface allows you to react to each step of the iteration process as implemented by the FeedbackIterator component. This is useful for implementing special iterations or solving transient simulations with a limited scope.

There are four main API features provided:

- Callback functions, which the FeedbackIterator calls at various points in each iteration
- Utility functions, which the Callback functions can use to extra properties of the containers being processed
- Output functions, which allow script debugging and supply additional messages
- Limited state management, which allows the script to store and retrieve state across callback functions and iterations

Callback and State API

State is managed in a limited fashion. All the API methods take a final dictionary argument. This dictionary is limited to using **string keys** and **number or string values**. Within this limitation, add new keys, read old keys, clear the dictionary, etc., and it will be persisted across functions calls and even across Updates. When the **Iterations Completed** property is **rest**, the dictionary is also cleared out.

It is advisable for the callback script to initialize the dictionary at Iteration 1.

All the API methods use a subset of the following arguments:

`iterationNumber`	An integer representing the current iteration. This always starts from 1.
`ContainerList`	A Python list of containers (DataContainerReference). This is the entire list of the coupled containers managed by the Feedback Iterator. You typically loop through them and using the utility methods listed below, identities them.
`Container`	A single container that is being processed. This is a DataContainerReference.
`State`	A read/write Python dictionary that is used to maintain state across function calls and iterations.

Only return values from `BeforeIterationEx` are processed. Returns from all other functions are discarded.

1. `BeforeIteration(IterationNumber, ContainerList, State)`: This method is called before each iteration. Ideally used to initialize the state dictionary, open editors as

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

required, or initialize setups as needed for each iteration.

2. `BeforeIterationEx(IterationNumber, ContainerList, State)`: Similar to the `BeforeIteration` method except that this allows you to control the number of iterations via the return value.
    - Return: `"more"` to request one more iteration
    - Return `"last"` to indicate that this is the last iteration
    - Any other return (including none) will be treated as a return of `"last"` and terminate iterations.
3. `AfterIteration(IterationNumber, ContainerList, State)`: This method is called after each iteration. This can be used to copy result files over, check results, implement any possible convergence calculations, logging of results, *etc*.
4. `BeforeContainerRefresh(IterationNumber, Container, ContainerList, State)`: Is called before each of the coupled containers is refreshed. The `Container` argument represents the container about to be refreshed.
5. `AfterContainerRefresh(IterationNumber, Container, ContainerList, State)`: Is called after each of the coupled containers is refreshed. The `Container` argument represents the container just refreshed.
6. `BeforeContainerUpdate(IterationNumber, Container, ContainerList, State)`: Is called before each of the coupled containers is Update (after a refresh). The `Container` argument represents the container about to be updated.
7. `AfterContainerUpdate(IterationNumber, Container, ContainerList, State)`: Is called after each of the coupled containers is updated. The `Container` argument represents the container just updated.

If the callback scripts uses other files to send commands to various containers (vb, js, apdl, python, etc.), all of those files are best saved under the user_files directory. This allows you to use the `FBGetUserFilePath(str)` to get the absolute path of the file and allows the files to be packaged with any created archive.

Utility Functions

`FBSystemForContainer` (container)	system	Given a container, returns the system it belongs to
`FBSystemDisplayName` (system)	string	Given a system, returns its display name on the schematic
`FBSystemID`(system)	string	Given a system, returns its ID. This is the same as the `UniqueDIrectory` for the system
`FBGetUserFilesPath` (relativePath)		Given a relative path located under the user_files directory, say "hello.py", this returns the absolute path of the file

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output/Debugging Functions

`FBAddInfoMessage (string)`	Adds an info message to the WB message window
`FBAddWarningMessage (string)`	Adds a warning message to the WB message window
`FBAddErrorMessage (string)`	Adds an error message to the WB message window
`FBMessageBox (string)`	Pops up a dialog box with the supplied string with an **OK** button

Example Scenarios for Feedback Iterator

This section describes several scenarios for using the Feedback Iterator:

Setting up Iteration with Feedback Iterator

Breaking Iteration Control

Starting an Iterative Update

Running a Single Iteration

Interrupting an Iterative Loop

Resuming an Interrupted Iterative Loop

Modifying any of the Systems Involved in Iteration (coupled clients)

Iterating to Convergence

Setting up Iteration with Feedback Iterator

Scenario	Steps to execute	Outcome
Electromagnetics product solution provides solution data to a single downstream setup	• Drop Feedback Iterator system on the setup component of Electromagnetics product OR • Create Feedback iterator system and connect	Electromagnetics product setup component is connected to the Iterator. Electromagnetics product solution and downstream setup and solution components are coupled as clients to the iterator. The iterator component is the coupled master and the rest are coupled clients. Any coupling changes schematic visuals: • Coupled cells including coupling master are colored differently from normal cells

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scenario	Steps to execute	Outcome
	Electromagnetics product setup component to the TwoWay iterator component AND User clicks on the Iterator component and sets the desired Feedback Iterator properties.	• Coupled cell icons change to reflect the icon of the coupling master • Coupled client cells no longer display the **Update** context menu item and cannot be updated via script commands • The cell states of all coupled cells (master and clients) are synchronized to be the most pessimistic state among any of them (e.g., after an update, any modification will set all of them to the "Modified" state).
Electromagnetics product solution provides to multiple downstream setup cells	same as above	All the downstream components (setup and solution in each downstream system) are coupled as clients. The update order in this case depends on any additional data-flow connections between the systems that consume the upstream Electromagnetics product solution data.
Electromagnetics product solution cell does not have any downstream targets	same as above	No changes occur beyond what is expected when you create a new connection (no coupling). Coupling only occurs when Electromagnetics product solution cell has a downstream connection. If a downstream connection is added after the link to the iterator, coupling will be performed as described in the first row.

Breaking Iteration Control

Scenario	Steps to execute	Outcome
Iterator coupled to Electromagnetics product solution cell and downstream setup/solution cells	• Break Electromagnetics product solution cell's provides link OR • Break Electromagnetics product setup link	In addition to what is normally expected from the user actions, all coupled clients are de-coupled (their icons and colors on UI are restored)

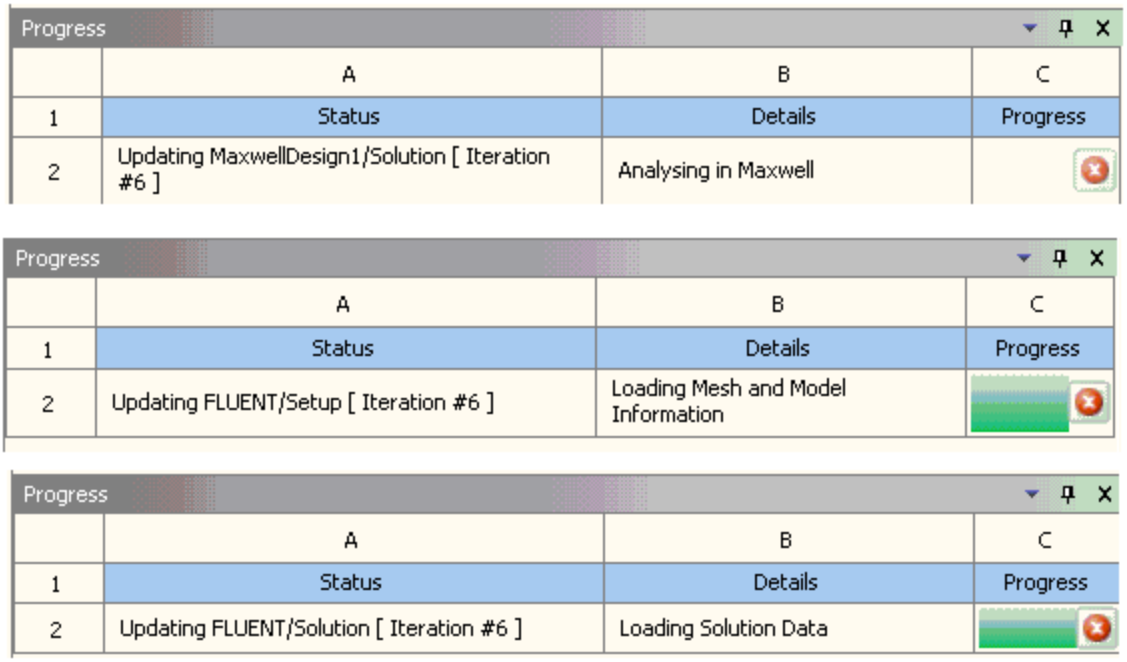
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scenario	Steps to execute	Outcome
	to iterator OR • Delete Electromagnetics product solution consumer system OR • Delete Iterator system	
Electromagnetics product setup cell is connected to iterator but Electromagnetics product solution does not provide anything. No components are coupled to the iterator	• Break Electromagnetics product setup link to iterator • Delete Iterator system	No coupling exists in this scenario so nothing visible changes beyond what is expected from the user actions

Starting an Iterative Update

Scenario	Steps to execute	Outcome
Standard update scenario	• RMB on iterator component and select Update GUI operation OR • Update Project OR • Update all design points OR • DX Update	For each iteration, the Electromagnetics product solution is updated followed by the downstream setup and the downstream solution. Each of the client component updates display their progress monitor and can be aborted (resulting in aborting the current iteration)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Running a Single Iteration

Scenario	Steps to execute	Outcome
NumCompleted < NumIterations	RMB on iterator component and select Single Iteration Update	One iteration is run. "Completed Iterations" property is incremented.
NumCompleted ≥ NumIterations	same as above	same as above

Interrupting an Iterative Loop

Currently, there is no special progress or interruption control. The progress monitor for individual components displays as they are updated and any control they choose to provide (interrupt, abort or both) is available. Choosing to abort any of the client component updates aborts the current iteration, and the Completed Iterations property remains unchanged from the previous iteration.

Resuming an Interrupted Iterative Loop

Scenario	Steps to execute	Outcome
Single step	RMB on iterator component and select Single Iteration Update	One iteration is always run and the Completed iterations property is incremented.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scenario	Steps to execute	Outcome
Run until completion	• RMB on iterator component and select Update OR • Select the Project Update menu option from the toolbar, etc.	If the user specified number of operations are already completed, nothing is done. Otherwise, the remaining iterations are run.

Modifying any of the Systems Involved in Iteration (Coupled Clients)

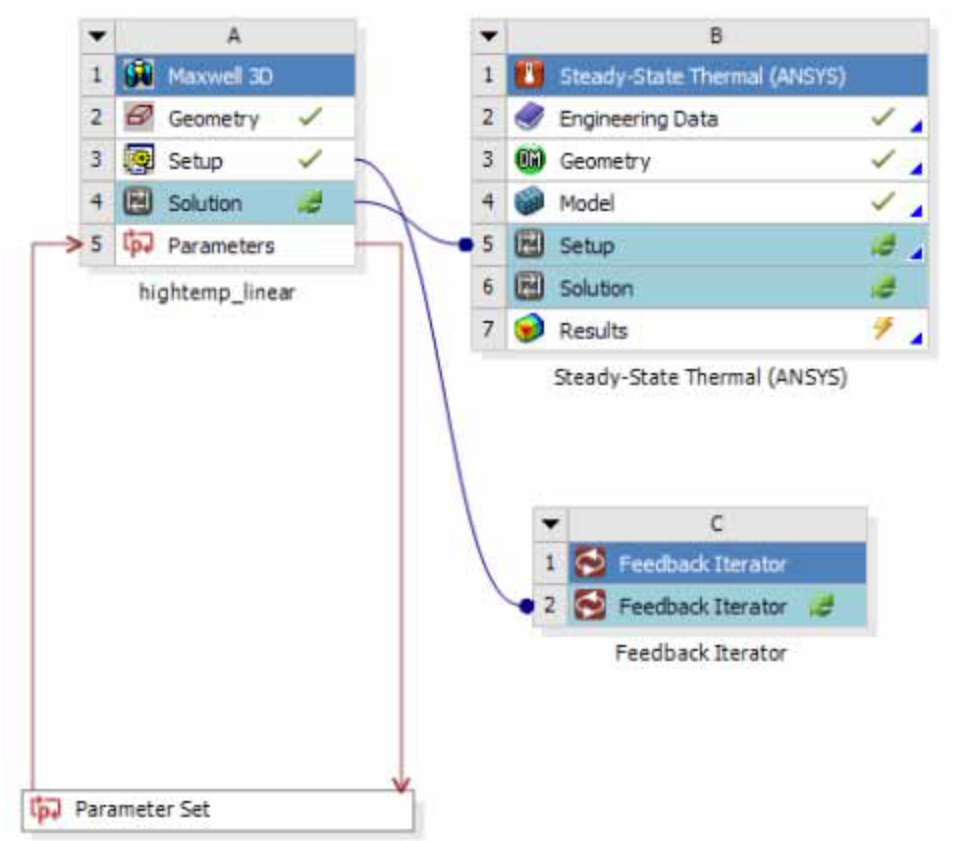
When any coupled client components are modified (either in Workbench or in an external editor), the coupled clients and master are marked as modified. The Completed Iterations property is set to 0.

Iterating to Convergence

Scenario	Steps to execute	Outcome
Standard update scenario	• Right-click on the Feedback Iterator component and select Update OR • Update Project OR • Update all design points OR • DX Update	For each iteration, the Electromagnetics product solution is updated, followed by the downstream setup and the downstream solution. Each of the client component updates display their progress monitor and can be aborted (resulting in aborting the current iteration)

The following temperature convergence example uses Feedback Iterator with a Maxwell design, coupled with a Steady-State Thermal component. Iterating to convergence operates similarly for HFSS projects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The example uses defaults of 5% for the convergence targets and a value of 100 max iterations. Recall that the max iterations value is set as a safety measure to ensure that iterations do not continue indefinitely if the solution does not converge. As we progress through the iterations, we observe the following:

End of Iteration #1

The first iteration, because it has no previous iteration, cannot return a meaningful delta value.

		started. (9:51:41 PM Jun 11, 2013)		
6	Informational	After iteration#: 1, Max temperature delta % = NaN, Max displacement delta % = NaN		6/11/2013 9:51:39 PM

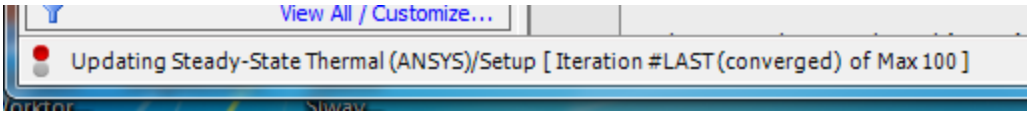
At the end of this iteration, the Latest Delta value is listed as Not Available.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

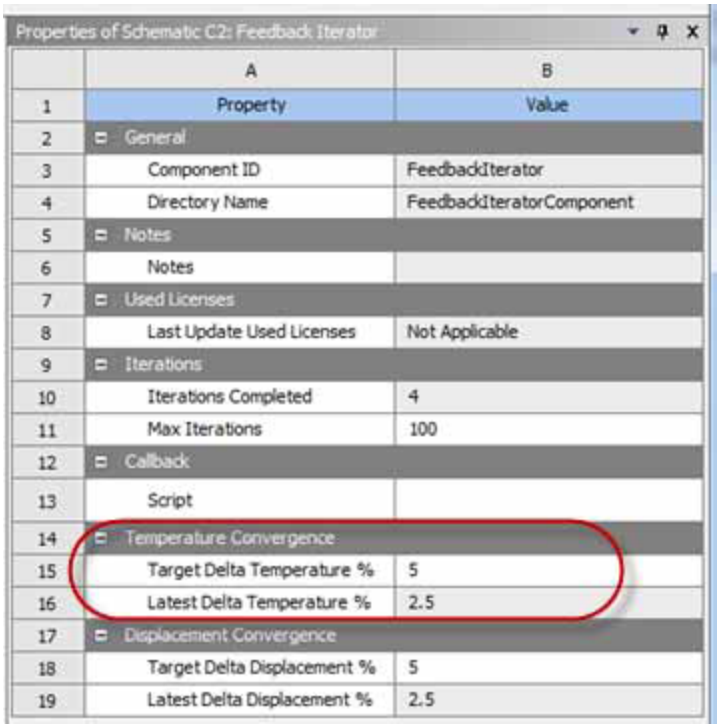
	A	B
1	Property	Value
2	General	
3	Component ID	FeedbackIterator
4	Directory Name	FeedbackIteratorComponent
5	Notes	
6	Notes	
7	Used Licenses	
8	Last Update Used Licenses	
9	Iterations	
10	Iterations Completed	1
11	Max Iterations	100
12	Callback	
13	Script	
14	Temperature Convergence	
15	Target Delta Temperature %	5
16	Latest Delta Temperature %	Not Available
17	Displacement Convergence	
18	Target Delta Displacement %	5
19	Latest Delta Displacement %	Not Available

Convergence Achieved

In this example, convergence has been achieved by the third iteration.

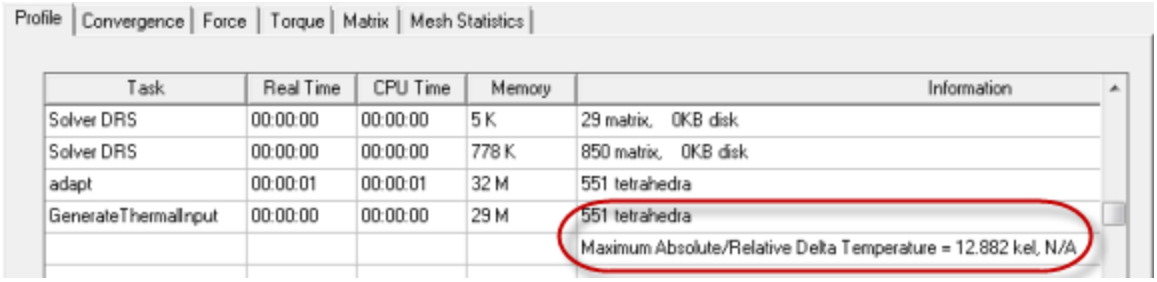


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When convergence criteria are met, the simulation stops. If you attempt to update after convergence has been achieved, the simulation will not be launched since the current solutions satisfy the convergence criteria.

You can monitor the iteration progress by opening the Maxwell editor and selecting **Results** > **Solution data**. Choose the **Profile** tab and keep the window open. As you solve each iteration, you can observe the reported Delta-T (with thermal feedback) and/or delta-displacement (with displacement feedback) and abort the iterations (or stop single iteration updates) when the values reach acceptable convergence levels.



Surface Force Density in HFSS

HFSS can calculate surface force density for Work Bench coupling and for post processing within HFSS. The whole approach resembles Maxwell except that HFSS must calculate the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

surface force density at the post processing stage while solver provide the surface force density in Maxwell.

The surface force density formulation is provided in a technical note here. Surface forces may exist on surfaces where one side is conductor but the other is not, or finite conductivity and layered impedance boundary. The computation of surface force density will be performed at field geometry instance level.

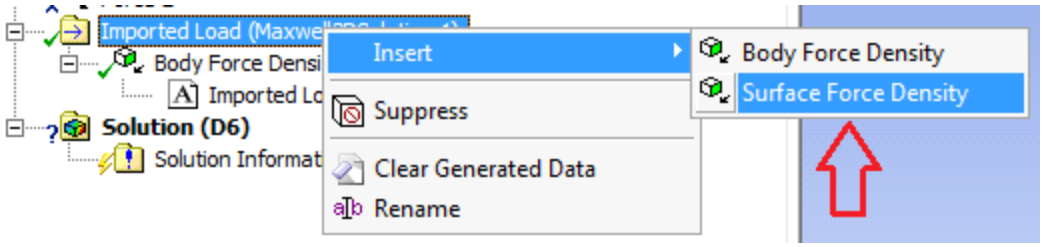
This feature is mainly for the purpose of mapping surface force density in HFSS to Work Bench Mechanical. We will also provide it for post processing within HFSS. This feature should work regardless of HFSS solution types as long as surface forces exist. Surface forces may exist when one side is conductor but the other is not, or finite conductivity and layered impedance boundary.

If an old coupling project happens to request surface force and is solved again, then the surface forces (likely nonzero) will be passed to mechanical side.

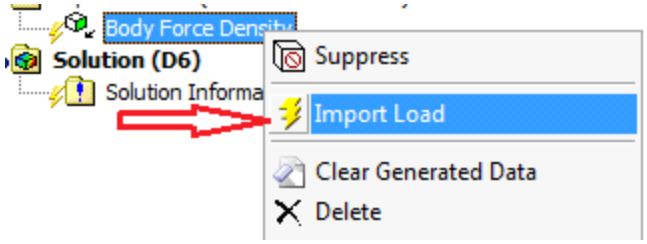
Mapping surface force density from HFSS to Work Bench Mechanical

The behavior is the same as all current couplings.

1. From the Mechanical side set up a surface force coupling to an HFSS design similar to Maxwell.



2. Import the load.



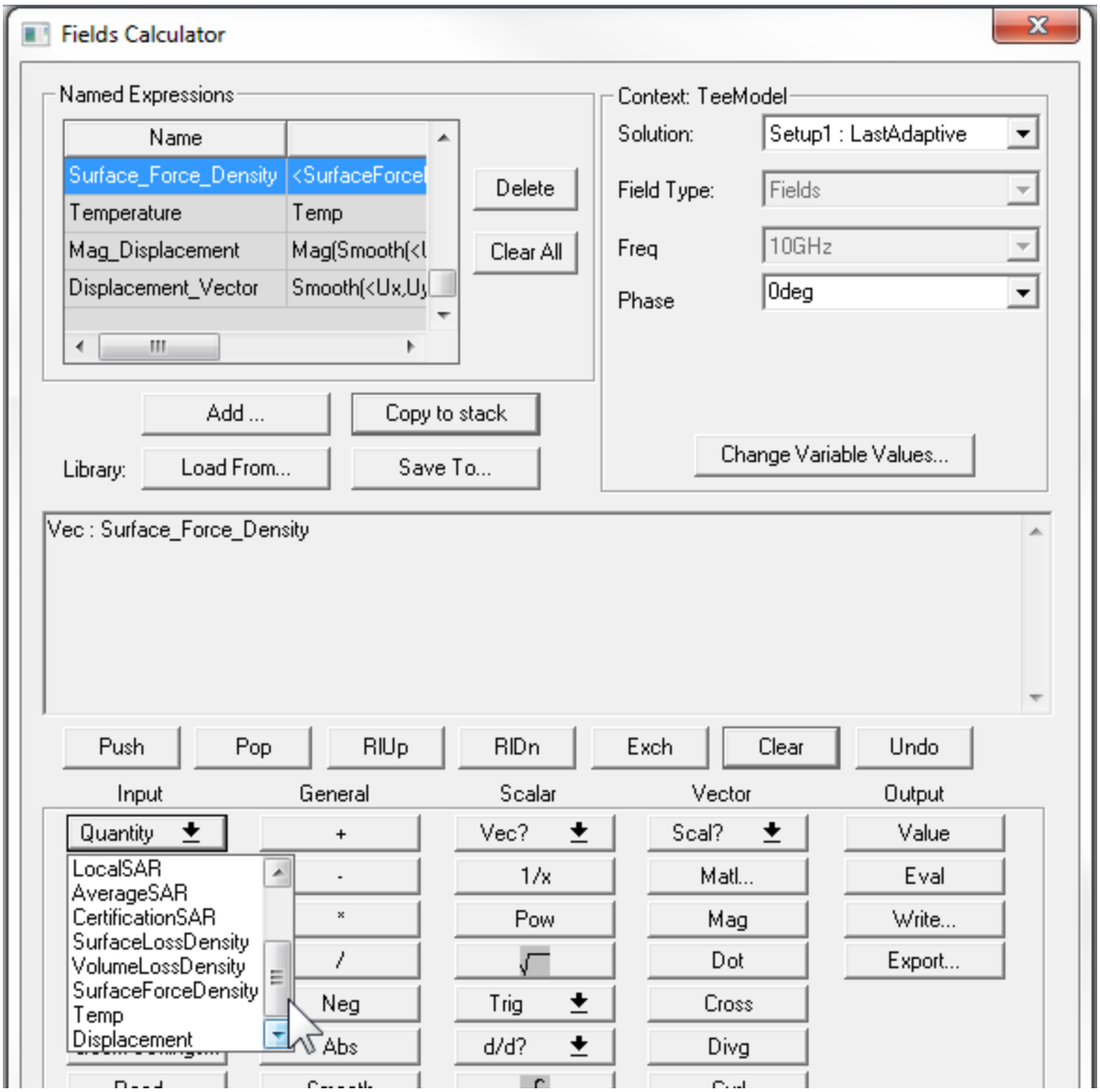
Behind the scenes Mechanical will pass necessary information to HFSS, points on surface to request surface forces and auxiliary files, HFSS will then calculate the surface force densities and pass them back to Mechanical.

3. Post process surface density within HFSS over any surfaces where surface forces exist.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Force Density will appear in the context menu in the Other category under Fields menu.

In the fields calculator Surface Force Density appears in the Named Expression list and the Quantity drop-down menu. So the user can use it the same way as other vector quantities.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

27 - Meshing In HFSS

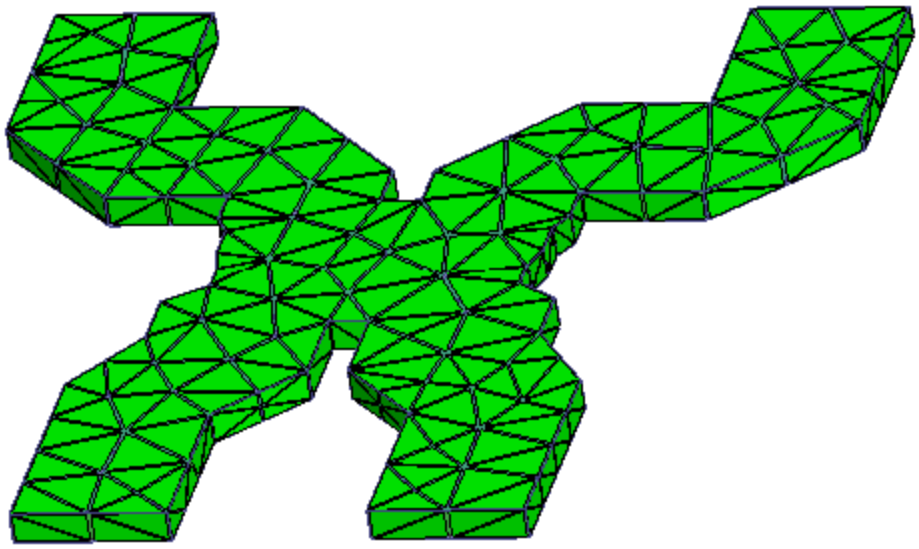
Automatic adaptive meshing is an essential and integral part of the simulation process. An appropriate mesh that supports electromagnetic FEM calculations is essential for obtaining accurate results. Meshes are produced automatically, ensuring efficient simulations in generating highly accurate results. During the simulation process, the initial mesh is created and then the adaptive refinement process begins in which Maxwell's Equations are solved and the mesh is intelligently refined. This adaptive refinement process continues until the solution is converged and accurate.

Introduction to Meshing in HFSS

The geometric model is automatically divided into a number of tetrahedra. The collection of tetrahedra forms the finite element mesh.

The partial differential equations that govern the behavior of the electromagnetic fields form an accurate mathematical model. The partial differential equations are Maxwell's Equations, valid everywhere in the three-dimensional space. The finite element formulation uses sophisticated mathematical techniques to satisfy the partial differential equations in the geometrical model, which represents some device or system of interest.

Collection of tetrahedra forming a finite element mesh that represents a waveguide combiner



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

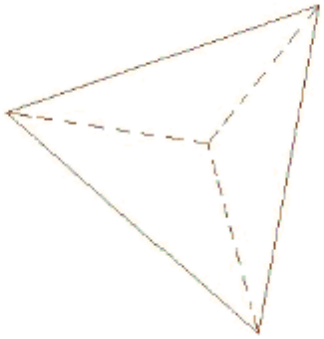
Mesh Generation

Mesh generation is the process by which space-discretization is implemented. During mesh generation, a geometric domain is divided into a number of elements.

The type of the mesh is defined by the shape of the element, which can be tetrahedral, hexahedral, polyhedral, pyramidal, prismatic etc.

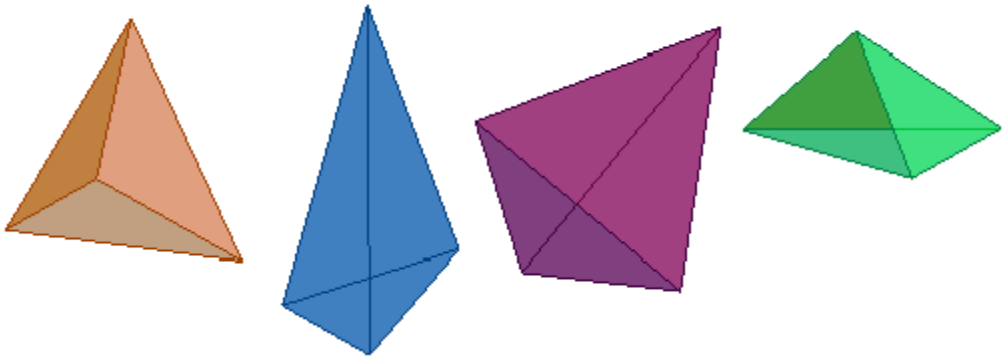
In HFSS, a mesh is a discretization of a continuous domain (geometry) into small three-dimensional tetrahedral and two-dimensional triangular elements. The following figure shows a tetrahedron, which is a solid composed of four triangular faces.

A tetrahedron



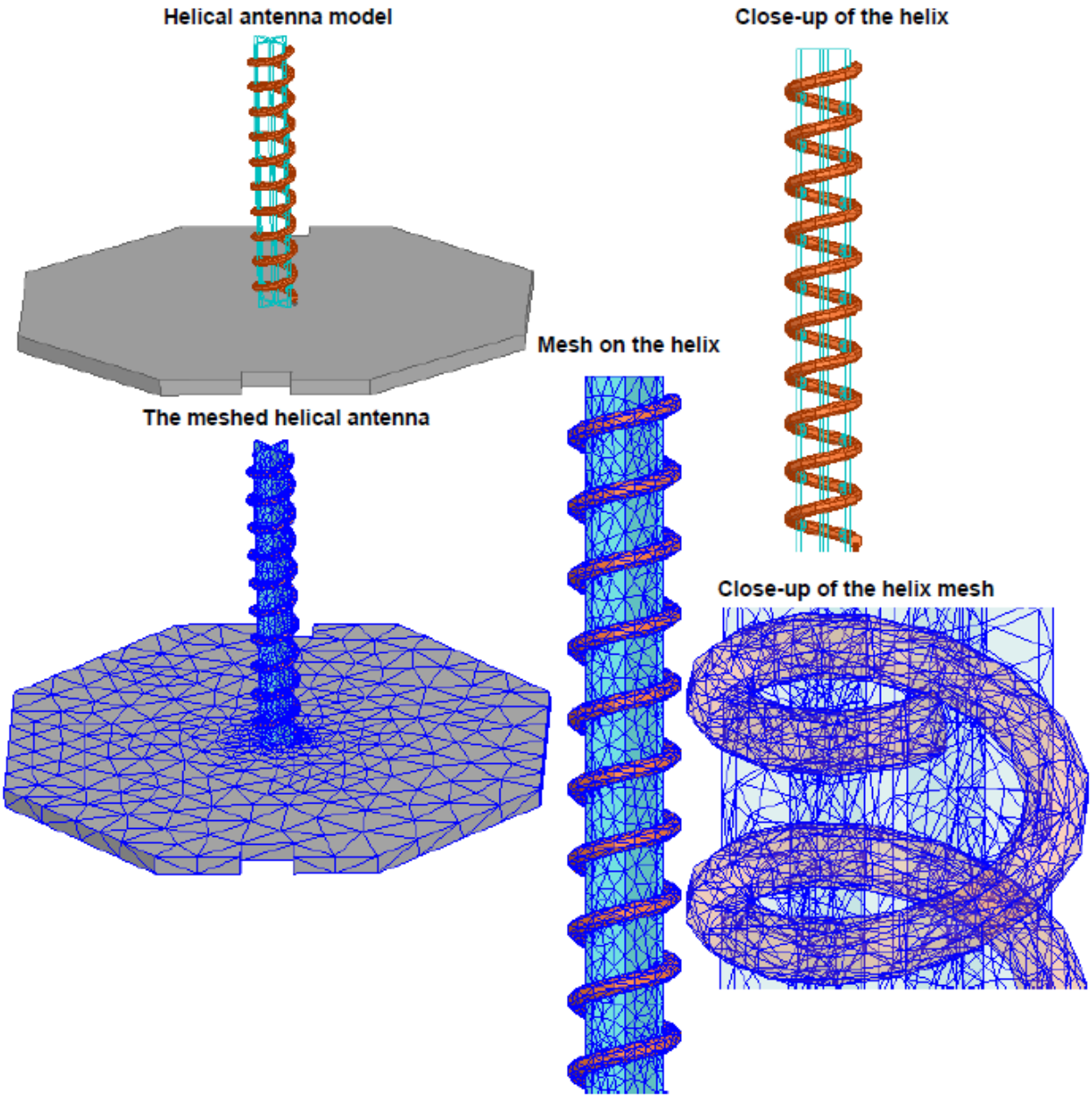
Tetrahedra are the simplex for R3 space, which means that any 3D shape can be decomposed into tetrahedra. Since the process of mesh generation is automatic in HFSS, this mathematical guarantee is necessary for successful mesh generation of arbitrary domains. Therefore tetrahedra (and triangles for two-dimensional surfaces) are used. Tetrahedra can be stretched and pulled (see the figure below), which allows them to fit any arbitrary 3D geometry; so any 3D shape can be decomposed into a number of tetrahedra.

Different kinds of tetrahedra



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following figures illustrate HFSS designs of a helical antenna before and after the mesh is created. Notice that the helical antenna model is well-represented by the mesh.



Adaptive Mesh Refinement

Adaptive mesh refinement automatically generates accurate solutions. This meshing technique helps you focus on setting up your design efficiently rather than spending time in determining and creating the best mesh. To set up the design, you need only to create the geometry and specify material properties, boundary conditions, excitations, and the solution frequency.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

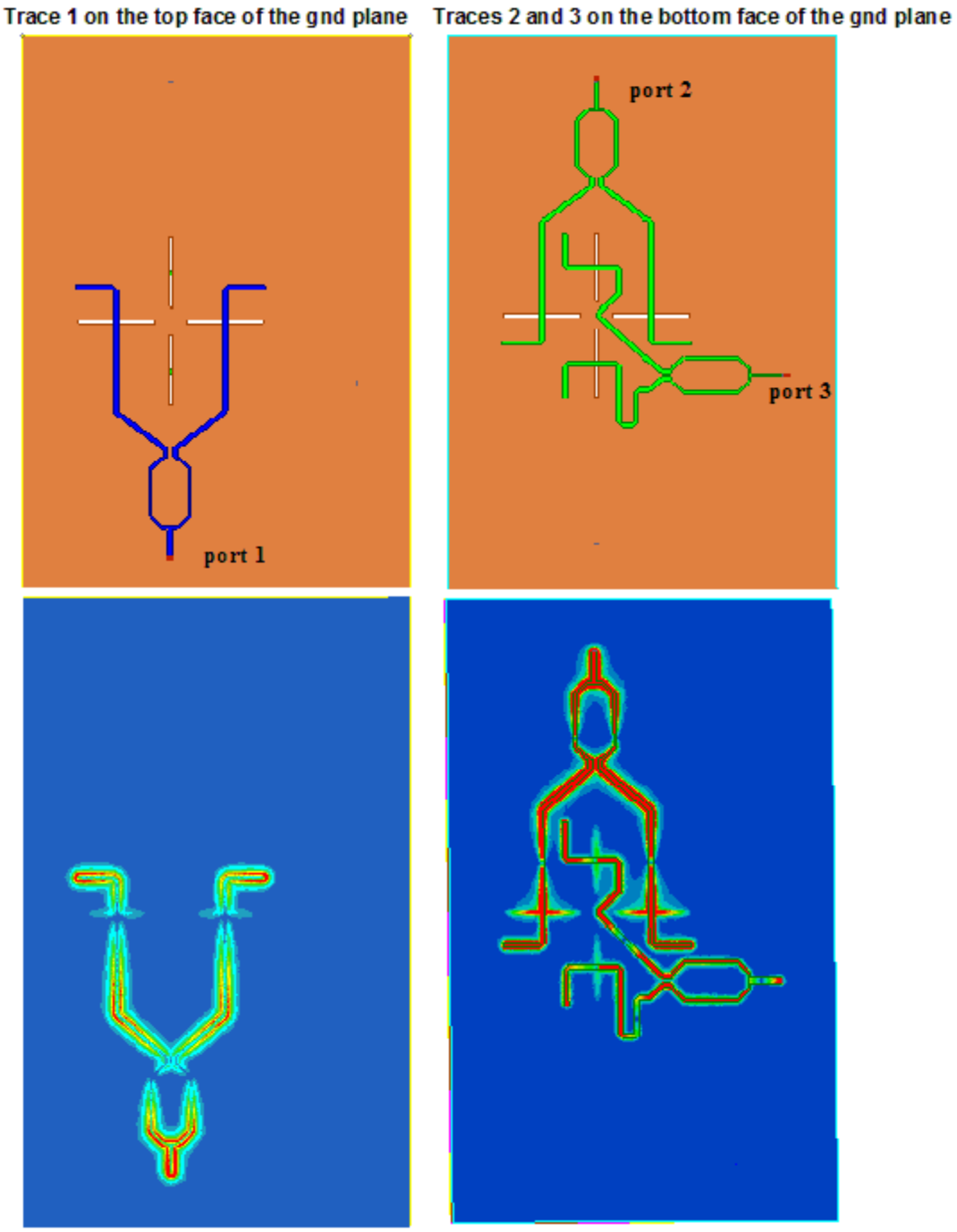
This automatic adaptive mesh refinement technique differentiates HFSS from most electromagnetic simulators, which typically require extensive user controls to ensure that the generated mesh is suitable for simulation. Without the correct mesh, results from such simulators may be erroneous.

In HFSS, however, the automatically generated mesh accurately represents the electromagnetic characteristics of the design. HFSS determines the size and density of tetrahedra for optimal efficiency and accuracy. An initial mesh is generated and solved to produce error estimates of the solution. These error estimates are used to refine the mesh in high-error regions. Using this refined mesh, HFSS generates another solution and recomputes the error. This adaptive refinement process continues until HFSS converges to an accurate solution. Convergence is determined by monitoring a parameter from one adaptive pass to the next. The most common convergence criterion is to ensure that the difference in the S-parameter value between two consecutive solves is less than the specified magnitude. If you need to increase the accuracy, you can easily tighten the convergence criteria, and HFSS further refines the mesh.

Adaptive refinement ensures that the mesh elements are sufficiently fine in those areas where strong electromagnetic fields exist and/or the field gradients are high. The mesh is coarser in the remaining areas. This process makes the mesh size appropriate and suitable for efficient simulations leading to highly accurate solutions.

To illustrate, an aperture coupled microstrip filter design with three traces is shown below. One of the traces is located on one side of the ground plane, while the other two traces are located on the opposite side. The figures also show the resulting electric fields that develop for the design after the simulation is completed. The plots show the traces of the microstrip on both sides of the ground plane illuminated by the electromagnetic fields.

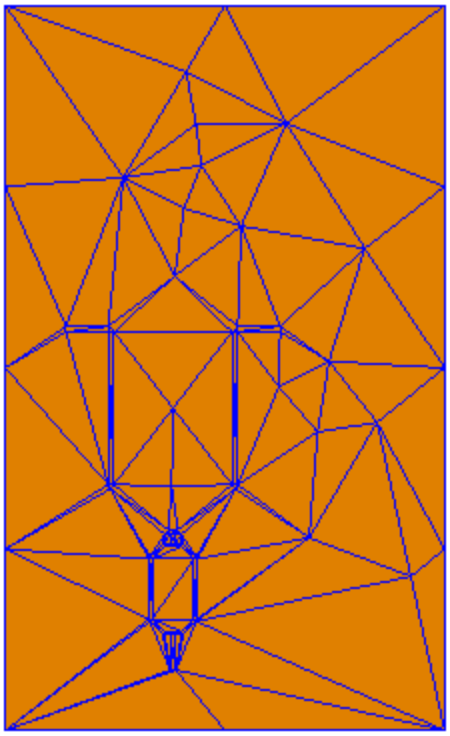
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



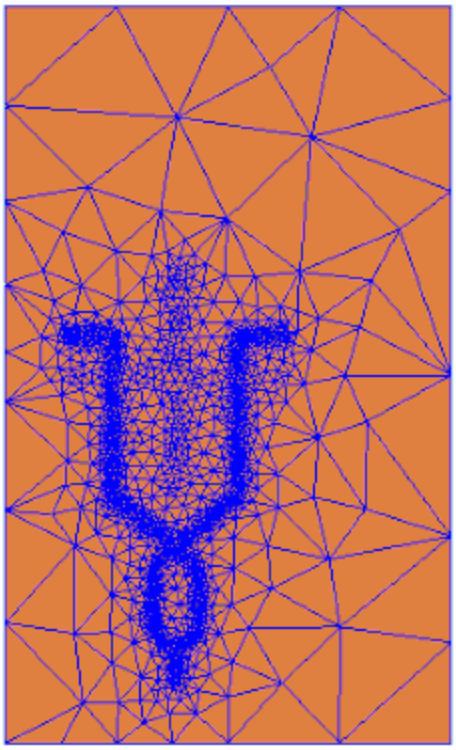
See both the initial and final meshes for these two layers and how they are automatically adapted by the adaptive refinement algorithm.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

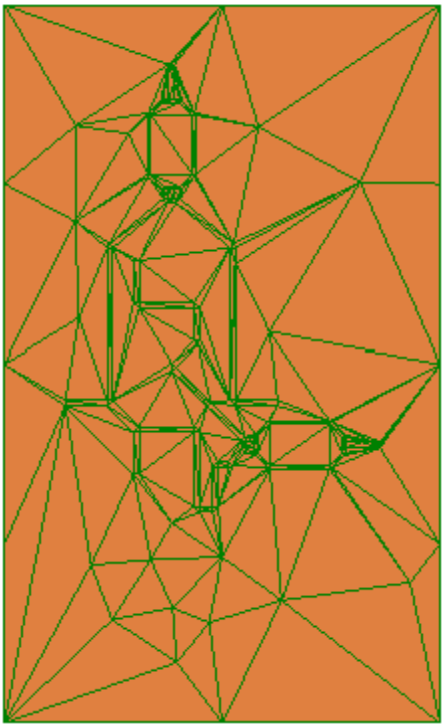
Initial mesh on the top face of the gnd plane



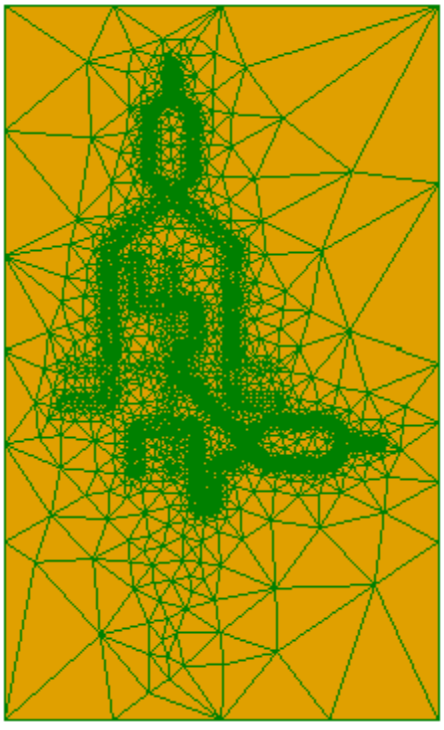
Final mesh on the top face of the gnd plane



Initial mesh on the bottom face of the gnd plane



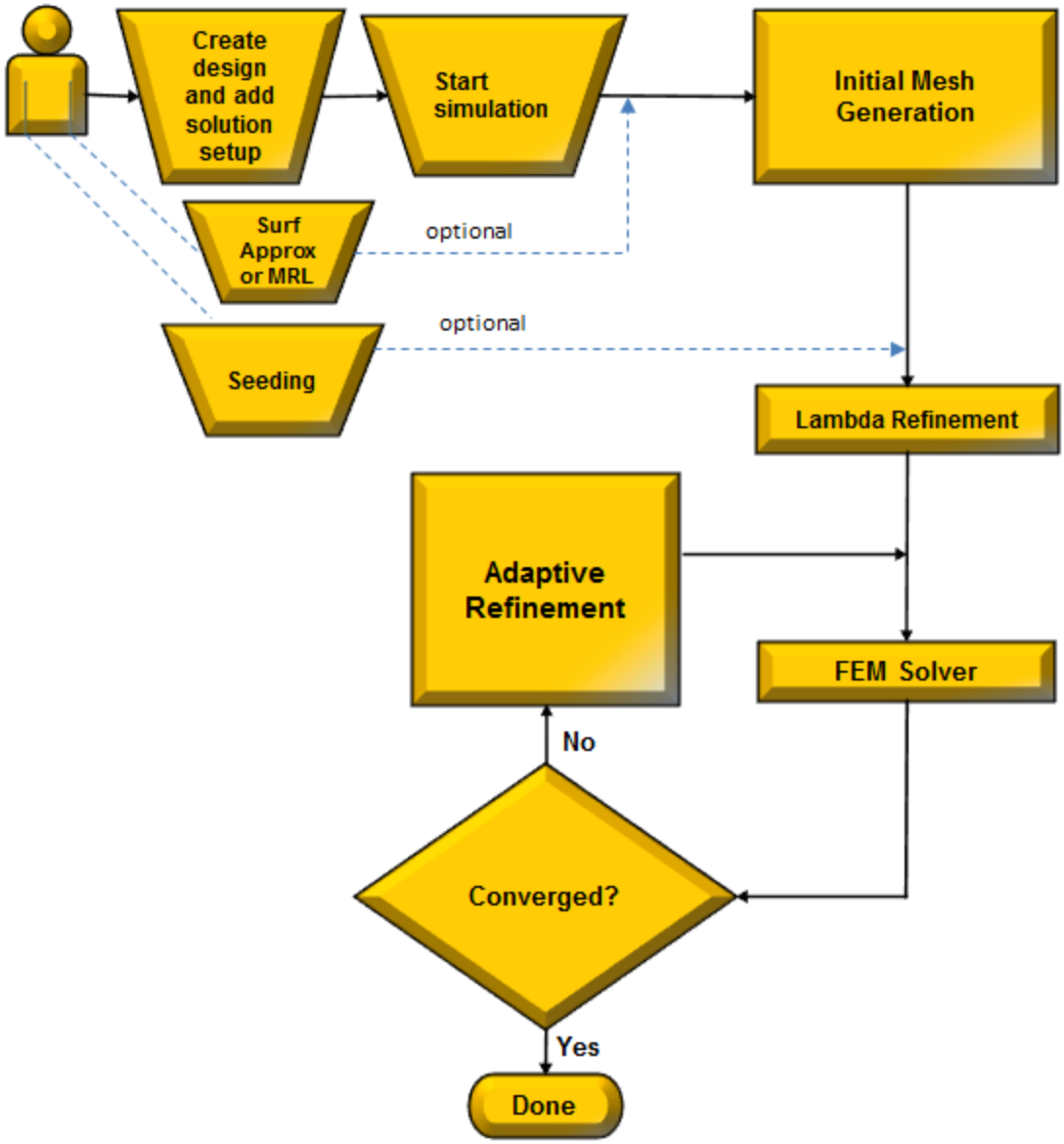
Final mesh on the bottom face of the gnd plane



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adaptive Mesh Refinement Process Flow

HFSS uses the adaptive mesh refinement technique and generates accurate solutions with minimal use of computational resources. The following diagram shows the adaptive mesh refinement process flow that can deliver accurate results with least user involvement.



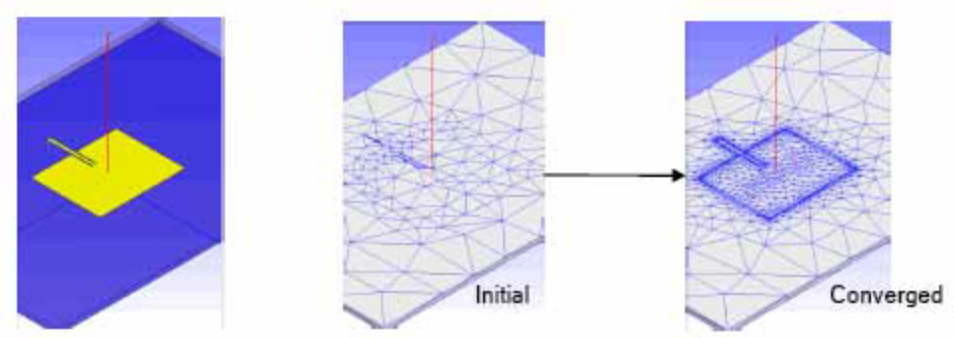
For mesh operations, defining surface approximation, model resolution length, and seeding are optional steps in the solution process, as shown in the flow diagram. This is because the automatic adaptive refinement process is adequate to produce accurate results. Only on rare occasions, adaptive refinement is used with mesh operations. For more information, see Optional Mesh Refinement Settings.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The process depicted by the flow diagram can be summarized as follows:

1. HFSS generates an initial mesh that can be controlled by surface approximation settings.
2. Seed refinement (if assigned) and Lambda refinement occur on the initial mesh.
3. Using the resulting mesh, HFSS computes the electromagnetic fields in the structure.
4. From the current solution, HFSS produces error estimates and refines the mesh in regions of high error and uses the refined mesh to generate another solution.
5. HFSS performs the iterative process of solve, error analyses, and adaptive refinement until the convergence criteria are satisfied or the maximum number of adaptive passes is completed.

The following figure illustrates that the initial mesh is refined until convergence criteria are satisfied and a suitable final mesh is generated.



Stages of Meshing

During mesh generation, HFSS divides the geometric domain into tetrahedra. The collection of tetrahedra forms the finite element mesh. The different stages of meshing are described as follows:

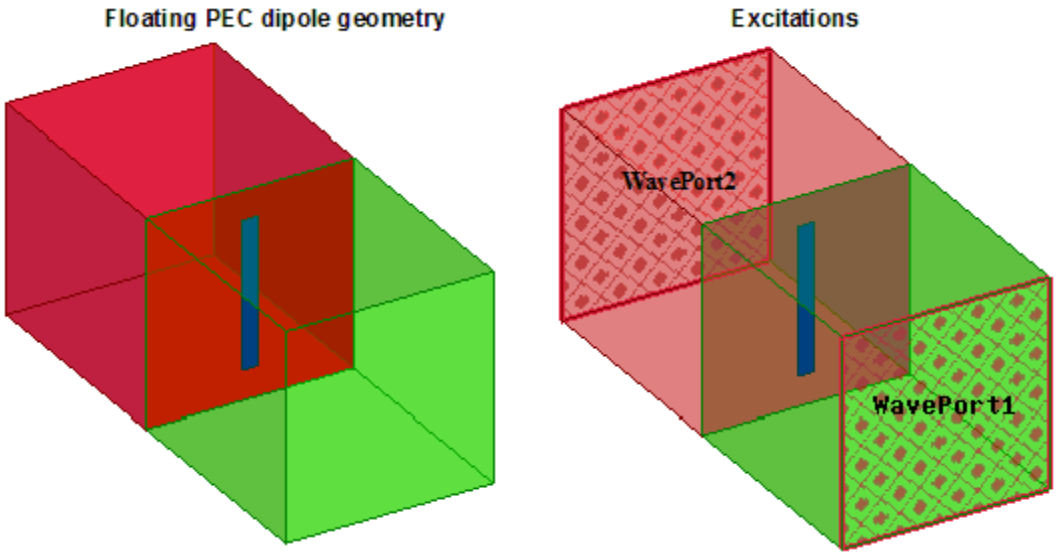
1. HFSS creates an initial mesh that can be controlled by the surface approximation settings.
2. The initial mesh is refined based on the material-dependent wavelength lambda refinement and any mesh operations that are defined.
3. HFSS iteratively refines the 2D mesh at any ports in the model.
4. The electromagnetic fields are computed inside the structure.
5. The current solution is used to estimate the tetrahedra where the solution has large error and those tetrahedra are refined. This results in a refined mesh.
6. The refined mesh is used to generate another solution.

HFSS continues with the iterative process (steps 4and 6) until the convergence criteria are satisfied and then generates accurate results.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example: Floating PEC Dipole

The following figures show a floating PEC dipole enclosed in a parallel plate waveguide excited at both ends by wave ports.



Although floating PEC dipole appears to be a simple design to solve, it is a canonical project for testing adaptive meshing algorithms. In this problem, the challenge is to efficiently refine the mesh for capturing the electromagnetic field singularities around the dipole.

The following figures illustrate how adaptive refinement addresses this challenge. First HFSS generates an initial mesh as shown in Figure 1. This initial mesh serves as a good starting point for adaptive refinement. Figure 2 shows the mesh plot after Adaptive Pass 1. This automatic adaptive refinement process continues until the convergence criterion is satisfied. Figure 3 shows an intermediate mesh during a later stage of the simulation. Figure 4 shows the final mesh that is produced after the solution converges.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

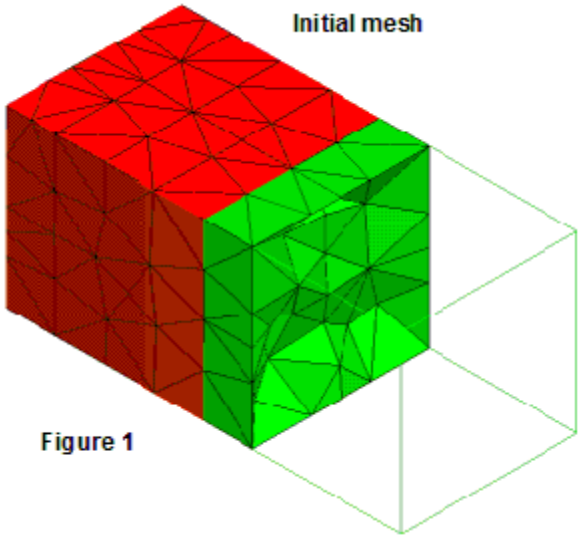


Figure 1

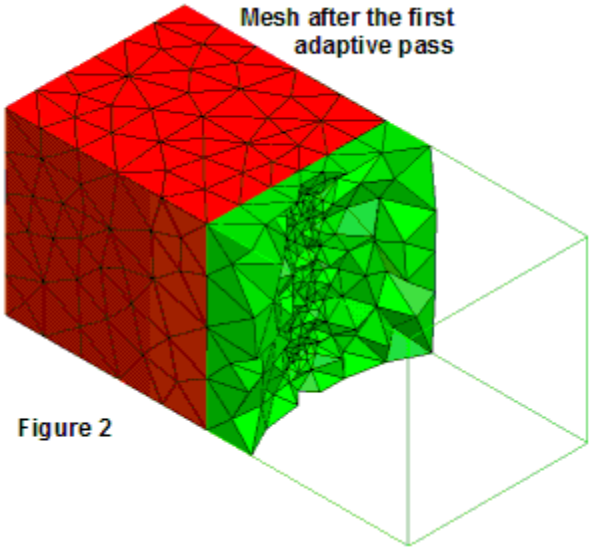


Figure 2

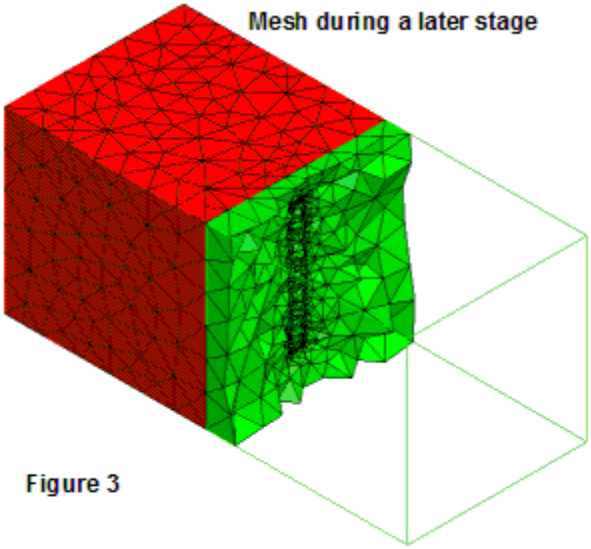


Figure 3

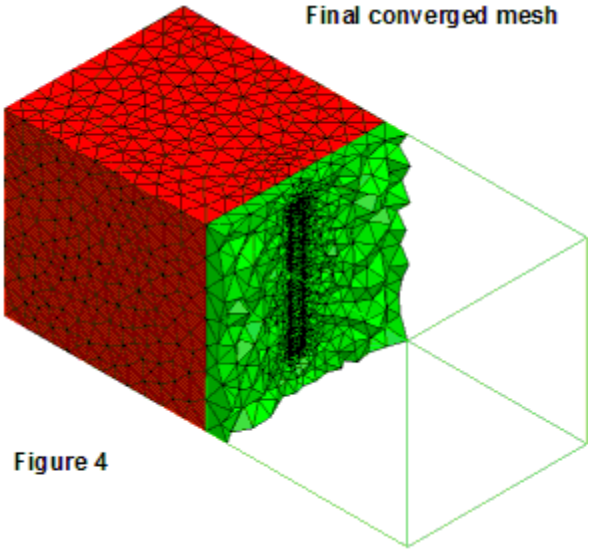
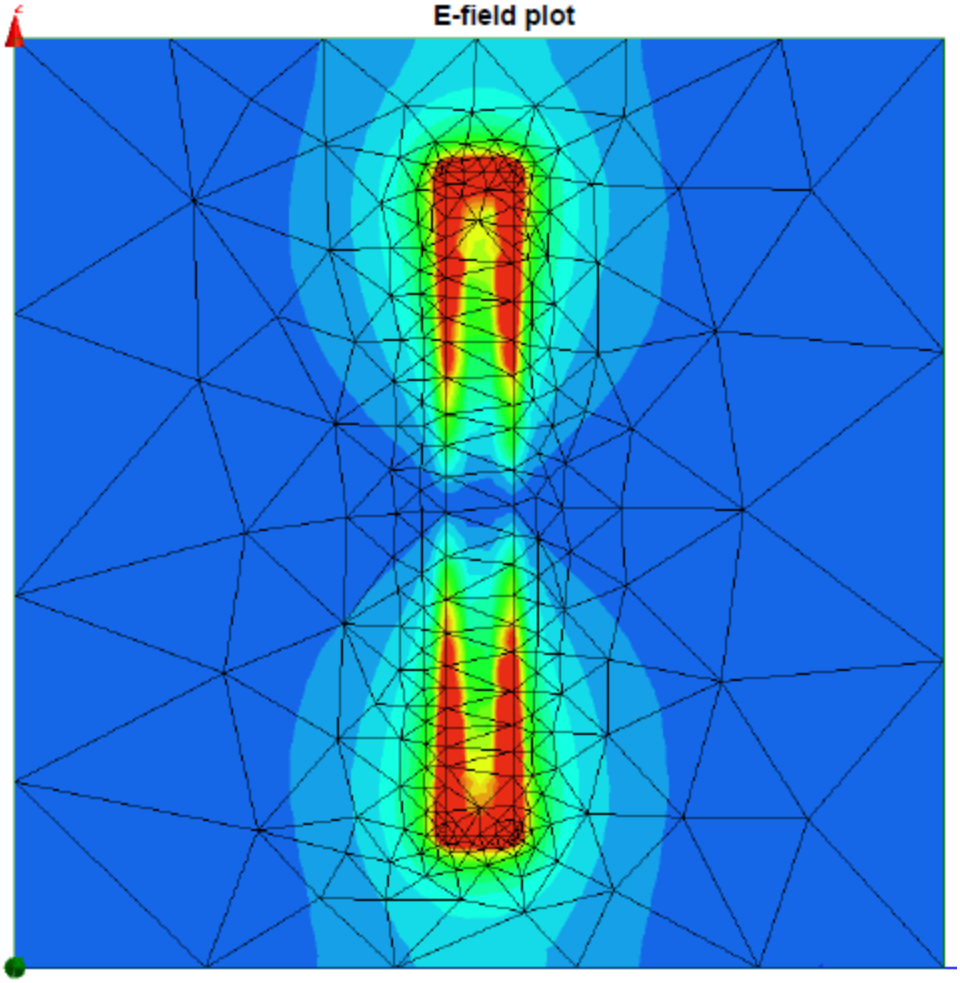


Figure 4

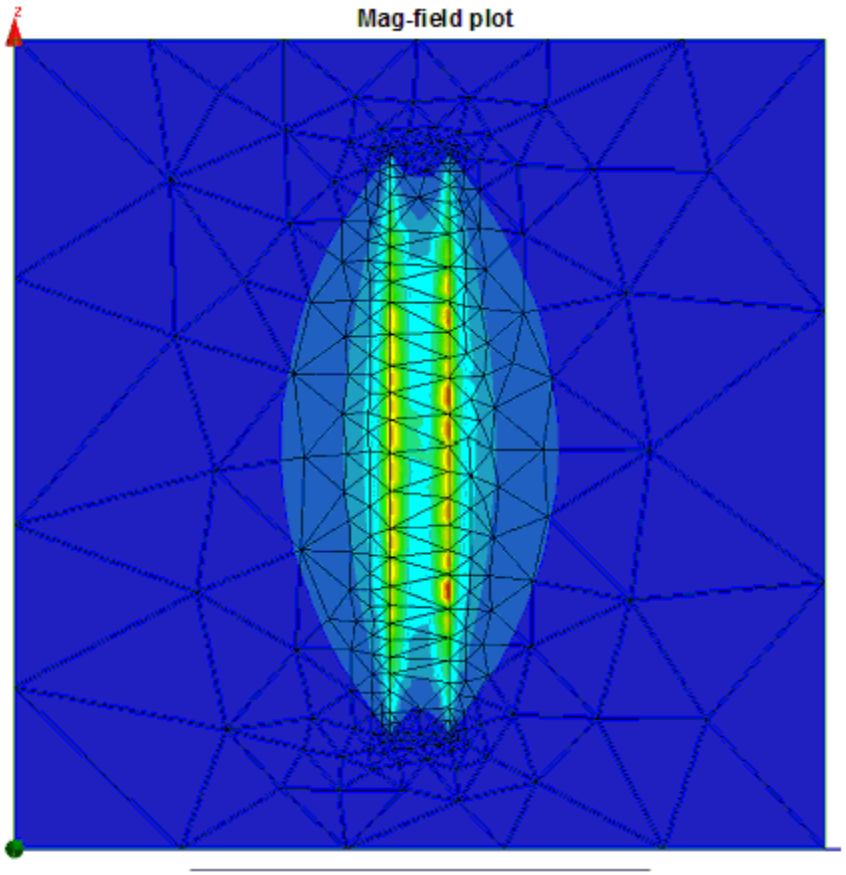
The following figure shows the electric fields with the mesh overlay.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The mesh is fine around the center of the dipole, although the electric fields are weak, because strong magnetic fields exist in that region. This is shown in the following figure of the magnetic field plot on the same mesh.

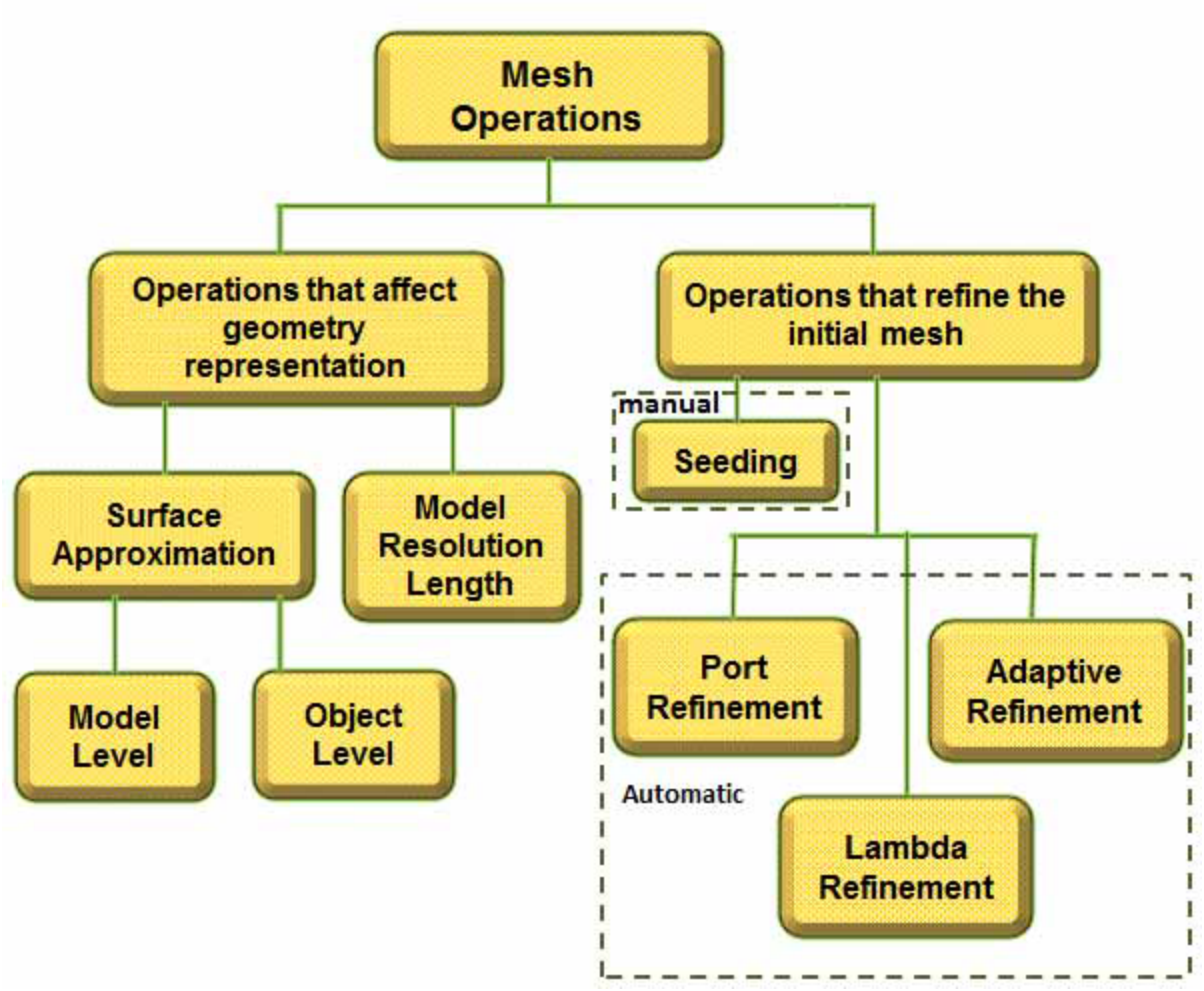
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Assign Mesh Operations

Mesh Operation settings are optional. The following figure provides a synopsis of the different mesh operations available in HFSS. The figure shows a classification only; it does not show the solution process flow. For more information about the solution process, see Stages of Meshing and Adaptive Refinement Process Flow.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The next sections describe these mesh operations.

Operations Affecting Geometry Representation

Surface Approximation and **Model Resolution Length** operations affect the initial mesh. If you modify any of the options on the **Surface Approximation** or the **Model Resolution Length** panels, the initial mesh is invalidated, and HFSS regenerates the initial mesh. The plausible reasons for controlling the initial mesh are:

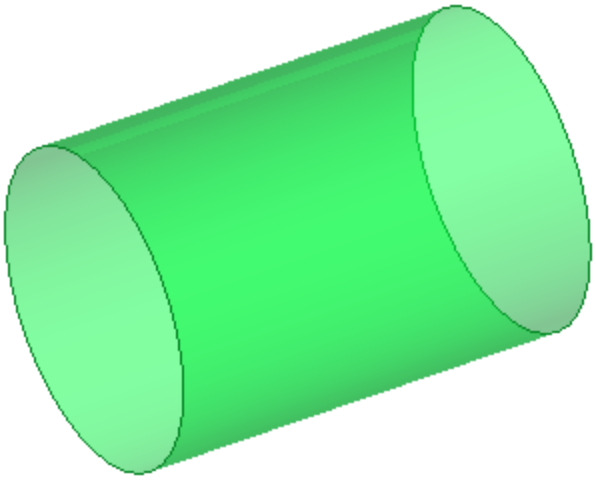
- if improved representation of curved surfaces is required.
- if initial mesh failures require workarounds.

> **Note:**
>
> See **Troubleshooting Mesh Failures** for more information.

Example: Improved Representation of Curved Surfaces

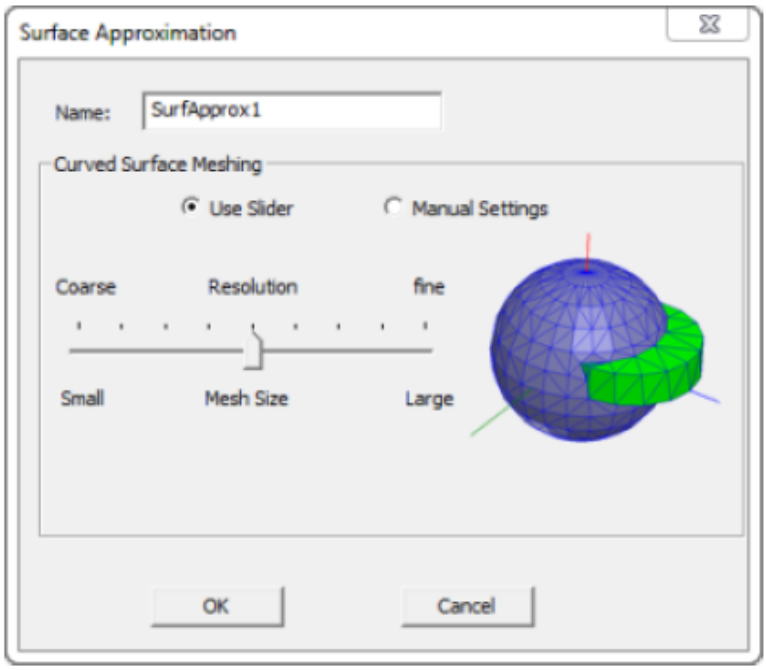
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Consider a cylinder shown in the following figure. The mesh representing the geometry of the cylinder can be modified by changing the surface approximation settings. HFSS uses these settings only during initial mesh generation.

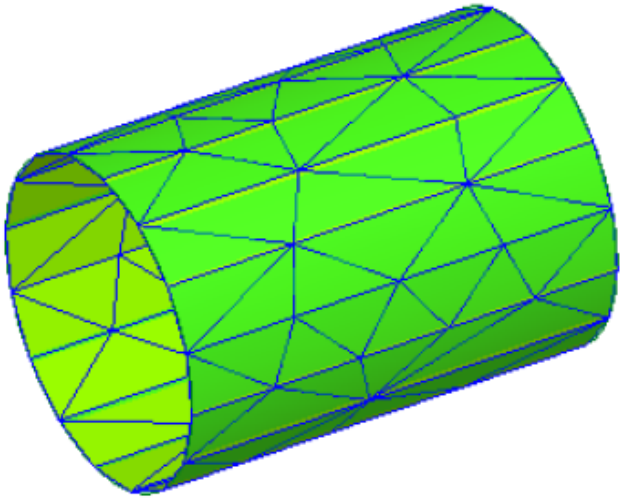


If you do not define any mesh operation, HFSS automatically uses default settings to create the initial mesh. If you define or modify a **Surface Approximation** setting, the current initial mesh is discarded and a new initial mesh is generated. The following figures illustrate the initial mesh for the default, and for a user-defined setting where maximum surface normal deviation angle is changed. Due to the reduced value of the maximum normal deviation angle, the initial mesh has more triangulations to represent the curved surface of the cylinder.

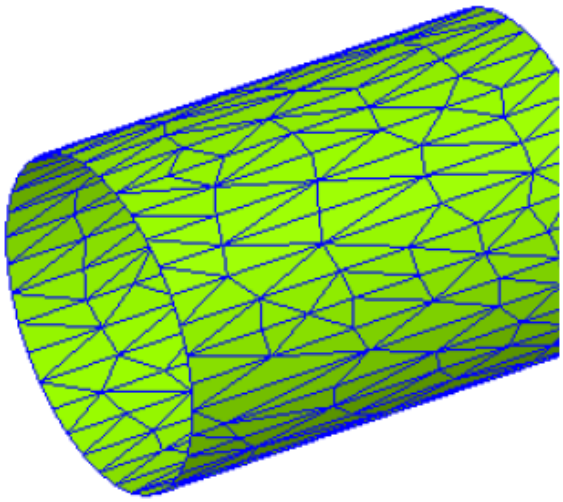
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Default SA settings



User-defined SA settings

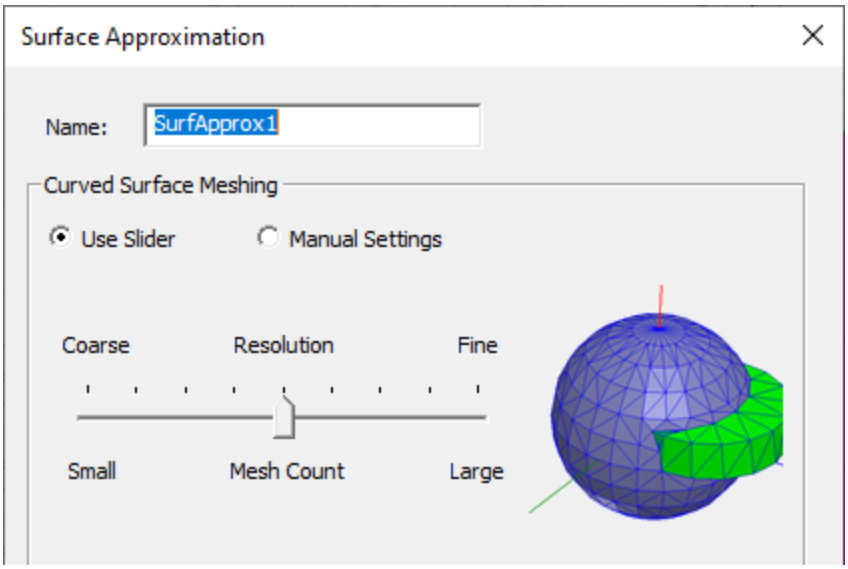


Surface Approximation

Surface Approximation is a mesh operation that controls the initial mesh generation. If you modify the default **surface approximation** settings, the initial mesh is invalidated. They are used for models with curved surfaces. In HFSS, you can access these settings in two places. When you select a curved surface, right-click and choose **Assign Mesh Operation** > **Surface Approximation** from the short-cut menu, you can define the settings for the current selection of

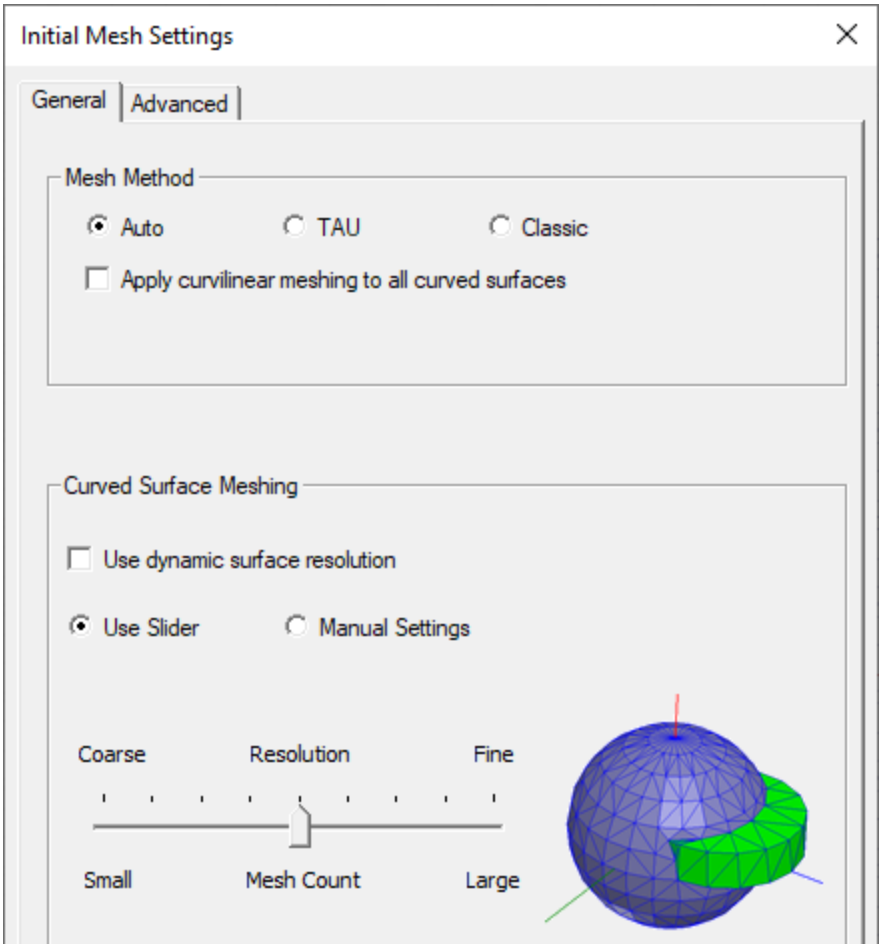
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the object. The slider permits you to set surface approximation in convenient way. The actual slider bar values applied can be interpreted differently by different solvers as is suitable in each case (for instance, it is more refined for IE Regions or SBR+ Regions than for FEM).



However, when you right-click **Mesh** in the project tree and select **Initial Mesh Settings**, you can define the settings at a global level. Additional options in this panel include **Mesh Method** and **Apply curvilinear elements** check box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

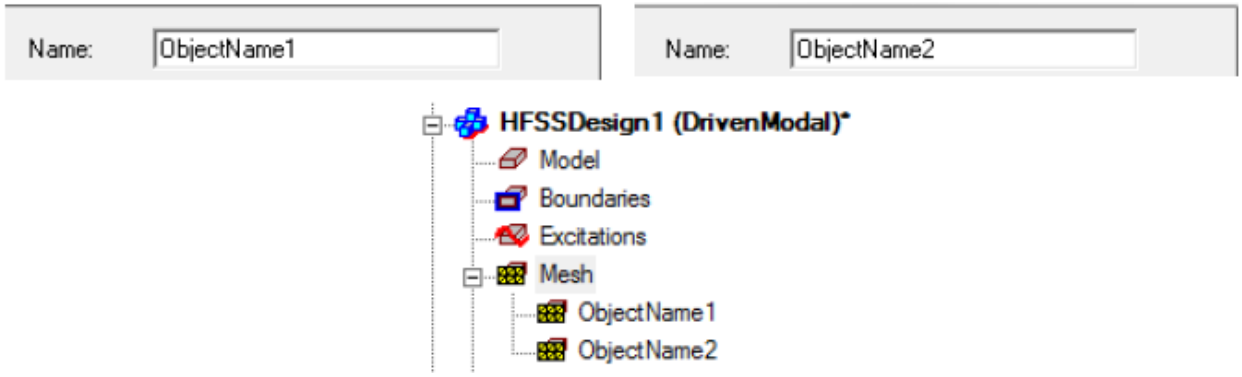


Note: Apply Curvilinear elements is only supported for the entire model whereas **Surface Representation Priority for Tau Mesh** is only supported for the selected object.

Name

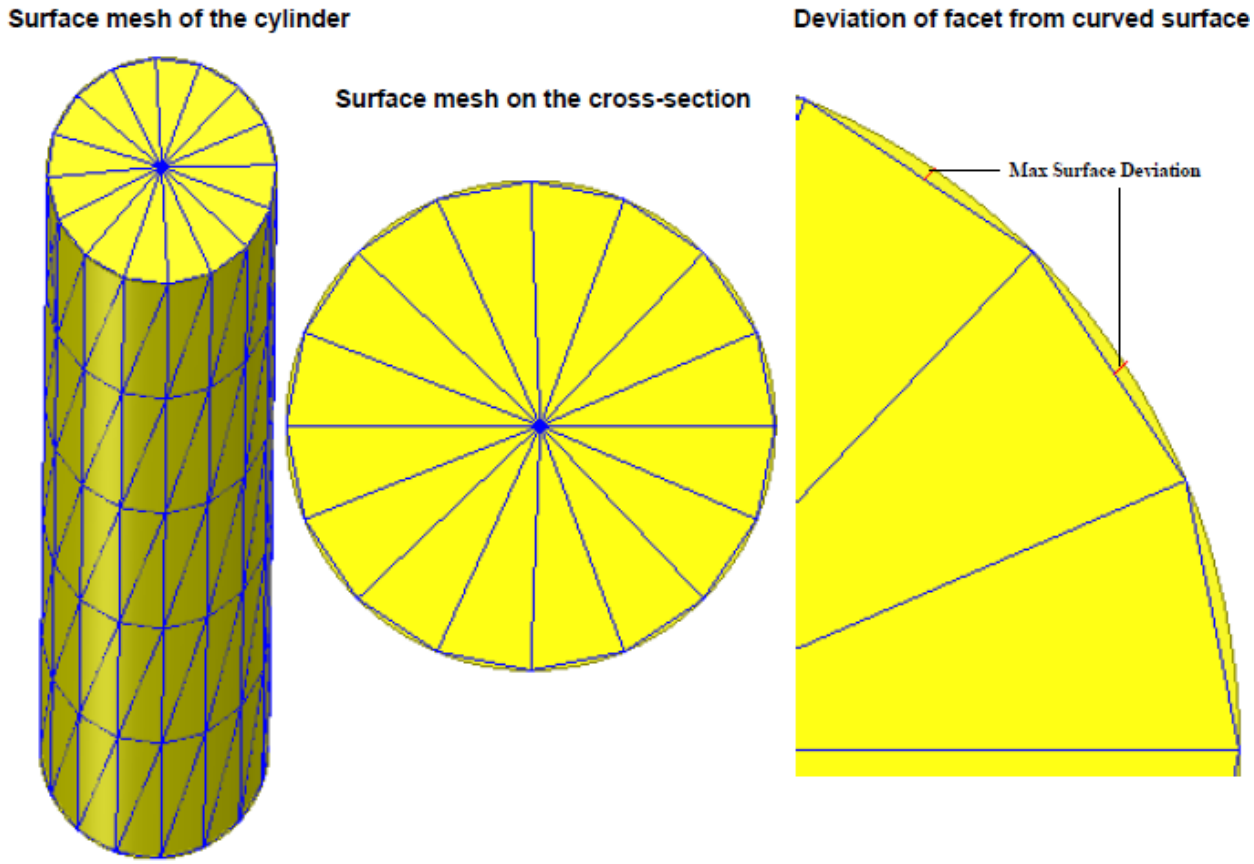
In this field, use the default or enter the name for the group of settings. Assign meaningful names so that you can easily distinguish between the different surface approximation assignments for the corresponding objects in the model. Accordingly, **Mesh** on the **Project Manager** window is populated as shown in the following figure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



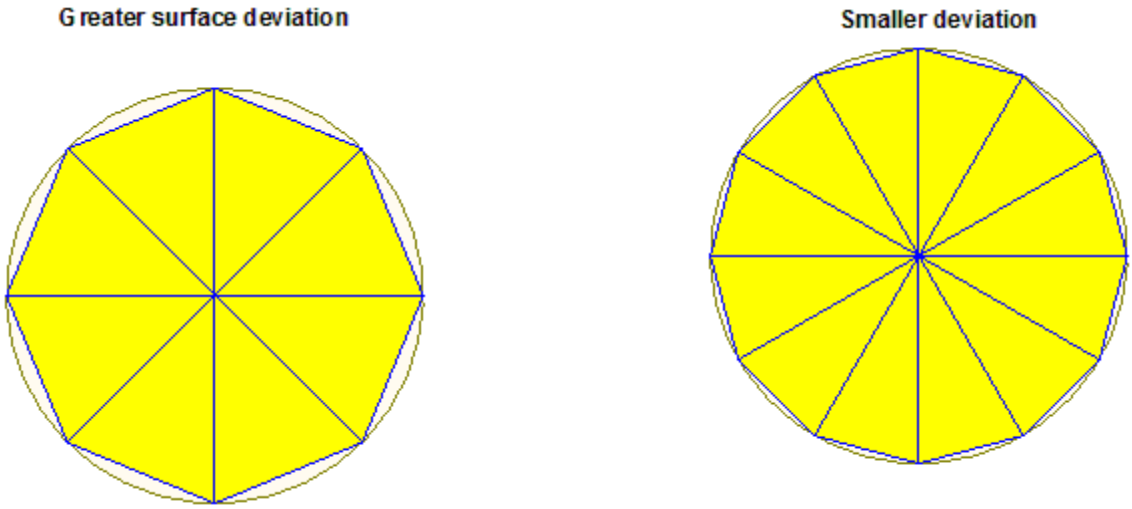
Maximum Surface Deviation (Length)

All curved surfaces are faceted during the meshing process. A facet is a collection of triangles. There is a small distance between the faceted surface and the curved surface, which is called surface deviation. **Maximum Surface Deviation** is the maximum value of this distance. The following figures show a meshed cylinder in HFSS, the cylinder cross-section, and the **maximum surface deviation** that occurs near the center of each faceted triangle. When you use an appropriate value for **Maximum Surface Deviation** (length), the curved surfaces are well-represented to realize the original surface.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

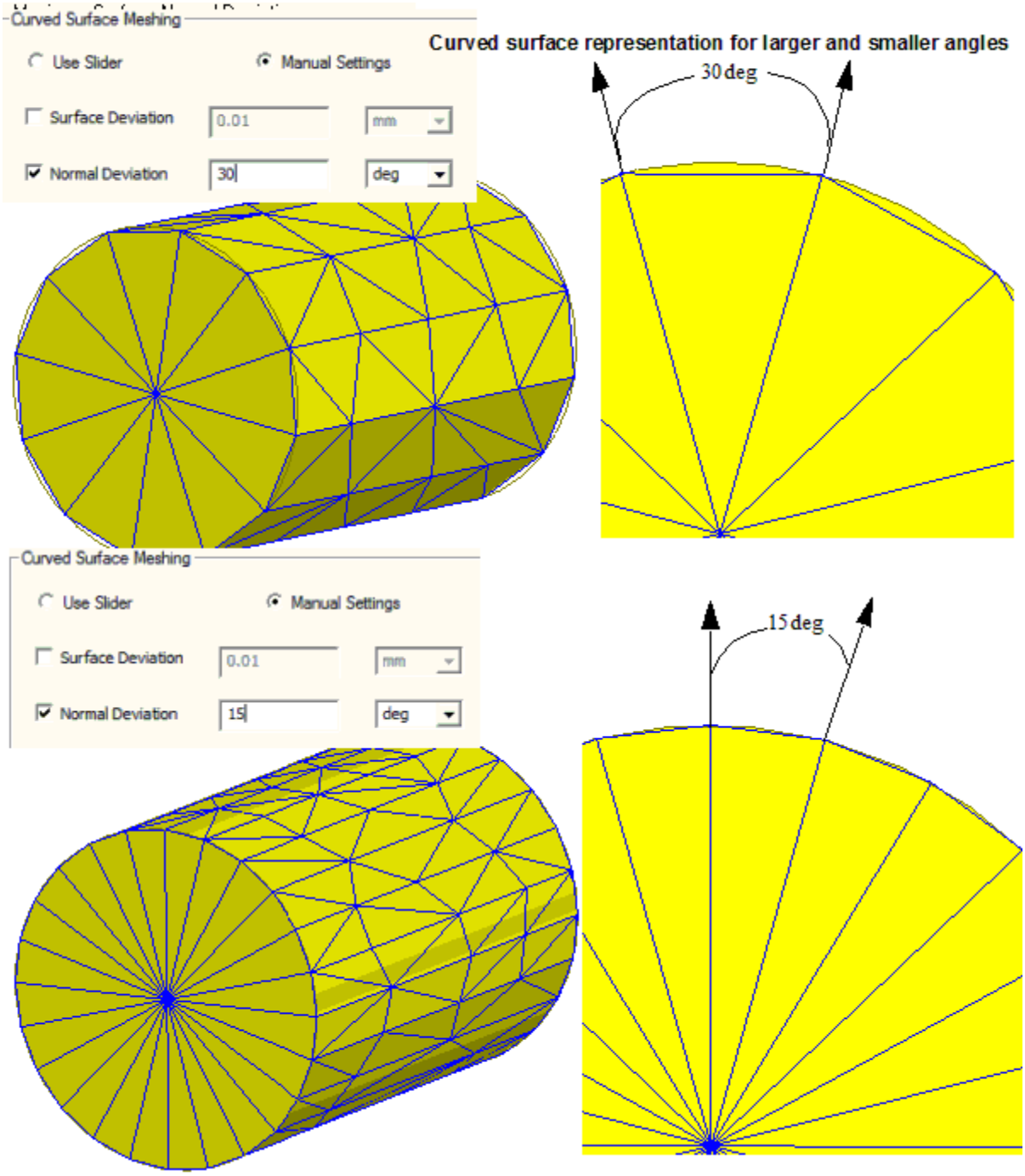
The following figures illustrate how the representation of a curved surface changes when different values of maximum surface deviation (length) are used. Smaller values of maximum surface deviation can result in greater number of faceted triangulations, for more accurate approximation of the curved surface. However, increasing the number of triangles causes more CPU time and memory consumption during simulation. For this reason, use the default settings, since they are adequate for most circumstances. In the following figure, the cylinder radius is 0.5 mm, height is 1.5 mm, and the normal deviation angle is kept constant at 45 degrees to illustrate the effects of different maximum surface deviation lengths.



Maximum Normal Deviation (Angle)

Maximum Normal Deviation is the maximum angular deviation between any two normals on a curved surface approximated by a faceted triangulation. The following figures illustrate the effect of using different maximum normal deviation angles on a cylindrical surface. In the following figure, the cylinder radius is 0.5 mm and its height is 1.5 mm.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



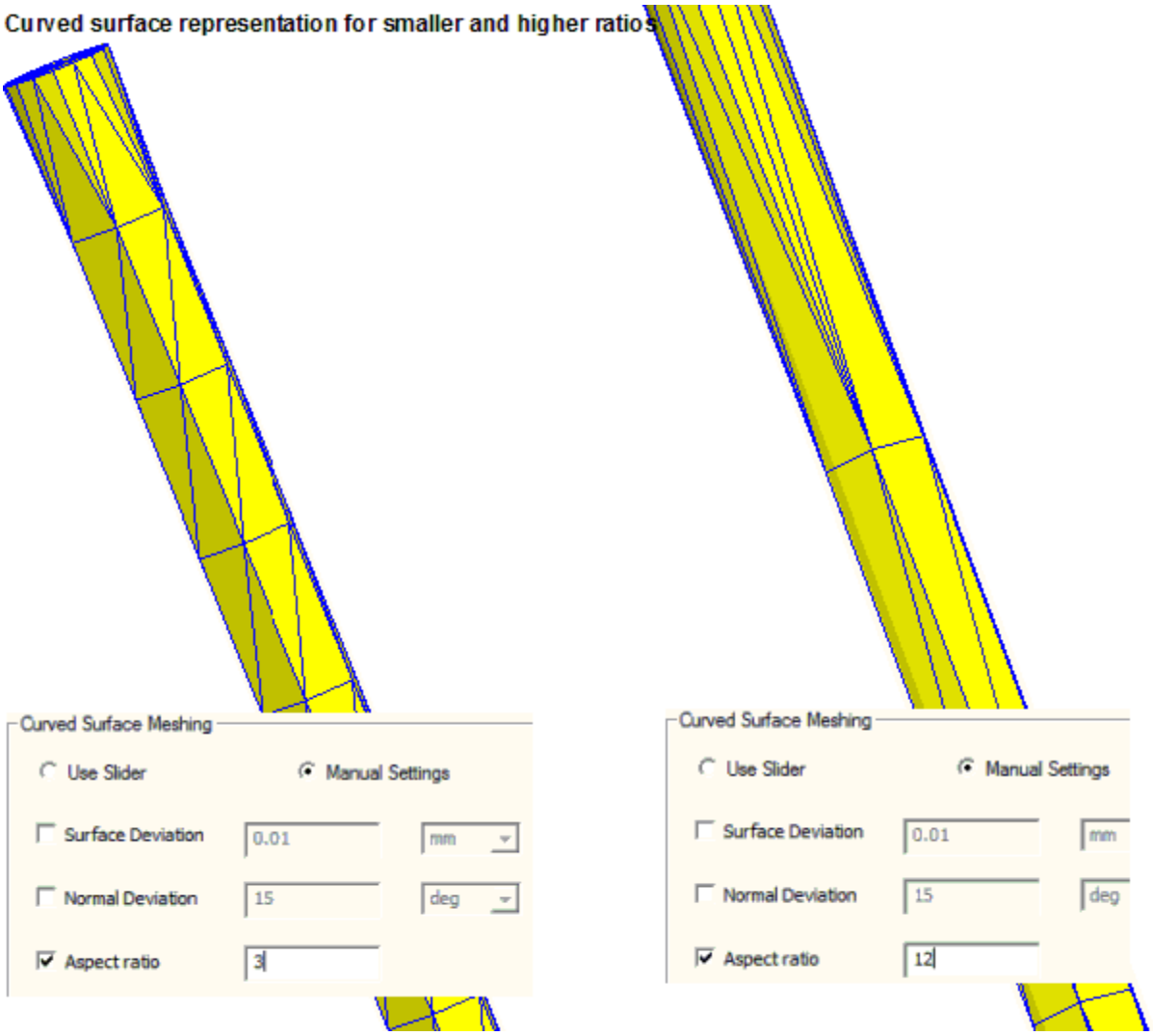
Maximum Aspect Ratio

Maximum Aspect Ratio controls the ratio of the longest side to shortest side in a triangular mesh. It is a type of mesh operation occurring at the object level, and it directly controls the surface representation of the object. Aspect ratio and the shape of the triangulated facet

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

representing the surface of the object, are related. As aspect ratio increases, the triangles become narrower. Setting aspect ratio to 1 does not guarantee equilateral triangles but produces triangles similar to isosceles right-angled triangles.

> **Note:**
>
> Default value for **Maximum Aspect Ratio** setting is recommended. Values in the range of 2 to 20 can produce suitable meshes.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

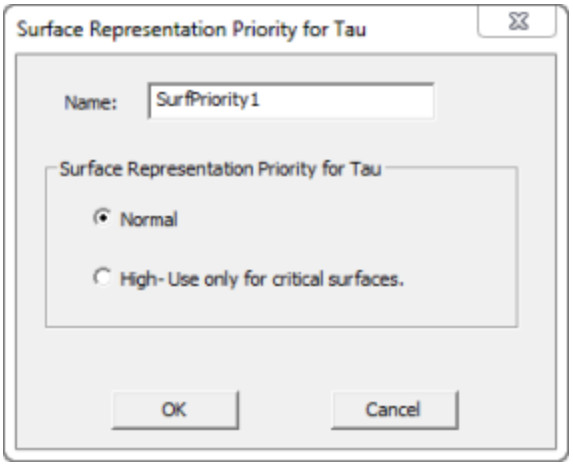
Surface Representation Priority for Tau Mesh (effective only for Tau Tolerant Mesh)

If a particular surface is very small in size and located close to large surfaces and/or this object is very important or has critical boundary conditions, you can set the Mesh Operation **Surface Representation Priority for Tau Mesh** to preserve it accurately. For example, a small antenna on the body of an aircraft may need high surface representation priority.

Tau mesh uses the dimension of the model as reference to specify the mesh tolerance. Generally, mesh tolerance is 1e-5 of the model dimension. In some designs, the dimension of the model is large while some critical objects are small, such as the framed antenna on a windshield embedded on a car or a small antenna on a huge aircraft.

In such designs, sometimes the small objects are assigned boundary conditions or they are critical to the simulation. Using global tolerance might not produce a highly accurate mesh on these small objects. Therefore for such designs when **Surface Representation Priority for Tau Mesh** is specified as high on the small objects, the mesher preserves every detail and by strict verification ensures that the mesh properly represents these objects. This mesh operation is effective only for the TAU (Tolerant) mesher. TAU (strict) follows the same rules as Classic mesh, which ensures a very accurate representation of the surface as defined by the surface approximation settings.

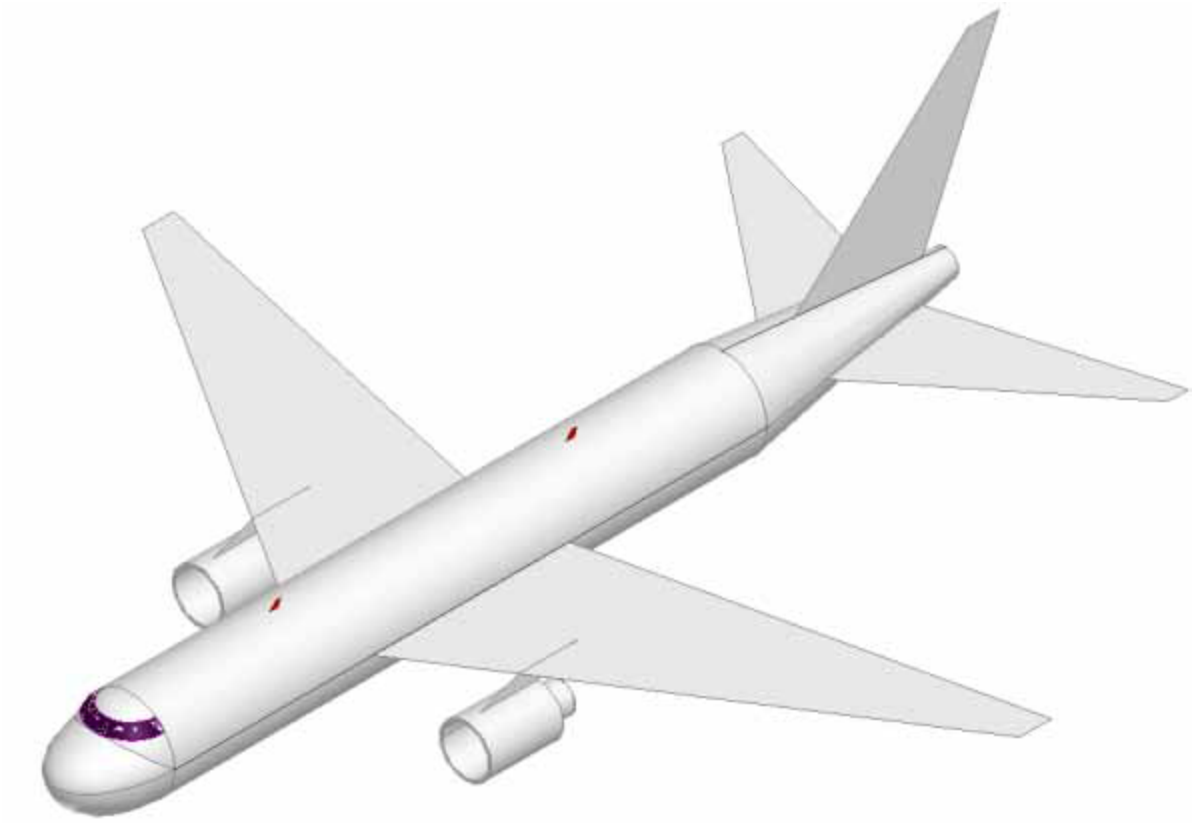
> **Important:**
>
> The consolidation of surfaces into a conformal mesh is skipped for dynamic and tolerant meshing, including for light weight geometries. This can lead to overlapping surfaces in SBR+ simulations. The user should carefully avoid overlapping surfaces or objects as SBR+ can produce unexpected results.

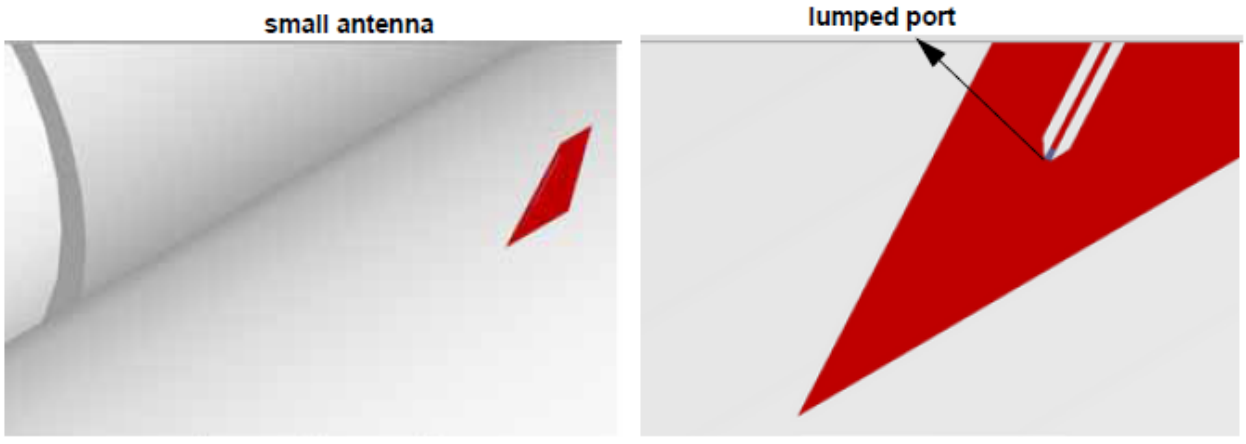


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

Use this mesh operation with caution. Use this option only if the model has tiny elements or critical boundary conditions since performs several additional strict mesh verification steps on the selected objects.

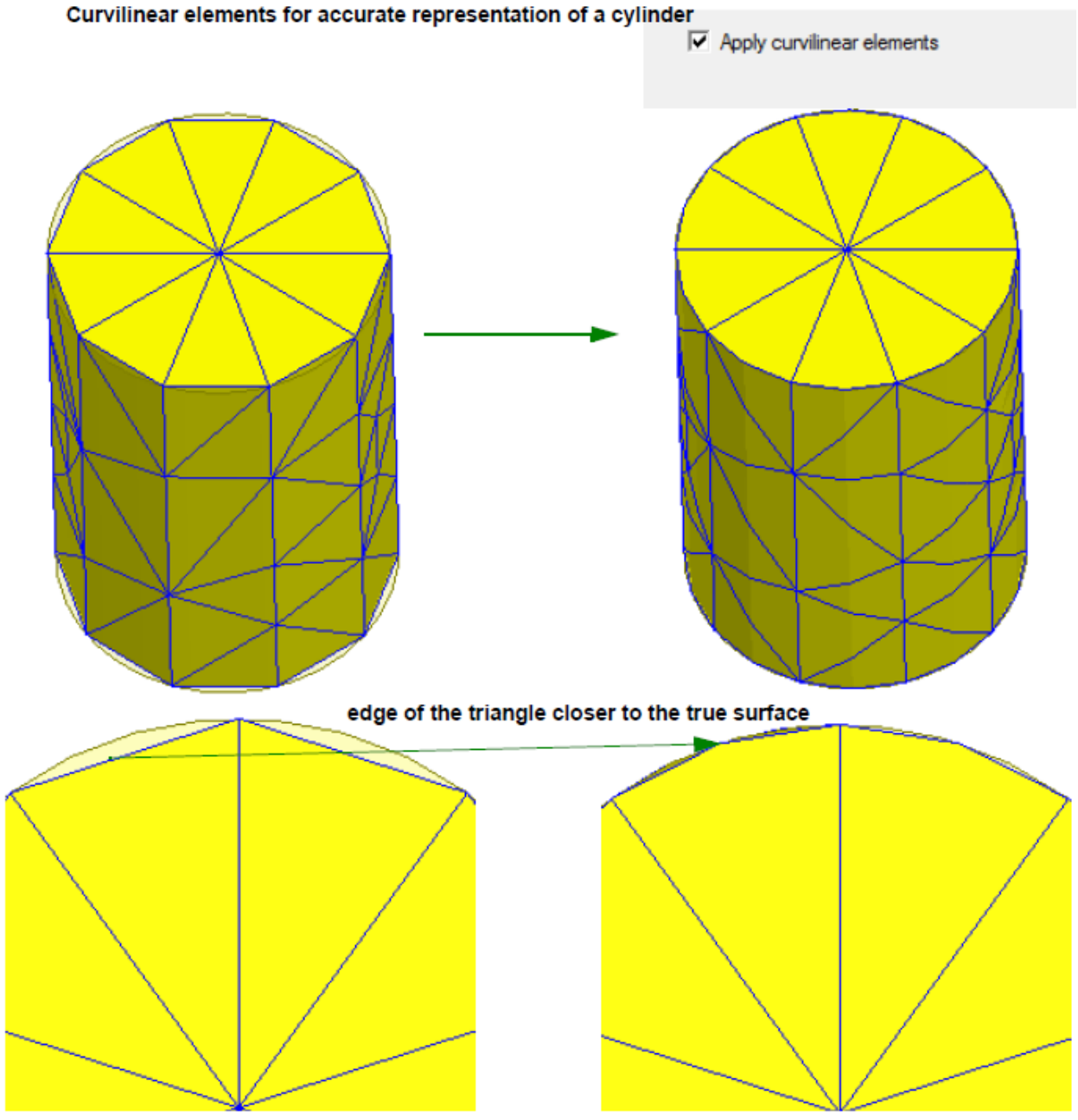




Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Apply Curvilinear Elements Discussion

This Mesh Operation is a global surface approximation setting, and it is used to enable curvilinear elements for all true surfaces in the model. If you select this Mesh Operation command, higher order or curvilinear elements are used to represent the geometry. The midpoints of the surfaces of the tetrahedra are pulled to the true surface ensuring better representation of the geometry as illustrated in the following figures.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Default surface approximation settings with curvilinear elements yield accurate results in an efficient way. The application of curvilinear elements allows you to maintain a relatively low facetation and therefore save on computational resources.

> **Note:**
>
> Curvilinear elements can be used on FE-BI, IE Region and SBR+ surfaces.

Meshing for Curved Surfaces

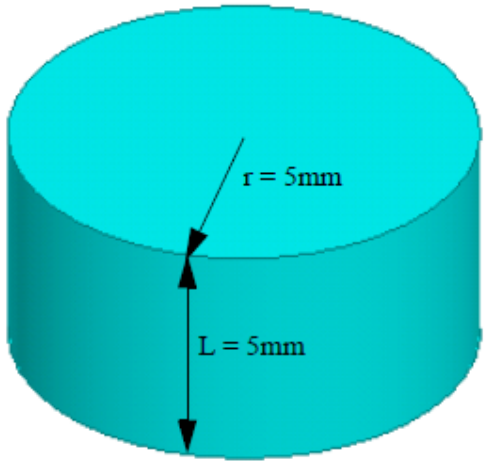
For curved surfaces, the initial mesh obtained on the surface of a geometry is the same initial mesh generated in accordance with the surface approximation settings. During the meshing process of curved surfaces, first the volumetric mesh is created based on Parasolid triangulation. For Classic and Tau (Strict), the surface triangulations on the volume mesh are very close to the Parasolid triangulation. The approximation can be further improved by modifying surface approximation settings. As the mesh is adapted and refined, new mesh points are added to the surface mesh. If the solution is rectilinear, these points are added to the faceted representation of the curved surfaces. On the other hand, if the solution is curvilinear, then newly added points are used to improve the curved surface representation in the mesh. The process of applying curvilinear elements is often referred to as “pulling points to the true surface,” which essentially means that edges of the triangulated facets are moved closer to the true surface, resulting in accurate representation of the curved surface.

Effective Surface Approximation For a Pillbox Resonator

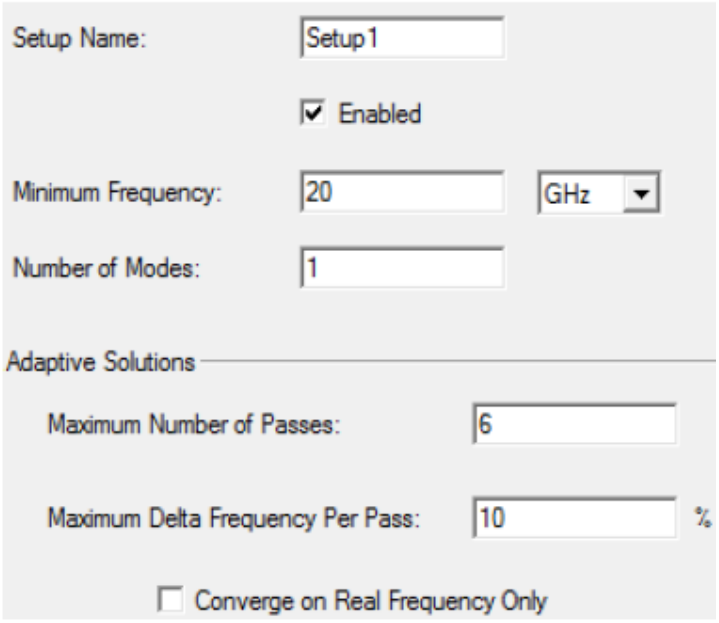
This section describes different surface approximation settings for meshing true surfaces and how they impact the accuracy of the solutions. To demonstrate these settings, a pillbox resonator is described whose dominant mode is estimated to be above 20 GHz. The eigensolver finds the specified number of modes that exist above the minimum frequency. In this example, the eigensolver finds the first mode above this frequency. The analytical resonant frequency is 22.9502 GHz.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pillbox Resonator design



Eigen solution setup options



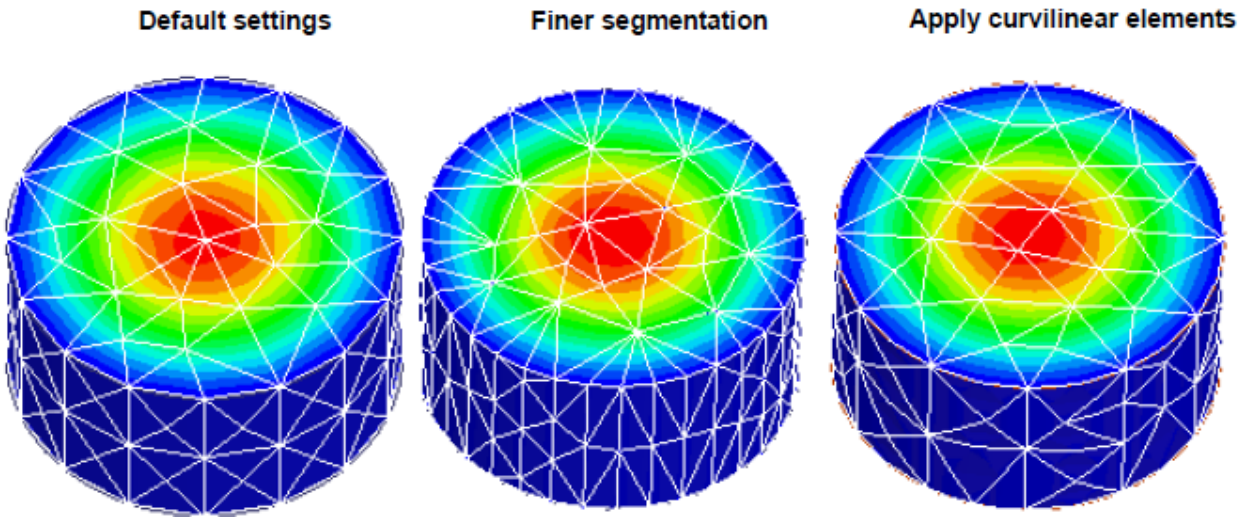
The pillbox resonator is solved using each of the following surface approximation settings:

- Default.
- Finer segmentation.
- Curvilinear elements.

For default settings, the calculated frequency is 23.2685 GHz. This frequency is higher than the analytical frequency (22.9502 GHz) because the effective volume that is being computed is smaller than the actual volume of the pillbox due to rectilinear elements. Default settings provide reasonable approximation, with relative error of 1.38%, and the degrees of freedom (matrix size) are 6024. To improve the accuracy, finer segmentation of the cylindrical surface is attempted by decreasing the maximum normal deviation angle to 10 degrees. In this case, the calculated frequency is 23.0121 GHz and the relative error is 0.026%. The accuracy is much better; the DOFs increase to 6140. In the third case, curvilinear elements are applied on a relatively coarse mesh with a maximum surface deviation angle of 30 deg, and the other default settings are retained. The calculated frequency is 22.9500 GHz, the relative error is 0.0008%, and the DOFs is only 4966. These results show that curvilinear elements on a coarse mesh ensure extremely accurate values in an efficient manner. This example showed that one way to approximate the curved surface better than the default is to decrease the maximum normal deviation angle, which results in more facets. Another method is to apply curvilinear elements, which performs surface approximation automatically and is also better than other improvement methods. For this reason, you can even start with a large normal deviation angle that will produce a coarse mesh; however, by applying curvilinear elements, the geometry is represented better and simulation yields more accurate results.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The field and the mesh plots for all the three cases are shown below.



Model Resolution Length

Model Resolution Length sets the threshold for the mesh maker to ignore features smaller than the default or user-specified length and captures major features in the model that are important for the simulation. Models that are imported from other sources may have extraneous details that do not contribute toward the electrical characteristics in the simulation process. If you specify an appropriate value for model resolution length, the mesh maker can selectively remove even tiny unneeded features. When you define the model resolution length for one or more objects, the mesh maker removes the triangulations for these objects.

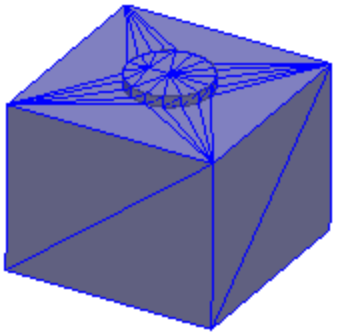
For objects with uniform thickness or with constant cross-sections, the value specified for **Model Resolution Length** can be less than 40% of the thickness. The default **Model Resolution Length** value is 10e-4 user units. For example, if the model is drawn in mm, all gaps, stubs, holes and other features smaller than 0.0001 mm can be removed by HFSS during the creation of the mesh. It is important that you choose units and values in such a manner that all important and critical features of the model are strictly greater than 0.0001 model units.

> **Note:**
>
> If you modify the model resolution length operation, the initial mesh is invalidated and HFSS triggers regeneration of the initial mesh.

Use Model Resolution Length

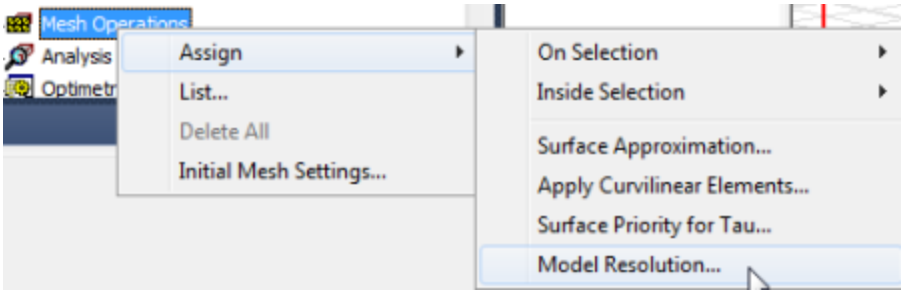
To illustrate model resolution length**,** a meshed model of a box that has a small cylindrical object on the top surface is shown in the following figure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you want to exclude the cylinder from meshing, specify an appropriate value for **Model Resolution Length** such that it is greater than the height of the box. To assign **Model Resolution** follow these steps:

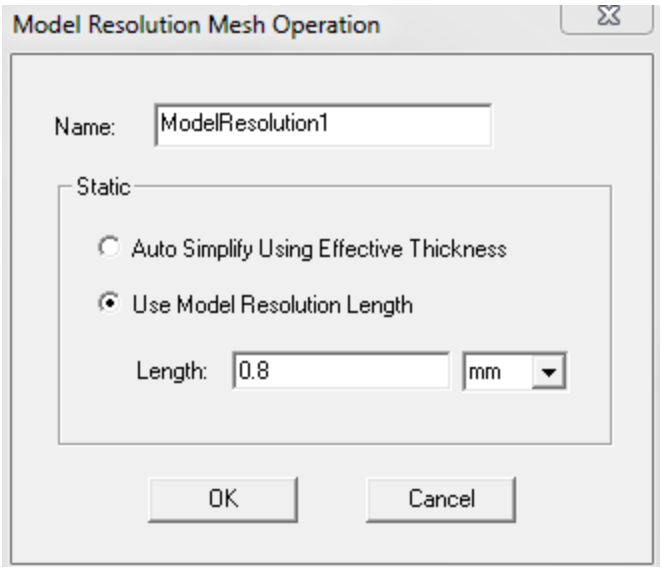
1. Press **Ctrl+A** to select the whole model.
2. Right-click **Mesh Operations** > **Assign** > **Model Resolution**.



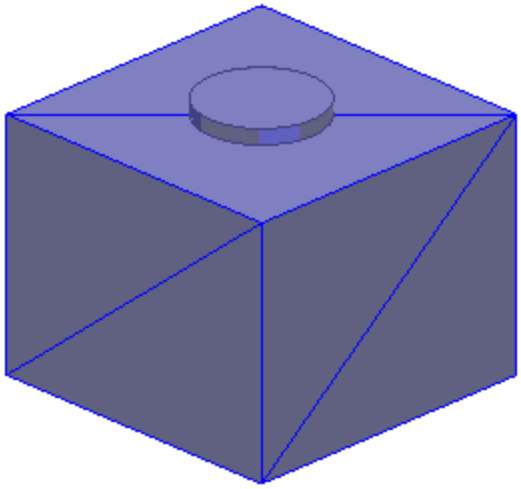
The **Model Resolution Mesh Operation** dialog box appears.

3. Set the value of the length greater than the box height and click **OK**. In this case it is 0.8mm.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



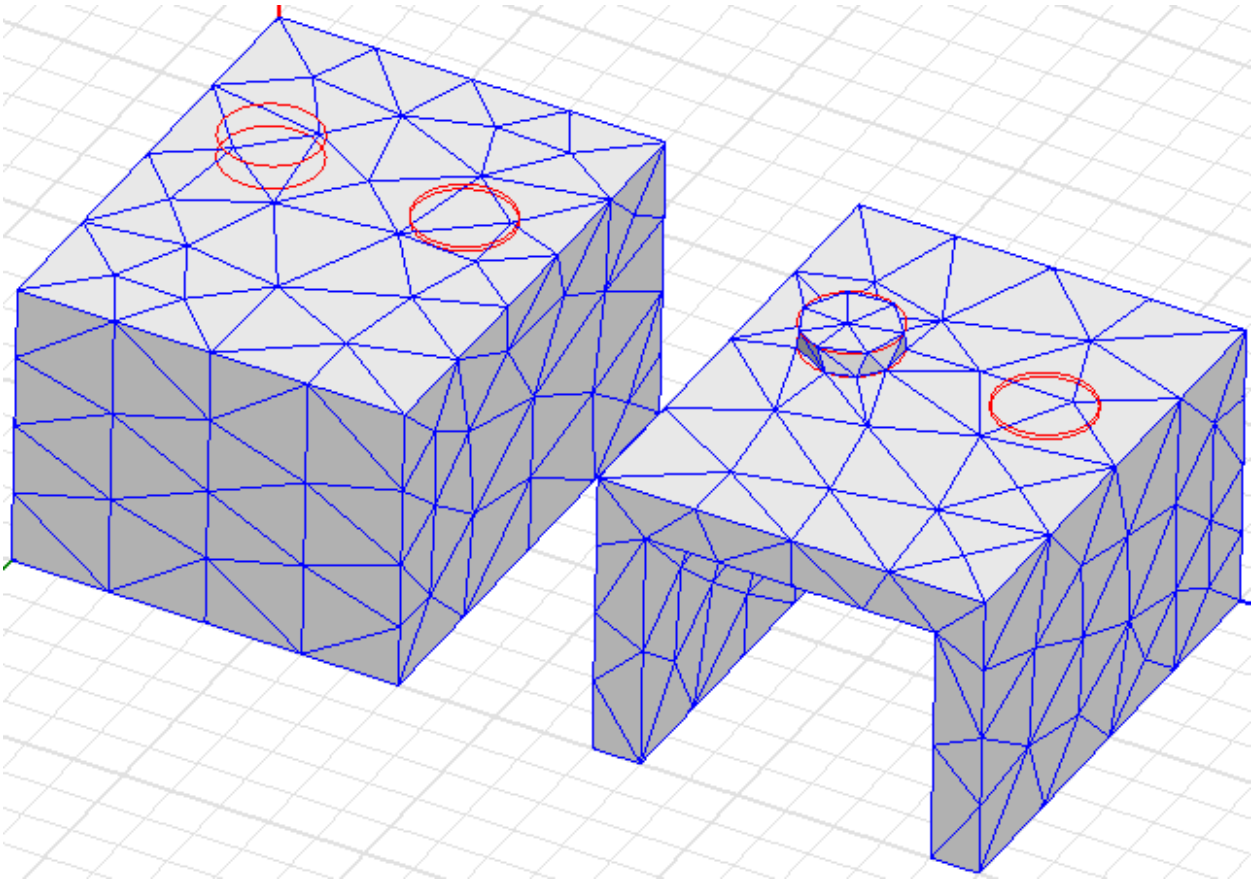
The meshed model excluding the cylinder looks as shown in the following figure.



Auto Simplify Using Effective Thickness

The following figures show the results of using the **Auto Simplify Using Effective Thickness** option on a model containing a box united with two cylindrical objects. The **Auto Simplify Using Effective Thickness** option appears in the **Model Resolution Length Mesh Operation** dialog box. The effective thickness of the body in the left is very high. Therefore applying automatic MRL defeatures both the cylinders out. The effective thickness of the body on the right is smaller so it defeatures the smaller stub and leaves the bigger stub.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

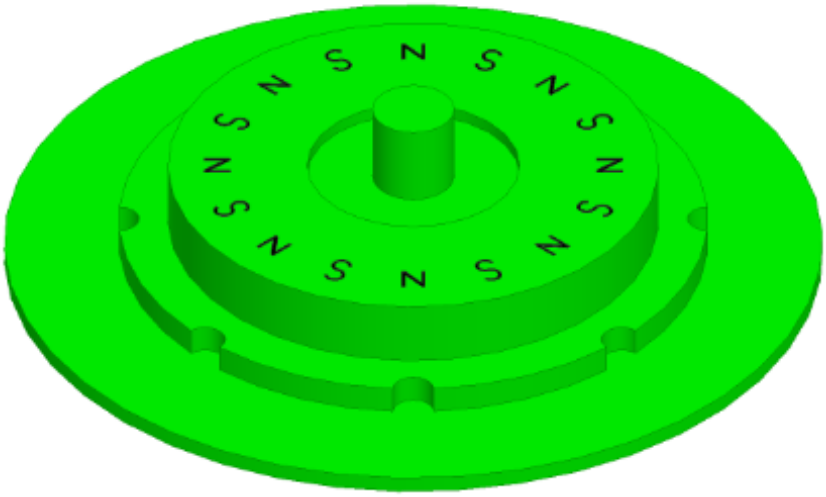


An MRL Scenario

When you define a **Model Resolution LengthOperation,** only major portions of the model are retained in the mesh. An MRL example is shown in the following figure: 3 D symbols S and N are stamped on the top surface of the original geometry.

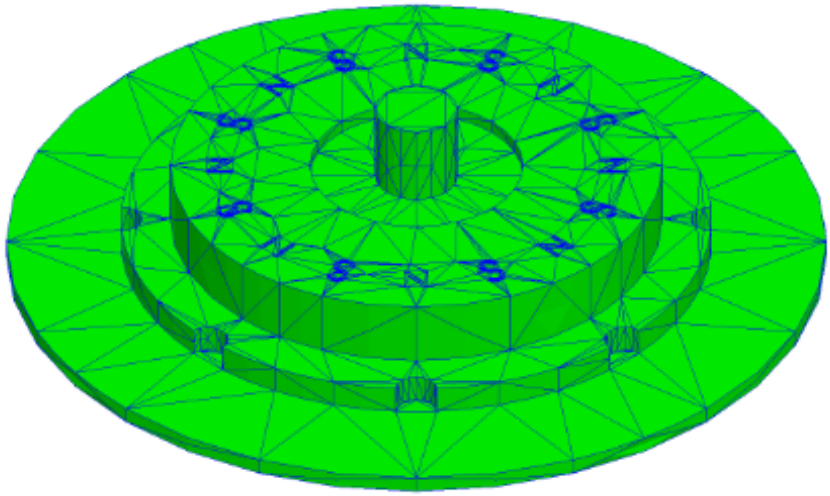
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Original Geometry

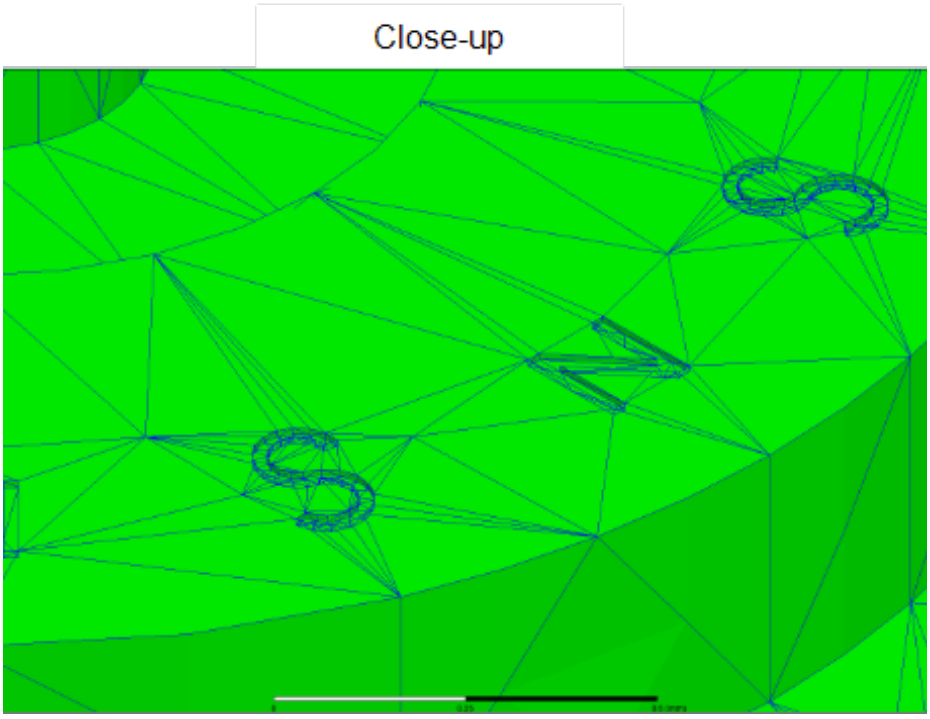


The following figures show the mesh plot without using **Model Resolution Length Operation**. The symbols S and N are magnified to show the mesh's appearance when no **model resolution lengthoperation**is applied. These tiny indentations on the top surface are actually not significant for the simulation. Their presence results in a denser and bigger mesh of the model, especially in the tiny regions as shown below.

Mesh with no MRL



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

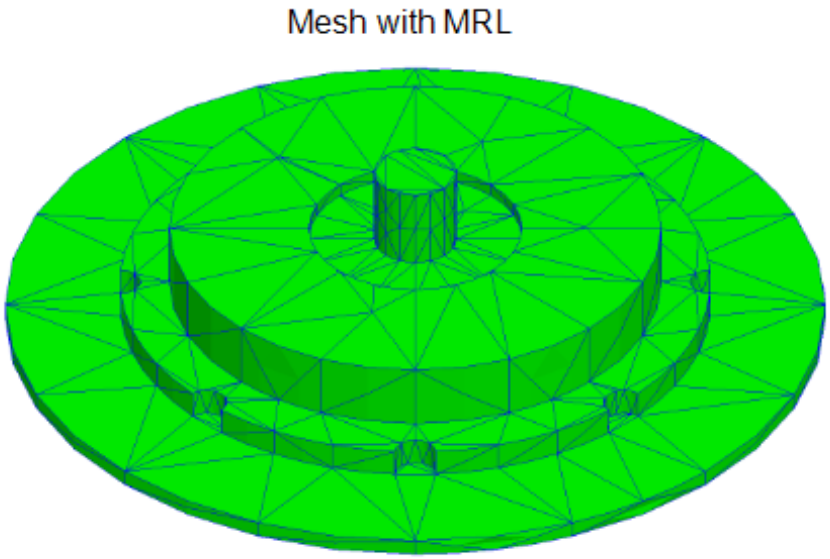


By defining a **model resolution length operation**, these tiny features or indentations can be removed from the mesh.

The following figure shows the mesh of the same model looks when you specify **model resolution length**. The mesh is coarse when compared to the previous mesh. Due to the **model resolution length operation,** the tetrahedrons representing the symbols for north pole and south pole have been removed.

Compare the figures and observe the difference between the meshes before and after the **model resolution length operation**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The removal of extraneous features improves mesh quality and simulation efficiency and requires fewer computational resources.

Operations Refining Initial Mesh

Seeding, port, lambda, and adaptive refinement modify the initial mesh. If you modify seed refinement, the changes are applied to the initial mesh saved by HFSS.

> **Note:**
>
> For more information see Mesh Refinement Settings (Optional)

Notwithstanding the availability of **Assign Mesh Operations**, use the automatic adaptive refinement technique of solving problems in most cases since it is sufficient to yield accurate results.

Mesh Refinement Settings (Optional)

Mesh operations are optional mesh refinement settings that provide HFSS with mesh construction guidance. They can give adaptive refinement a head-start and may accelerate the convergence of the solution. Although assigning mesh operations like seeding can further refine the mesh and may help in generating rapidly convergent results, for most practical purposes, the adaptive meshing technique alone is sufficient to produce accurate results. You must know beforehand where mesh operations apply; otherwise the mesh will be needlessly refined.

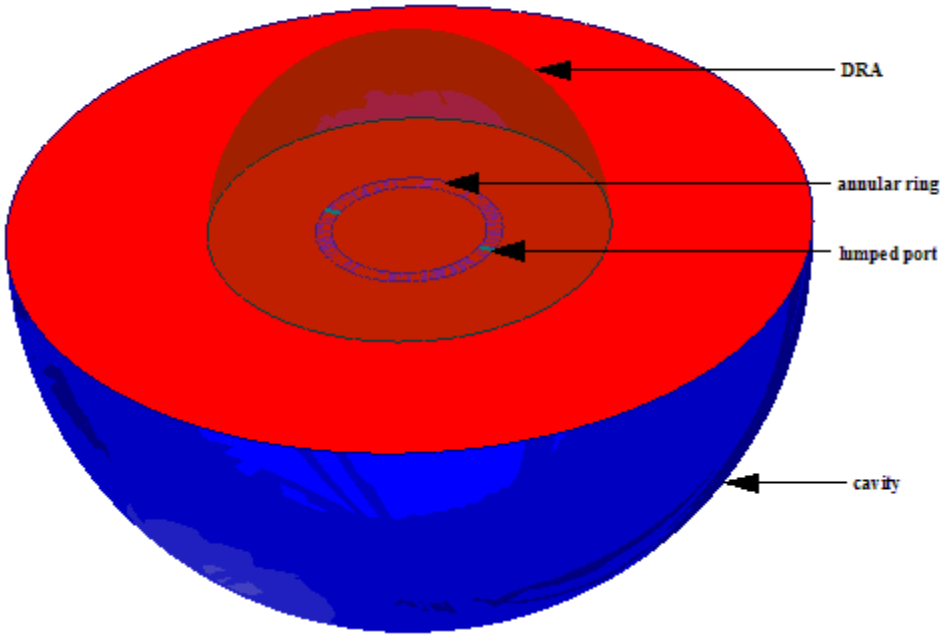
The following examples demonstrate adaptive mesh refinement occurring with mesh seeding.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example 1: Dielectric Resonator Antenna

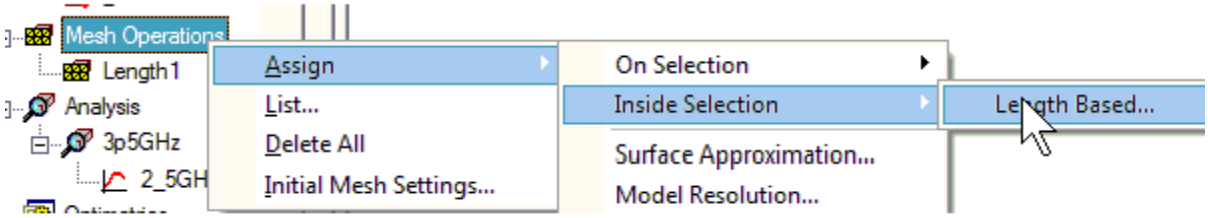
The following figure shows a dielectric antenna resonator. The antenna is cavity-backed with an annular-slot-fed dielectric resonator. The model is fed with lumped port across the annular slot. The design's operating frequency is 3.5 GHz.

Dielectric resonator antenna design

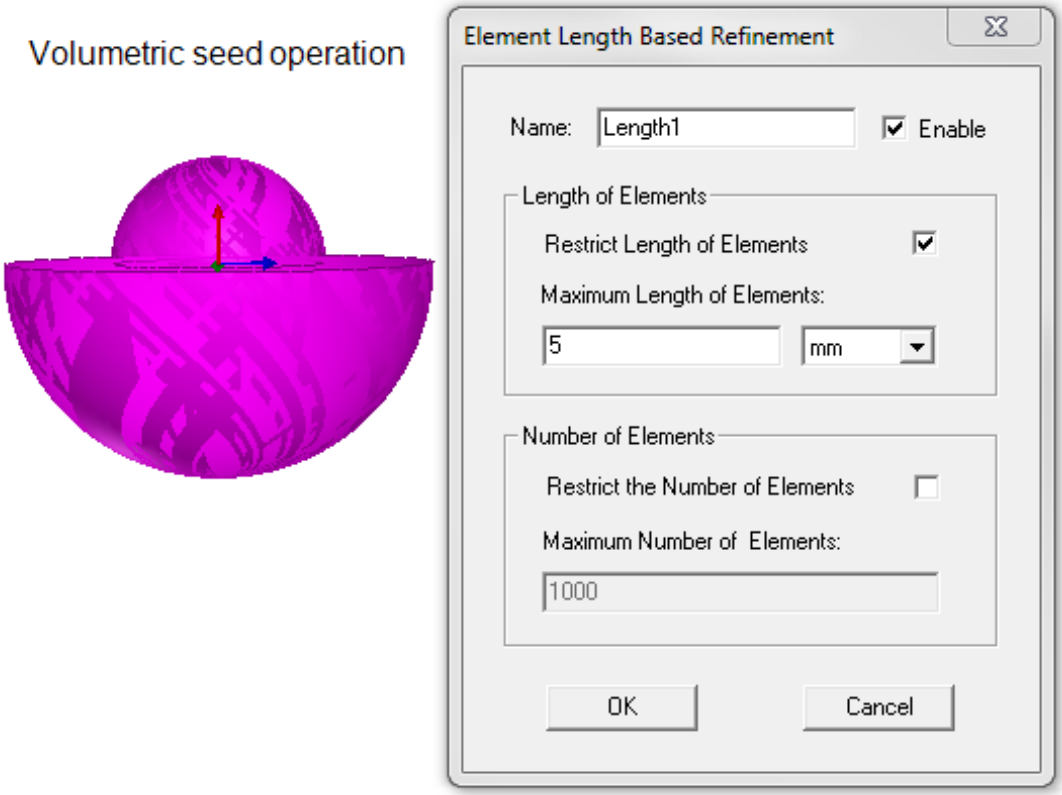


Seeding this model can improve the appearance of the field plots in the desired areas. For this design, mesh seeding operations for the model are assigned as shown below.

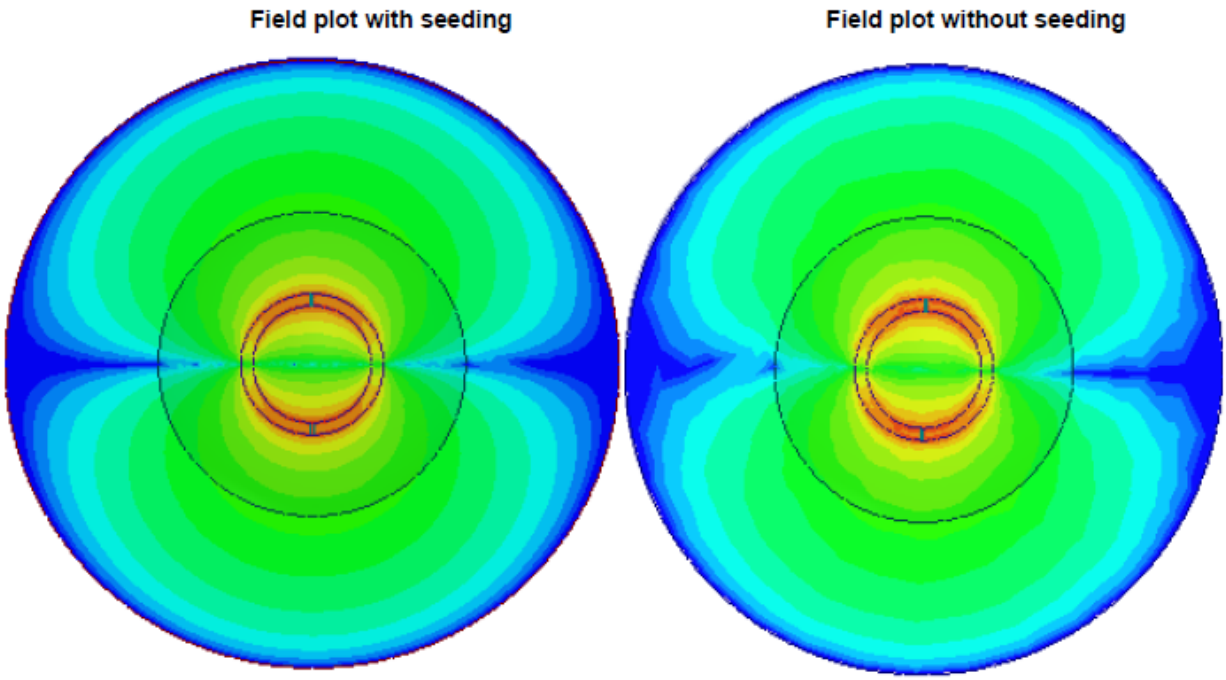
UI controls for assigning seed operation



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



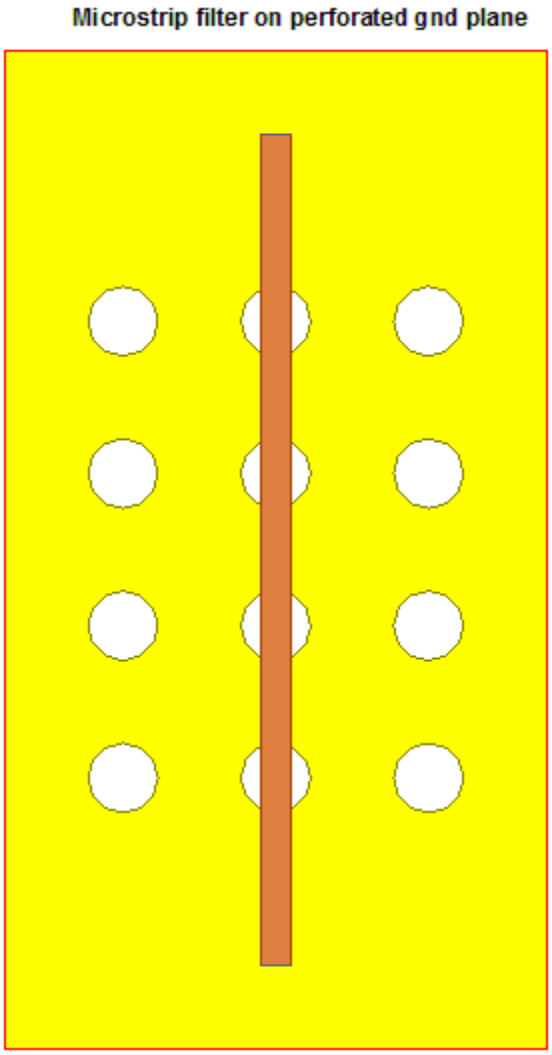
After HFSS solves the design, the resulting field plots on the ground plane are shown in the following figures. In this case, seeding the mesh ensures a high resolution plot.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example 2: Microstrip Filter on a Perforated Ground Plane

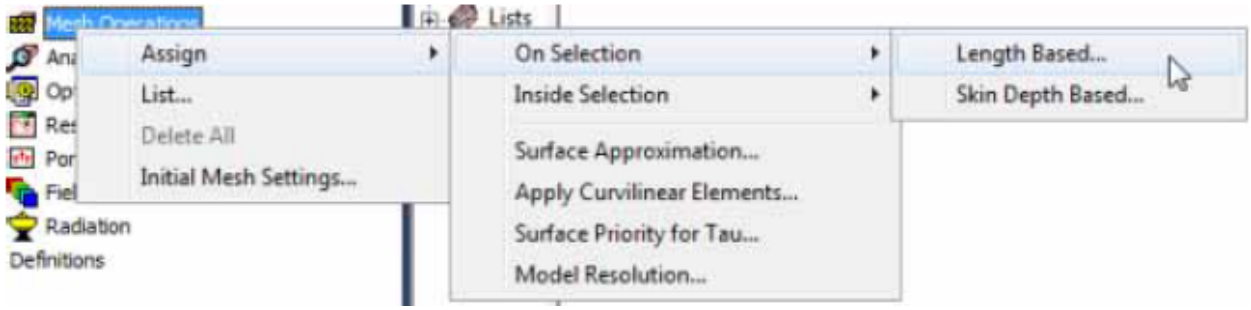
Consider a microstrip filter with a single trace and perforations in the ground plane as shown in the following figure. The design is set up to run at an adapt frequency of 14 GHz. Using the default adaptive refinement process, the mesh consists of 64681 solved elements and the matrix size is 556699. The simulation uses 3.85 GB of memory. The solution converges in 4 minutes, 7 seconds after running 9 adaptive passes using a 12 core machine. The total time taken to solve the problem including the interpolating sweep, is 26 minutes, 44 seconds.

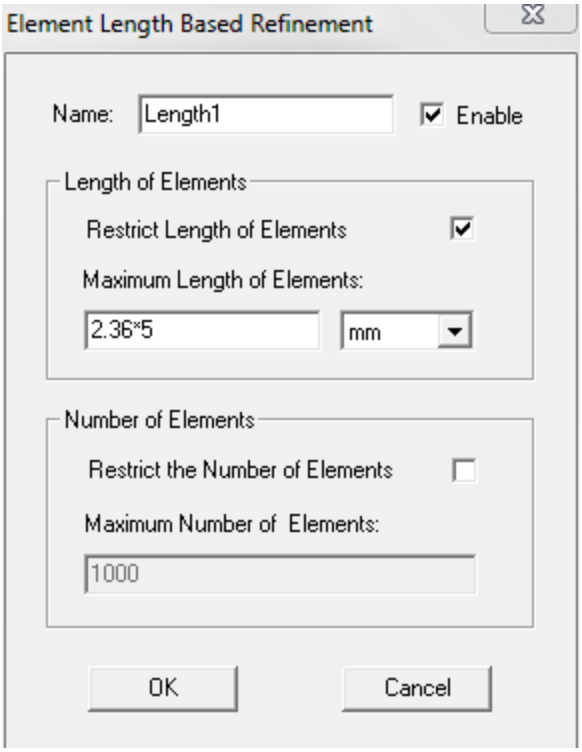


For this design, the trace seeding target length used is five times the width of the trace (see the following figure).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UI controls for assigning seed operation on the trace





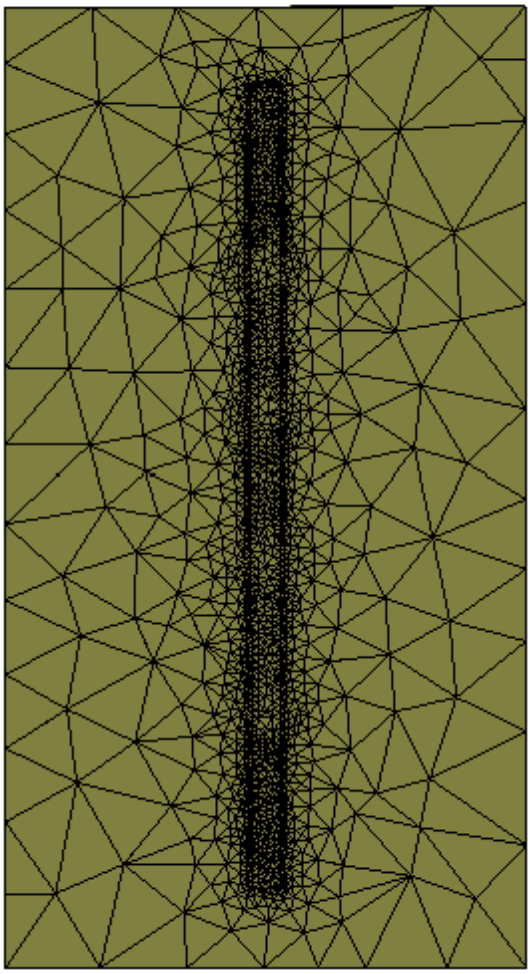
Adaptive refinement with seed operation generates a mesh of 49300 solved elements and a matrix size of 422017. The solution converges in 3 minutes after running 8 adaptive passes and the simulation uses 2.79 GB of memory. The total time taken to solve the problem including the interpolating sweep is 17 minutes, 7 seconds.

For this project, seed refinement ensures that the solution converges faster with fewer adaptive passes, uses less memory, and creates smaller matrix size than when using adaptive refinement without any seed operation.

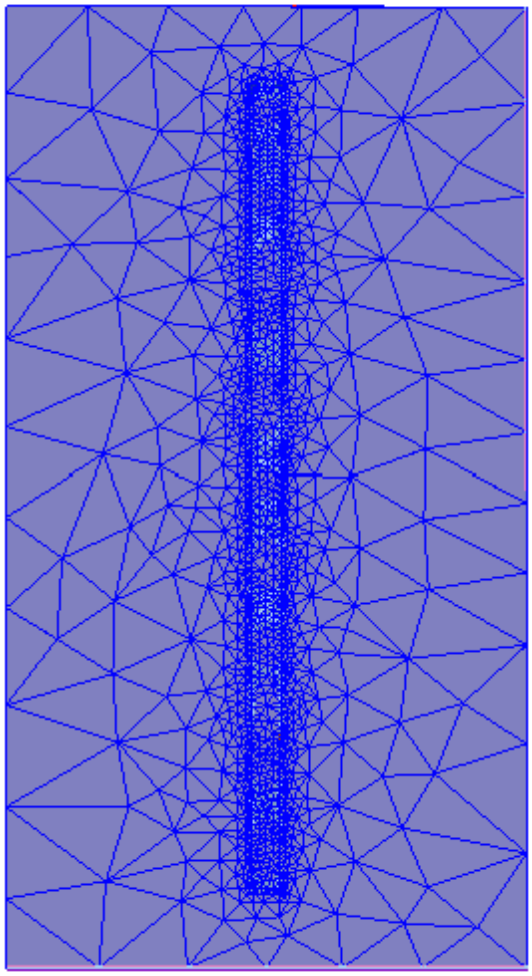
The following figures show the resulting mesh plots and field plots.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh plot without specifying seed operation.



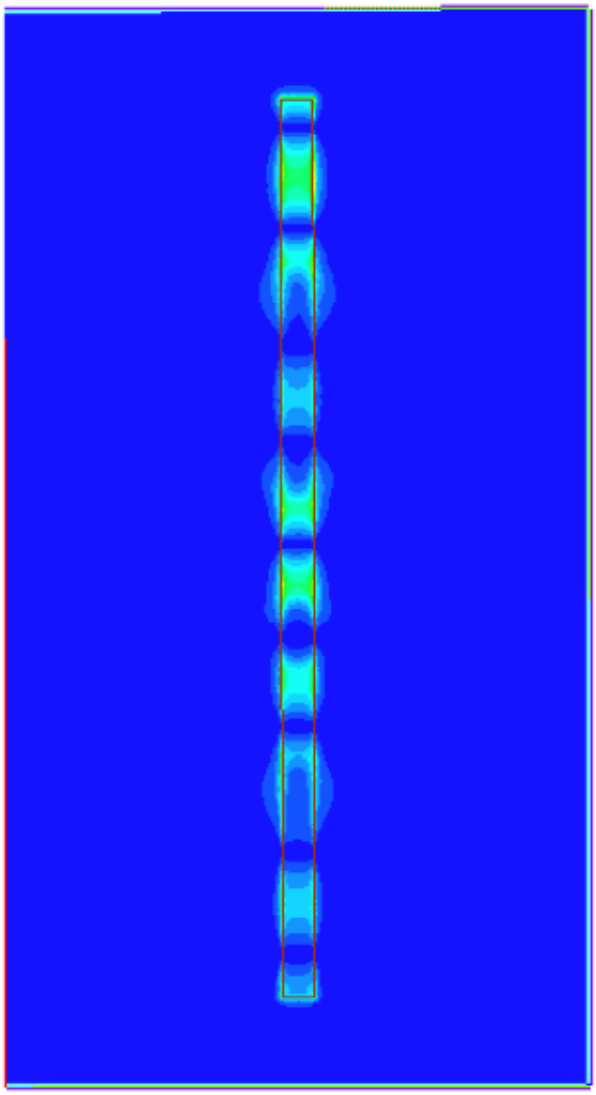
Mesh plot after seeding the trace.



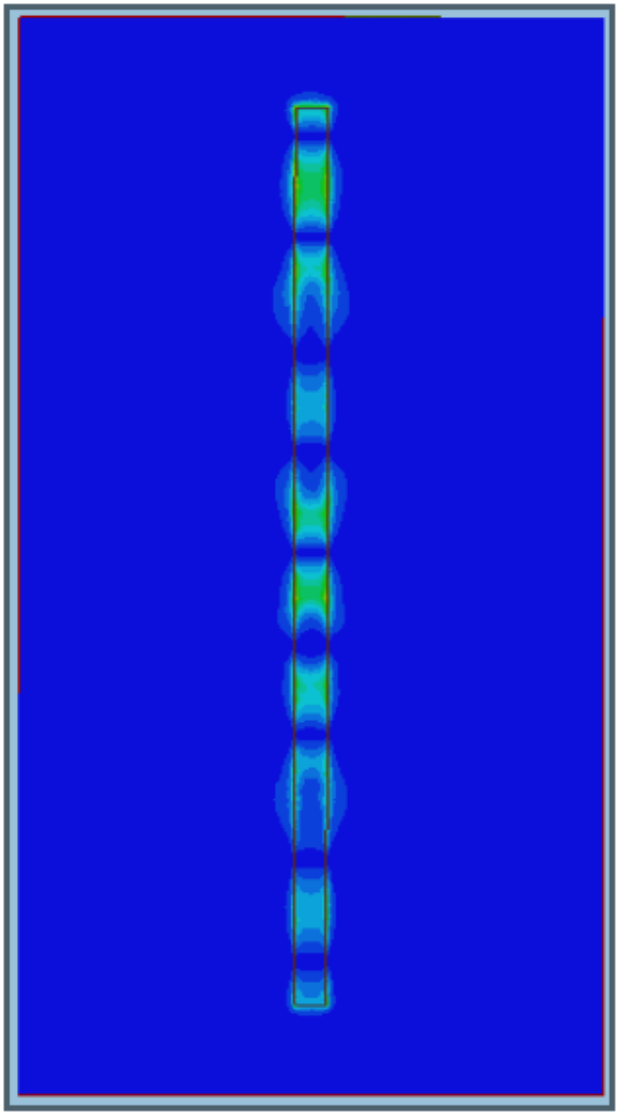
The following figures show the E-field plot for both cases.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

E-field plot without specifying seed operation.

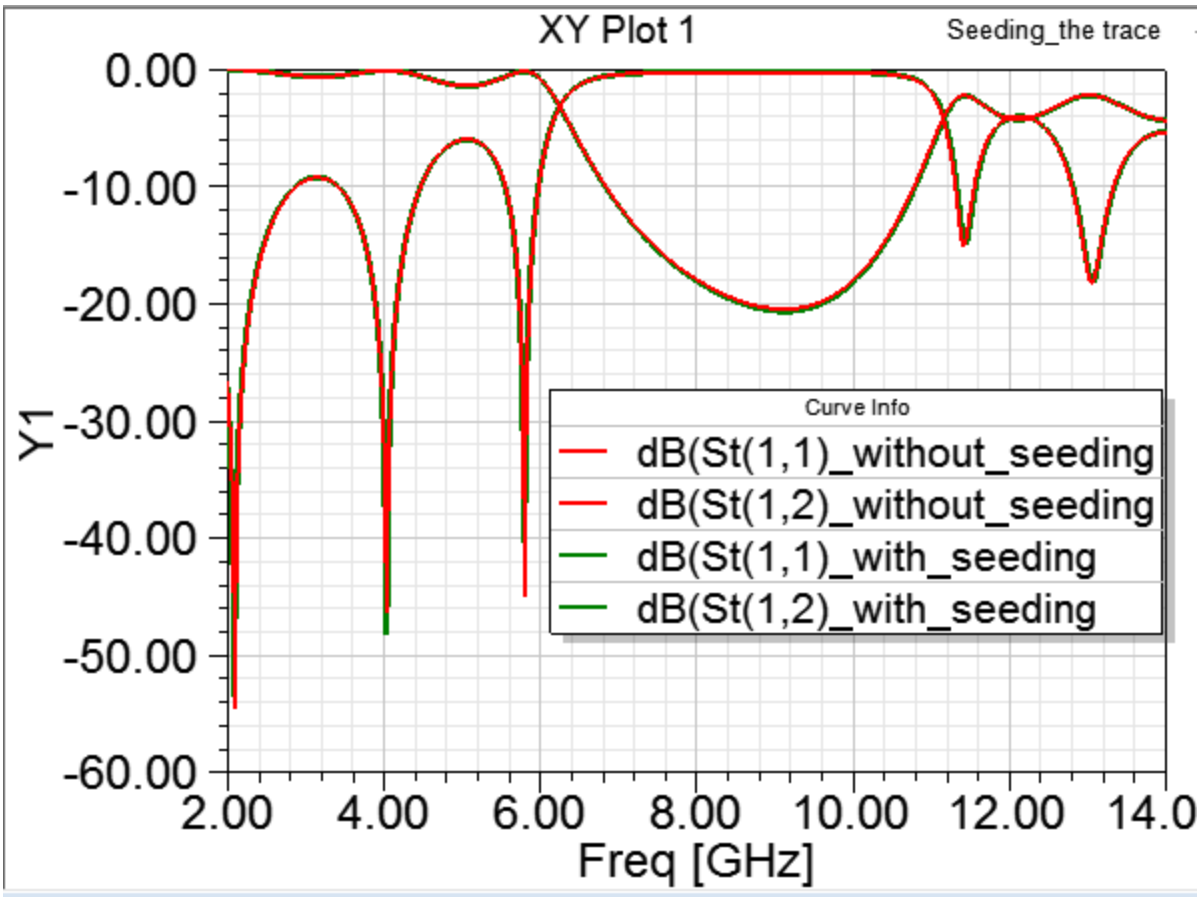


E-field plot after seeding the trace.



The S-parameter plots for both cases are shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



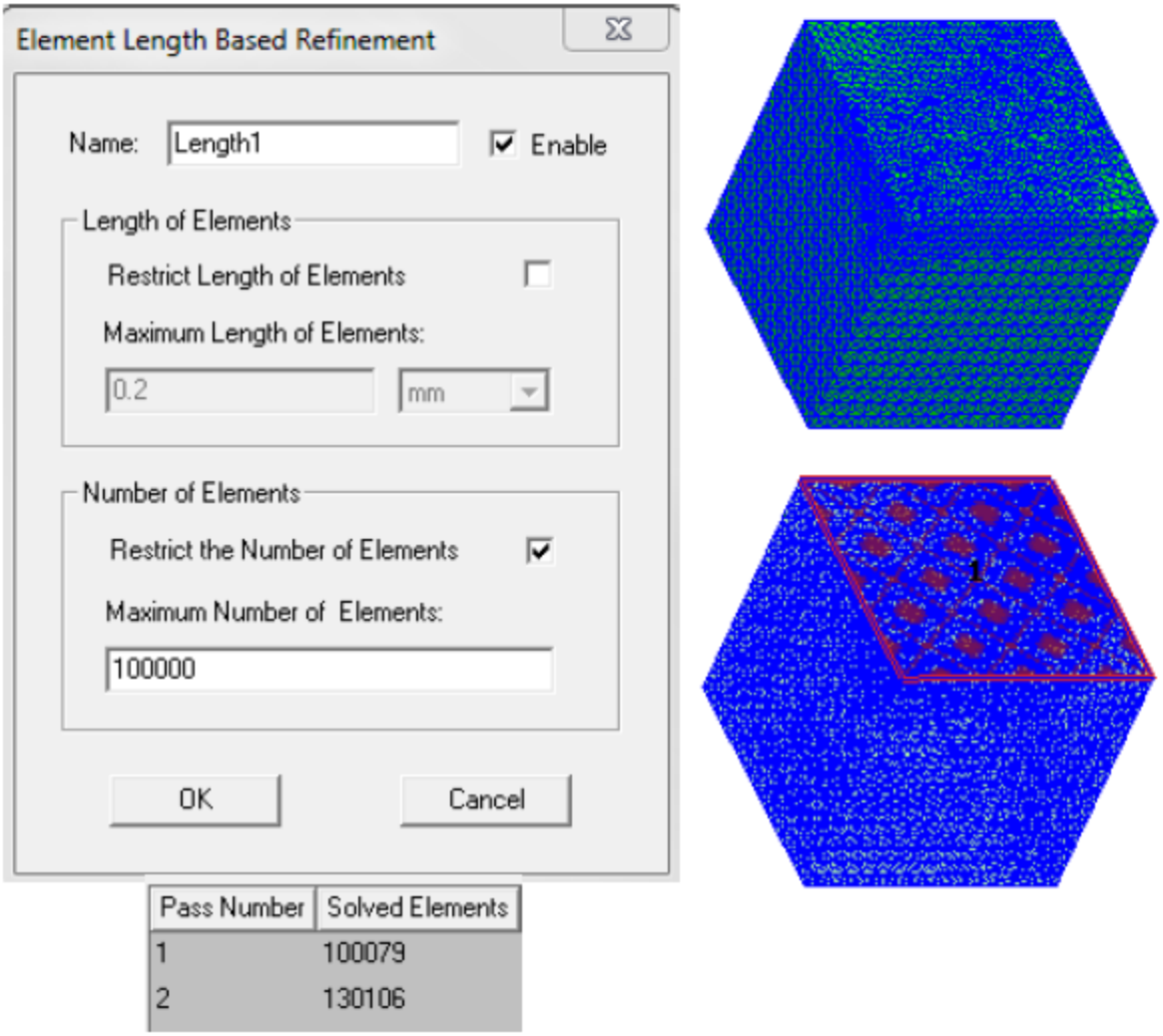
The results from the two simulations (with and without seeding) agree. The automatic meshing feature of adaptive refinement is effective and produces accurate results. The advantage obtained by seeding is not significant enough to warrant the time and effort needed to determine the seed values and define the seed operation. It is extremely challenging to improve the efficiency by defining a seed operation.

Mesh Seeding Controls

Seeding is the technique of guiding HFSS through the construction of the mesh. HFSS starts with an initial mesh and by adaptive refinement refines the mesh. However, if you know before starting the simulation where such refinement is needed in your model, seeding can provide a more refined starting mesh for the adaptive refinement process, which may result in obtaining the converged mesh faster. The following figure shows a type of mesh seeding called element based refinement.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

empirical example showing a heavily seeded mesh representing a cube

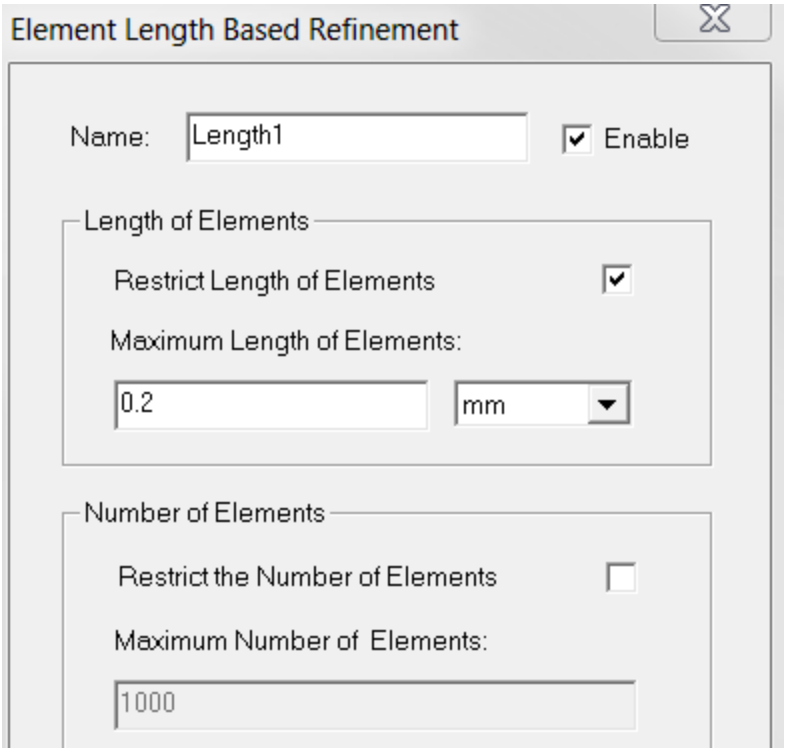


Since the value of Maximum Number of Elements is restricted to 100000, the number of solved elements is in excess of 100000.

Length-Based Mesh Refinement

Length based mesh refinement controls the length of the tetrahedral elements and the growth of the whole mesh. You can set a ceiling on the mesh growth with the option **Restrict the Number of Elements** to ensure a mesh of reasonable size after the length refinement.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When you define a mesh operation on a selection of objects (more than one) and set a maximum length, the operation occurs on the whole mesh. If you select just one object, when you refine its boundary, elements are added both inside and outside of that object.

When there are no more elements to refine or when no more elements can be added, the refinement process stops because it has either met the growth limit or the element length target. However, there are more quality improvement swaps, smoothing, and matching boundary issues, performed at the finishing stage that might slightly alter the mesh size.

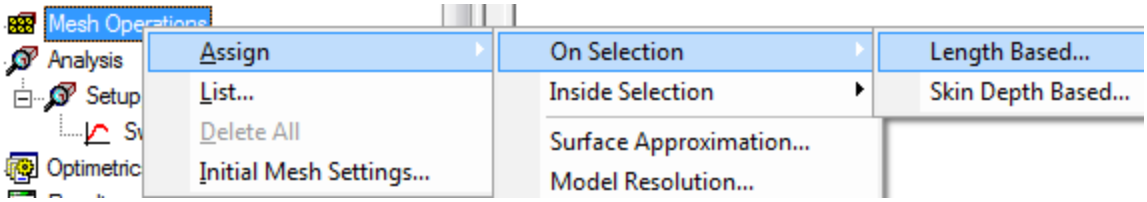
> **Important:**
>
> Be careful about specifying length targets for voluminous structures. This is because element count and target length have a cubic relationship. The relationship between length target and element count is inversely proportional, to the order of n^3 (that is length target = 1/n^3) where ***n*** is the element count.

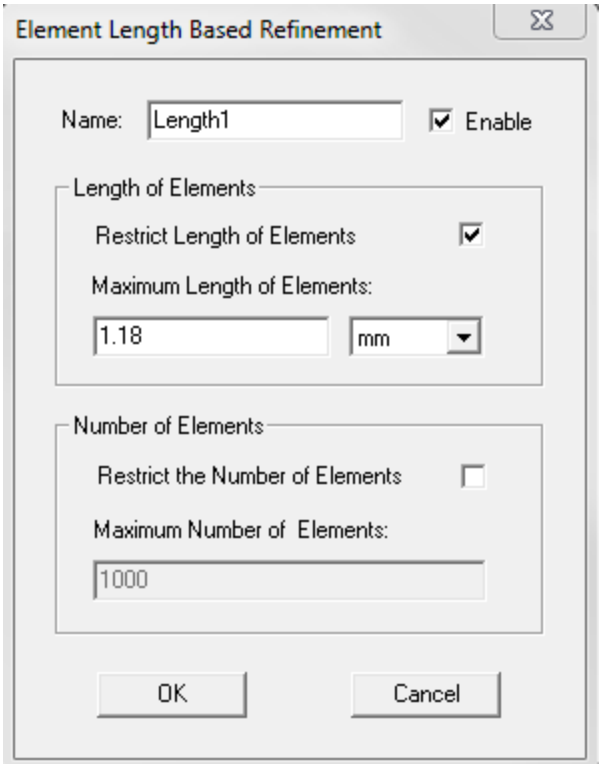
Seed Refinement for Mesh Growth

Seed refinement for mesh growth is performed by defining either the **Number of Elements** and/or the **Length of Elements** option. When either one of these specified criteria on the **Element Length Based Refinement** window is met, seed refinement stops. To access the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

settings for seed refinement, . Depending upon your selection, you can either seed inside the object or on the object surface.





Mesh Refinement Scenarios

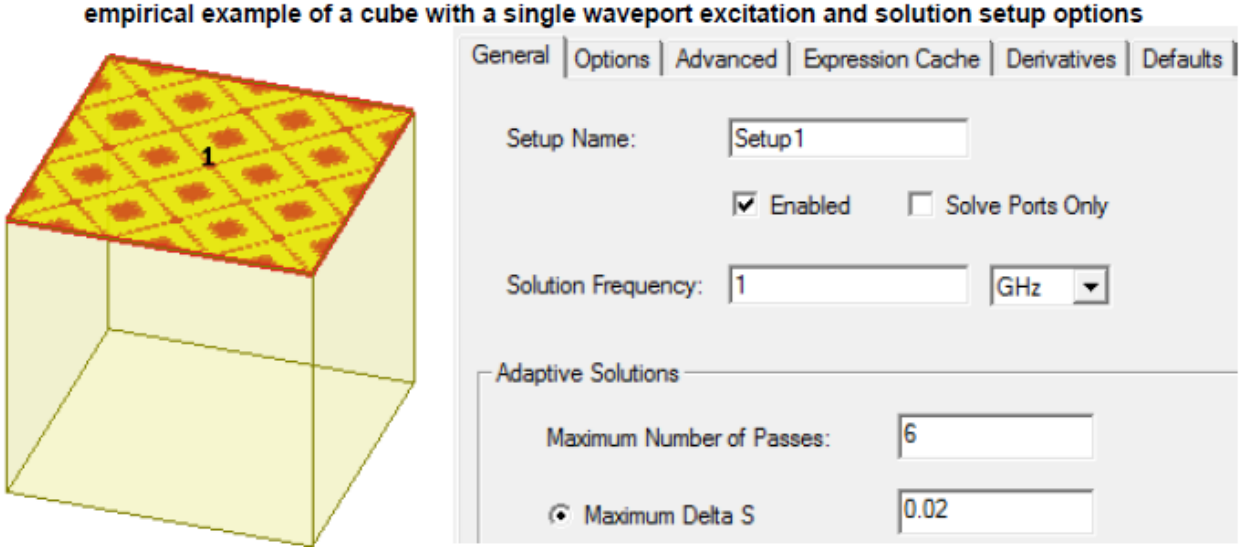
This section describes how mesh refinement settings affect your simulation. You can refine a mesh at any stage and then solve from there.

Scenario 1: Assign Mesh Operations after the First Pass

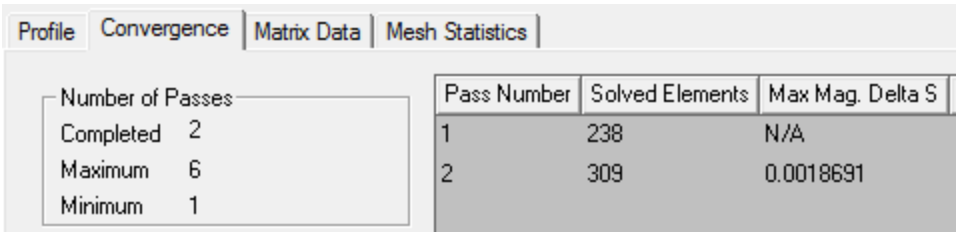
The following figure shows a cube with vacuum as the material assigned. The cube is 1 mm in size with a wave port defined on a face. Default options in the solution setup dialog box are used with **Maximum Number of Passes** =***6***.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

empirical example of a cube with a single waveport excitation and solution setup options

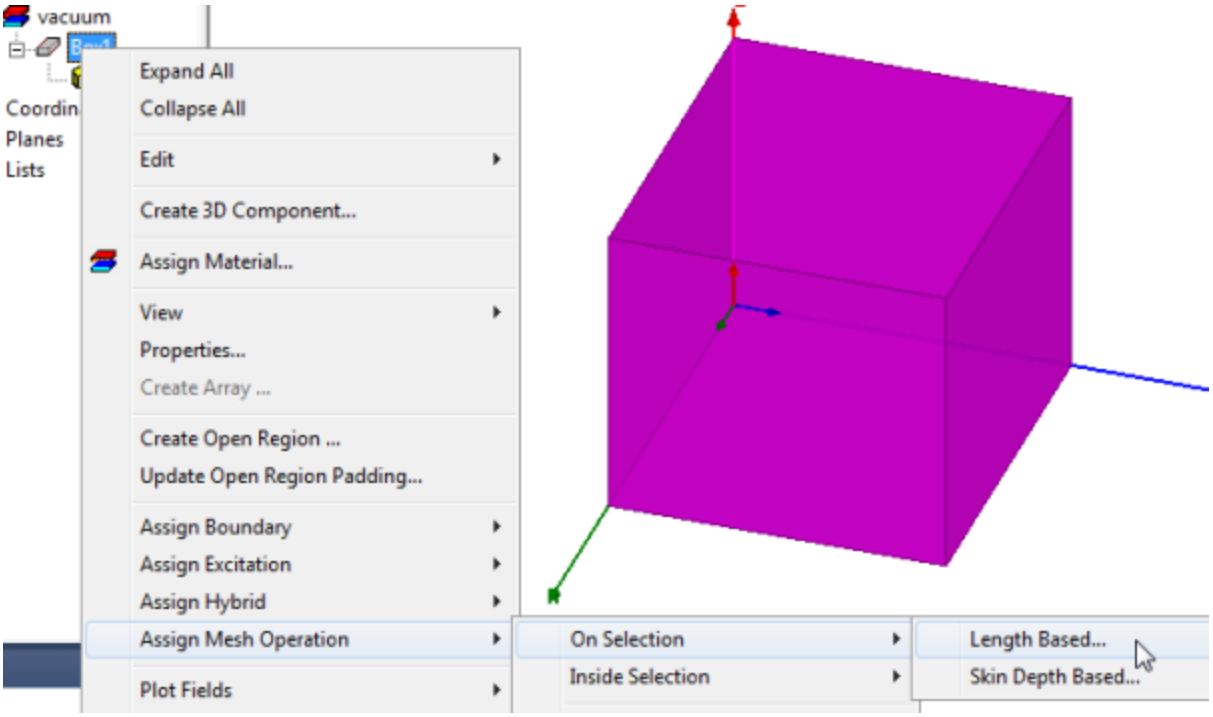


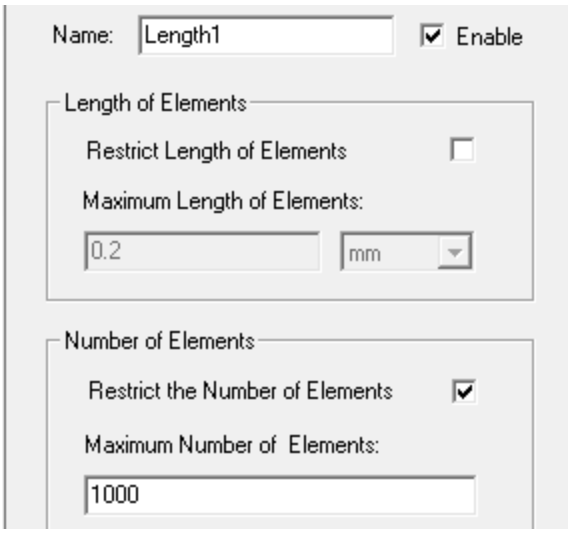
When you solve this design, it runs 2 passes and creates over 300 solved elements on the second pass as shown below.



At this stage, suppose mesh operations are applied as shown in the following figures.

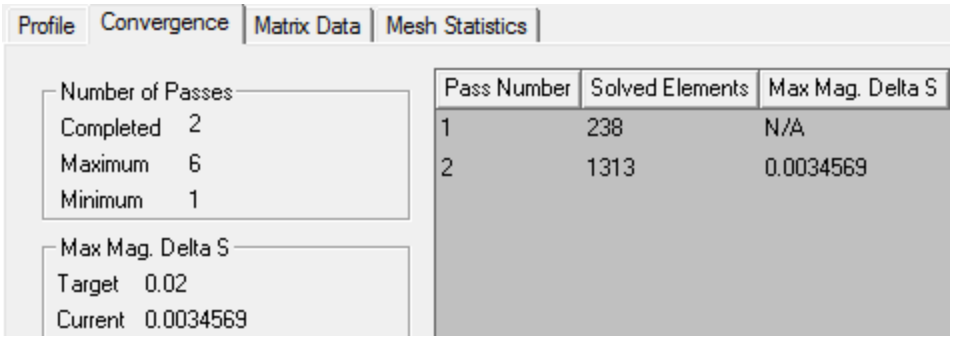
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



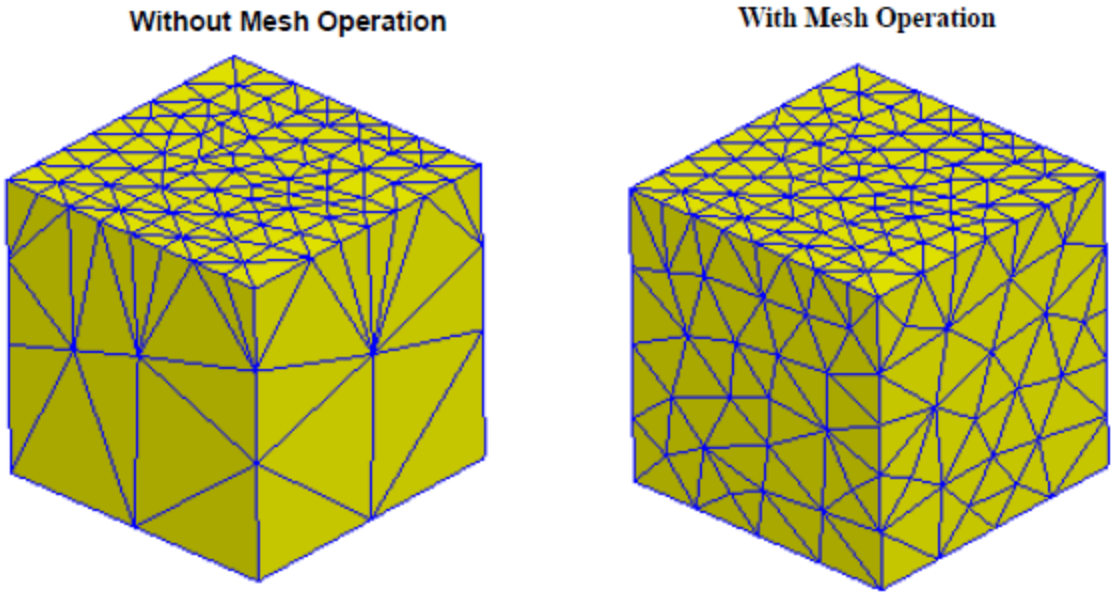


If you solve the design with **Maximum Number of Elements** = 1000, approximately 1000 elements are added in Pass 2 as shown by the column **Solved Elements** on the **Convergence** panel in the figure below. Since the mesh operation was defined after the first pass, the specified value for the maximum number of elements that is 1000 is added to the second pass.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

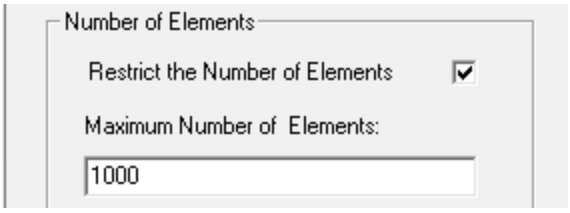


The corresponding mesh plots are given below.



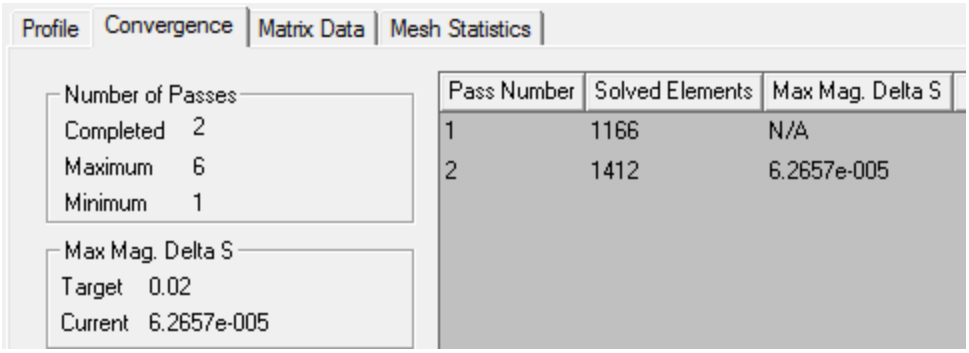
Scenario 2: Assign Mesh Operations Upfront

In this scenario for the same cube, the mesh operation is performed upfront.

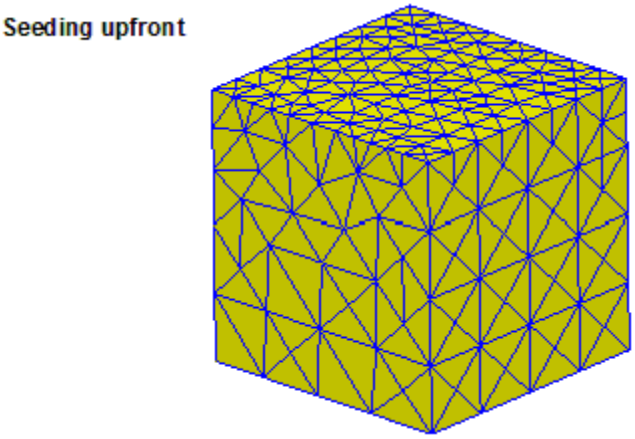


Therefore the assigned mesh operation is applicable for the first pass and 1000 elements are added in the initial mesh as shown in the following figure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The mesh plot for this operation is shown in the following figure.



Note:

The number of solved elements is directly affected only for that pass where seed operation begins. For example, when seeding is performed upfront, only the first pass gets affected and elements are added according to the value defined in the mesh for the first pass only, and subsequent passes are solved but they remain unaffected directly by the seed operation. Similarly, if you seed from the nth pass, the operation affects only the nth pass and the subsequent passes from n+1 are solved from there without being affected directly by the seed operation. For example, if you set **Maximum Number of Elements** = M for a particular pass N, M elements are added on the Nth pass only. In the (N+1)th pass, again M extra elements are not added.

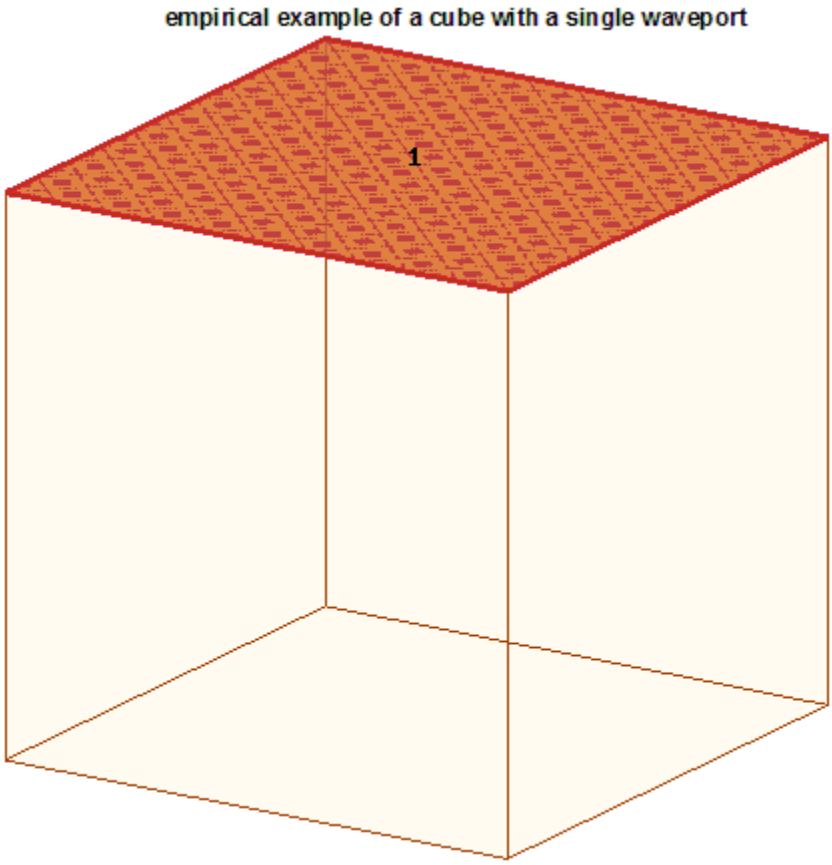
Restrict Length of Elements

Selecting the **Restrict Length of Elements** check box enables you to set the stopping criteria for seed refinement. When the longest edge of a tetrahedral element under the specified value

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

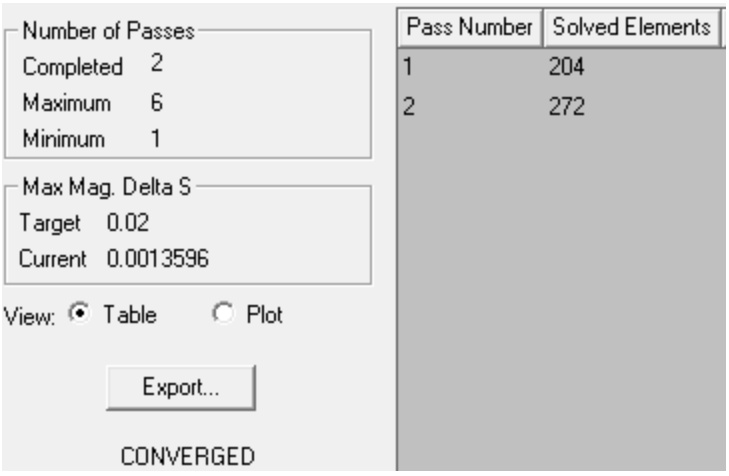
of **Maximum Length of the Tetrahedral Element** is reached, seed refinement stops.

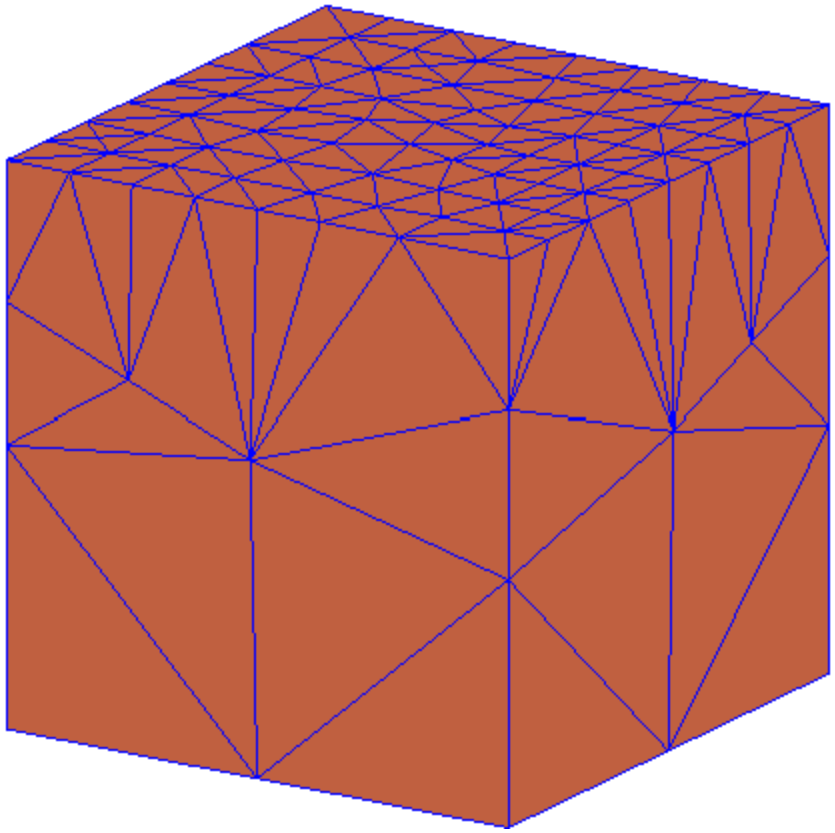
The following example of a cube 1000 mm in size and material vacuum illustrates seed refinement when it is specified by a value of the **Maximum Length of The Tetrahedral Element** as the stopping criteria. The cube has a wave port assigned on one of its faces and the adaptive frequency is 0.25 GHz.



First the design is solved with no seed refinement. The mesh plot on the cube after convergence is shown in the following figures.

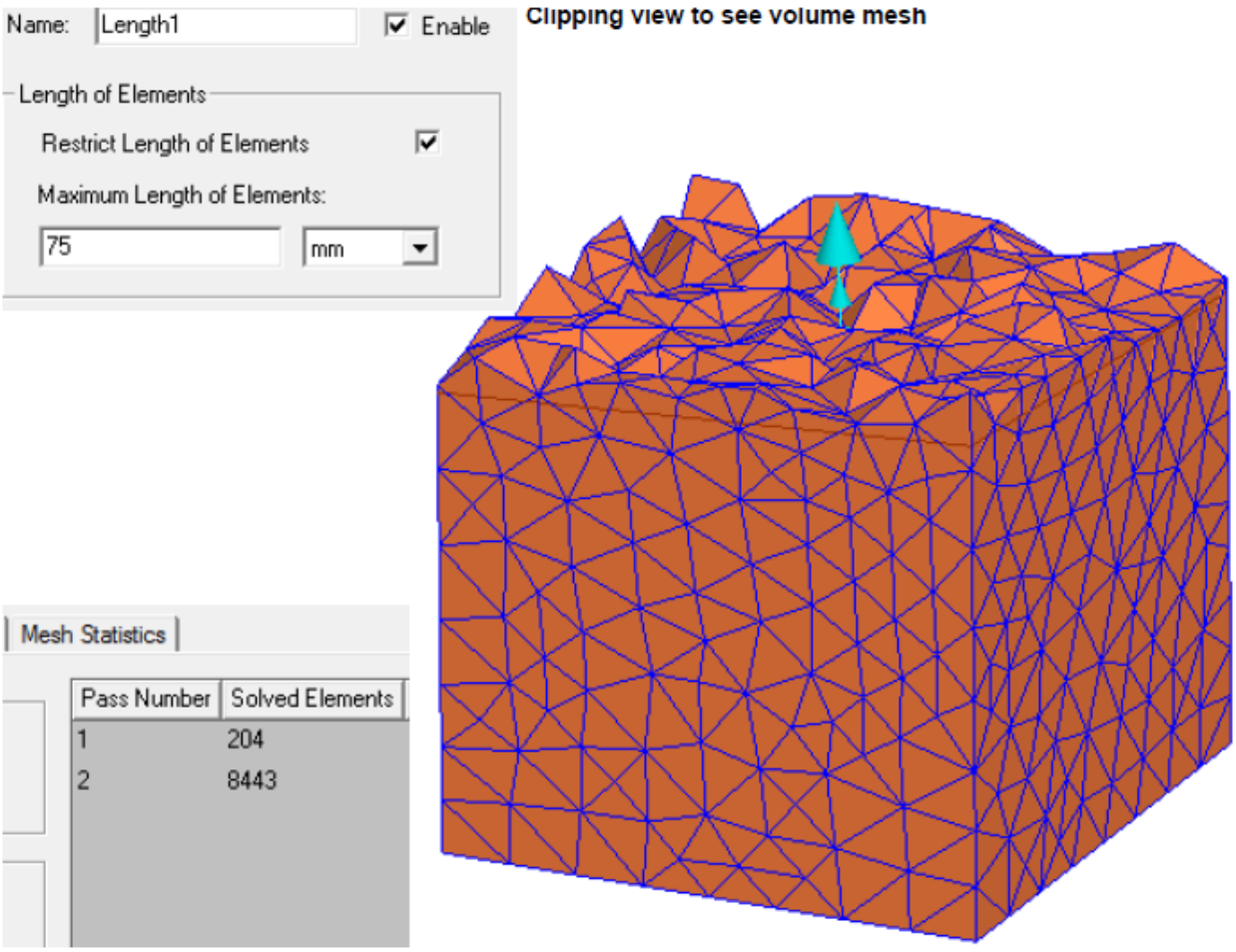
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





A plane is drawn at the center of the cube and the normal to the plane is along the Z axis. Using the **Clipping** view feature, the cube is virtually clipped along the normal. After seed refinement, the mesh plot of the cube along the normal is shown in the following figure.

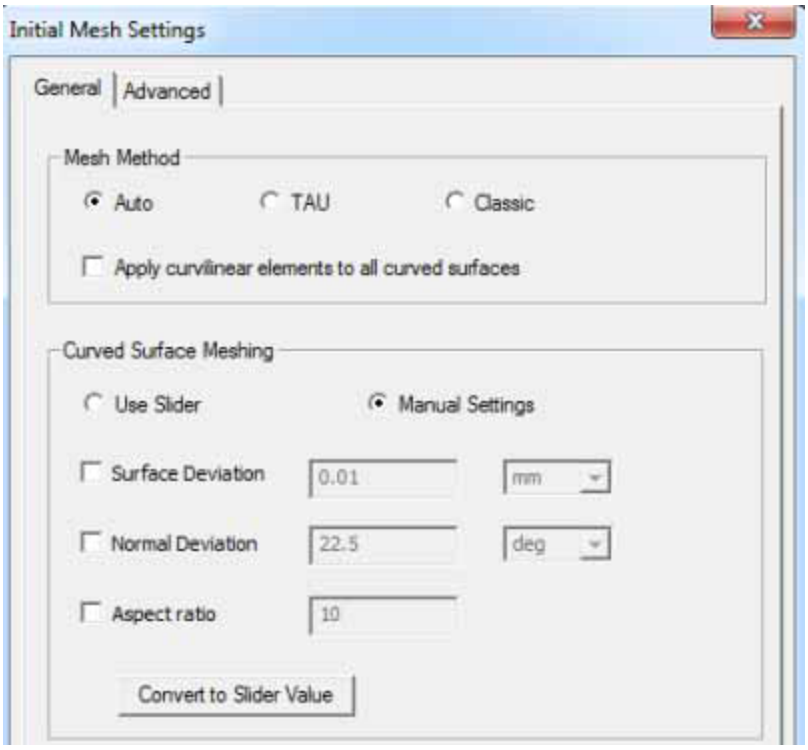
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Defining a Model-Level Mesh Operation

A model-level mesh operation is applied to all objects in the model. To illustrate, a model that has two cylindrical objects is used. For model-level mesh operation, the two important controls for the Manual settings are **Normal Deviation** and **Surface Deviation**. Model-level operations with respect to these **Manual Setting** controls are described below.

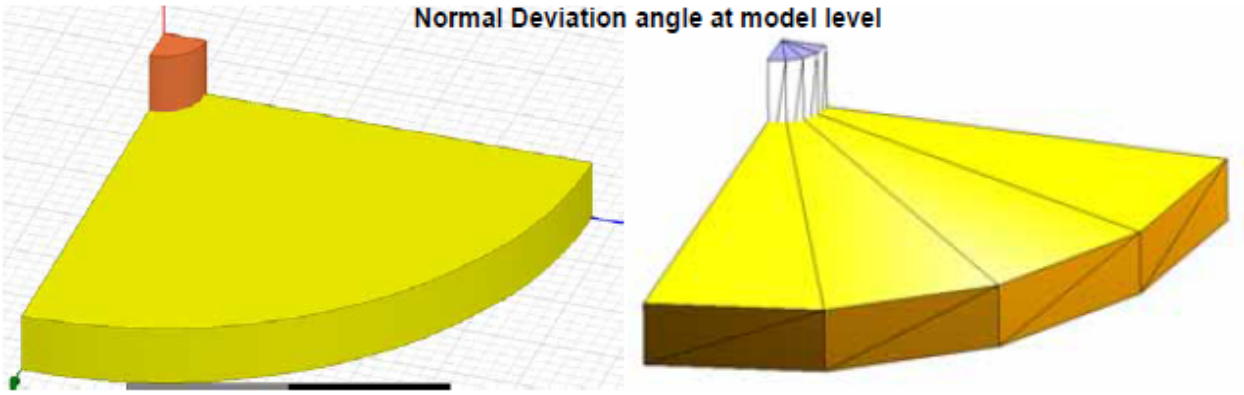
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Normal Deviation Angle

By default, **Normal Deviation** is 22.5 degrees in HFSS. When specifying model-level mesh operation, consider the sizes and shapes of all curved objects to obtain the desired faceted triangulation model for the mesh.

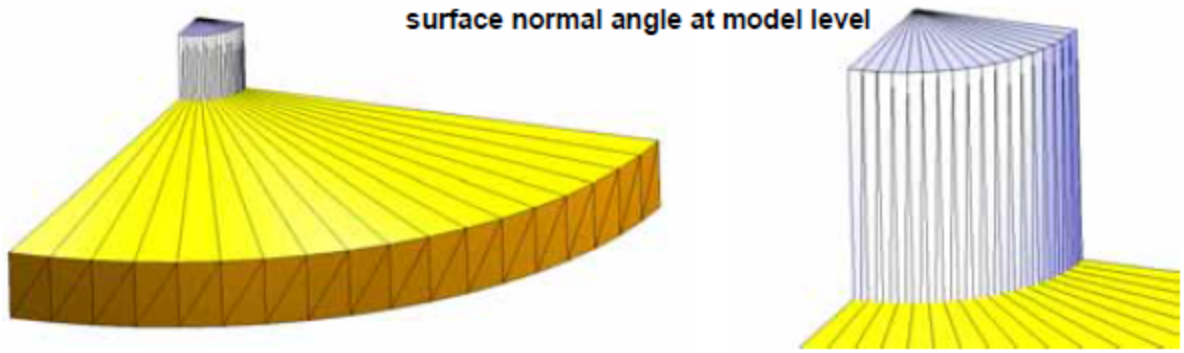
The following illustrations show a model that has two cylindrical objects.



In HFSS, the 90 degree sweeping curves are faceted as 4 segments since **Normal Deviation** is 22.5 degrees by default. If you specify the surface normal angle as 5 degrees at the model level, there are 18 segments on both cylindrical curved surfaces as illustrated in the following figures.

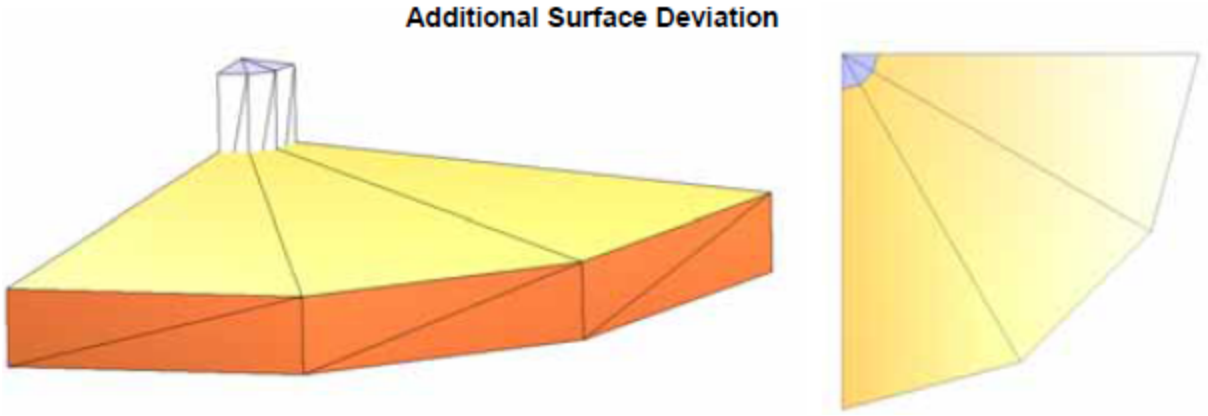
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Such a level of granularity is useful for large curved surfaces, but for small curved surfaces, it creates an extremely dense mesh. Therefore, use caution to assign a model-level mesh operation with small **Normal Deviation** unless all the curved objects in the model have equivalent curvatures and sizes.



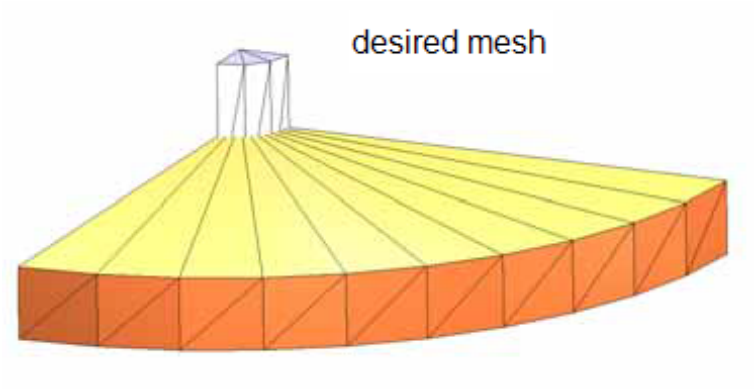
Maximum Surface Deviation Length

An appropriate value of **Surface Deviation** supplements the **Normal Deviation** setting. When you set **Surface Deviation** in addition to the **Normal Deviation** setting, it can further improve the geometry representation. By specifying **Surface Deviation** you can control the difference between the faceted face and the real curved face. **Surface Deviation** is defined in user unit. Consider the same model with two cylindrical objects. In this mode, if you specify 30 degrees as the **Normal Deviation**, there are 3 segments on both cylindrical surfaces as expected.



Since the radius of the large cylindrical surface is 6mm, set 0.06mm as the **Surface Deviation** in the model-level mesh operation. This ensures we have the desired faceting geometric model.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To summarize, the **Surface Normal Angle** of 30 degrees provides one facetation of the geometry, and the **Surface Deviation** of 0.06 mm controls the difference from the real curvatures. Since the small faceted cylindrical surface already has the difference smaller than 0.06 mm, the surface is maintained as is. However, since the large cylindrical surface has a gap greater than 0.06mm, the facetation is made finer to adhere to the specified length.

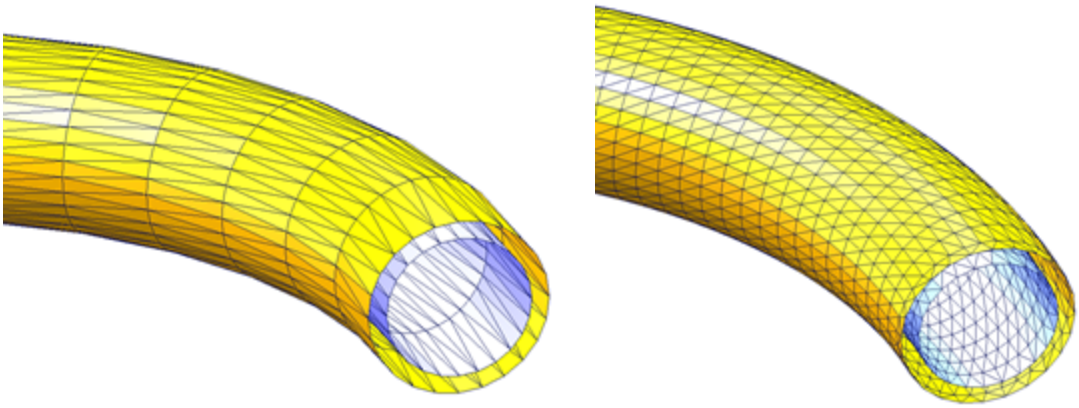
As a rule of thumb, large surface normal angle can control the small curved objects while small Surface Deviation can control the large curved objects in the model-level mesh operation. **Surface Normal Angle** is effective for skewed curved surfaces.

Aspect Ratio

Aspect Ratio *is the ratio of the longest length of an element to its shortest length.* **Aspect ratio** controls the faceting triangle ratio. It is needed only when you want a highly uniform mesh on the curved surfaces or when the curved object is either thin or has a narrow gap. By default, **Classic** mesher uses 20.0 as the **Aspect Ratio** whereas **TAU** mesher uses 8.0 as the **Aspect Ratio**.

When there is a narrow gap between two curved surfaces, a small value of aspect ratio ensures the gap to be uniform. The figures below illustrate a geometric model with the aspect ratio of 20 and 2. Smaller aspect ratio ensures better representation of the curvature for skewed curved surfaces, especially in thin curved layers.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



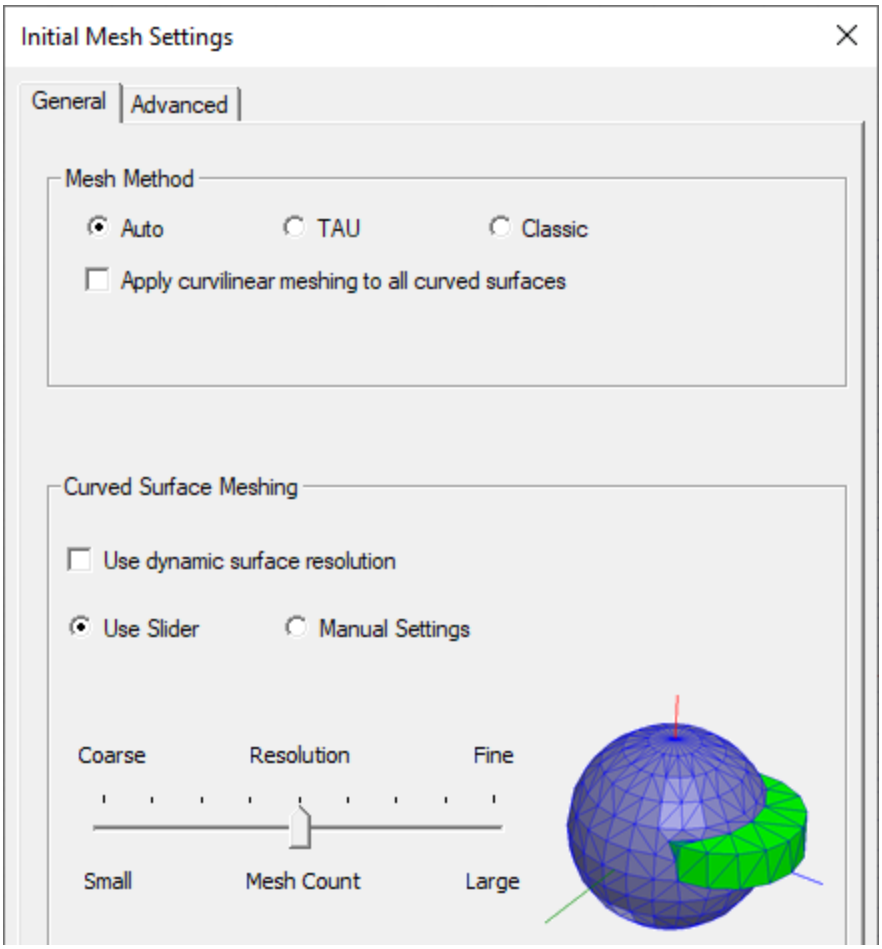
Use caution when assigning a small aspect ratio, since it will produce a dense mesh at many other curved surfaces which may not require the denseness.

Initial Mesh Settings

The initial mesh settings dialog box has the following panels:

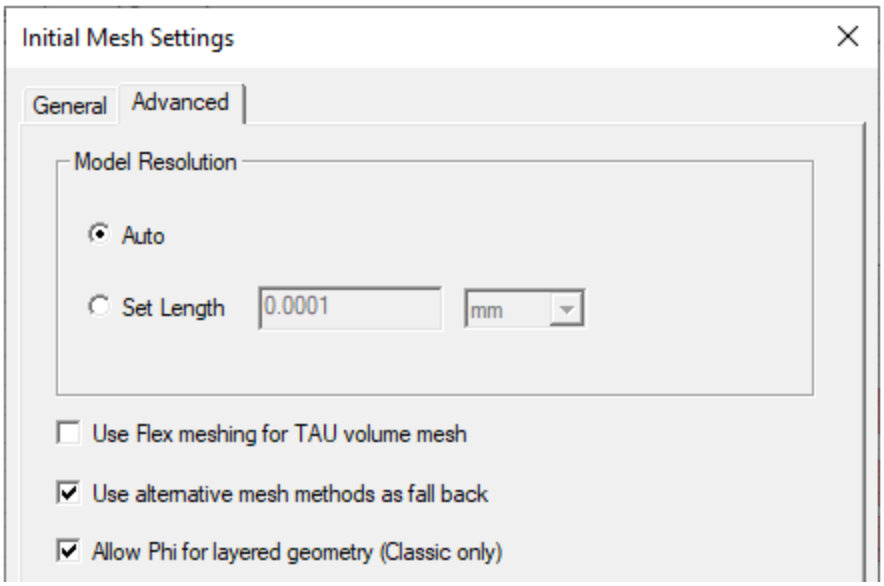
- Curved Surface Meshing
- Meshing Method

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Advanced** tab lets you optionally specify a **Set Length** for **Model Resolution**. This is for experienced users who have a good understanding of how particular values will affect their models. In general, the **Auto** setting provides good results.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For details, on using these features, see Specifying Initial Mesh Settings.

Meshing Method

HFSS has four meshers available to generate the initial mesh: the **Auto**, **Tau**, **Classic**, and **Phi Plus** meshers.

The **Auto** selection automatically chooses a suitable mesh method for the design based on geometry characteristics (e.g., if the design contains bondwires, **Phi Plus** is selected).

The **Classic** mesher uses a Bowyer algorithm to create compact meshes for the model with very large length scales. It represents the model very accurately.

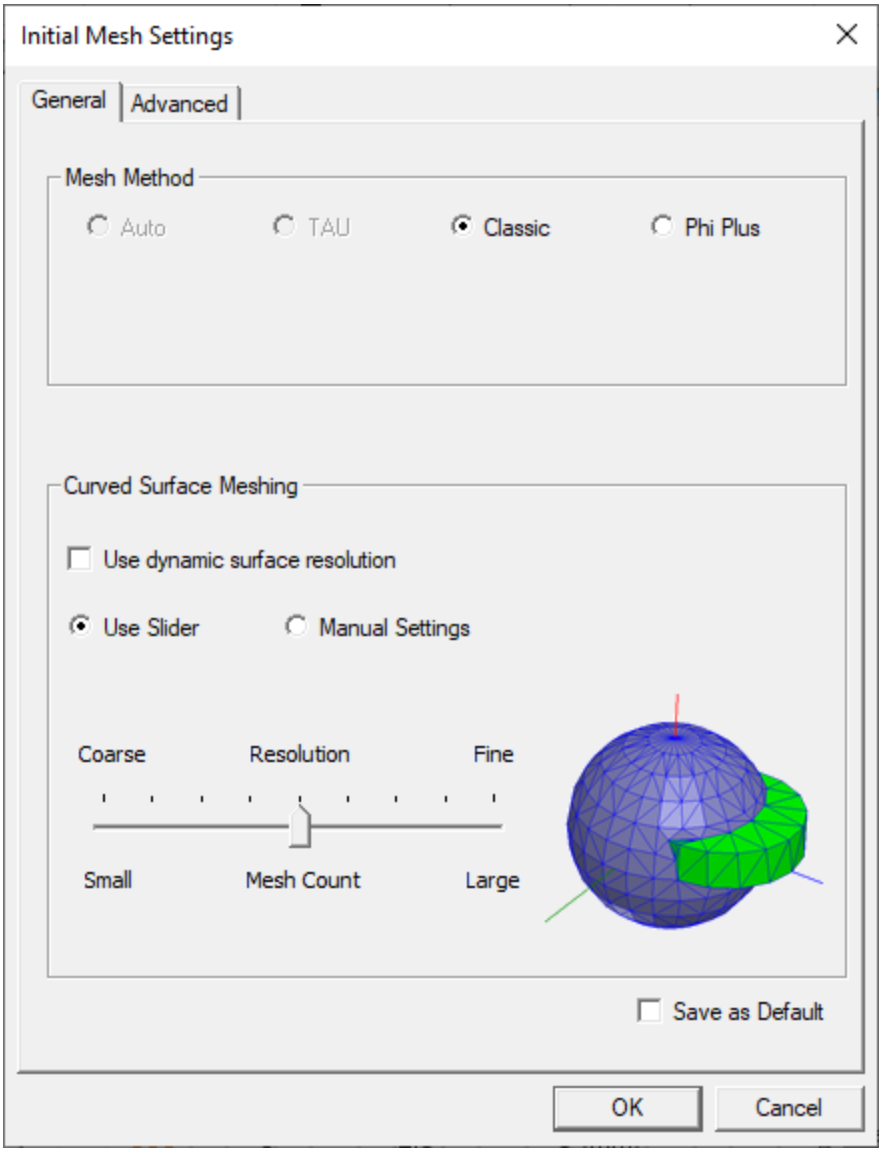
The **Tau** mesher produces very high quality meshes. It uses a volume decomposition method to create the initial mesh and coarsens it to achieve small element count. It has the ability to adaptively relax the tolerance to overcome the inaccuracies and misalignments in the input model. This mode of operation is called the **Tau** tolerant. The other version is called **Tau** strict.

Both meshers are designed to handle all geometries. However, if you select **Auto**, HFSS picks either the **Classic** or **Tau** mesher based on the analysis of the geometry. You can override the selection and choose either **Classic** or **Tau** to make the initial mesh. Depending on the model, sometimes **Classic** converges with fewer resources whereas **Tau**may converge faster. Whichever mesh is selected, the converged results are accurate. Ansys recommends selecting **Auto** since HFSS automatically determines the correct choice of the mesher for a given geometry.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important:

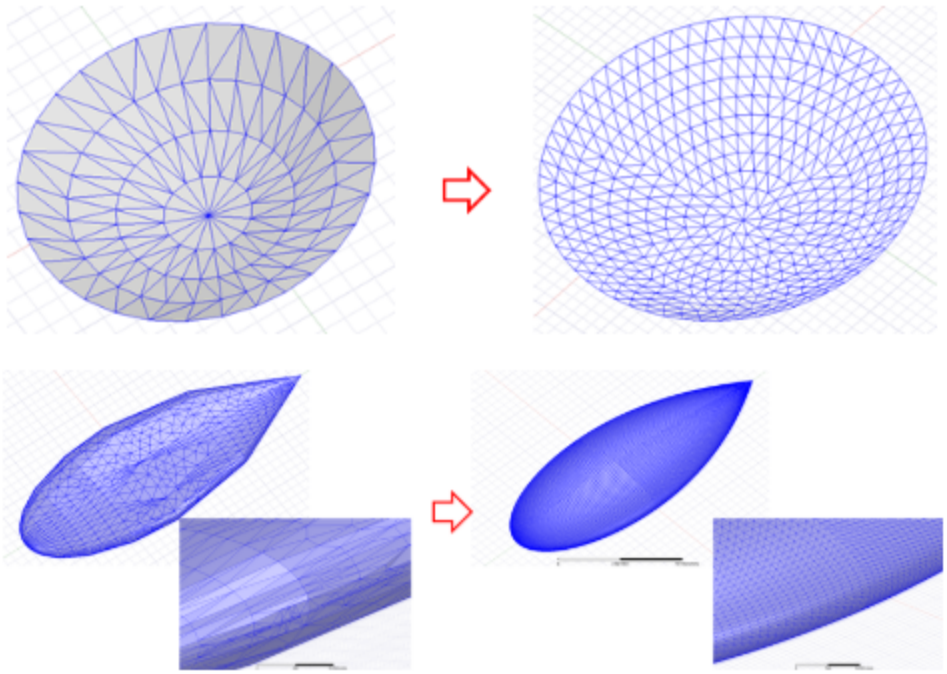
The consolidation of surfaces into a conformal mesh is skipped for dynamic and tolerant meshing, including for light weight geometries. This can lead to overlapping surfaces in SBR+ simulations. The user should carefully avoid overlapping surfaces or objects as SBR+ can produce unexpected results.



If the model contains IE Regions, the Tau mesher uses a surface mesh routine to detect which curve faces should be remeshed, rather than using the faceting triangles. This produces a high quality mesh, uniform, including the top, sides and bottom. (See the following figure for

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

examples of the benefit.) Other faces that cannot be remeshed still get faceting triangles. In the final output, the remeshed surface mesh and faceting surface mesh are coupled to ensure the mesh is watertight and conformal.

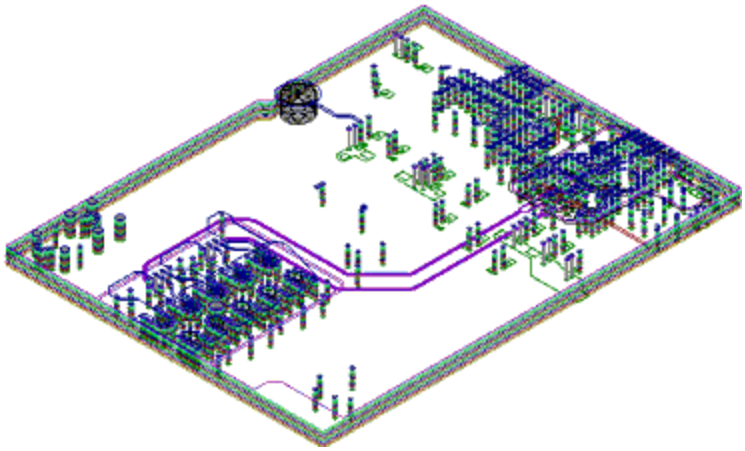


The Phi Mesher

The Phi mesher is a layout-based meshing technology, available in the HFSS 3D Layout interface. This advanced meshing technology is capable of rapidly generating an initial mesh ensuring faster simulations that can be further accelerated and enhanced by using high performance computing.

While the traditional Classic mesher is suitable for HFSS 3D MCAD and for certain geometries solved in HFSS 3D Layout, the Phi Mesher works on the HFSS 3D layout interface where principally 2.5D geometry is comprised of z-extruded layers that have no oblique facets. Therefore package models, for example, containing bondwires cannot take advantage of this meshing technology. Phi mesher uses a unique method to generate the initial mesh from the a priori knowledge of the layered nature of the geometry. It generates a volume mesh directly from the layout data saving tremendous amount of time, requires less user clean up, and eliminates several situations that might cause potential kernel errors in the traditional 3D modeler flow. A typical complex layout package is shown below.

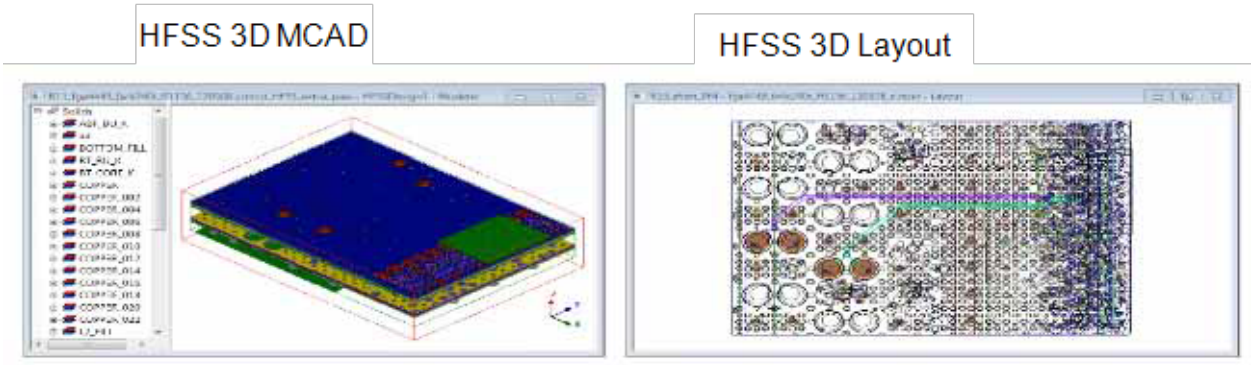
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When compared to the traditional Classic mesh, the Phi mesher is several orders of magnitude faster, including from the initial meshes the adaptive meshing algorithm produces to the final convergence. Another advantage of the Phi mesher is its ability to mesh larger and more complex designs previously unmeshable from the traditional 3D interface or with the Classic mesher in the 3D layout interface. In cases where Phi produces large meshes, you can use the HPCsolver feature to leverage more cores and distributed memory to neutralize the time that would have been otherwise required for the simulation to complete.

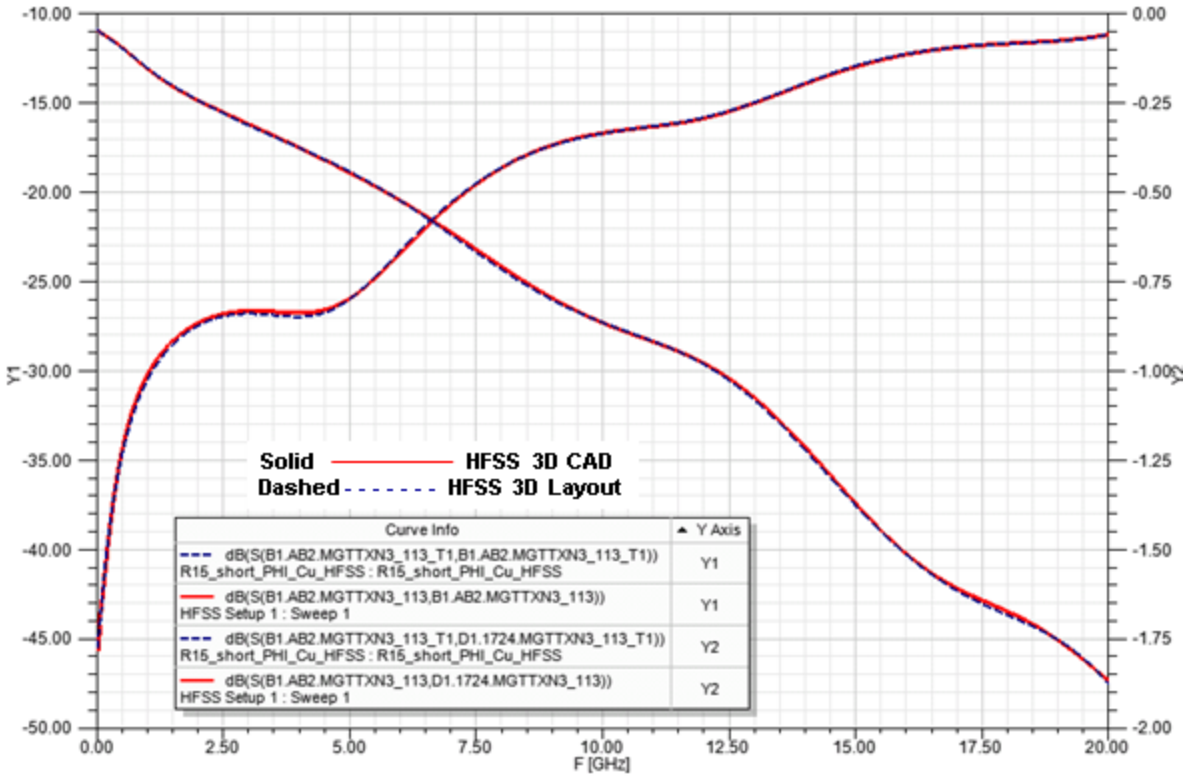
Phi Mesher for Multilayer Complex Package Layout

In this section, the simulation of a multilayer complex package layout is used to demonstrate the feature of the Phi mesher in HFSS 3D layout. To illustrate the advantages of a Phi mesher, a task-based comparison is performed between a regular HFSS 3D MCAD and the same geometry which is imported, set up, and solved through the HFSS 3D Layout GUI.



The same HFSS solver engine is used to solve both the geometries.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The HFSS MCAD results marked in red agree with the HFSS 3D Layout results in dashed blue.

The comparison of the two solutions is shown in the following table.

	HFSS 3D CAD	HFSS 3D Layout
Initial Mesher	Classic	Phi
Initial Mesh Size (tets)	763, 953	712, 536
Initial Mesh Time	1hr 20min	4min
Number of Adaptive Passes	6	7
Number of Tets - Final	1, 617, 954	1, 620, 969
Adaptive Passes Run Time	4hrs 1min	1hr 5min
RAM	47.6 GB	49.6 GB
HFSS Model number of Ports	8	8

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initial mesher in the HFSS 3D CAD interface takes 1 hour and 20 minutes to generate the initial mesh. The entire simulation takes just over 4 hours to complete in 6 adaptive passes, and the total number of tetrahedra generated in the final mesh is 1.6 million.

On the other hand, the Phi initial mesher creates a very comparable initial mesh size of 712000 tetrahedra in only 4 minutes. The entire simulation takes one extra pass to solve; however, for the same number of solved tetrahedra, 1.6 million, it required only 1 hour and 5 minutes. This example shows the benefits of using the HFSS 3D layout and its ability to leverage the Phi mesher to mesh an extremely complex geometry. Typically, the Phi mesher requires a small amount of time ranging from only 5 to 10 minutes to create the initial mesh. The Phi mesher provides a great advantage for solving any arbitrary package or PCB layout structures.

Troubleshooting Mesh Failures

Common causes of mesh failures are improper geometry, incorrect design, or mesh settings. The following checklist can be used to address mesh failures and ensure that a suitable mesh is created to solve the problem:

1. Select the model units in such a way that the smallest critical features in a design are at least 0.001 units long. Critical features include traces, gaps, vias, and contacts between objects.
2. Override the **Auto** option on the **Initial Mesh Settings** panel to see if that helps. For example, set **Classic Mesh** as the initial mesh if the **TAU Mesh** fails and vice versa.
3. Specify an appropriate value for the option **Surface deviation (length)** on the **Surface Approximation** panel (Manual Settings), if spline surfaces are present in the geometry. Typically, five percent of the thickness of the bounding box of the surface is a good choice.
4. Either tighten or loosen the settings on the **Surface Approximation** dialog box to check if that helps.
5. Select the **Mesh Feedback** tab on the **Model Analysis** dialog box to view the error log reported by the mesher. Go to **Modeler > Model Analysis > Show Analysis Dialog > View Mesh Feedback** to access this tab.

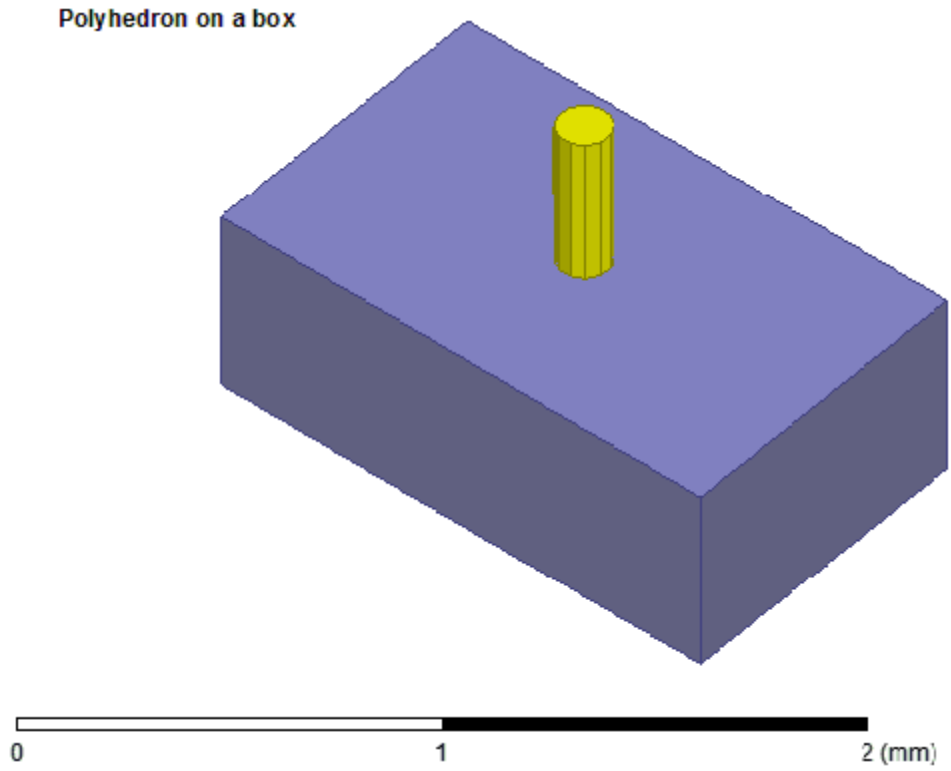
The HFSS mesher can automatically repair and correct a poorly translated and healed model to generate a reliable mesh. You can adjust the surface approximation settings if you want to further modify the corrections. The **Profile** reports when mesh repairs are made and the results of these repairs are displayed for every object in the mesh statistics panel.

The following sections provide troubleshooting tips to correct potential mesh failures on simple geometries:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

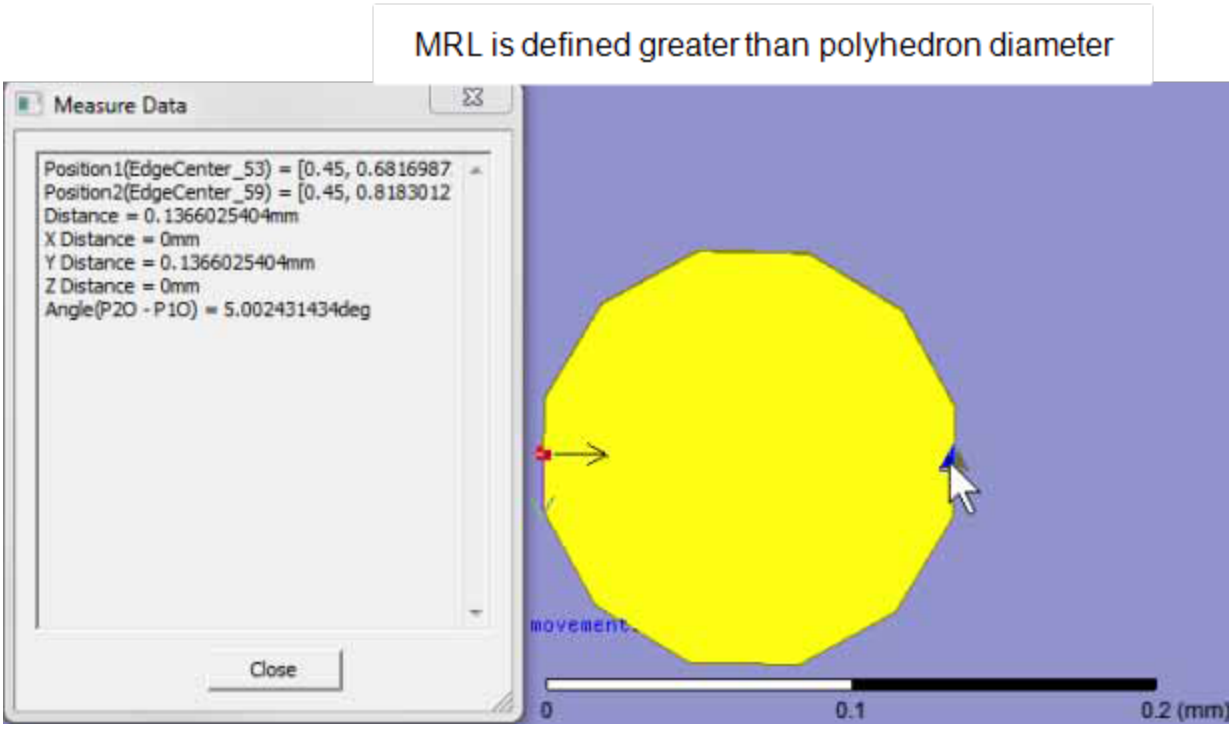
Scenario 1:Model Resolution has Caused Significant Changes in Object Dimensions

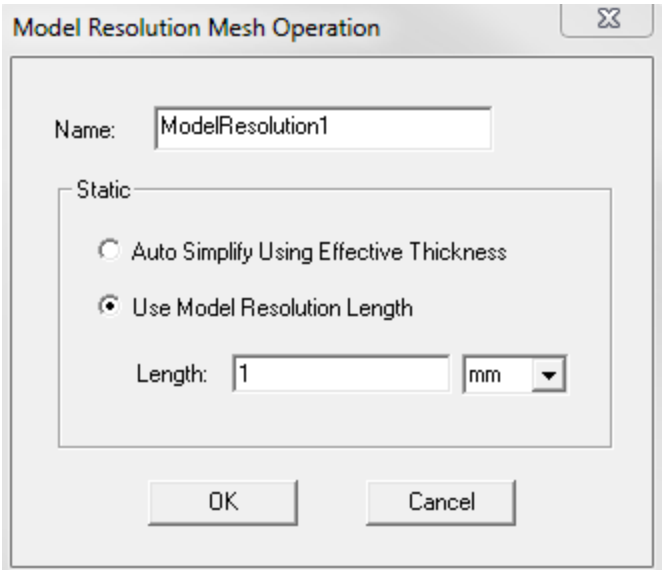
The following figure illustrates a scenario where improper MRL length can cause mesh failures. The geometry consists of a regular polyhedron on the top of a box.



The diameter of the polyhedron is 0.136 mm and the **Model Resolution Length** defined for the polyhedron is larger than the diameter itself as shown in the following figure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





HFSS generates the following warning messages:

Model Resolution has caused significant changes in object dimensions or contact between objects. Please compare the MRL set on the bodies with the dimensions of the bodies. Please ensure the unites selected such that the smallest critical features of the model are at least 0.001

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

units long. Specific details may be found in Modeler > Model Analysis > Show Analysis Dialog > View Mesh Feedback.

Geometry stitch failed. Failing back to backup process.

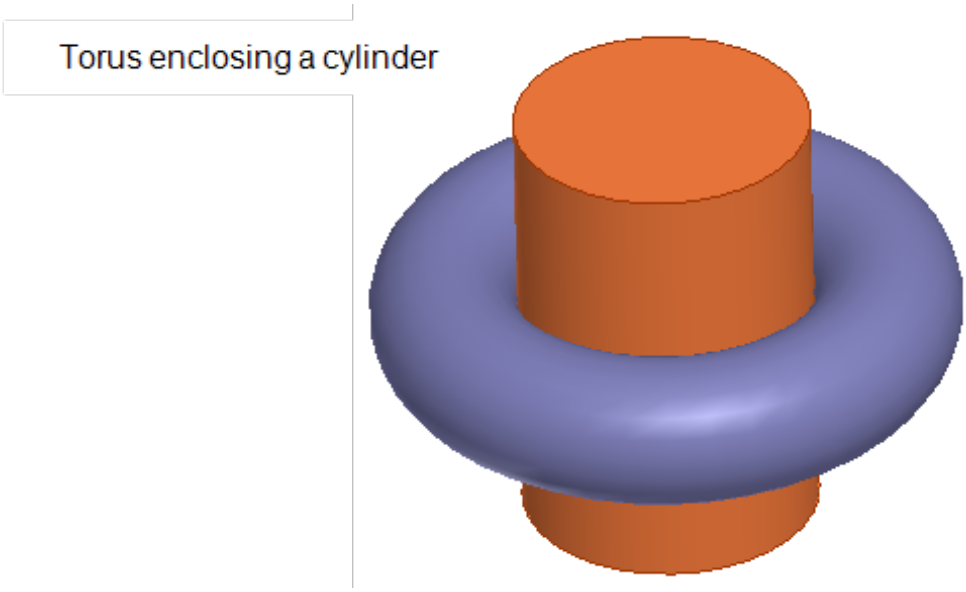
To diagnose the problem, first see the **Model Analysis** dialog box. For more details of the error type, select the **Display Mesh Error Log** check box.

For this particular geometry, a model resolution length of 0.045mm (one-third of the diameter of the polyhedron) created the correct mesh.

In general, select the model units such that the smallest critical features are at a minimum 0.0001 units long to address such warning messages. For a circuit board with a 1-micron trace width and vias, solder balls, bondwires, and gaps, each greater than 1 micron, the smallest critical feature is the trace; if the model is drawn in mm, the smallest critical feature can be 0.001 mm. However, if the model is drawn in centimeters, the smallest critical feature is 0.0001cm, which is way below the minimum permissible value 0.001. Such improper unit selections can cause mesh failures. Choose the units such that the entire solution domain is less than 1000 units.

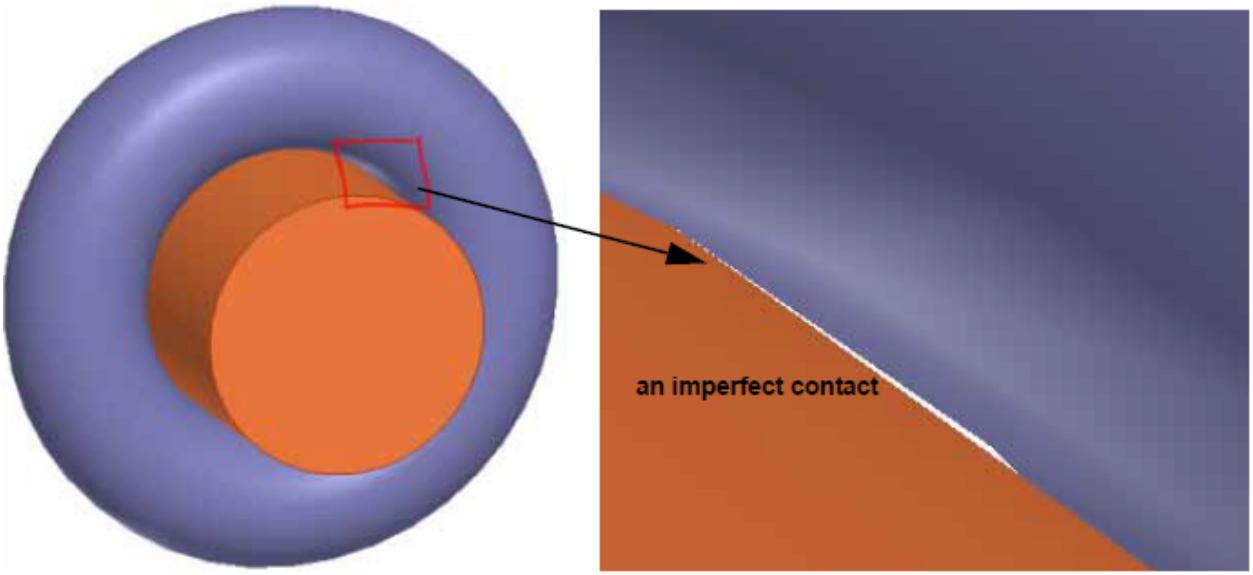
Scenario 2: Contact Area Change Too Large

Imperfect tangential contacts between curved surfaces can cause mesh failures. In the following figure, the radius of the cylinder is 0.1 mm, the major radius of the torus is 0.15 mm, and the minor radius is 0.05mm.

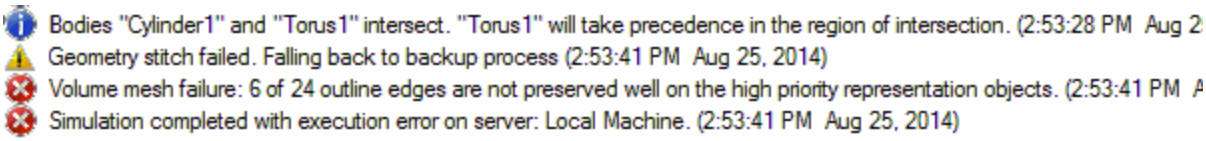


The axis of the cylinder is slightly tilted relative to the axis of the torus causing significant change in contact between objects.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

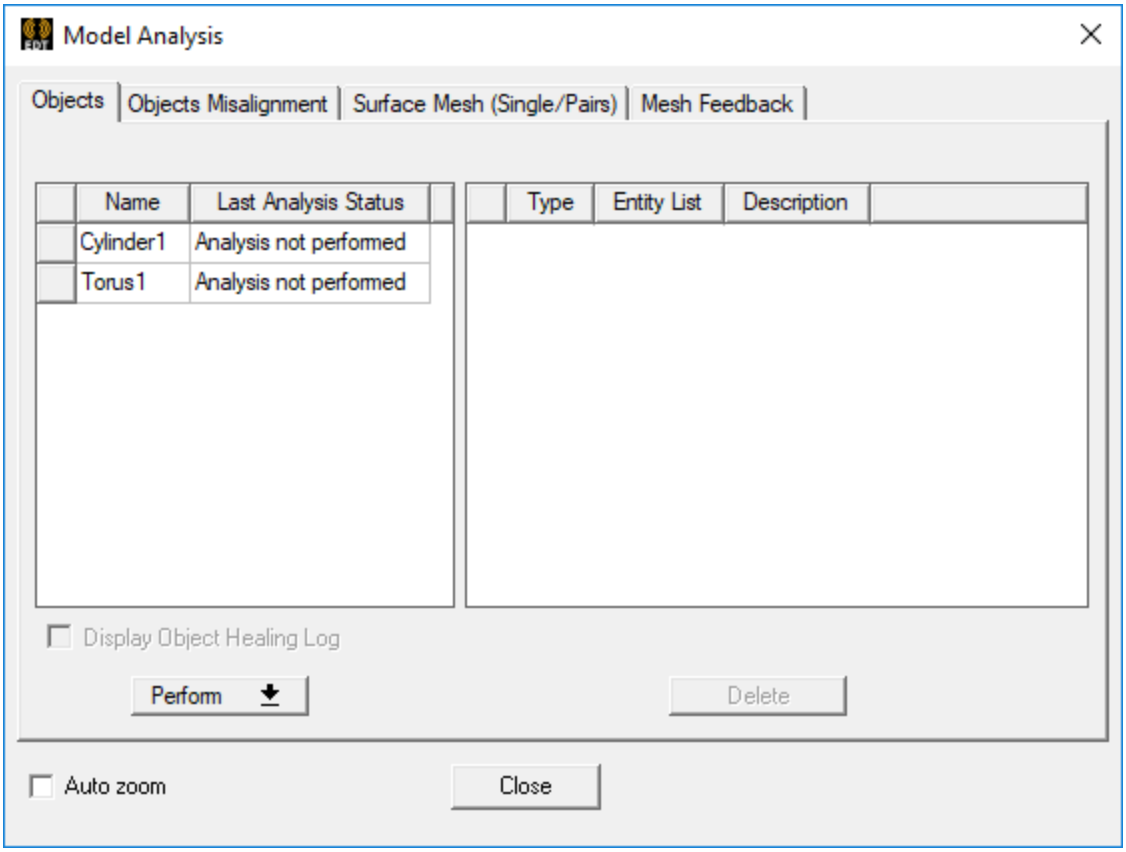


A mesh cannot be generated for this incorrect geometry. When analysis of this design is attempted, the following informational, warning, and error messages are reported in the Message Manager window.



On the **Model Analysis** dialog box, the last analysis status for each of the objects is listed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For the simulation to complete successfully, the axis of the cylinder relative to that of the torus must be aligned properly.

UI Controls for Mesh Operations

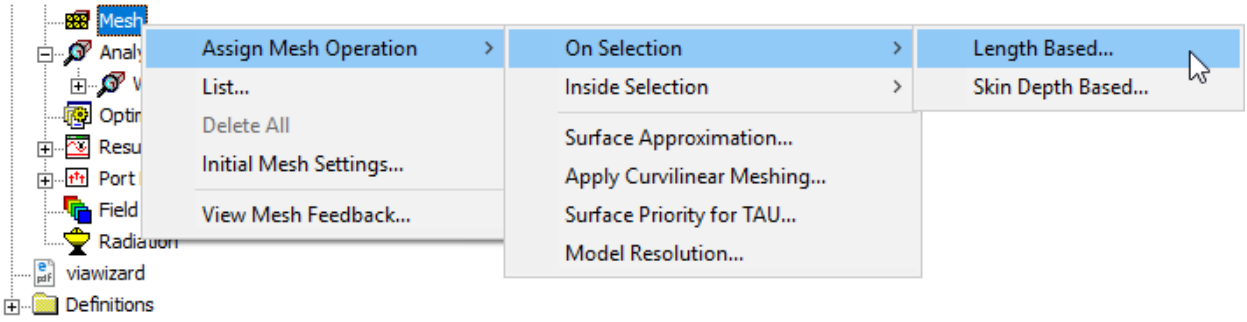
This section describes the various UI controls to access and define mesh operations. Each of these controls is described below.

Mesh Operations

You can define **Mesh Operations** from various places. From the Project Manager window, define mesh operations as follows.

1. Press F to enter face-selection mode, and select the surface of interest.
2. In the **Project Manager** window, right-click **Mesh** to access all the controls.

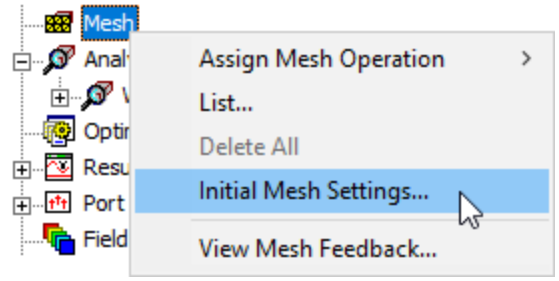
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Mesh Operations are also available on the **HFSS** menu item.

Assign Curvilinear Elements

To apply curvilinear elements on a surface, right-click **Mesh** and select **Initial Mesh Settings** from the short-cut menu as shown in the following figure.



The **Initial Mesh Settings** dialog box has the controls for **Surface Approximation**.

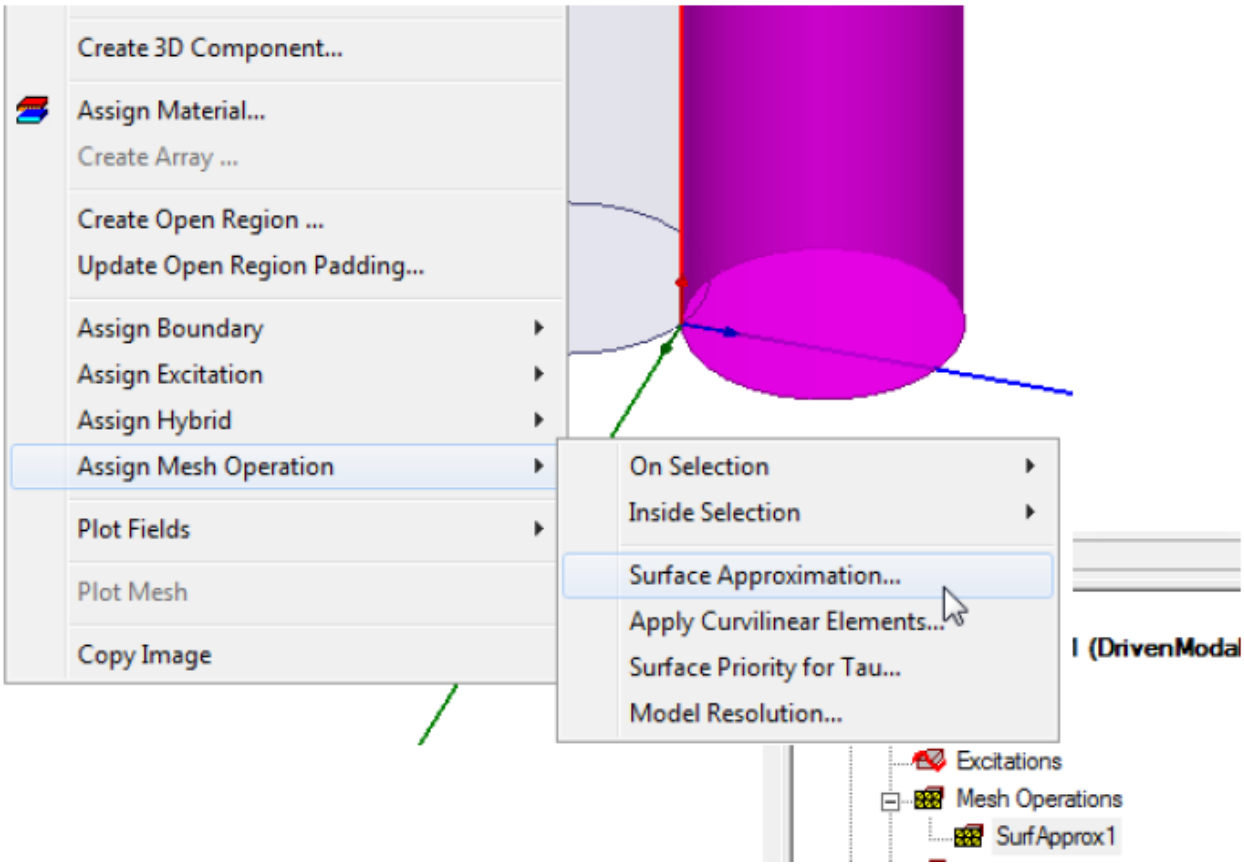
Access Surface Approximation (Object Level)

To access **Surface Approximation** at the object level, follow these steps:

1. Click **Edit** > **Selection Mode** > **Object** to enter object-selection mode.
2. Select the objects for which you want to apply **Surface Approximation**.
3. Right-click in the modeler field and select **Assign Mesh Operation**>**Surface Approximation**

   The **Surface Approximation** dialog box appears.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

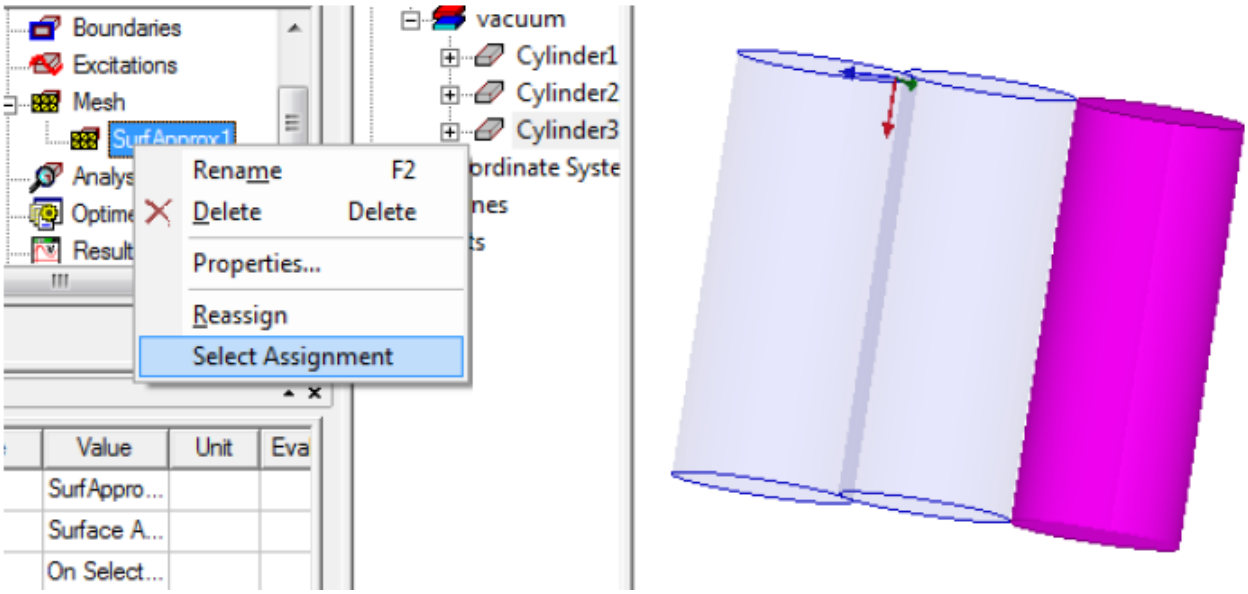


Note:

Surface Approximation performed at the object level overrides Surface Approximation performed at the model level. Also, once you define Surface Approximation at the object level, the setting name appears under **Mesh** in the Project Manager window. In the figure it is the default name SurfApprox1. If you want to apply Surface Approximation on specific surfaces of the object, you must be in Face Selection mode.

To determine where in the geometry Surface Approximation setting was applied, right-click the particular surface approximation setting of interest from the project tree and choose Select Assignment from the short-cut menu as shown in the following figure. This feature is especially useful when there are several surfaces that are assigned these approximation settings and you want to see which setting corresponds to which surface in the geometry.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Mesh Assembly

With mesh assembly you can specify that multiple design modal FE-BI or IE/PO, SBR+ or FEM components and native regions are meshed independently to ensure very robust and rapid meshing. If a design contains multiple instances of a component, that component need only be meshed once. Parametric variations for one or more instances only affects the mesh of those instances.

The solver supports the following types of mesh assembly components.

- Volume component containing FEBI boundary
- Surface component (IE/PO)
- SBR+ component
- Volume components without FEBI.

Some of the benefits of mesh assembly are as follows:

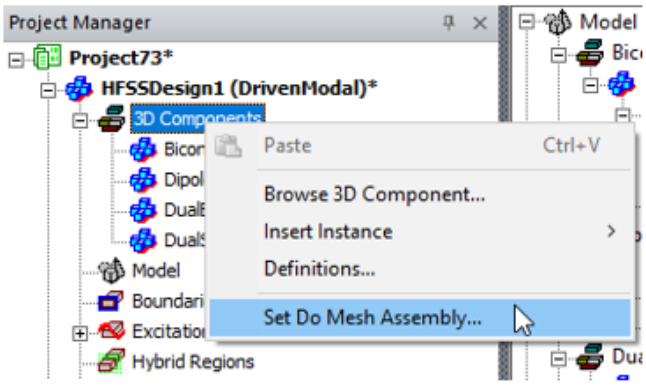
- Systems with large variation in length scales can be successfully meshed. For example, in the case of a small 3D antenna touching an aircraft, you can mesh both the bodies separately and ensure a conformal mesh between them. That is, the triangulations in the contact area between the volume mesh of the 3D antenna component and the surface mesh of the aircraft are aligned resulting in a conformal mesh.
- You can select different meshers for each component via the **Component Meshing** properties. That is, you can pick the Tau mesher for a given component and the Classic mesher for another component, in order to ensure system meshing is successful.

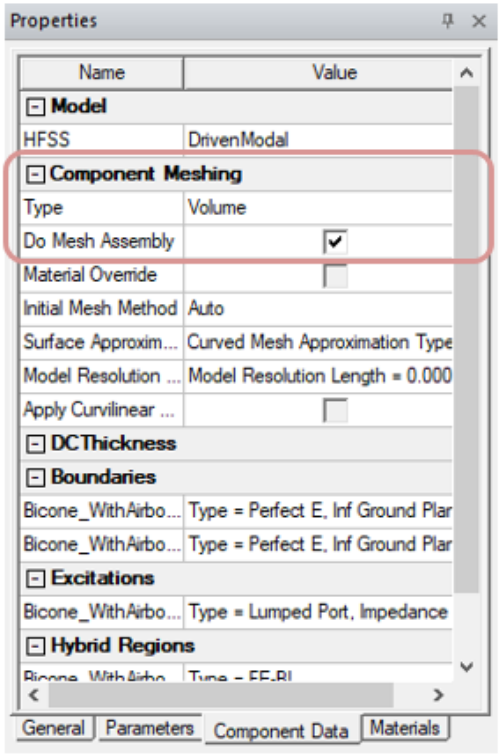
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Faster meshing for multiple instances of the same component is possible: only one initial mesh is created. For instance, when multiple identical antennas appear in a design, only one needs to be meshed.
- Mesh resuse enables fast placement studies. If you move a component, no re-meshing needs to be done. Also only the last adaptive mesh is used and no adaptive meshing of the component needs to done.
- Reused initial meshes are not impacted by any parametric variation. Suppose there are multiple components that are independently meshed and you make a parametric variation in one component, the remaining components are not remeshed. In this situation, the last adaptive mesh cannot be reused. Some examples of parametric variation can include increasing or decreasing the dimension of a geometry or changing the material properties.

- Mesh Assembly can be distributed.

You can access the **Do Mesh Assembly** feature through the **Project** tree shortcut menu for **3D Components**, through the **Advanced** tab of the Solution Setup, or through **Component Meshing** properties.

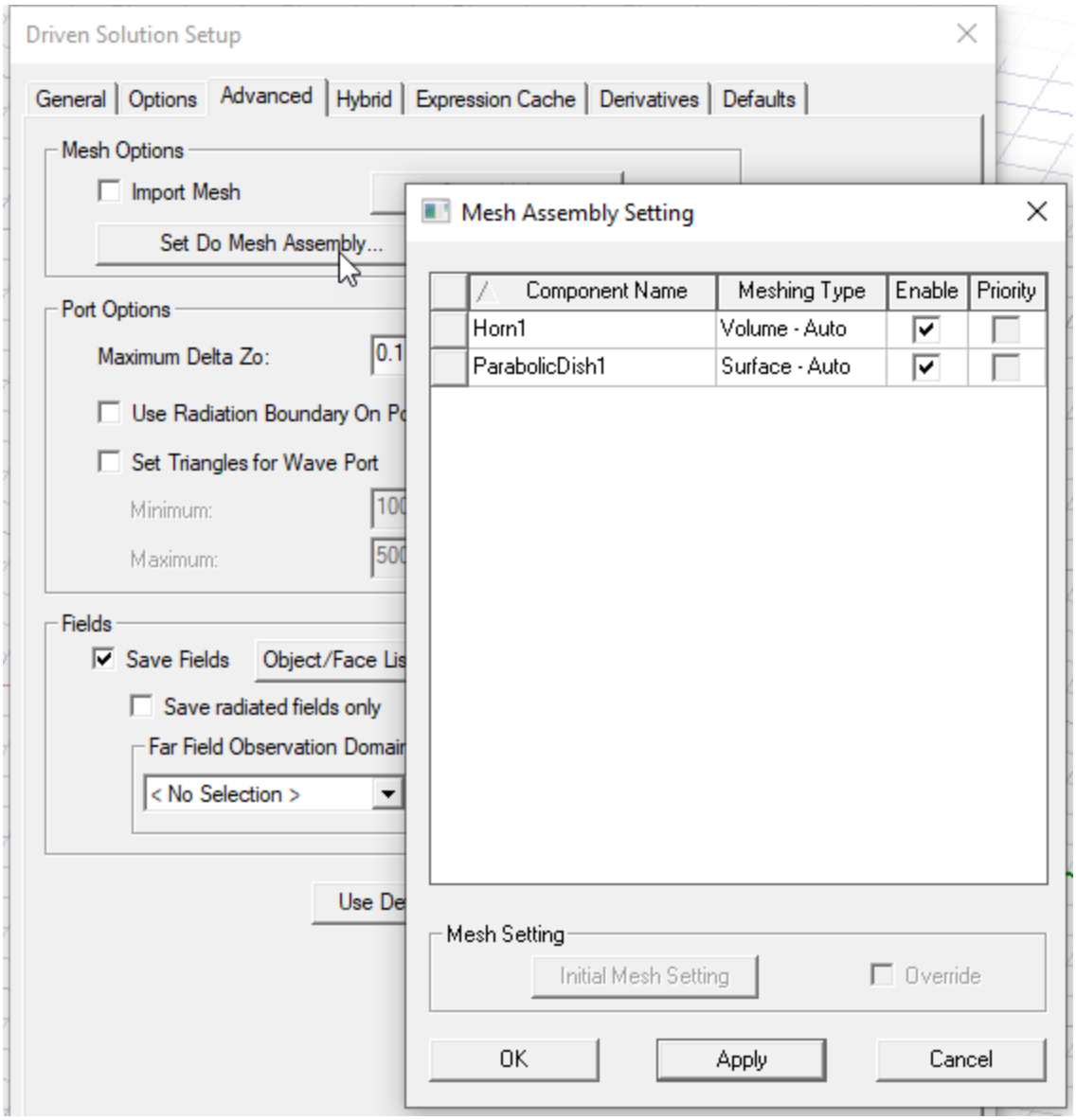




Selecting **Set Do Mesh Assembly...** opens a dialog which lists eligible components. Selecting a component enables the fields.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also access the **Do Mesh Assembly Setting** dialog through the **Advanced** tab of the Solution Setup.



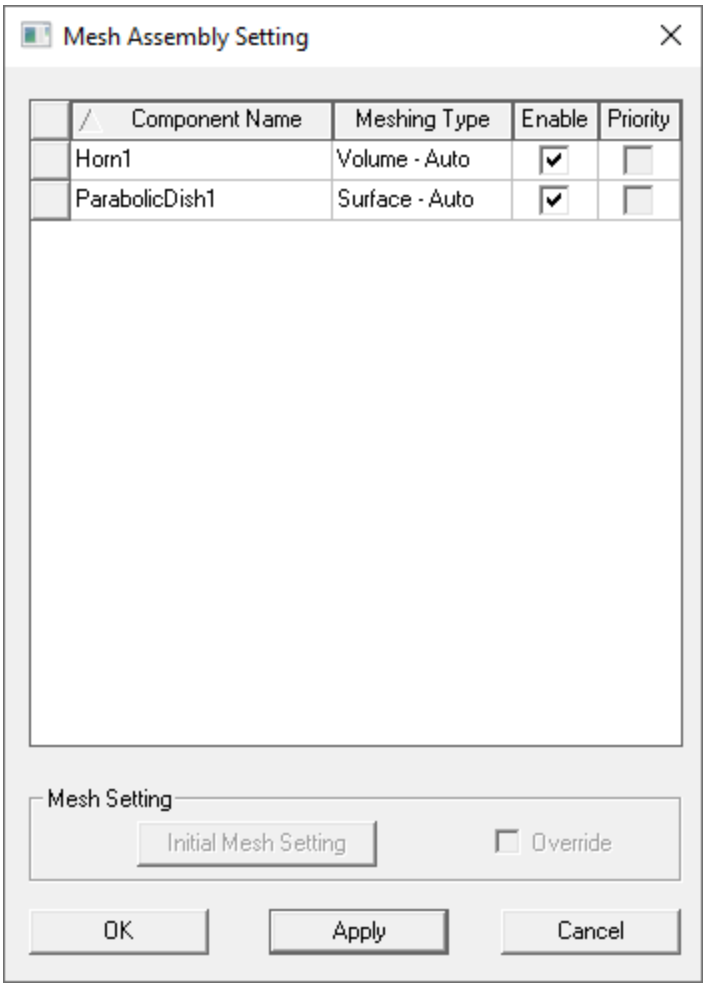
If a component does not meet the requirement for mesh assembly, the feature is disabled in the **Do Assembly Setting** dialog and the **Component Meshing** properties.

Mesh Method Override

You can override the mesh method from the component definition in the **Do Mesh Assembly Setting** dialog box, accessed either through the 3D Component shortcut menu in the **Project** tree, **3D Components>Set Do Mesh Assembly**, or the **Advanced** tab of the an Advanced

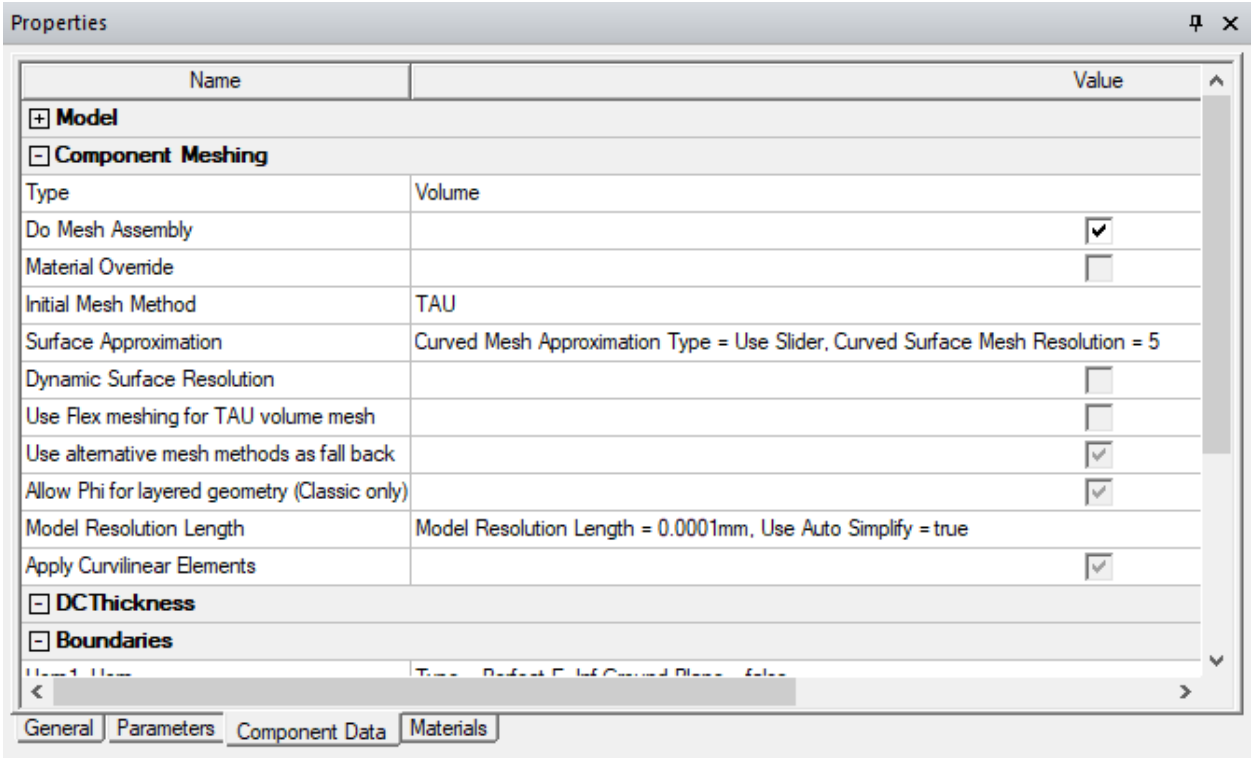
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution Setup. This lets you deal with potential mesh failure based one specific mesh method when doing assembly meshing. If a failure happens using one method, you can easily switch the mesh method and try again. Selecting a component in the **Do Mesh Assembly Setting** dialog enables Override checkbox. Checking the box enables the **Intial Mesh Setting** button, which opens the **Initial Mesh Settings** dialog box, where you can select and Apply, Auto, TAU, or Classic. For convenience, the dialog also displays each component's meshing method in its Meshing Type column. The Component Priority checkboxes will be enabled if the components are valid for that feature.



The overriding Component Mesh settings are shown in the Component **Properties** window, Component Data tab.

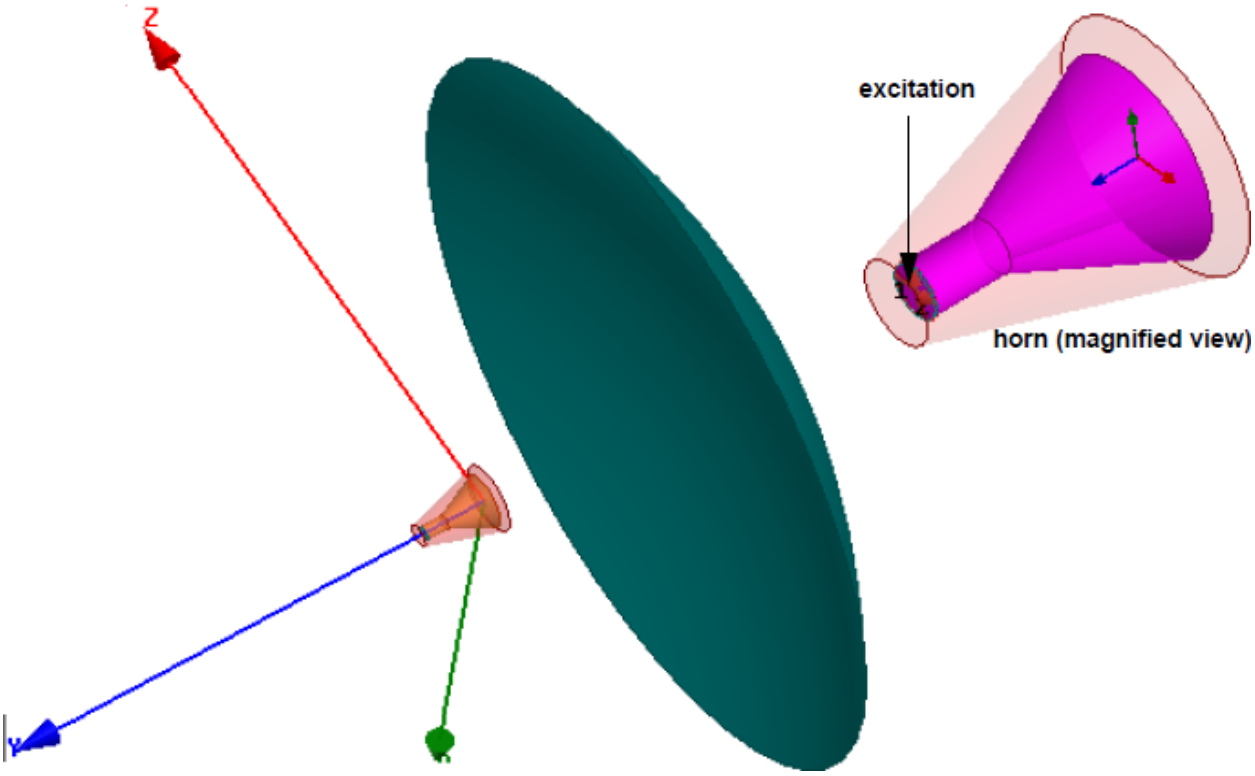
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



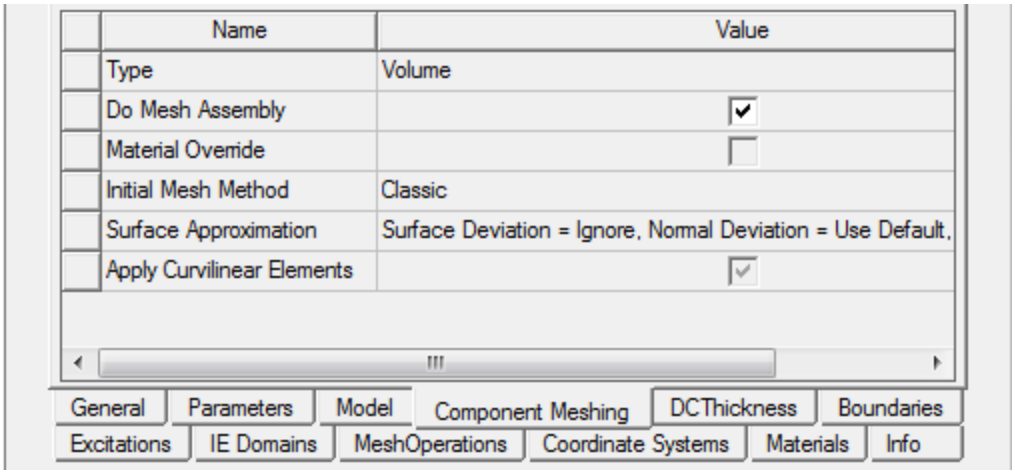
Example: Horn Antenna and a Dish

This example shows an electrically large problem for the placement study of a horn antenna illuminating a dish, when the antenna is moved along the y-axis. For such designs, meshing the horn and the dish separately makes the simulation faster since only the currents on the dish and on the radiation boundary of the horn need to be modeled. Meshing the volume of the region between the horn and the dish to model coupling occurring due to the currents on the dish and the radiation volume bounding the horn is not required. With mesh assembly, you can reuse the meshes of the individual components for performing placement studies. The diameter of the reflector is 2000 mm and the adapt frequency is defined to be 5 GHz. A design variable offset is defined to vary the placement of the horn along the y axis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To use the mesh assembly feature, select the **Do Mesh Assembly** option for the individual components of the design as shown in the following figures:

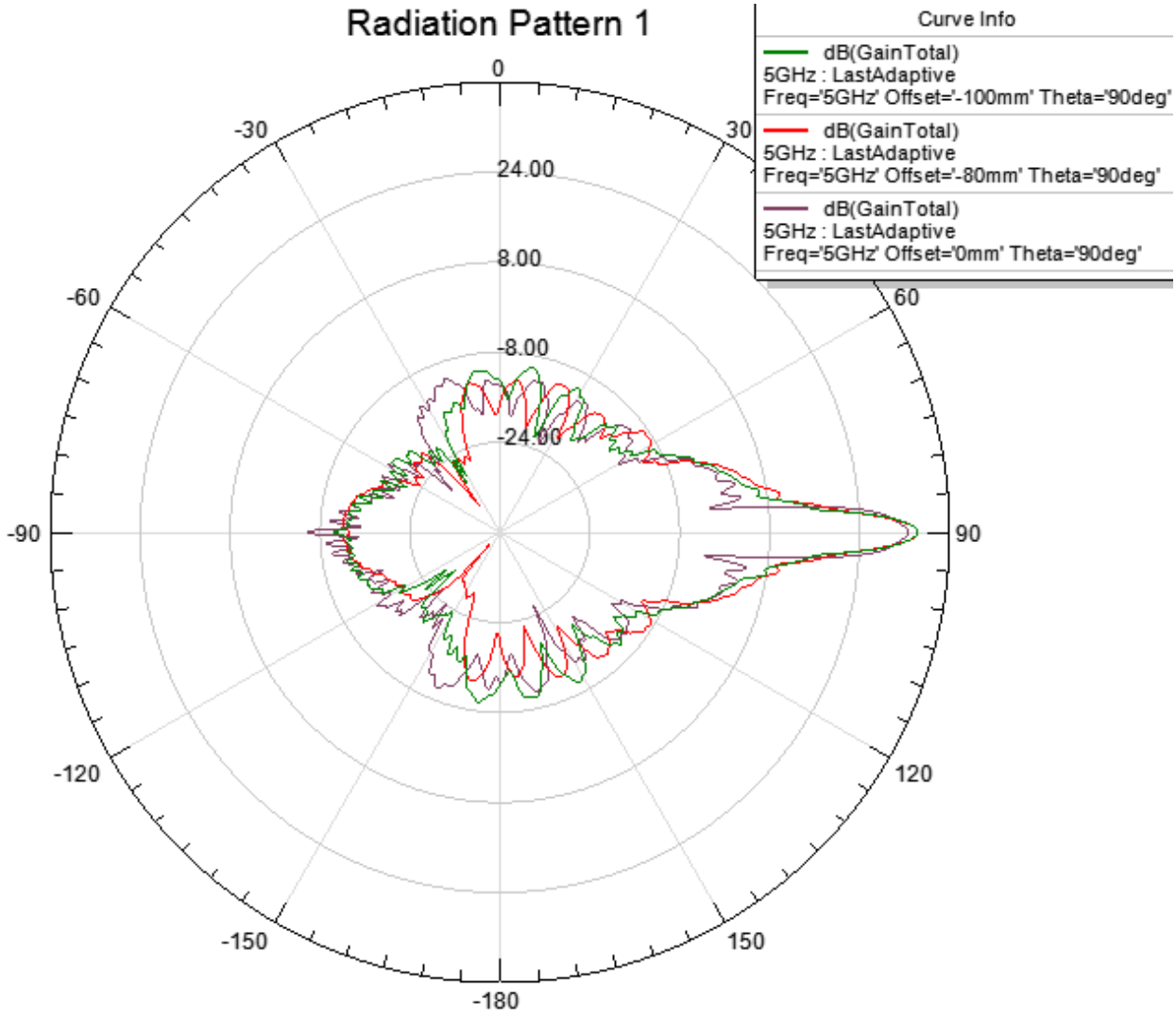


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

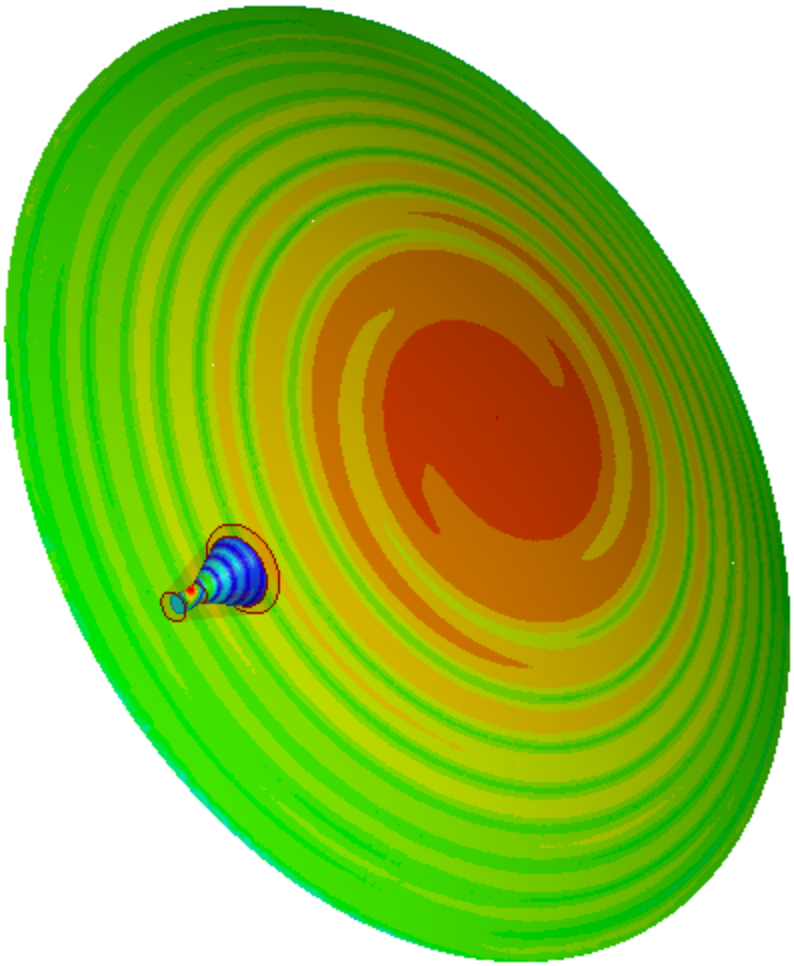
To access this feature, select component in the Project tree and select the **Component Meshing** tab in the **Properties** window. You can also view and set mesh assembly for all valid components in a design by using **Set Do Mesh Assembly** to open the **Do Mesh Assembly** dialog box, as shown above.

The radiation pattern for the three design variations is shown below.



The plot for one such design variation, offset at 0 mm, is shown in the following figure.

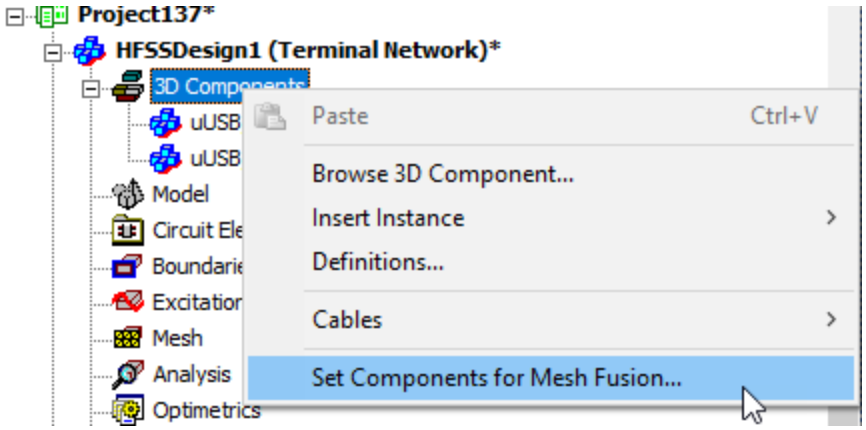
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



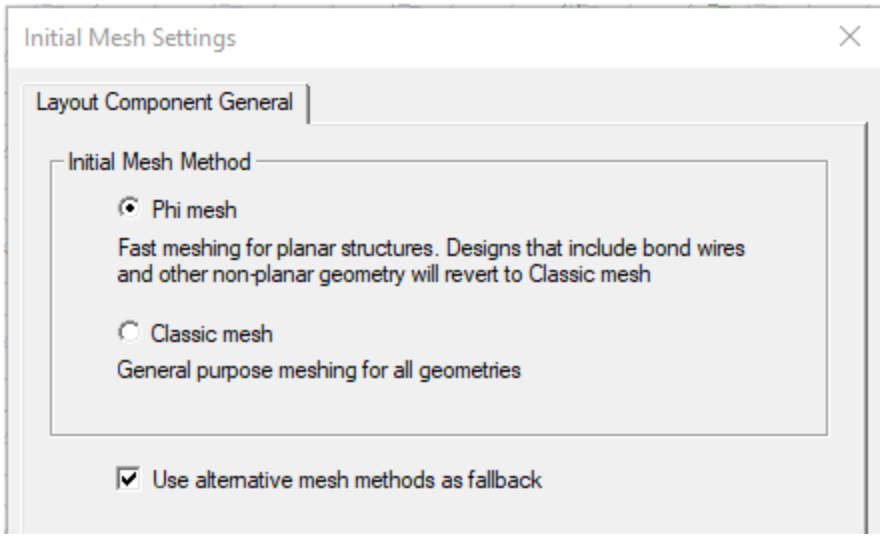
Mesh Fusion

With mesh fusion you can specify that multiple design modal FEM components and native regions are meshed independently to ensure very robust and rapid meshing. If a design contains multiple instances of a component, that component need only be meshed once. Parametric variations for one or more instances only affects the mesh of those instances. You can access the **Do Mesh Fusion** dialog through the **Project** tree shortcut menu for 3D Components, using the **Set Components for Mesh Fusion...** command. You can use the dialog to manage settings for all components in the project.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

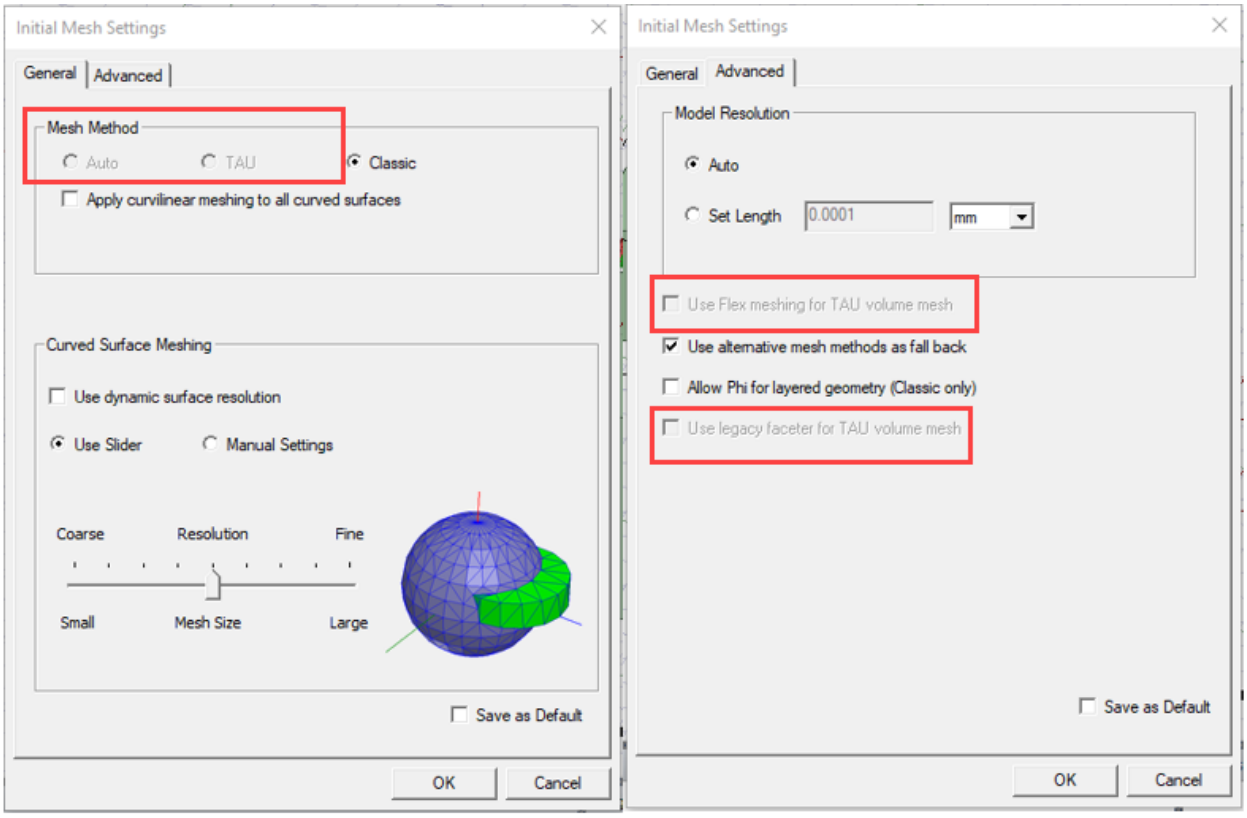


The solver supports mesh fusion for FEM volume and HFSS 3D Layout components in HFSS design type. This type of volume component cannot coexist with any FEBI, and/or surface and/or SBR+ components. Similar to 3D components, HFSS 3D layout components, when marked as mesh fusion, also support mesh envelope padding and mesh setting override. The mesh setting override dialog for layout components are different from that for 3D components and can be seen in the figure below.



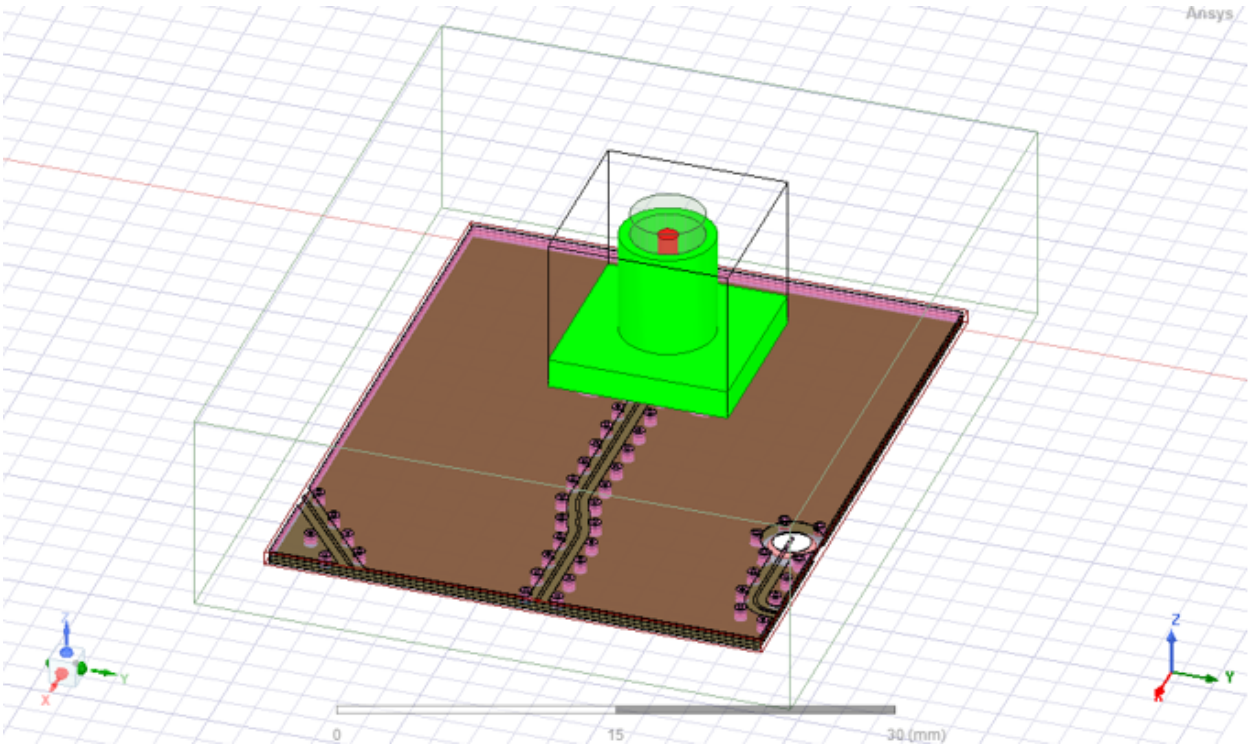
In this dialog, you can choose different mesh methods specific to layout components. This dialog is fully modeless, which loads different pages when you select layout or 3D components. In addition to this dialog, the **Initial Mesh Setting** dialog for the native design is also modified. If there are layout components meshed in native, the **Initial Mesh Setting** dialog will disable the Auto and TAU mesh method. The options related to TAU in the advanced setting page are also disabled.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With the proper setting, the design can be meshed and solved. Layout components marked as mesh fusion will have full interaction with other layout components, 3D components and native geometry.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

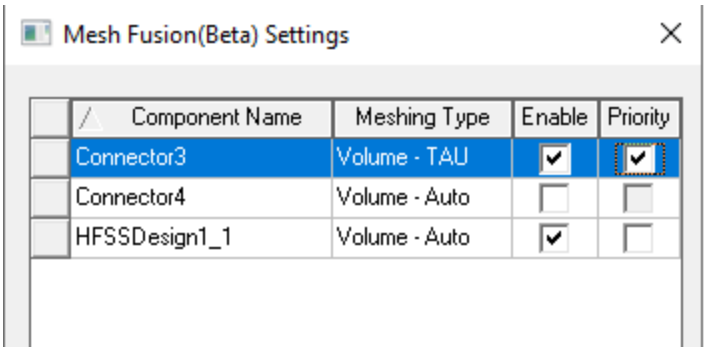


Some of the benefits of Mesh Fusion are as follows:

- Systems with large variation in length scales can be successfully meshed. For example, in the case of a small 3D antenna touching an aircraft, you can mesh both the bodies separately and ensure a conformal mesh between them. That is, the triangulations in the contact area between the volume mesh of the 3D antenna component and the surface mesh of the aircraft are aligned resulting in a conformal mesh.
- You can select different meshers for each component via the Component Meshing properties, or directly . That is, you can pick the Tau mesher for a given component and the Classic mesher for another component, in order to ensure system meshing is successful.
- The iterative solver option is available for mesh fusion projects as an Advanced Solution Setup option, except for 3D component array in both 3D layout and MCAD. The Iterative solver uses 1.e-6 as the default.
- Faster meshing for multiple instances of the same component is possible: only one initial mesh is created. For instance, when multiple identical antennas appear in a design, only one needs to be meshed.
- Mesh resuse enables fast placement studies. If you move a component, no re-meshing needs to be done. Also only the last adaptive mesh is used and no adaptive meshing of the component needs to done.

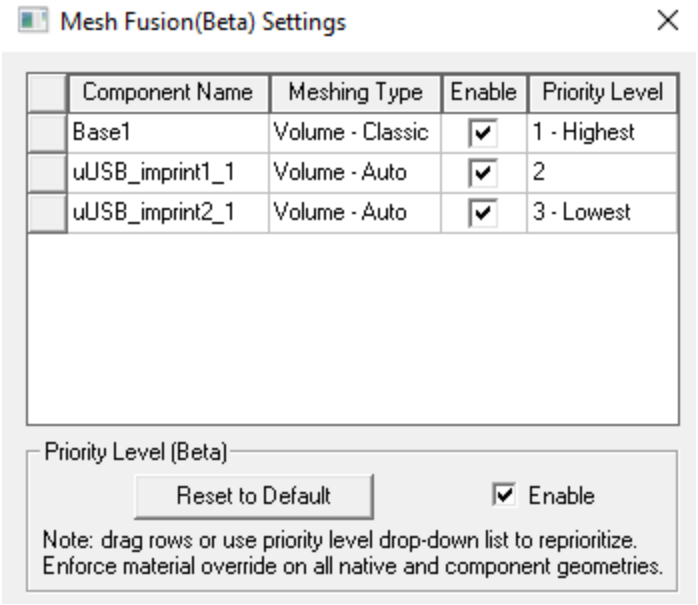
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Reused initial meshes are not impacted by any parametric variation. Suppose there are multiple components that are independently meshed and you make a parametric variation in one component, the remaining components are not remeshed. In this situation, the last adaptive mesh cannot be reused. Some examples of parametric variation can include increasing or decreasing the dimension of a geometry or changing the material properties.
- Mesh fusion can be distributed, including support for distributing Iterative Solver Excitations.
- Mesh Fusion components can be marked as Priority in the **Mesh Fusion Settings** dialog box. It is common for a large project to have some native objects intersect with mesh fusion components due to user error or modeling issue. These intersections are usually small and would not affect simulation result if ignored, but they can stop simulations and cost much time to fix. To improve this problem, you can mark any mesh fusion components as priority.



The dialog also contains a section to enable Priority Level ranked from Highest to Lowest for each component in the design. This feature aims to develop a better intersection resolution mechanism that could help you resolve most of the intersections with ease and customization for both regular and mesh fusion applications.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

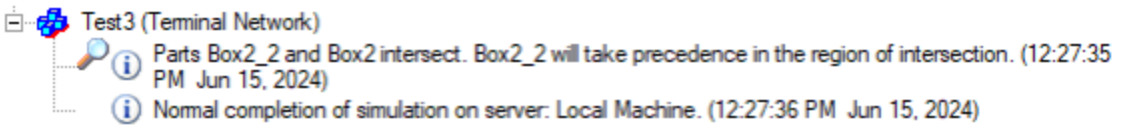


Once the Priority Level is enabled, a column named “Priority Level” replaces the “Priority” column. In this setting, all the components in the design are ranked from highest priority to the lowest based on rules for default priority. You can drag a row to change the priority level of a component. Alternatively, you can also click the cell in the Priority Level column for a row to open a drop-down list. You can then pick a level in the list and assign it to that component.

Additionally, you can click the **Reset to Default** button to assign a default priority level to all components. For details on how the default priority is calculated, refer to the **Rules for Default Priority** section below.

Most of the intersections will be resolved automatically during simulation according to the priority levels. All the resolutions will happen behind the scenes, and you will not notice any difference due to objects being cut-off in the post processing. This priority levels setting is only used for resolving intersections and not for containment. This priority levels setting will enforce the material setting to turn on.

Once you enable Priority Level, the design validation reports the intersection resolution result.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the priority levels setting is not enabled, HFSS uses the old behavior to handle any intersections.

Rules for Ranked Priority Level

If you enable Priority level and use the default rankings the rules for Default Priority Level apply.

Rules for Default Priority Level

Default Priority level uses the following rules.

1. Starting from the first rule, the dialog applies each rule to compare the priority levels of two components until the winner can be determined.
2. Components that are checked for mesh fusion have higher priority than those that are not.
3. Layout/encrypted 3D components have higher priority than regular 3D components.
4. If both components are checked for mesh fusion, the component with smaller mesh envelop has higher priority.
5. If both components are not checked for mesh fusion, the component that is inserted later has higher priority.
6. Geometries that do not belong to a component have the lowest priority.

Rules for Resolving Intersection between Two Objects

If two objects intersect, and the priority level is enabled, HFSS tries to resolve the intersection using the following rules. Starting from the first rule, HFSS applies each rule until the winner can be determined.

1. If one object is conductor and the other object is dielectrics, conductor wins in the region of intersection.
2. If both objects are conductors or dielectrics, the object that belongs to the component with higher priority wins.
3. If both objects are conductors or dielectrics and have the same component priority level, but their material is different, it is not valid. The code will return validation error.
4. For the rest of the case, the object with the smaller volume wins. If their volumes are equal, the object with smaller ID wins.

For example:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Two objects, red and blue, intersect. Both are conductors. The program determines that the red object has higher priority according to the rules.

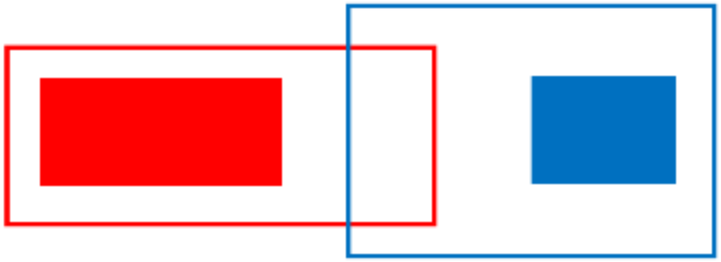
The red object will take precedence in the region of intersection.

Rules for Resolving Intersection between Two Components' Mesh Fusion Envelope

If two components are marked as mesh fusion, and their mesh envelopes intersect, the envelope that belongs to the component with higher priority level takes precedence in the region of intersection. The intersecting portion is subtracted from the envelope with lower priority. The mesh fusion envelope includes both the user defined mesh region and auto-generated mesh envelope.

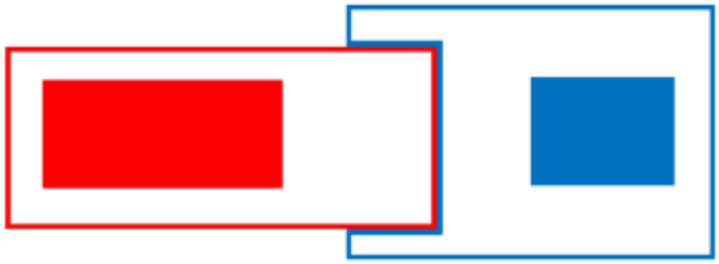
For example:

There are 2 components in red and blue. The rectangular outline represents the mesh fusion envelop of each component, and the solid rectangle represents the actual geometry of that component. The two components' mesh fusion envelopes intersect, and the red component has a higher priority level.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HFSS resolves such intersection by subtracting the intersecting portion from the blue component's mesh fusion envelope.



Rules for Resolving Intersection between an Object and Another Component's Mesh Fusion Envelope

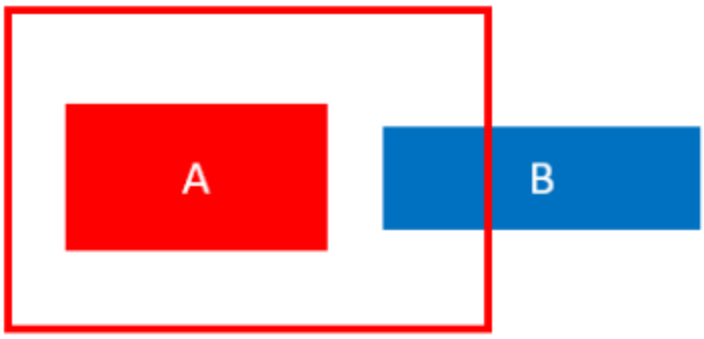
If one object intersects another component's mesh fusion envelope, the intersecting portion is cut off from the object and included with the other component. This cutout portion is meshed with that component.

If there are any boundaries or mesh operations assigned on the object, they are shared to the cutout portion. However, the following case is not valid and will cause a validation error.

- The object belongs to a layout component or an encrypted 3D component
- The cutout portion is assigned with circuit elements or excitations

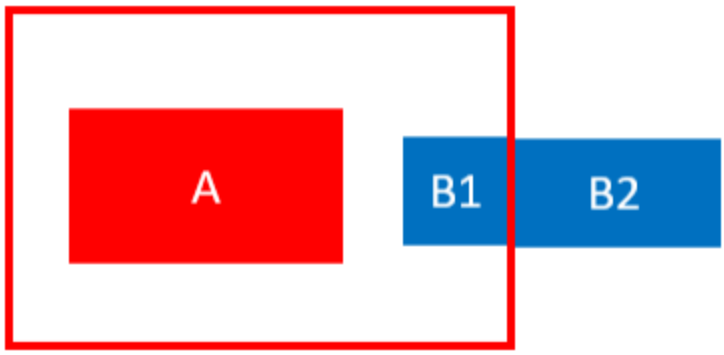
For example:

There is a component A whose mesh fusion envelope intersects with object B.



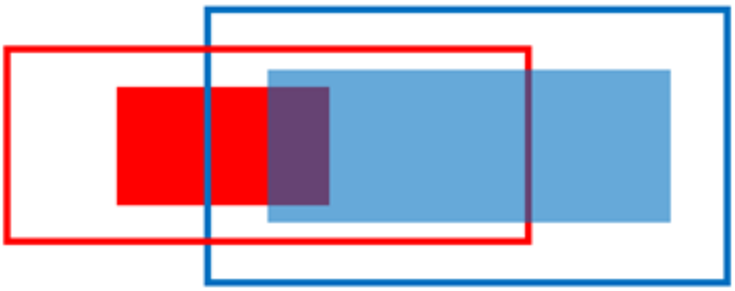
The intersecting portion is cut off from Object B. Object B is now split into 2 parts, B1 and B2. B1 is meshed with component A.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



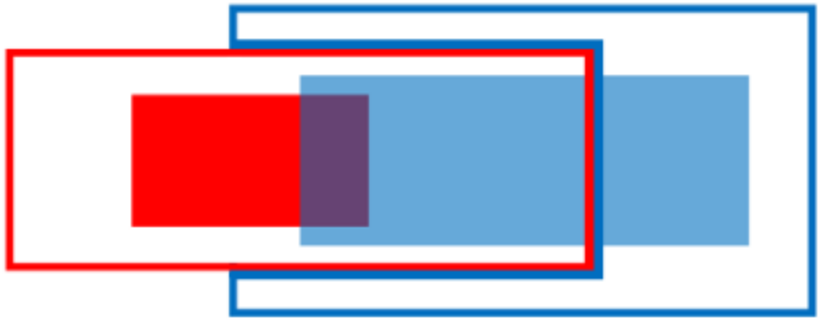
Putting it all together:

In the below example, there are 2 components in red and blue. The rectangular outline represents the mesh fusion envelop of each component, and the solid rectangle represents the actual geometry of that component. The two components' mesh fusion envelopes intersect, so do their actual objects. The red component has a higher priority level.

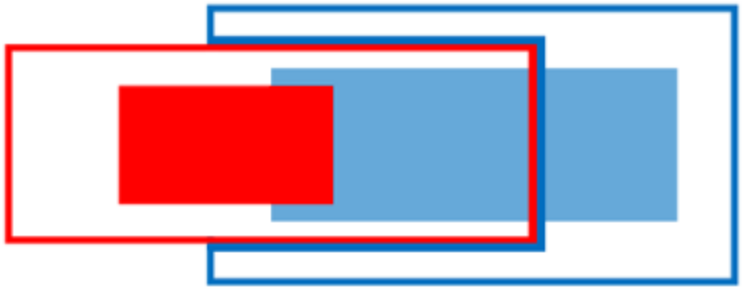


The envelope-envelope intersection is resolved by subtracting the blue component's envelope. After this, the blue component's envelope no longer intersects with the red component's actual object. However, the red component's envelope still intersects with the blue component's actual object. This intersection is resolved by cutting off the intersecting portion from the blue object and including it with the red component.

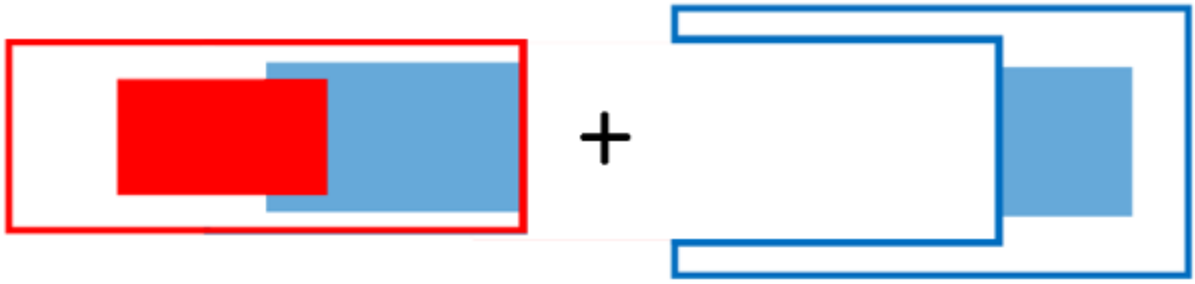
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The cutout portion from the blue object still intersects with the red component's actual object. This intersection is resolved by letting the red object take precedence in the region of intersection.

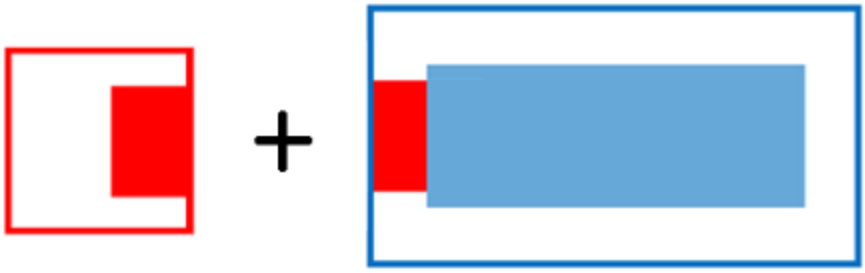


With all the intersection resolved, the two components are processed and split into two new groups of geometries that are meshed separately using mesh fusion.



On the other hand, if the blue component has higher priority instead, the two components will become the following after intersection resolution.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

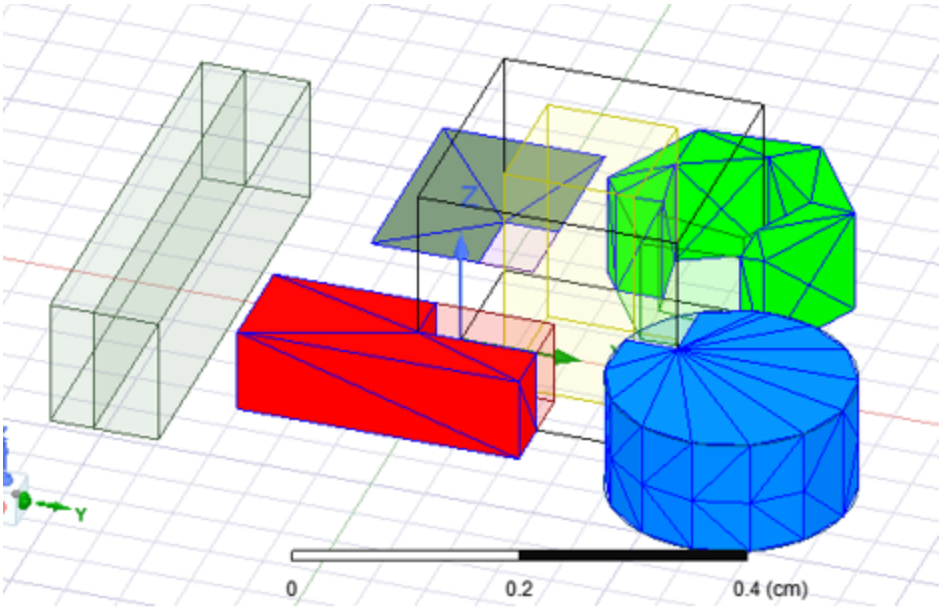


Rules to Resolve Containments

The Priority Level feature does not introduce any new rules to resolve the case in which one object contains another object. HFSS follows the old behavior for resolving such containment or reporting error. However, the Priority Level feature resolves a special containment case in which one component's mesh fusion envelope contains another component's mesh fusion envelope. This case was invalid in prior releases but can now be supported by letting the smaller envelope subtract the larger envelope. This leaves a cavity on the larger envelope, and the smaller envelope resides in the cavity.

Rules for Simple Priority

Using Simple Priority, if a priority mesh fusion component intersects a native object, the intersecting portion is subtracted from the native object, so that the simulation continues.

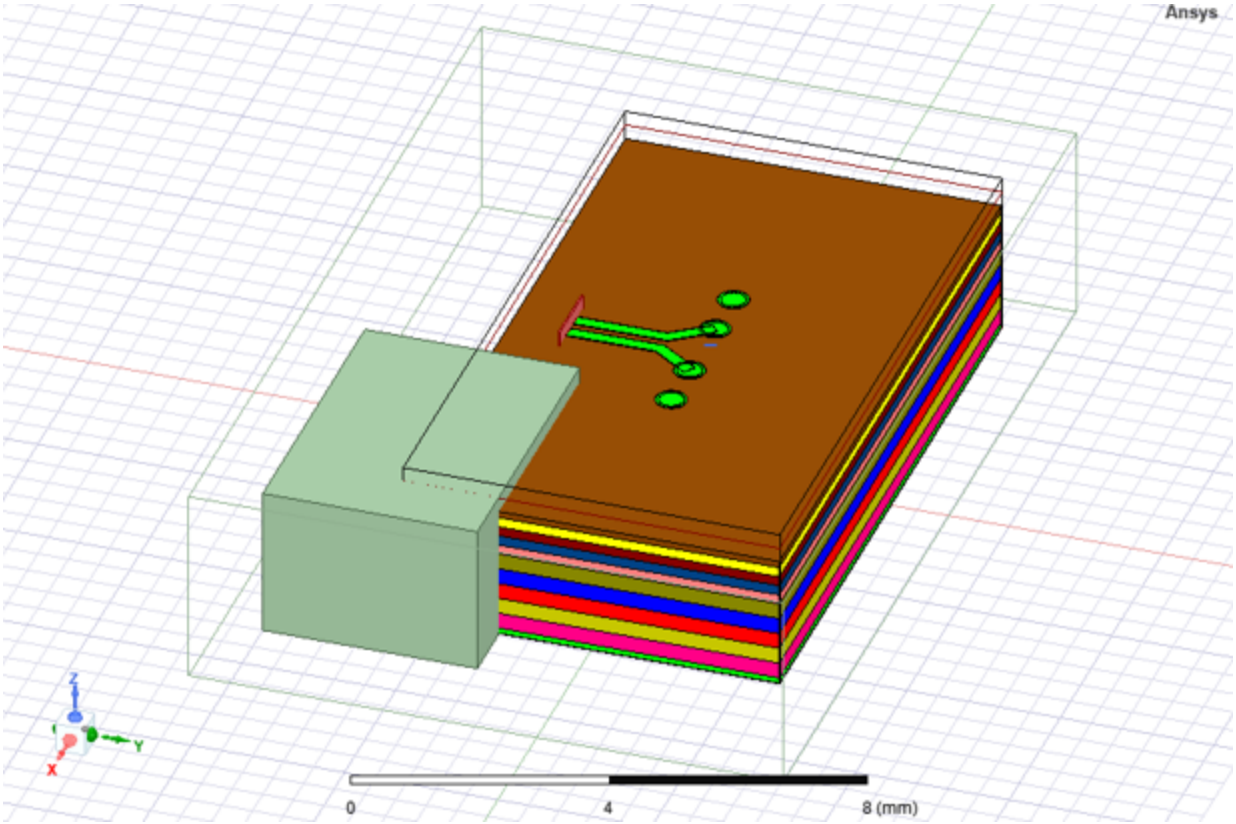


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - If the intersection portion has boundary assignment, the simulation will still report error
    - If two mesh fusion components intersect each other, the simulation will still report error
- Any overriding mesh setting appears in each component's property window.
- Mesh Fusion can share surface approximation operation between imprint face pairs that have curved outlines. Take surface approximations assigned on face, on body, and on the component level for both faces of the imprint pair and combine them to create the operation that results in finest mesh. This will only be done for face pairs that have classic mesh method on both sides. (TAU mesher does not support assigning surf approximation on planar faces, even if they have curved edges.) You can automatically assign 15 degrees normal angle to all shared mesh operations by env var ASSIGN_15_DEG_NORMAL_DEV_TO_CURVE_IMPRINT. This approach does not support array in native.
- You can generate field plots using data from either side of the boundary surfaces, using the "Adjacent side" check box on the Field Plot dialog, despite the sides belonging to different meshes and domains.

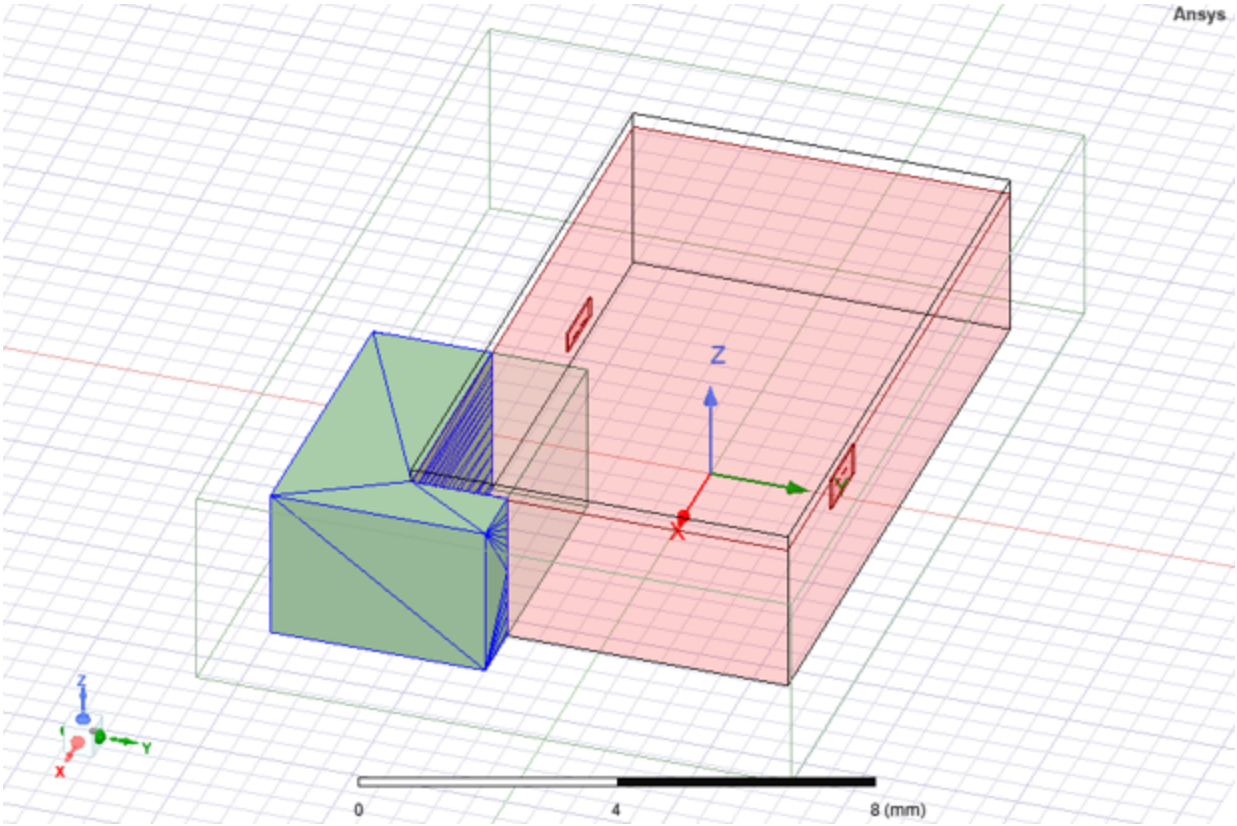
Layout components that are marked as Mesh Fusion also support the priority option. The priority option provides a way to resolve component intersection conflicts with native objects. If such intersection appears, the intersecting portion will be subtracted from the native object, so that the components marked as priority stay intact and the simulation could continue. For example, consider a design with layout component being intersected by a native box.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If the priority option is not marked for this layout component, the simulation reports error. If the component is marked as priority, then the intersecting portion is subtracted from the native object, and the simulation can complete. You can plot mesh on the native objects to confirm that the subtraction is applied, and the mesh for the layout component marked as priority is intact.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

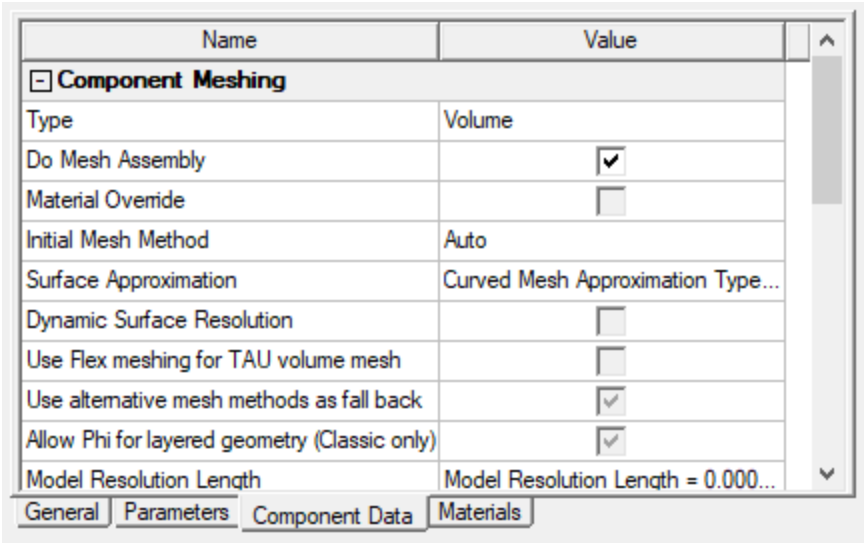


Some limitations of mesh regions include:

- Unless marked as Priority, Mesh region cannot intersect with each other or any non-assembly geometries.
- Mesh region cannot contain each other or any non-assembly geometries.
- Mesh region must have all planar faces.
- Any face of a mesh region cannot contact following boundaries:
  - port
  - boundaries with shell element enabled
  - boundaries containing spatial or temperature dependent material

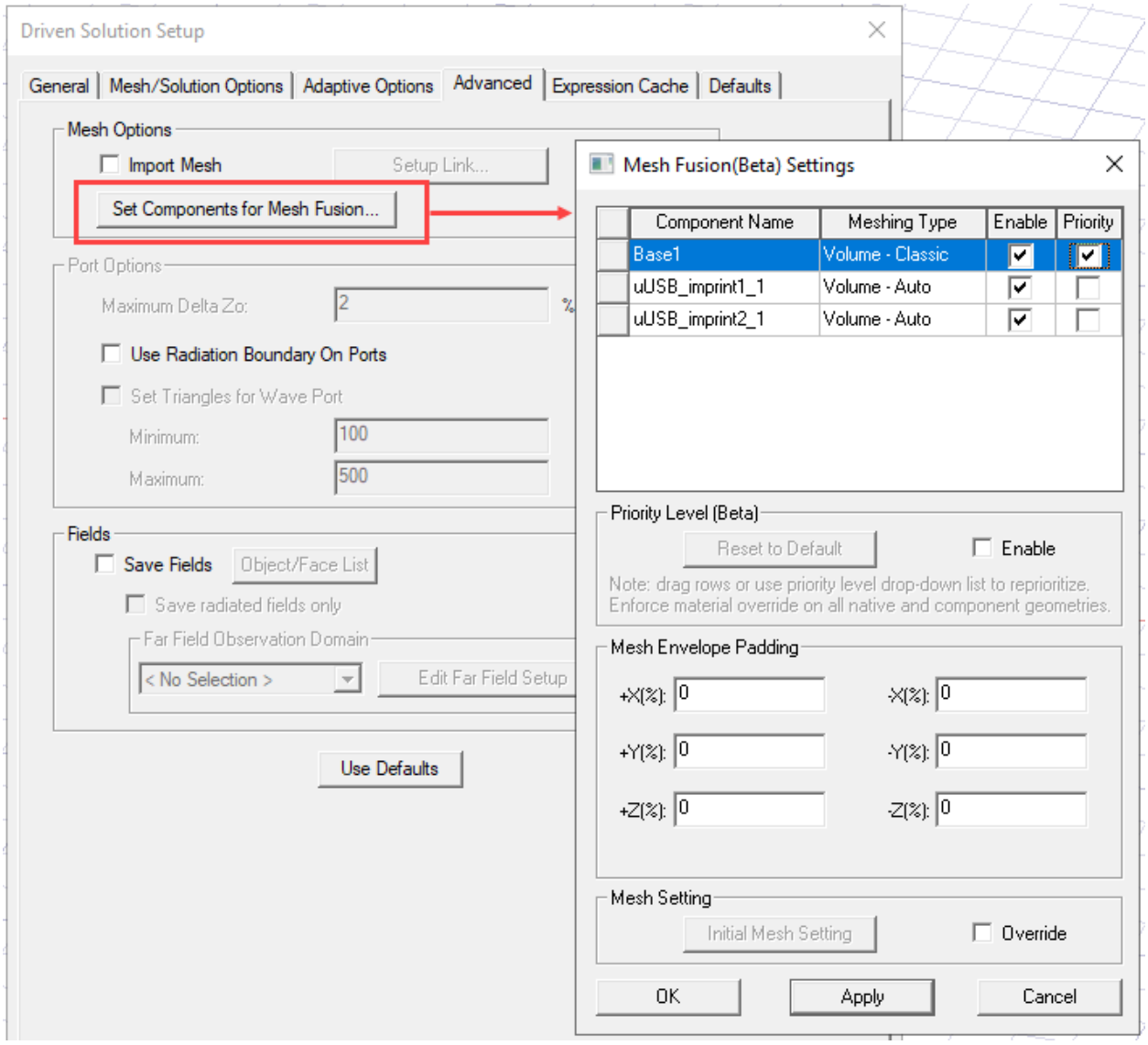
You can access the setting through **Component Meshing** properties.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can also access Mesh Fusion through the **Advanced** tab of the Solution Setup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

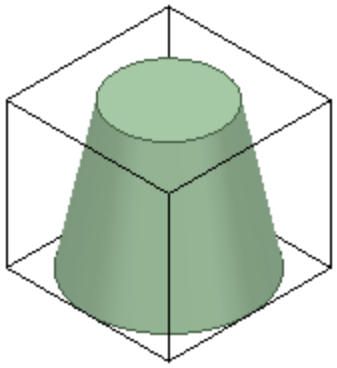


If a component does not meet the requirements for **Mesh Fusion**, the feature is disabled in the **Do Mesh Fusion** dialog and the **Component Meshing** properties.

Mesh Envelope Padding for Components without a User-defined Mesh Envelope

For each component used in mesh fusion, a tight bounding box is automatically created in the component's target coordinate system. This tight region box is referred as 'mesh envelope'. You can visualize such mesh envelope from the modeler window. See the example below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

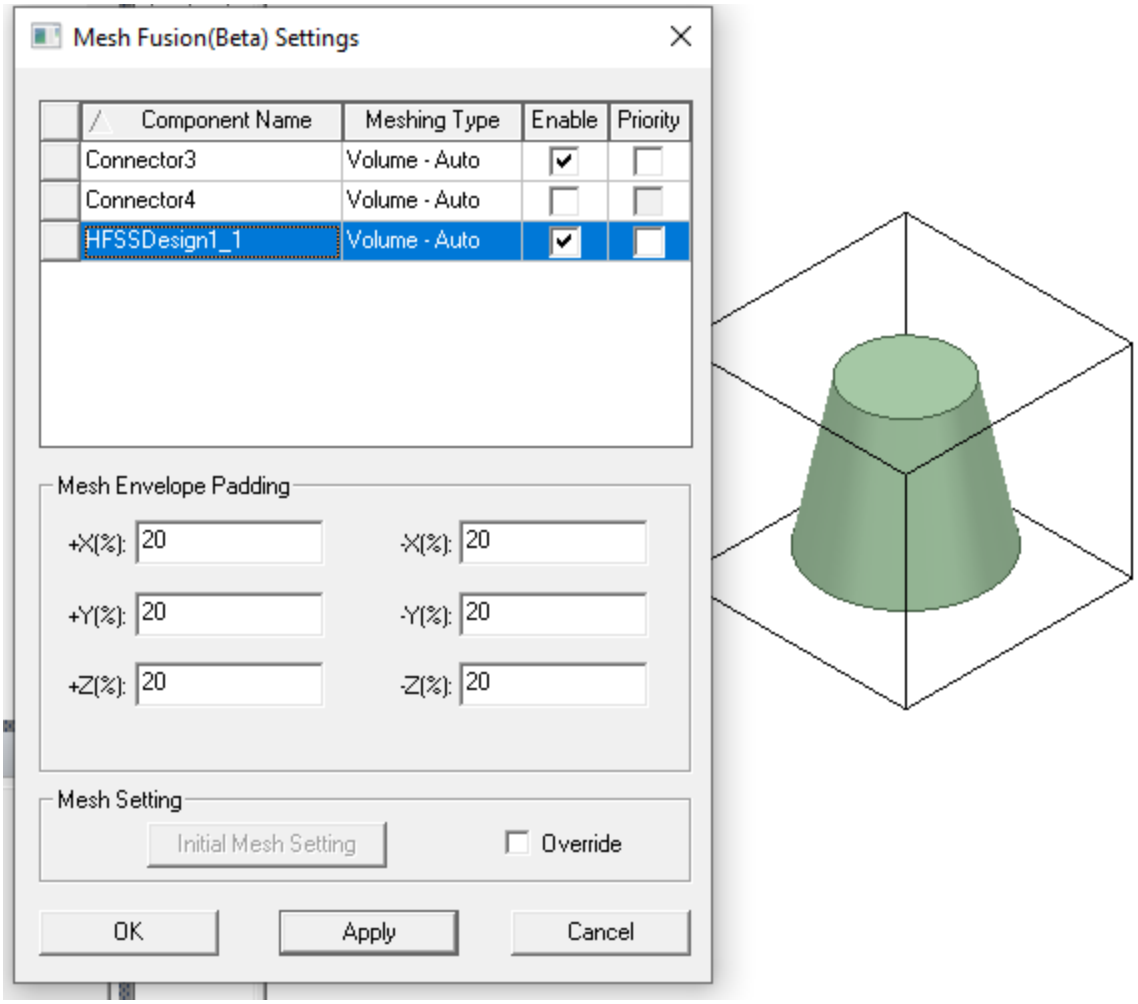


To override this automation, users can manually assign mesh region to a single solid solve-inside model object which must contain all geometries in the design. Manual mesh region is needed in following scenarios:

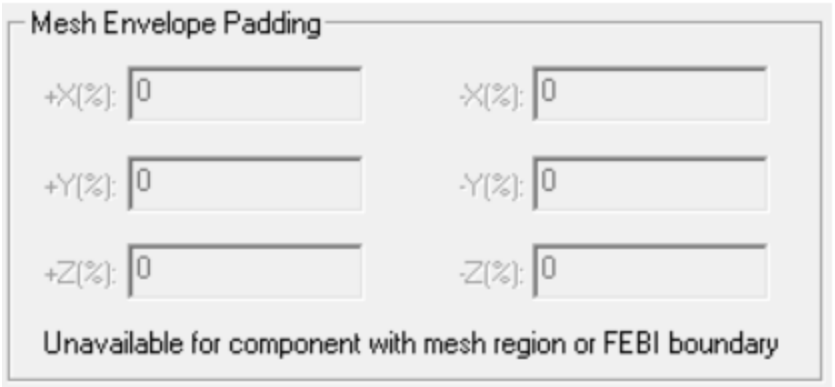
- To avoid overlap with native geometry or another component
- To avoid contact with internal faces when possible. This will improve mesh quality leading to better solver performance.

For each component without a manually assigned mesh envelope, users can add padding to its mesh envelope in all 6 directions of its target coordinate system. The example below shows the same component discussed above with 20% padding added to all directions of its mesh envelope.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



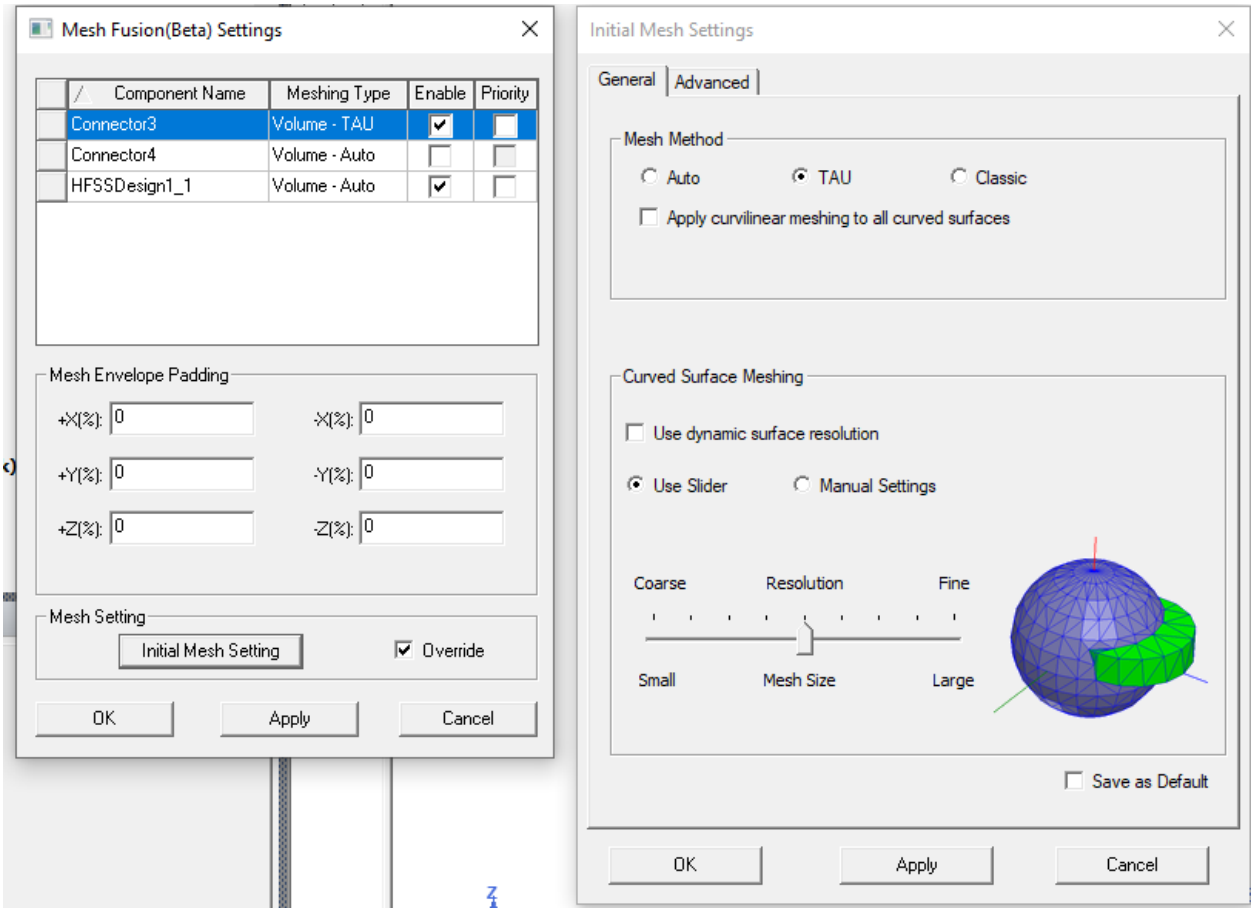
If a component has user-defined mesh envelope or is not a FEM volume component, the mesh envelop padding is disabled with a warning message.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

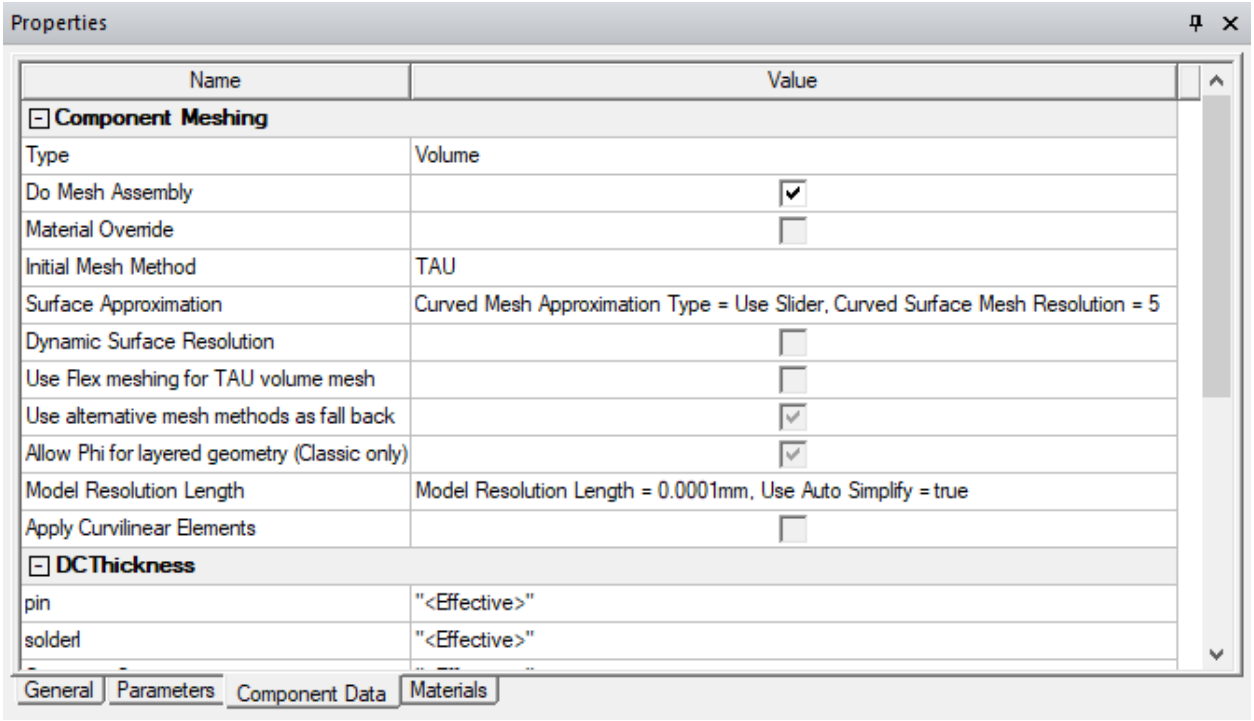
Mesh Method Override

You can override the mesh method from the component definition in the **Mesh Fusion Setting** dialog box, accessed either through the **3D Component** shortcut menu in the **Project** tree, or the Advanced tab of the an **Advanced Solution Setup**. This lets you deal with potential mesh failure based one specific mesh method when doing assembly meshing. If a failure happens using one method, you can easily switch the mesh method and try again. Selecting a component in the **Mesh Fusion Setting** dialog enables the region, where can check Override, and then select, Auto, TAU, or Classic. For convenience, the dialog also displays each component's meshing method in its Meshing Type column.



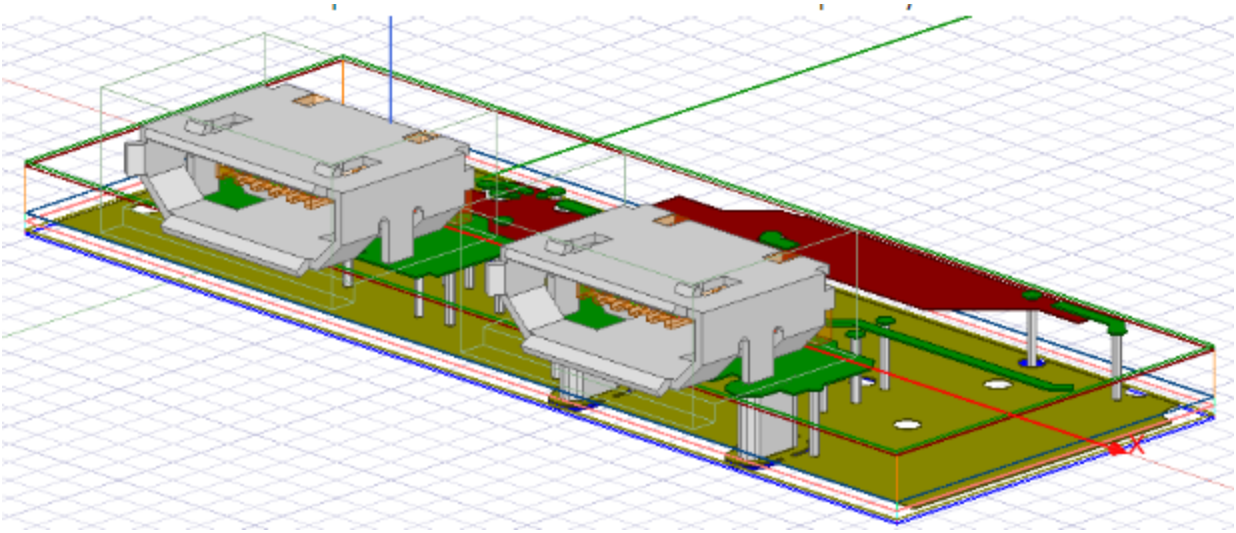
The Component Mesh settings will appear in the Component **Properties** window, **Component Data** tab.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



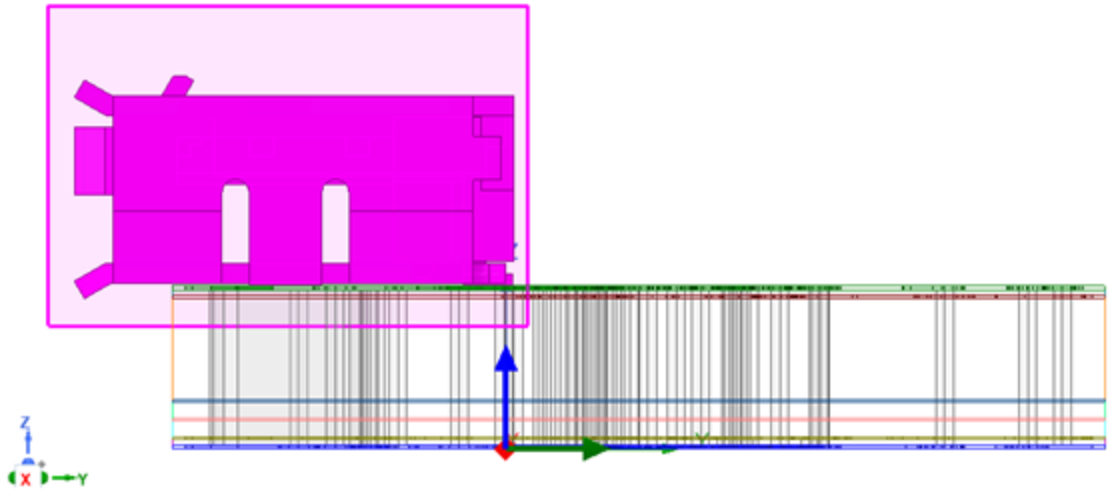
Example 1: USB Socket using Priority Level

In the following example design, there are two micro-USB's mounted on a mother board. The 2 USB's are from two 3D components, and the mother board is from a third 3D component. Users would like to mark each of the three components as mesh fusion to mesh them separately.

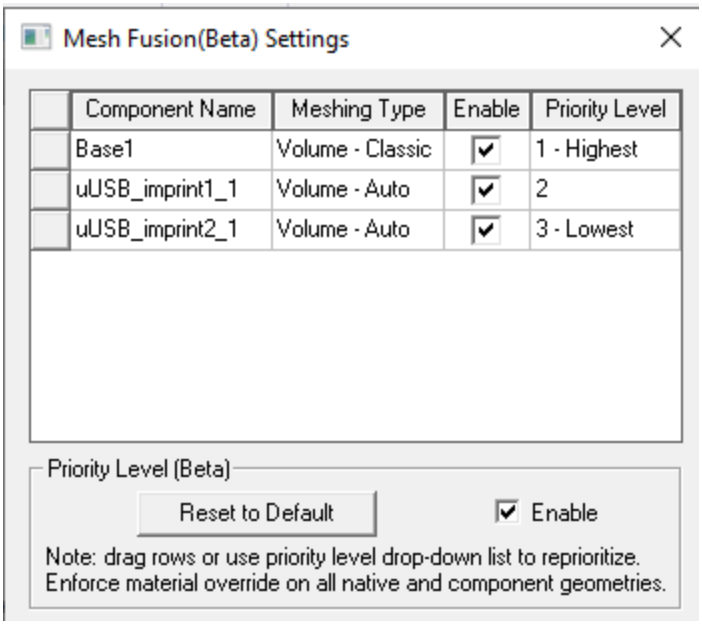


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

However, there are intersections in the design that must be resolved to use the mesh fusion technology. The below figure shows the right-side view of the design. The USB components have mesh fusion envelopes that intersect with the mother board. To resolve such intersections, the old way would be to manually cut out some bottom portion of the USB's mesh fusion envelope.

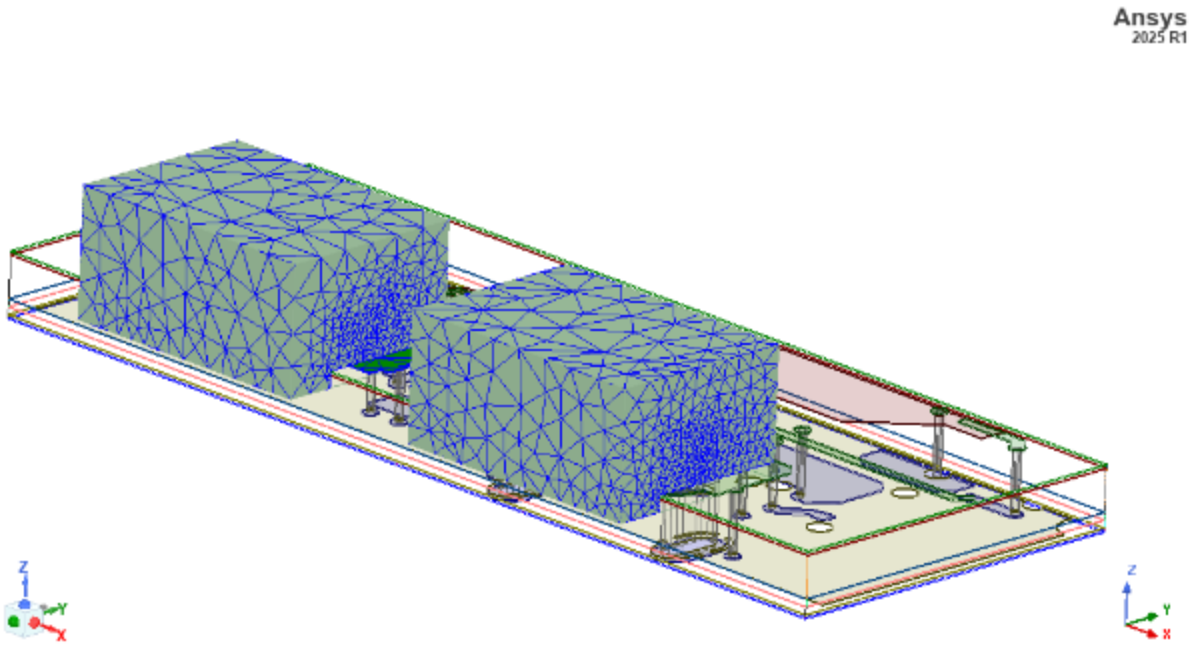


Using Priority Level, resolving such intersections is made easy. You can open the **Set Do Mesh Assembly** dialog and enable the component priority level. Then, you can assign the mother board component with the highest priority level as follows.



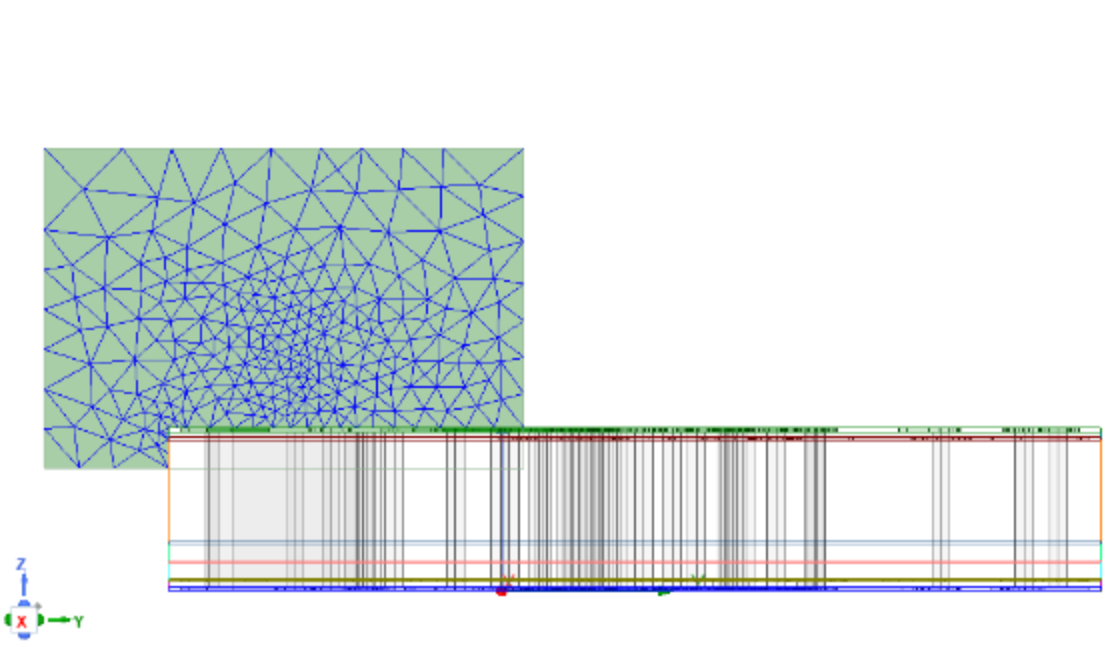
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By doing so, the intersection is automatically resolved during simulation. According to the rules discussed in the earlier section, the mesh fusion envelope of the mother board component will cut off the mesh fusion envelopes of the USB components. The mesh plot of the USB components; envelopes is shown as follows.

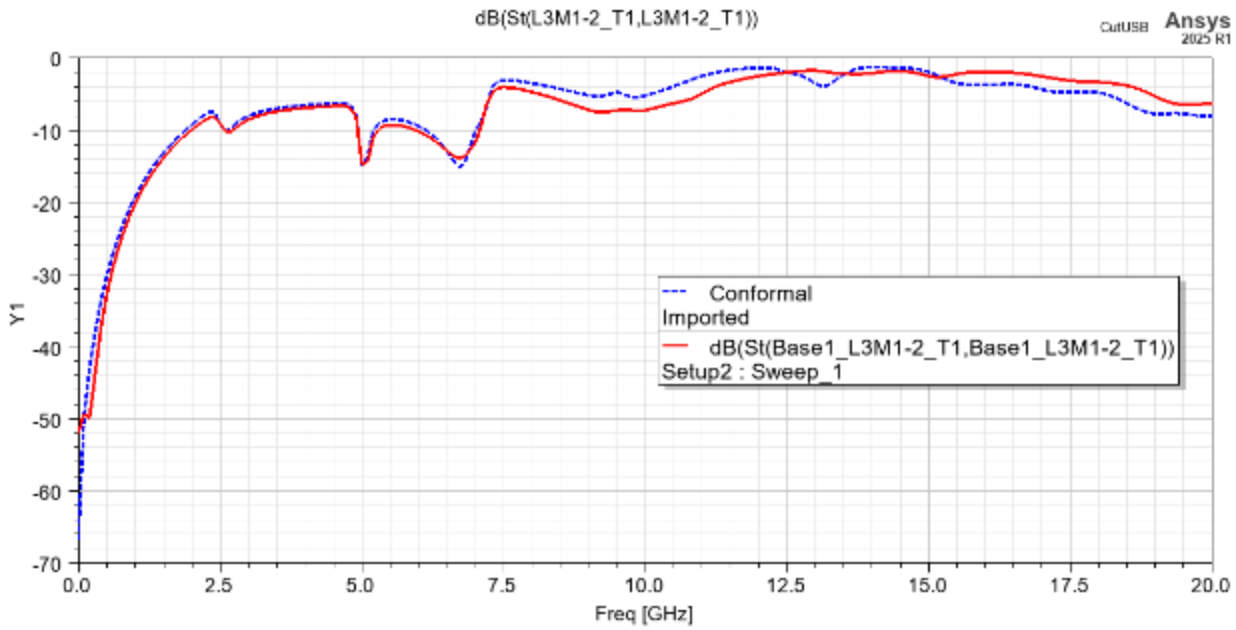


It is clear that the intersection is resolved by cutting the intersecting portion off the USB components' envelopes.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



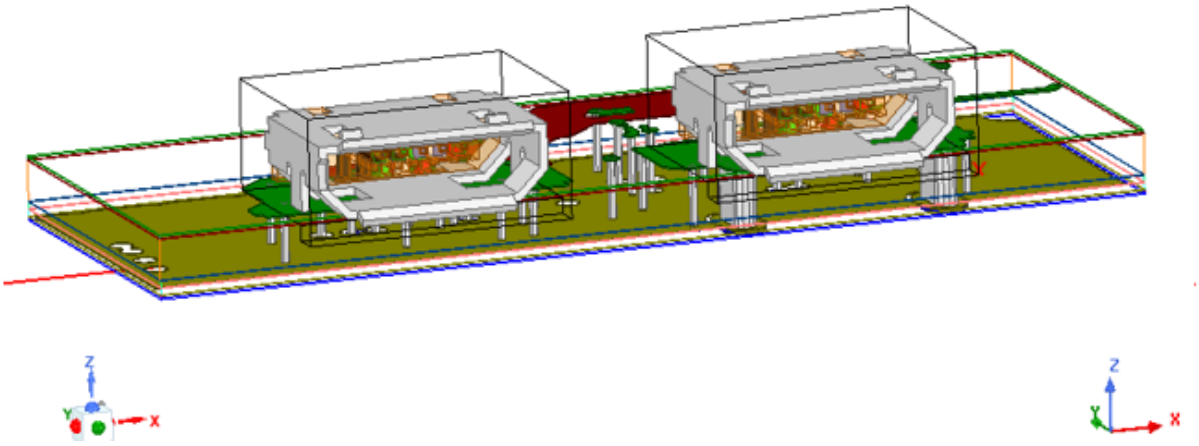
The simulation result of this case highly agrees with the result where all mesh fusion is turned off.

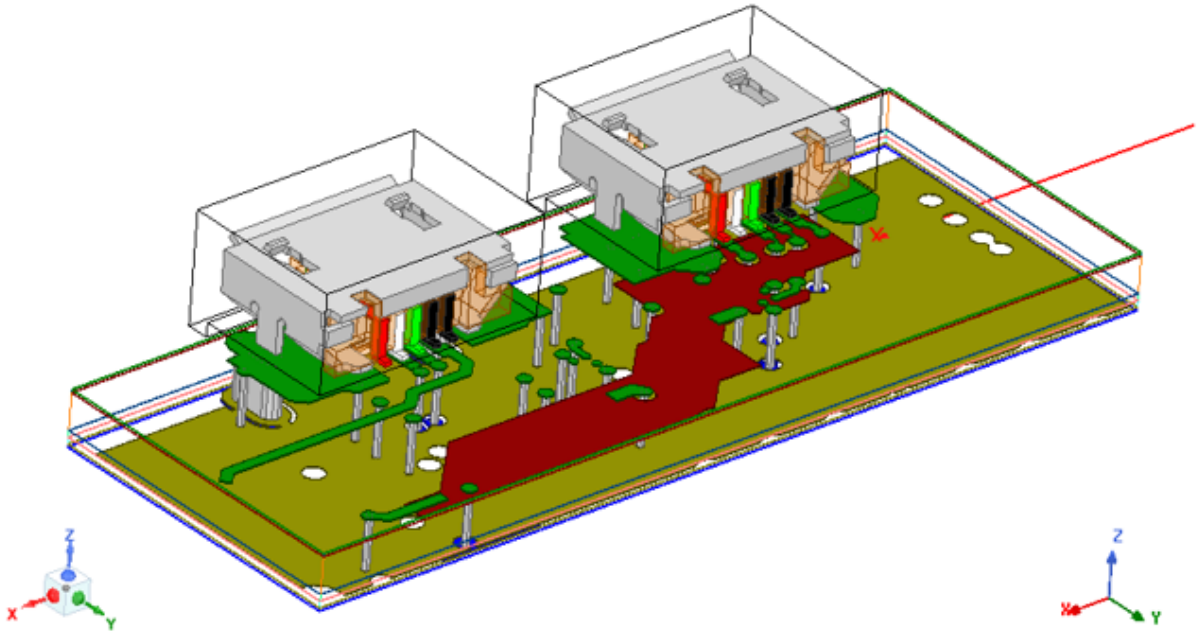


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example 2: USB Socket using Simple Priority

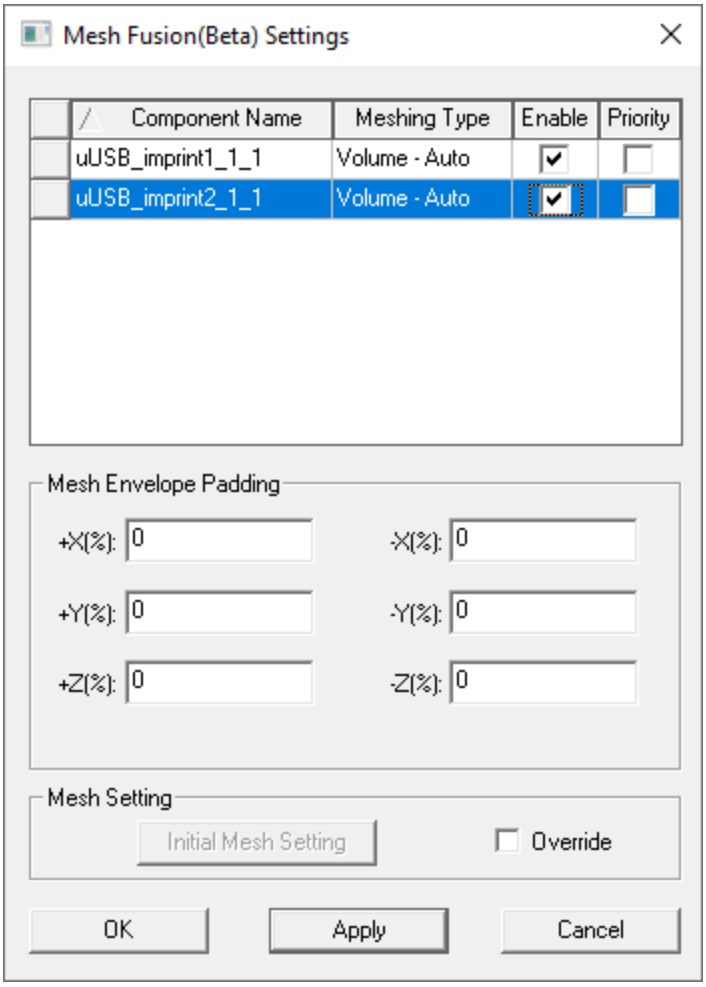
This example shows a circuit board with two USB sockets. The USB sockets are FEM components with custom defined air box as mesh regions.





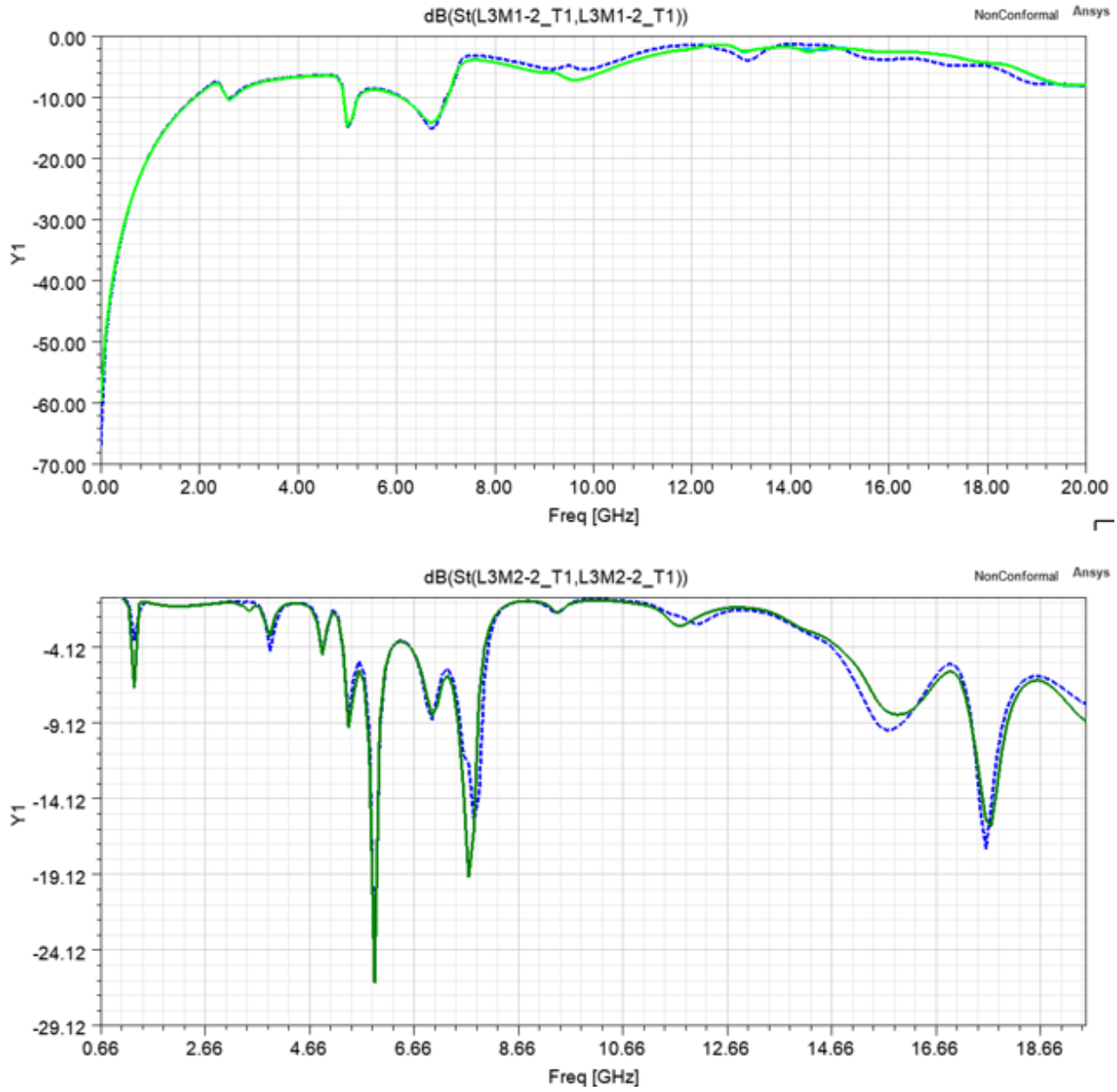
You can set the **Do Mesh Fusion** property of each components by accessing the **Do Mesh Fusion** dialog through the **Project** tree shortcut menu for 3D Components.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The plots below show the comparison of terminal S parameters between designs in which the components are set for mesh fusion (solid line) or not (dashed line).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

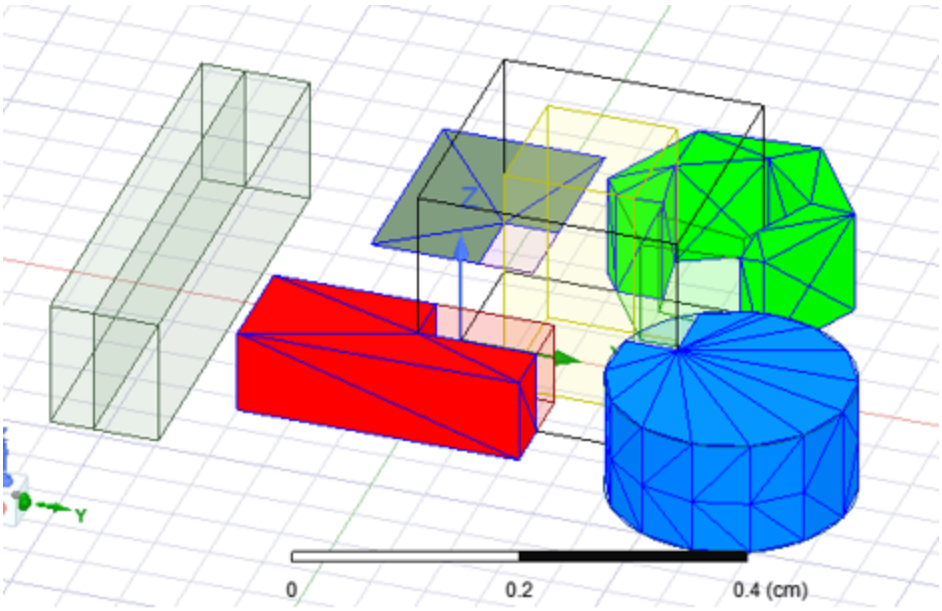


Component Priority

Component Priority provides a way to resolve component intersection conflicts with native objects. It is common for a large project to have some native objects intersect with mesh fusion/mesh assembly, or encrypted 3D components, that is components with a defined mesh region, due to user errors or modeling issues. Such intersections are usually small and would not affect simulation result if ignored, but they can stop simulations and cost much time to fix. To improve this problem, you can mark any of these components as priority. If such intersection

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

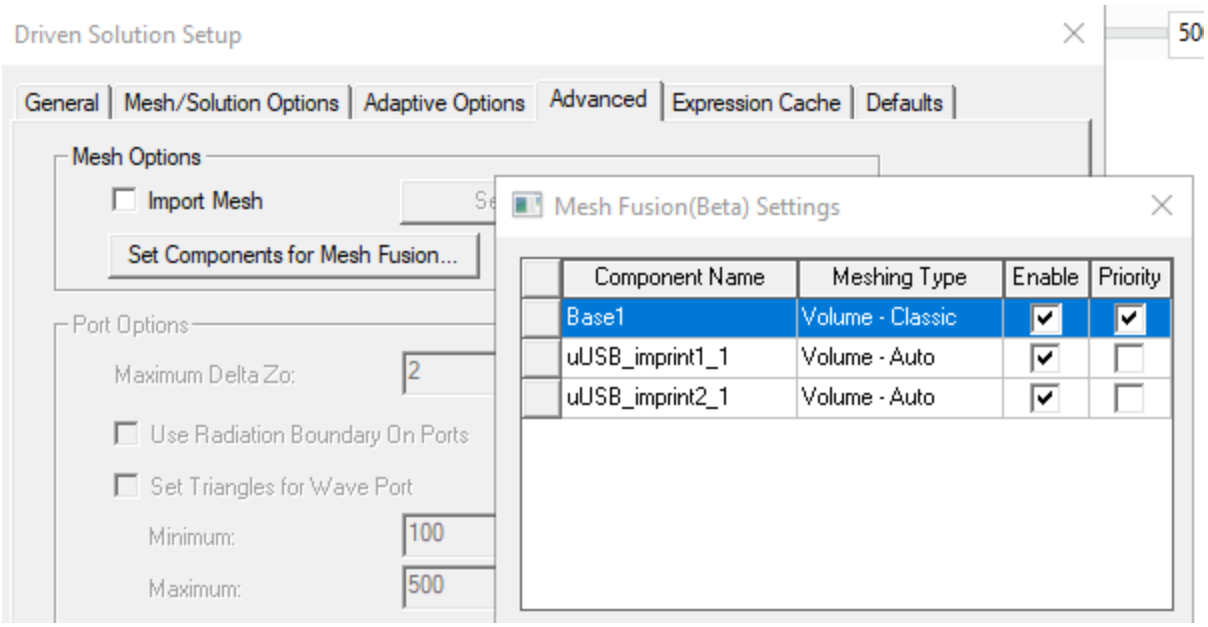
appears, the intersecting portion will be subtracted from the native object, so that the components marked as priority stay intact and the simulation could continue.



You can mark any component as priority as long as it has defined mesh region. This feature does not support array or SBR+ solution type. You access the feature through either the dialogs for the Set Components for Mesh Assembly... command or the Set Components for Mesh Fusion... commands.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also access these dialogs through the **Advanced** tab of the Solution Setup.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The components for which you can set priority include:

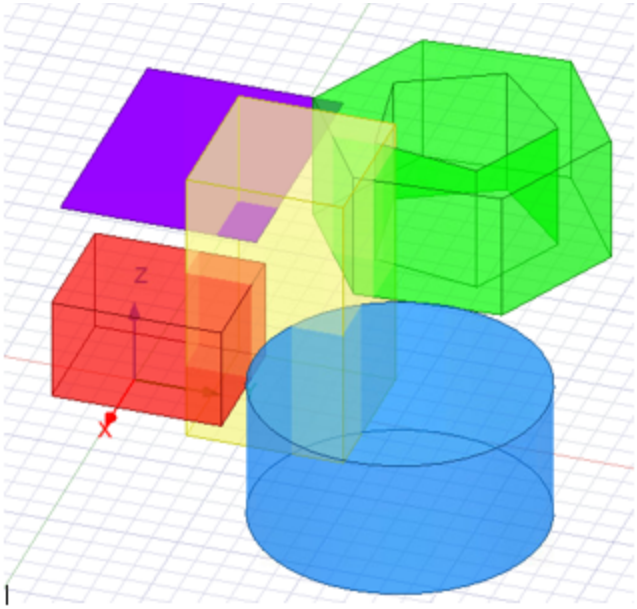
- Components with mesh region manually assigned. This feature also supports encrypted components because the subtraction will not expose any of their internal data.
- Components marked as mesh fusion (They will have mesh envelopes automatically generated).
- Layout components in 3D if they are marked as mesh fusion.

The subtraction only works on normal native objects.

- If the priority component is intersecting with an object from a component marked with priority as well, or from a component marked as mesh fusion/mesh assembly, or from an encrypted component, then the setup is invalid, and the simulation will report error.
- If the intersection portion has boundary assignment, the setup is invalid, and the simulation will report error
- The priority setting only affects intersections. It does not change the behavior if one object is fully contained by another object.
- This feature does not support array or SBR+ solution type.

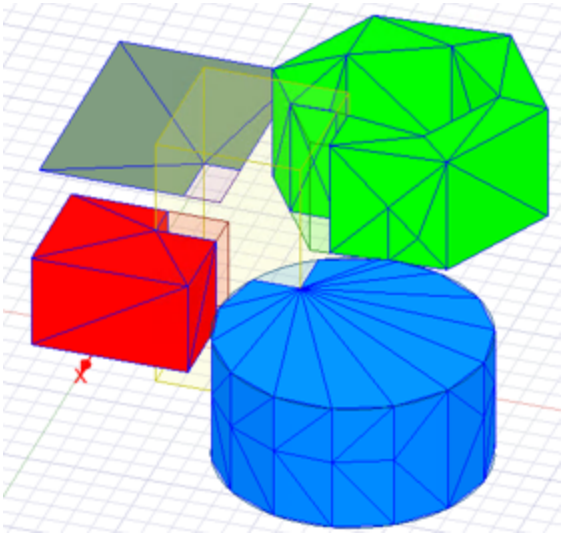
The steps to process priority components will be saved to a file in the initial mesh folder and the current mesh folder, together with steps to generate mesh fusion imprints (if any). This file will be used to generate identical mesh in the future.

For example, consider a simple design with a component (yellow box) being intersected by many native objects.

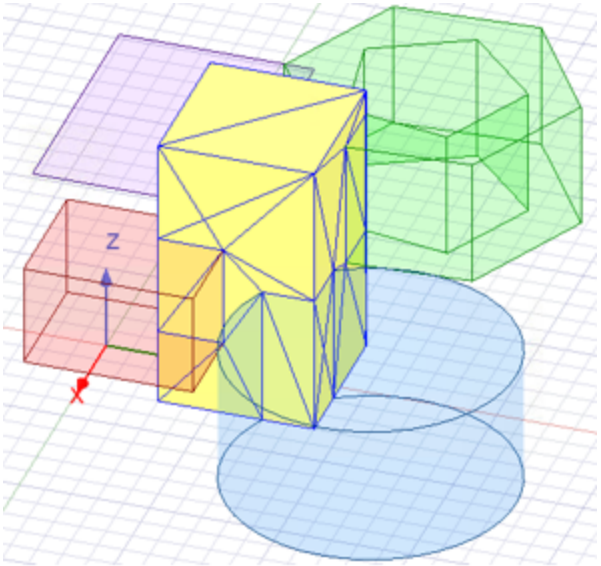


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the priority is not marked for this component, and that component intersects native object, the simulation will report error. If the component is marked as priority. Then the intersecting portion will be subtracted from the native object, and the simulation could complete. Users can plot mesh on the native objects to confirm that the subtraction is applied.



The mesh for the component marked as priority is intact.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28 - HFSS Technical Notes

The simulation technique used to calculate the full 3D electromagnetic field inside a structure is based on the finite element method. Although its implementation is largely transparent, a general understanding of the method is useful in making the most effective use of HFSS.

The HFSS Technical Notes provide an overview of the finite element method and its implementation in HFSS. They also describe how modal S-parameters are computed from the simulated electric and magnetic fields and how they can be converted to "nodal" or "voltage" based pseudo-S-parameters used in circuit theory.

Information is included on the following:

- The Finite Element Method
- The HFSS Solution Process
- Domain Decomposition
- S-Parameters
- Radiated Fields
- Geometric Objects
- Boundaries
- Derivatives
- Integral Equation Method Used in IE Regions
- HFSS SBR+ Technical Overview
- Transient Solution Theory
- Excitations
- Materials
- Best Practices for Accurate and Causal Broad Band Frequency Sweeps
- Parametric Analysis
- Sensitivity Analysis
- Tuning Analysis
- Modes to Terminals Conversion
- Large Scale DSO Theory

The Finite Element Method

In order to generate an electromagnetic field solution, HFSS employs the finite element method. In general, the finite element method divides the full problem space into thousands of smaller regions and represents the field in each sub-region (element) with a local function.

In HFSS, the geometric model is automatically divided into a large number of tetrahedra, where a single tetrahedron is a four-sided pyramid. This collection of tetrahedra is referred to as the finite element mesh.

Representation of a Field Quantity

Rectilinear Elements and Curvilinear Elements

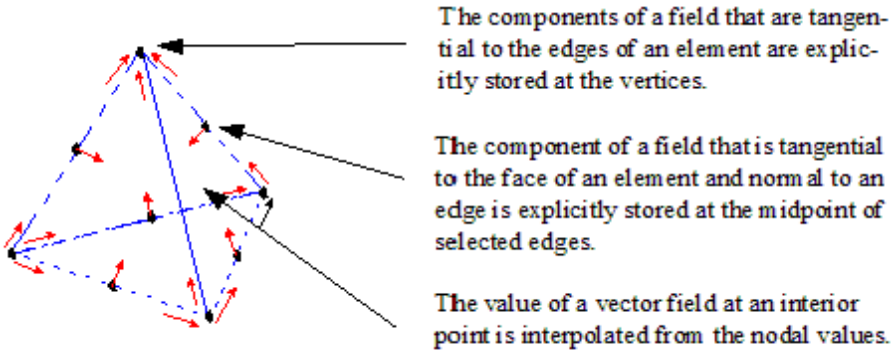
Basis Functions

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Size of Mesh Vs. Accuracy

Representation of a Field Quantity

The value of a vector field quantity (such as the H-field or E-field) at points inside each tetrahedron is interpolated from the vertices of the tetrahedron. At each vertex, HFSS stores the components of the field that are tangential to the three edges of the tetrahedron. In addition, HFSS can store the component of the vector field at the midpoint of selected edges that is tangential to a face and normal to the edge (as shown below). The field inside each tetrahedron is interpolated from these nodal values.



By representing field quantities in this way, the system can transform Maxwell's equations into matrix equations that are solved using traditional numerical methods.

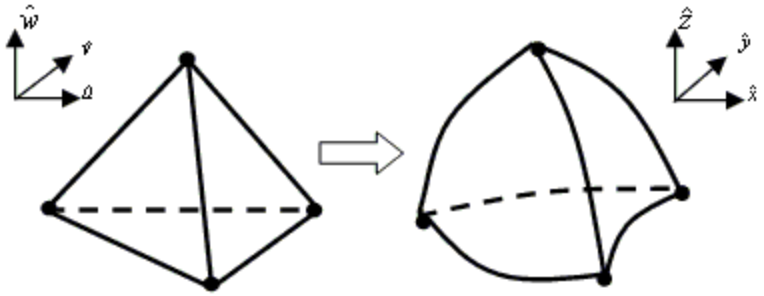
Rectilinear Elements and Curvilinear Elements

In the Finite Element Method (FEM), the physical domain is subdivided into many small elements. In the global coordinates (x, y, z), the field quantities to be solved are expanded by a set of basis functions. Since the basis functions are associated with each element, it is more convenient to write them in the local coordinates (u, v, w), which are independent of the shape of the element. The local coordinates and the global coordinates are linked by a transformation through a so-called Jocobian.

If the transformation is linear, the elements are called rectilinear elements, with straight edges and planar faces; if the transformation is non-linear, such as quadratic or cubic, the elements are called curvilinear elements, with curved edges and/or curved faces. So the curvilinear elements are a more general type of elements than their rectilinear counterparts. While they may not always conform exactly to the curved boundaries, the curvilinear elements conform better with less number of elements than the rectilinear ones. As a result, they generate more accurate and usually faster FEM solutions.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$J = \begin{bmatrix} \frac{\partial x}{\partial u} & \frac{\partial y}{\partial u} & \frac{\partial z}{\partial u} \\ \frac{\partial x}{\partial v} & \frac{\partial y}{\partial v} & \frac{\partial z}{\partial v} \\ \frac{\partial x}{\partial w} & \frac{\partial y}{\partial w} & \frac{\partial z}{\partial w} \end{bmatrix}$$



Basis Functions in HFSS

Various interpolation schemes, or basis functions, can be used to interpolate field values from nodal values.

- A first order tangential element basis function interpolates field values from both nodal values at vertices and on edges.

First order tangential elements have 20 unknowns per tetrahedron.

- A zero order basis function makes use of nodal values at vertices only — and therefore assumes that the field varies linearly inside each tetrahedron.

Zero order tangential elements have six unknowns per tetrahedron.

- A second order tangential element interpolates field values from nodal values at vertices, on edges and on faces.

Second order tangential elements have 45 unknowns per tetrahedron.

- Mixed order assigns basis function elements based on the need for higher accuracy in different parts of the model.

Size of Mesh Vs. Accuracy in HFSS

There is a trade-off among the size of the mesh, the desired level of accuracy, and the amount of available computing resources.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The accuracy of the solution depends on the size of each of the individual elements (tetrahedra). Generally speaking, solutions based on meshes using thousands of elements are more accurate than solutions based on coarse meshes using relatively few elements. To generate a precise description of a field quantity, each element must occupy a region that is small enough for the field to be adequately interpolated from the nodal values.

However, generating a field solution involves inverting a matrix with approximately as many elements as there are tetrahedra nodes. For meshes with a large number of elements, such an inversion requires a significant amount of computing power and memory. Therefore, it is desirable to use a mesh fine enough to obtain an accurate field solution but not so fine that it overwhelms the available computer memory and processing power.

To produce the optimal mesh, HFSS uses an iterative process, called an adaptive analysis, in which the mesh is automatically refined in critical regions. First, it generates a solution based on a coarse initial mesh. Then, it refines the mesh in areas of high error density and generates a new solution. When selected parameters converge to within a desired limit, HFSS breaks out of the loop.

The HFSS Solution Process

To calculate the S-matrix associated with a structure with ports, HFSS does the following:

- Divides the structure into a finite element mesh.
- Computes the modes on each port of the structure that are supported by a transmission line having the same cross-section as the port.
- Computes the full electromagnetic field pattern inside the structure, assuming that one mode is excited at a time.
- Computes the generalized S-matrix from the amount of reflection and transmission that occurs.

The resulting S-matrix allows the magnitude of transmitted and reflected signals to be computed directly from a given set of input signals, reducing the full 3D electromagnetic behavior of a structure to a set of high frequency circuit parameters.

The Mesh Generation Process

Reverting to the Initial Mesh

Seeding the Mesh

Guidelines for Seeding the Mesh

Length-Based Mesh Refinement

Skin Depth-Based Mesh Refinement

Surface Approximation Settings

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Guidelines for Modifying Surface Approximation Settings

Meshing Region Vs. Problem Region

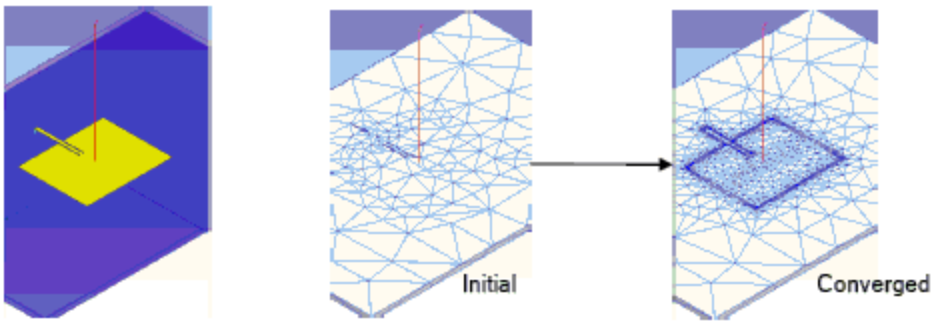
Mesh Refinement on Ports

Model Resolution

The Mesh Generation Process

Following is the general mesh generation process:

1. An initial mesh is generated and includes surface approximation settings. If necessary, the mesher will automatically perform any repairs needed to recover an accurate mesh representation of a model. The solution profile will indicate when mesh repairs have been made, and the results of these repairs will be displayed per object in the mesh statistics panel.
2. If lambda refinement was requested, the initial mesh is refined based on the material-dependent wavelength.
3. Any mesh operations that were defined are used to refine the mesh.
4. If ports were defined, the 2D mesh is refined iteratively.
5. Using the resulting mesh, the electromagnetic fields are computed that exist inside the structure when it is excited at the solution frequency.
6. If you are performing an adaptive analysis, the current finite element solution is used to estimate the regions of the problem domain where the exact solution has strong error. Tetrahedra in these regions are refined.
7. Another solution is generated using the refined mesh.
8. The error is recomputed, and the iterative process (solve — error analysis — adaptive refinement) repeats until the convergence criteria are satisfied or the maximum number of adaptive passes is completed.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. If a frequency sweep is being performed, then the problem is solved at the other frequency points without further refining the mesh. An adaptive solution is performed only at the specified solution frequency.

> **Note:**
>
> An initial mesh is not generated each time it starts the solution process. The initial mesh is generated only if a current mesh is unavailable.

Seeding the Mesh

In HFSS, mesh operations are optional mesh refinement settings that enable you to provide HFSS with engineering guidance based on your knowledge of the parts of the model geometry that are critical to the structure's electromagnetic performance. Providing such guidance prior to beginning the adaptive analysis process can reduce (sometimes extensively) the number of passes necessary to converge upon a field solution as well as the final number of tetrahedra in the mesh for that solution. Although adaptive analysis convergence targets areas where field behavior is found, refining the mesh using more than the standard criteria, such as material characteristics, can result in finding areas of critical field behavior as soon as the first few passes are solved.

The technique of guiding the mesh construction is referred to as "seeding" the mesh. Seeding is performed using the **Mesh** commands on the **HFSS** menu.

You can instruct HFSS to refine the length of tetrahedral elements on a surface or within a volume until they are below a certain value (length-based mesh refinement) or you can refine the surface triangle length of all tetrahedral elements on a surface or volume to within a specified value (skin depth-based mesh refinement.) These types of mesh operations can be defined at any time. If you apply them before the adaptive solution process, they are used to refine the initial mesh after it has been generated. You can also choose to generate a mesh without solving, in which case mesh operations are applied to the current mesh.

In a few circumstances, you may also want to define a mesh operation that modifies the surface approximation settings for one or more faces. Surface approximation settings are only applied to the initial mesh.

Guidelines for Seeding the Mesh

While seeding the mesh is not required, it is useful in the following conditions:

- Seeding the mesh inside a volume in the model geometry where regions of strong electric or magnetic fields (with strong capacitive or inductive loading) are expected. Examples include a capacitively loaded gap in a resonant structure, sharp waveguide angles or corners, or gaps between multi-coupled lines in filter structures.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Seeding the mesh on every face of higher aspect ratio boundaries, such as long PCB traces or on the surfaces of long wires. Spacing the mesh points roughly equal to the trace width of the wire diameter enables you to more accurately capture the behavior of the high-aspect structure from the first adaptive pass.

Length-Based Mesh Refinement Process

When you request length-based mesh refinement, you instruct HFSS to refine the length of tetrahedral elements until they are below a specified value. The length of a tetrahedron is defined as the length of its longest edge.

You can specify the maximum length of tetrahedra on faces or inside of objects. You can also specify the maximum number of elements that are added during the refinement. When the initial mesh has been generated, the refinement criteria you specified will be used to refine the initial mesh.

Skin Depth-Based Mesh Refinement

When you request skin depth-based mesh refinement, you instruct HFSS to refine the surface triangle length of all tetrahedral elements on a face to within a specified value. A layered mesh is created based on the surface mesh. The layers are graded based on the skin depth and number of layers you specify.

During skin depth-based mesh refinement, HFSS creates a series of layers that are planes parallel to the object face, and that are spaced within the specified skin depth. For each point on the surface of the face, a series of points (P0, P1, P2, ..., Pn) are added to the mesh, where n is the number of layers. P0 is the point on the surface and the distance from P0 to Pn is the skin depth. The points are spaced in a non-uniform manner, with the distance between them decreasing in a geometric progression, as you move from Pn to P0.

For example, if

Skin Depth:	12 mm
Number of Layers of Elements:	4

then

Distance [P0,P1]:	0.8 mm
Distance [P1,P2]:	1.6 mm
Distance	3.2 mm

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[P2,P3]:	
Distance [P3,P4]:	6.4 mm
Distance [P0,P4]:	0.8 + 1.6 + 3.2 + 6.4 = 12 mm

The skin depth-based refinement first satisfies the surface triangle edge length criterion, then introduces the series of points to each additional layer. If a limit has been placed on mesh growth, one of the following happens:

- The limit is set high enough to complete the skin depth refinement.
- The limit is set high enough to satisfy the surface triangle edge length criterion, but not high enough to complete the depth seeding.
- The limit is not set high enough to satisfy even the surface triangle edge length criterion.

Because refining by skin depth can add many seeding points, you should first refine the surface of the object using length-based mesh refinement to obtain an accurate count of the number of points HFSS will add when refining by skin depth. This allows you to reach the surface edge length criterion and approximate the number of elements in the mesh and the number of points on the surfaces before proceeding to skin depth seeding.

The refinement criteria you specified are used to refine the current mesh.

Surface Approximation Settings

Object surfaces in HFSS may be planar, cylindrical or conical, toroidal, spherical, or splines. The original model surfaces are called *true surfaces*. To create a finite element mesh, HFSS first divides all true surfaces into triangles. These triangulated surfaces are called faceted surfaces because a series of straight line segments represents each curved or planar surface.

For planar surfaces, the triangles lie exactly on the model faces; there is no difference in the location or the normal of the true surface and the meshed surface. When an object's surface is non-planar, the faceted triangle faces lie a small distance from the object's true surface. This distance is called the *surface deviation*, and it is measured in the model's units. The surface deviation is greater near the triangle centers and less near the triangle vertices.

The normal of a curved surface is different depending on its location, but it is constant for each triangle. (In this context, "normal" is defined as a line perpendicular to the surface.) The angular difference between the normal of the curved surface and the corresponding mesh surface is called the *normal deviation* and is measured in degrees.

The *aspect ratio* of triangles used in planar surfaces is based on the ratio of circumscribed radius to the in-radius of the triangle. It is unity for an equilateral triangle and approaches infinity as the triangle becomes thinner.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can modify the surface deviation, the maximum permitted normal deviation, and the maximum aspect ratio of triangles settings on one or more faces at a time in the **Surface Approximation** dialog box. (Click **HFSS** > **Mesh Operations** > **Assign** > **Surface Approximation**.)

The surface approximation settings are applied to the initial mesh.

> **Note:**
>
> For the initial mesh, all the vertices of the triangles lie on the true surfaces. During mesh refinement all the points are first added to the faceted surface for all products, not to the true surfaces. Then, for all Maxwell projects and for curvilinear meshing enabled HFSS projects , the points on the faceted surfaces will be moved to the curved surface locations when such movement does not degrade the quality of the mesh.

Guidelines for Modifying Surface Approximation Settings

If you intend to modify the surface approximation settings for an object face or faces, keep the following guidelines in mind:

- When necessary, override the default surface approximation settings to represent curved surfaces more accurately. More accurate representation will increase the mesh size and consume more CPU time and memory. The default settings are adequate for most circumstances.
- If you want to obtain a faster solution by using a cruder representation of curved surfaces, set the coarser setting for the whole object, not just a single face.
- It is difficult to satisfy aspect ratio demands if the aspect ratio value is set close to 1 because an arbitrary shape cannot be filled with only equilateral triangles. Therefore, setting the aspect ratio to 1 can lead to unreasonably large meshes. The aspect ratio is limited to 4 for planar objects and 1.2 for curved objects.

Meshing Region Vs. Problem Region

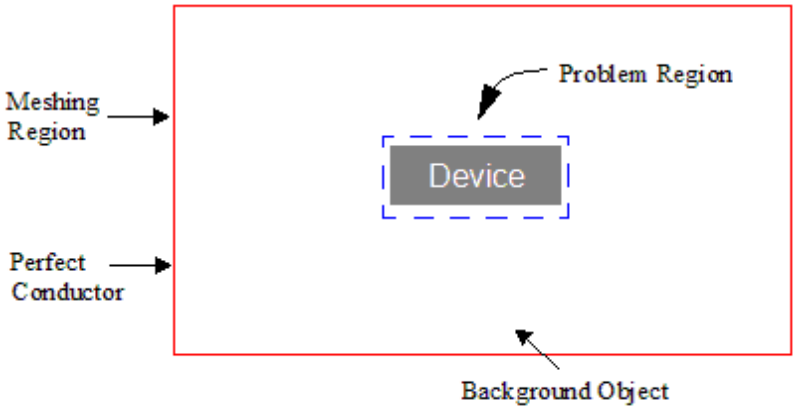
HFSS distinguishes between the problem region and the meshing region. The *problem region* is the region in which the solution is generated and the mesh is refined. The *meshing region*, which includes the problem region, is the area in which an initial mesh is generated. After an initial mesh is generated, the mesh is refined only in the problem region.

The problem region encompasses an area that is just large enough to include the entire design, but no larger. HFSS automatically defines the problem region during the solution process. If you are interested in effects outside of the structure, such as radiated effects, then you can create a virtual object to expand the size of the problem region to include these areas.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The meshing region, like the problem region, is a box that completely encloses the structure. However, the meshing region must be at least 10 times larger than the model. The part of the meshing region not occupied by objects is considered to be the *background* object. The background extends to the boundaries of the meshing region and fills in any voids not occupied by objects. Because the background object is defined as a perfect conductor, no solution is generated inside the background even though an initial mesh is generated for it. HFSS automatically defines the meshing region during the solution process.

The problem region and the meshing region are illustrated below.



Model Resolution

Model Resolution is a setting that determines the smallest details of a model that the mesher should capture and represent in the mesh.

Many times the analysis starts with the geometry already drawn in a different tool for different purpose. Some tools are designed for manufacturing and the resulting models contain lots of extra details not needed for electromagnetic analysis. If the user removes such details in the original tool the results will be better. But if the user does not have access to the original drawing tool or redrawing the model without these details is not possible, **Model Resolution** is another way to remove the details from analysis.

When the user sets the model resolution length to be L, the mesher will start with a surface representation of the model accurate to the modeler's tolerance limit. Then it will progressively remove edges, move points, merge points etc., within the allowable model resolution limit and simplify the surface mesh. During this process, tiny fillets, rounds, and chamfer protrusions are removed.

Other common model translation anomalies are also handled using **Model Resolution**. For example, some geometry engines will blindly export all of the surfaces as splines. When a user imports such a model for analysis, it would result in very large number of triangles. If the surface

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

can be represented by a smaller set of triangles using Model Resolution, this procedure would reduce the number of triangles in the surface mesh.

The user can start with a model resolution length around 0.1*wavelength. If the model resolution length chosen by the user is too large, the mesher will detect it and report it as an error. The model resolution length is specified in the user units of the modeler. It can be set on selected bodies only. The default value is 100* the tolerance limit of the modeler.

Port Solutions

The excitation field pattern at each port must be calculated before the full 3D electromagnetic field inside a structure can be calculated. HFSS calculates the natural field patterns (or modes) that can exist inside a transmission structure with the same cross-section as the port. The resulting 2D field patterns serve as boundary conditions for the full 3D problem.

- Excitation Fields
- Wave Equation
- Mesh Refinement on Ports
- Modes
- Multiple Ports on the Same Face
- Port Accuracy
- Calculating Characteristic Impedance
- Complex Propagation Constant

Excitation Fields

HFSS assumes that each port is connected to a uniform waveguide that has the same cross-section as the port. A waveguide supports an infinite number of modes with a given field pattern and propagation constant.

Therefore, the excitation field is one of the modes specified for a port where mode *m* can be expressed by:

$$\boldsymbol{E}_m(x,y,z,t) = \Re\left[\boldsymbol{E}_m(x,y)e^{j\omega t-\gamma_m z}\right] \quad (1)$$

where

- $\Re$ is the real part of a complex number or function.
- **Em**(x,y) is the electric field mode pattern of mode *m*.
- $\gamma_m=\alpha_m + j\beta_m$ is the complex propagation constant of mode *m*, where
- α_m is the attenuation constant of mode m.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- β_m is the propagation constant of mode m that determines, at a given time *t*, how the phase angle varies with z.
- $\omega = 2\pi f$ is angular frequency of oscillation,
- *j* >is the imaginary unit, $\sqrt{-1}$.

In this context, the x- and y-axes are assumed to lie in the cross-section of the port; the z-axis lies along the direction of propagation.

Wave Equation

The field pattern of a traveling wave inside a waveguide can be determined by solving Maxwell's equations. For each mode m, the port solver solves the following equation obtained from Maxwell's equation.

$$\nabla\times\left(\frac{1}{\mu_r}\nabla\times\boldsymbol{E}_m(x,y)e^{-\gamma_m z}\right)-k_0{}^2\varepsilon_r\boldsymbol{E}_m(x,y)e^{-\gamma_m z}=0 \qquad (1)$$

where

- $k_0 = \omega/c$ is the wave number of free-space
- $\mu_r(x,y)$ is the complex relative permeability.
- $\varepsilon_r(x,y)$ is the complex relative permittivity.

By solving the equation, the electric field mode pattern $E_m(x,y)$ and the propagation constant γ_m are both obtained for all the modes specified for a given port. For the current based impedance calculations, the port solver also solves independently for $H_m(x,y)$ using a corresponding equation for the magnetic field.

Also note that the excitation field pattern computed by the port solver is valid only at a given frequency. A different excitation field pattern is computed for each frequency point of interest.

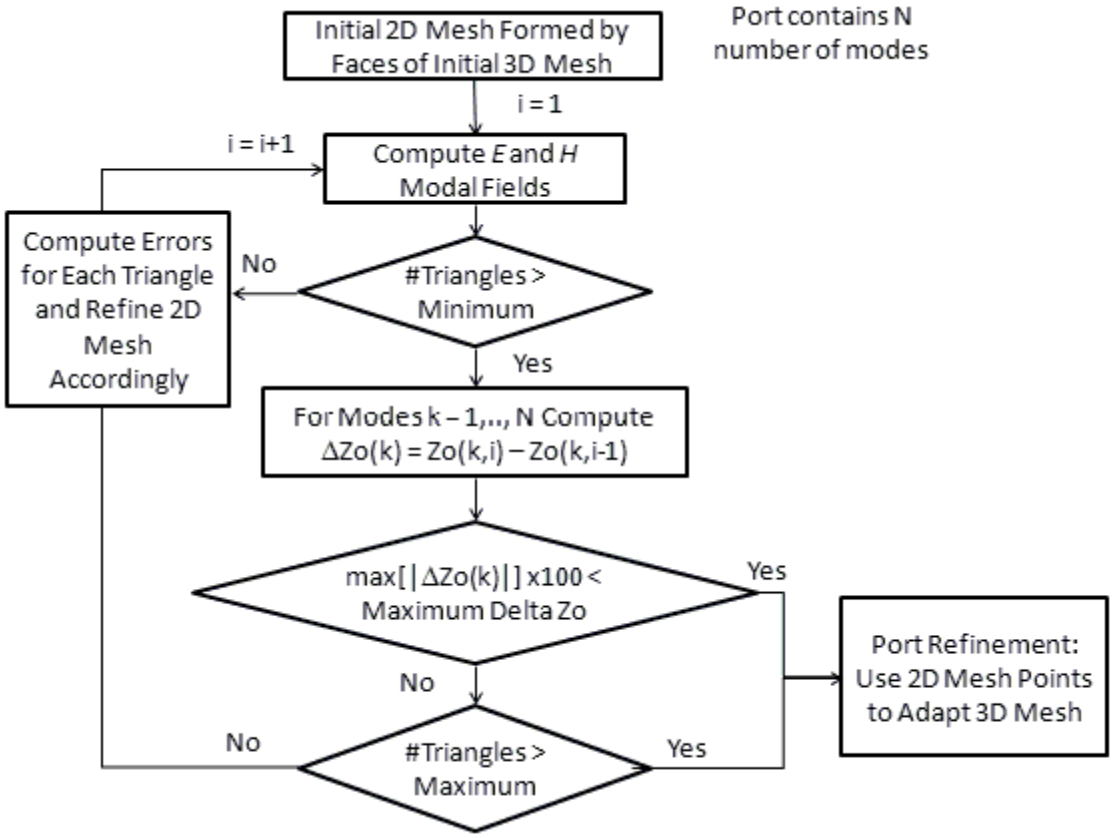
Mesh Refinement on Ports

The port solver computes mode patterns and propagation constants based on solutions of a finite element system of equations. The finite element mesh associated with each port is a 2D mesh of triangles corresponding to the faces of the tetrahedra that lie on the planar port surface. The port solver performs an iterative refinement of the triangular mesh.

User settings for the iterative refinement can be found under Port Options in the **Advanced** tab of the **Solution Setup** window. Those settings are reflected in the flow chart below that describe the refinement procedure where #Triangles refer to the number of triangles of the current mesh in the refinement process. The numerical implementation of $\Delta Zo(k)$ in the flow chart is actually based on relative difference when $|Zo(k)| > 1.0$. Moreover, for terminal projects the comparison

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is done for each entry of the terminal characteristic impedance matrix rather than the modal impedances such that the worst entry in the impedance matrix satisfies the user specified value.



For a good technical discussion of the port solution procedure, refer to the following:

Jin-Fa Lee, Din-Kow Sun, and Zoltan J. Cendes, "Full-Wave Analysis of Dielectric Waveguides Using Tangential Vector Finite Elements," *IEEE Transactions on Microwave Theory and Techniques*, vol. 39, No 8, August 1991.

Modes

For a waveguide or transmission line with a given cross-section, there is a series of basic field patterns, or modes, that satisfy Maxwell's equations at a specific frequency. Any linear combination of these modes can exist in the waveguide. A user is responsible to specify the number of modes to be included in a wave port but not for lumped ports since they are restricted to contain only one mode. In general, in order to obtain accurate S-parameters the number of modes defined for a given port should be the same or greater than the number of propagating modes at the highest frequency of interest. Note that the port solver automatically determines what modes should be included based on the specified number of modes. The modes included in a port are based on a sorting criteria to extract the dominant set of modes out of the infinite set

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of possible modes. If there is no prior knowledge to the number of propagating modes, the user can do a ports only solution while specifying a reasonably large number of modes and view the 'Gamma' values in the Matrix Data panel to determine what modes are propagating.

If a port is defined on a surface where there is a geometrical discontinuity in the normal direction of the port, it may not be enough to include only the propagating modes for accurate S-parameters. If possible, it is recommended to avoid this case by modeling a short section of the port transmission line such that all the higher order modes have decayed and do not contribute to the field on the port surface. The length of the constant cross-section segment that has to be included in the model depends on the value of the mode's attenuation constant, α.

Degenerate Modes

Degenerate modes refer to the case when a set of modes have the same complex propagation constant which means that any linear combination of those modes is still a valid set of natural (or eigen) modes. When you become aware that a wave port contains degenerate modes as a part of the requested modes, you should use the Mode Alignment feature in HFSS.

Multiple Ports on the Same Face

Visualize a port face on a microstrip that contains two conducting strips side by side as two separate ports. If the two ports are defined as being separate, the system simulates the case in which the two ports are connected to uncoupled transmission structures. It is as if a conductive wall separates the excitation waves. However, in actuality, there will be electromagnetic coupling between the two strips.

To model this coupling accurately, analyze the two ports as a single port with multiple modes. In general, if there are N disconnected conductors in the port cross-section, at least N - 1 modes are required for an accurate solution. For example, if the port consists of two adjacent microstrip lines surrounded by a conducting enclosure, N = 3; therefore at least two modes should be defined on the port. Assign an equal number of terminals as modes. Refer to Assigning Wave Ports for Terminal Solutions for more information.

If the multi-conductor port plane is near discontinuities within the 3D model, additional modes beyond *N* - 1 may be necessary. However, if you define terminals on a multi-conductor port, the presence of non-quasi transverse electromagnetic (TEM) modes will adversely affect the entries of any computed terminal matrices. Therefore, rather than increase the number of modes beyond the required *N* - 1, extend the port outward until any higher-order modes have sufficient attenuation to be omitted from consideration.

Port Accuracy

Generally, the port accuracy defaults**,** specified under the Port Options in the Advanced tab of the **Solution Setup** window, are adequate. You may want improved port accuracy under the following conditions:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You are interested primarily in the port impedances. Port impedances are computed as part of the port solution.
- You need to lower the noise floor to catch S-parameters that are expected to be in the –70 dB range.

The port accuracy settings are only used for port adapt. If those settings are modified after port adapt nothing will happen. Therefore, to specify new port accuracy settings, add another solution setup and generate a new solution.

Refining the mesh at the ports causes HFSS to refine the mesh for the entire structure as well. This occurs because it uses the port field solutions as boundary conditions when computing the full 3D solution. Therefore, specifying too small a port field accuracy can result in an unnecessarily complex finite element mesh.

Calculating Characteristic Impedance

Each port in a structure being analyzed can be viewed as a cross-section of a transmission line. HFSS computes the characteristic impedance of each port in three ways — as Zpi, Zpv, and Zvi impedances. You have the option of specifying which impedance will be used in the renormalization calculations.

- For TEM waves, the Zvi impedance converges on the port's actual impedance and should be used.
- When modeling microstrips, it is sometimes more appropriate to use the Zpi impedance.
- For slot-type structures (such as inline or coplanar waveguides), Zpv impedance is the most appropriate.

HFSS will always calculate Zpi impedance, the impedance calculation using power and current, which are well-defined for a port because they are computed over the area of the port. Zpv and Zvi are not calculated by default. This is because V is computed by integrating along a user-defined integration line. To renormalize the solution to a Zpv or Zvi characteristic impedance, you must have defined an impedance line.

Under the **Matrix Data** tab of the **Solution Data** dialog box, the characteristic impedance can be displayed as magnitude/ phase, real/ imaginary, magnitude, phase, real, or imaginary.

For more information on the computation of impedances, refer to the following:

Bruno Bianco, Luigi Panini, Mauro Parodi, and Sandro Ridella, "Some Considerations about the Frequency Dependence of the Characteristic Impedance of Uniform Microstrips," *IEEE Transactions on Microwave Theory and Techniques*, vol. MTT-26 No. 3, March 1978.

Edward F. Kuester, David C. Chang, and Leonard Lewin, "Frequency-Dependent Definitions of Microstrip Characteristic Impedance," *International URSI Symposium on Electromagnetic Waves*, Munich, 26-29 August 1980, pp. 335 B/1-3.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculating the PI Impedance

The Z_{pi} impedance is the impedance calculated from values of power (P) and current (I):

$$Z_{pi} = \frac{P}{I \cdot I} \quad (1)$$

The power and current are computed directly from the simulated fields. The power passing through a port is equal to the following:

$$P = \oint_S E \times H ds \quad (2)$$

where the surface integral is over the surface of the port.

The current is computed by applying Ampere's law to a path around the port:

$$I = \oint_l H \bullet dl \quad (3)$$

While the net current computed in this way will be near zero, the current of interest is that flowing into the structure, I^-, or that flowing out of the structure, I^+. In integrating around the port, HFSS keeps a running total of the contributions to each and uses the average of the two in the computation of impedances.

> **Note:**
>
> Zpi calculation is modified when a symmetry E plane touches a port. The current on the symmetry E plane is included in the Zpi calculation for a port with more than one conductor, where as for a single conductor port, such as a waveguide, the current on the symmetry E plane is not included in the Zpi calculation.

Calculating the PV Impedance

The Z_{pv} impedance is the impedance calculated from values of power (P) and voltage (V):

$$Z_{pv} = \frac{V \bullet V}{P} \quad (1)$$

where the power and voltage are computed directly from the simulated fields.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The power is computed in the same way as the Z_{pi} impedance. The voltage is computed as follows:

$$V = \int_l E \bullet dl \quad (2)$$

The result over which HFSS integrates is referred to as the impedance line — which is defined when the ports are set up. To define the impedance line for a port, select the two points across which the maximum voltage difference occurs. You must define an integration line to specify where the maximum voltage difference will be.

Calculating the VI Impedance

The Z_{vi} impedance is given by:

$$Z_{vi} = \sqrt{Z_{pi} Z_{pv}}$$

For TEM waves, the Z_{pi} and Z_{pv} impedances form upper and lower boundaries to a port's actual characteristic impedance. Therefore, the value of Z_{vi} approaches a port's actual impedance for TEM waves.

Impedance Multipliers

If a symmetry plane has been defined (allowing the model of a structure to be cut in half), the impedance computations must be adjusted by specifying an impedance multiplier. The need for this multiplier can be understood by looking at how the use of symmetry affects the computation of Z_{pv}.

In cases where a perfect E plane of symmetry splits a structure in two, only one-half of the voltage differential and one-half of the power flow can be computed by the system. Therefore, since the Z_{pv} impedance is given by:

$$Z_{pv} = \frac{V \bullet V}{P}$$

the computed value is one-half the desired value. An impedance multiplier of 2 must be specified in such cases.

In cases where a perfect H plane of symmetry splits a structure in two, only one-half of the power flow is seen by the system but the full voltage differential is present. Therefore, structures split in half with perfect H symmetry planes result in computed impedances that are twice those for the full structure. An impedance multiplier of 0.5 must be specified in such cases.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If multiple symmetry planes are used or if only a wedge of a structure is modeled, you must adjust the impedance multiplier accordingly.

It is important to be careful with the impedance multiplier when using symmetry planes in order to get the right answer. In general, there does not exist a simple rule for a multiplier and it is problem dependent. However, in most cases the following values can be used for the impedance multiplier:

- If the structure has a perfect E plane of symmetry, use 2. Such models have one-half of the voltage differential and one-half of the power flow of the full structure, resulting in impedances that are one-half of those for the full structure.
- If the structure has a perfect H plane of symmetry, enter 0.5. Such models have the same voltage differential but half the power flow of the full structure, resulting in impedances that are twice those for the full structure.
- If the structure has a combination of perfect H and perfect E boundaries, adjust accordingly. For example, you do not have to enter an impedance multiplier for a structure with both a perfect E and perfect H boundary since you would be multiplying by 2 and 0.5.

Calculating Terminal Characteristic Impedance Matrix

See Modes to Terminals Conversion.

Complex Propagation Constant

Each port is assumed to be connected to a transmission structure that has the same cross-section as the port. The complex propagation constant, γ, of these transmission lines is computed by HFSS, and is given by $\gamma = \alpha + j\beta$, where:

- α is the attenuation constant of a signal in the transmission structure. It is the real component of the propagation constant and has units of nepers per meter.
- β is the phase constant associated with the wave. It is the imaginary component of the propagation constant and has units of radians per meter.

Under the **Matrix Data** tab of the **Display Items Dialog**, gamma can be displayed as magnitude/ phase, real/ imaginary, magnitude, phase, real, or imaginary.

Calculating the Effective Wavelength (Lambda)

The effective wavelength, λ_{eff}, is calculated from

$$\lambda_{eff} = \frac{2\pi}{\beta} \quad (1)$$

where β is the phase constant associated with the wave.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Under the **Matrix Data** tab of the **Solution Data** dialog box, lambda is displayed when **Gamma** is selected as the matrix type.
s

Calculating the Relative Permittivity (Epsilon)

The relative permittivity, ε_r, is calculated using

$$\lambda_{eff} = \frac{c}{\sqrt{\varepsilon_r} f} \quad (1)$$

where

- λ_{eff} is the effective wavelength given in meters.
- *c* is the speed of light.
- *f* is the frequency of the wave.

Under the **Matrix Data** tab of the **Solution Data** dialog box, epsilon is displayed when **Gamma** is selected as the matrix type.

Maximum Delta S

For designs with ports.

The delta S is the change in the magnitude of the S-parameters between two consecutive passes. If the magnitude and phase of all S-parameters change by an amount less than the **Maximum Delta S Per Pass** value from one iteration to the next, the adaptive analysis stops. Otherwise, it continues until the requested number of passes is completed.

For example, if you specify **0.1** as the **Maximum Delta S Per Pass**, HFSS continues to refine the mesh until the number of requested passes is completed or until the magnitude of the complex delta of all S-parameters changes by less than **0.1**.

The maximum delta S is defined as

$$Max_{ij}\left|S_{ij}^{N} - S_{ij}^{N-1}\right| \quad (1)$$

where:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- *i* and *j* cover all matrix entries.
- *N* represents the pass number.

> **Note:**
>
> Delta S is computed on the appropriate S-parameters - modal or terminal - after the S-parameters have been de-embedded and renormalized.

Maximum Delta E

For designs with voltage sources, current sources, or incident waves.
Not applicable to designs with ports.

The delta E is the difference in the relative energy error from one adaptive solution to the next. It is a measure of the stability of the computed field values from pass to pass. As the solution converges, delta E approaches zero.

The **Maximum Delta E Per Pass** value is a stopping criterion for the adaptive solution. If the delta E falls below this value, the adaptive analysis stops. Otherwise, it continues until the convergence criteria are reached.

The data represents the delta E for all tetrahedra.

Percent of Tetrahedra Refined Per Pass

The value you set for **Percent Refinement Per Pass** determines how many tetrahedra are added at each iteration of the adaptive refinement process. For instance, entering **30** causes the mesh to increase approximately 30 percent each pass. The tetrahedra with the highest error will be refined. If your mesh consisted of 1000 elements, the tetrahedra would be refined so that 300 new elements are added to the mesh. Generally, you can accept the default value.

Magnitude Margin

For solutions in which convergence criteria for specific S-matrix entries were specified.

For each element in the S-matrix, the magnitude margin is the difference between the S-parameter delta magnitude and the target delta magnitude, as specified in the **Matrix Convergence** dialog box. The magnitude margin reported under the **Convergence** tab is the maximum of these values over the entire matrix. The magnitude margin is defined as

$$Max_{ij}\left[\left|S_{ij}^{N}-S_{ij}^{N-1}\right|-\left|M_{ij}\right|\right] \quad (1)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where M_{ij} is the matrix convergence entry. It indicates the solution's proximity to the target delta magnitude. If the solution has converged within the target delta magnitude, a value of zero will be reported for the pass.

Phase Margin

For solutions in which convergence criteria for specific S-matrix entries were specified.

For each element in the matrix, the phase margin is the difference between the S-parameter delta phase and the target delta phase, which was specified in the **Matrix Convergence** dialog box. The phase margin reported under the **Convergence** tab is the maximum of these values over the entire matrix. The phase margin is defined as

$$Max_{ij}\left[\left|phaseS_{ij}^{N} - phaseS_{ij}^{N-1}\right| - phaseM_{ij}\right] \quad (1)$$

where M_{ij} is the matrix convergence entry. The phase margin indicates the solution's proximity to the target delta phase. If the solution has converged within the target delta phase, a value of zero will be reported for the pass.

> **Note:**
>
> When the Mag S becomes small (near to zero) its phase becomes indefinite and insignificance due to mathematical issue so that Phase Margin will be discarded.

Maximum Delta Frequency

For Eigenmode solutions.

At any time during the solution process, you can view the percent difference in the resonant frequencies from one adaptive solution to the next, or the maximum delta frequency. This is a measure of the stability of the computed frequencies from pass to pass and is available only two or more adaptive passes are completed.

For lossless problems, the maximum delta frequency is the largest percent change in the real part of the frequency for any of the calculated modes. For lossy problems, the maximum delta frequency is the greater of two quantities: the largest percent change in the real part of the frequency over all the modes, and the largest percent change in the imaginary part of the frequency.

Max Delta (Mag S)

For solutions in which convergence criteria for specific S-matrix entries were specified.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Max delta(Mag S) is the maximum difference of S-Matrix magnitudes between two consecutive passes. If the difference in magnitudes of the S matrices change by an amount less than the **Maximum Delta Mag S** value from one pass to the next, this satisfies the part of the convergence criteria.

$$Max\left|S_{ij}^{N} - S_{ij}^{N-1}\right| \quad (1)$$

Max Delta (Phase S)

For solutions in which convergence criteria for specific S-matrix entries were specified.

The Max delta(Phase S) is the maximum difference of S-Matrix phase between two consecutive passes. If the difference in phase of the S matrices change by an amount less than the **Maximum Delta Phase S** value from one pass to the next, this satisfies this part of the convergence criteria.

$$Max(PhaseS_{n} - PhaseS_{n-1}) \quad (1)$$

Matrix Solvers

HFSS includes two solvers. The default solver and the Iterative Matrix Solver. See Enable Iterative Solver for a discussion of when to use the Iterative Matrix solver. For details on the default solver see Direct Matrix Solver. For details on the Iterative Matrix solver see *Technical Notes*: Iterative Matrix Solver.

Direct Matrix Solver

Direct methods obtain an exact solution to the following linear system of equation

$$Ax = b \quad (1)$$

where A is a matrix, b a right hand side and x the solution. A matrix solver using a direct method is called a direct matrix solver.

The most widely known direct method is Gaussian elimination, which uses elementary matrix operations to compute the solution. Most other direct solution techniques are either a variant of Gaussian elimination or are based upon a particular factorization of the equations that will allow an exact computation. One of the most commonly used such techniques is the LU decomposition, which is introduced below.

LU decomposition is both a factorization approach and closely related to Gaussian elimination. It is based upon the assumption that A can be decomposed into a product of two matrices or factors.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$A = LU \quad (2)$$

where L is a lower triangular matrix (has elements only on the diagonal and below) and U is an upper triangular matrix (has elements only on the diagonal and above).

With the help of (2), we can equivalently solve (1) by first solving for y such that

$$Ly = b \quad (3)$$

then solving for x such that

$$Ux = y \quad (4)$$

The advantage of decomposition (2) is that the solution of triangular set of equations (3) and (4) is quite trivial. Namely, they can be solved directly by forward substitution and by backward substitution respectively. Furthermore, the factors in (2) are computed only once and can be reused in (3) and (4) for different right hand sides.

In LU decomposition, the major storage is for matrix A, factor L and factor U. The major operation is the decomposition (2), which is roughly equivalent to a matrix-matrix multiplication; while the operation in (3) and (4) combined, is equivalent to a matrix-vector multiplication. The total computational cost is S+mT with S>>T in general, where S is the number of operations for decomposition (2), m the number of right hand sides, and T the number of operations for both forward substitution and for backward substitution.

Direct methods are best used for solving system matrix equations with moderate size or with a large number of right hand sides. As the system to be solved becomes larger, the overhead associated with the more complicated iterative methods becomes less of an issue, and the iterative methods should outperform the direct methods. For sparse systems, the use of direct methods is complicated by the possible introduction of more nonzero entries (fill-in) such that the L and U factors become much denser than the original matrix A.

Both the distributed and non-distributed direct solvers automatically trade speed for memory when that is necessary.

Iterative Matrix Solver

This section contains information on the Iterative Matrix Solver.

- Guidelines for Using the Iterative Solver
- Multiprocessing and the Iterative Solver
- Iterative Solver Technical Details

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Guidelines for Using the Iterative Solver

1. The iterative solver works most efficiently when it is enabled for designs that do not contain many excitations. (For example, the number of excitations is less than twice the number of processors.)
2. If you choose to take advantage of the iterative solver, and your analysis includes interpolating sweeps or discrete sweeps, the adaptive solution should be well converged at the higher end of the frequency band.
3. The **Relative Residual** provides a stopping criteria. The residual measures the convergence of the iterative solver to the solution of the matrix equation. Its value affects the performance of the iterative solver as follows:

- Default is 1E-6. This gives accurate S-parameters and fields, indistinguishable from those generated by the direct solver. Ansys recommends this residual.
- With a larger residual, for example, 1E-3 or 1E-2, the iterative process will stop with fewer iterations and the solution will be less converged. S-parameters won't differ much from those of a direct solution, for example, a difference in third or second digit. Fields and antenna patterns are visually the same.
- A residual of 0.1 can be used for quick adaptive mesh refinement early in the adaptive process, but S-parameters will be noticeably different.
- A residual of 1 should never be used. The interface will not allow a residual above 0.1.

Multiprocessing and the Iterative Solver

In computing the pre-conditioner, the iterative solver uses multiple processors, if set under **Tools>Options>HFSS Options**, just like the multi-frontal solver does. The operations involved are very similar. After that, there is one iterative process per excitation. Each iterative process can only use one processor. If you have one excitation and multiple processors, you will see that HF3D.exe uses only one processor at this stage. If you have multiple excitations and multiple processors, HF3D.exe will use multiple processors at this stage. For example, with four processors and eight excitations, HF3D will perform the iterative processes for four excitations first, and for the other set of four excitations next.

Iterative Matrix Solver Technical Details

The iterative solver always saves significant memory - easily a factor two with simulations of intermediate size, and more with larger simulations. It is also faster than the multi-frontal solver with large simulations.

The following table shows asymptotic behavior with large simulations. N = number of unknowns. This shows that the iterative solver has a better asymptotic behavior both in RAM and in time.

	Time	RAM
Multi-frontal	$N^{1.7}$	$N^{1.3}$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solver		
Iterative Solver	N $^{1.2}$	N $^{1.0}$

The iterative solver uses a preconditioner that is based on the next-lower order. Therefore, there is no iterative solver option when you solve with zeroth order.

Consider the matrix equation

$$Ax = b \quad (1)$$

where *A* is a matrix, *b* a right hand side and *x* the solution.

When $A^{-1}$is computationally expensive or the exact solution *x* is impossible, an alternative is to seek an approximation $\bar{x}$ to *x*, with an error $e = x - \bar{x}$. The exact solution can therefore be rewritten as

$$x = \bar{x} + e \quad (2)$$

Substituting (2) into (1) results in the so called residual equation

$$Ae = r \quad (3)$$

where *r* is the residual defined by

$$r = r - A\bar{x} \quad (4)$$

As aforementioned, the exact solution for *e* in (3) is impossible since it requires A^{-1}. However, if an approximation $M \approx A$ is available, the error *e* can be approximated in (3) by

$$\bar{e} = M^{-1}r \quad (5)$$

$\bar{x}$

$$\bar{x} \leftarrow \bar{x} + \bar{e} \quad (6)$$

It is (4)-(6) that form the foundation of the iterative solution method. A matrix solver using the iterative solution method is called an iterative matrix solver. The method starts with an initial guess $\bar{x} = x_0$ and repeats (4)-(6) until the approximation $\bar{x}$ to *x* is within tolerance, or the number of iterations exceeds a given number. In the former case, it is said the solution converges; while in the latter, it doesn't.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The residual *r* is used for measuring the closeness of $\bar{x}$ to *x*. Since *A* and *b* in (1) can be scaled by the same factor without altering *x*, so does the residual *r* in (4). It typically makes more sense to replace ||*r*|| as the stopping criterion with the **relative residual**:

$$res = \frac{\|r\|}{\|b\|} \quad (7)$$

$\| \|$

M in (5) is called a preconditioner of *A*. A good preconditioner greatly reduces the number of iterations. The following makes *M* a good preconditioner: *M* is very similar to *A* in eigen spectrum and M^{-1} is computationally cheap. Some of the classic iterative matrix methods include: the Jacobi method where *M* is the diagonal of *A,* the Gauss-Seidel method where *M* is the lower triangular or upper triangular matrix of *A* and the successive over-relaxation method (SOR) where *M* is a weighted combination of the lower triangular and upper triangular matrix of *A.*

In the iterative method, the major storage is for matrix *A* and for preconditioner *M.* The major operation is a matrix-vector multiplication in (4). The computational cost is *S*+*mnT,* where *S* is the number of operations for setting up the preconditioner*,m* the number of right hand sides, *n* the average number of iterations per right hand side and *T* the number of operations for each iteration*.*

Details of the iterative solver in HFSS are given in:

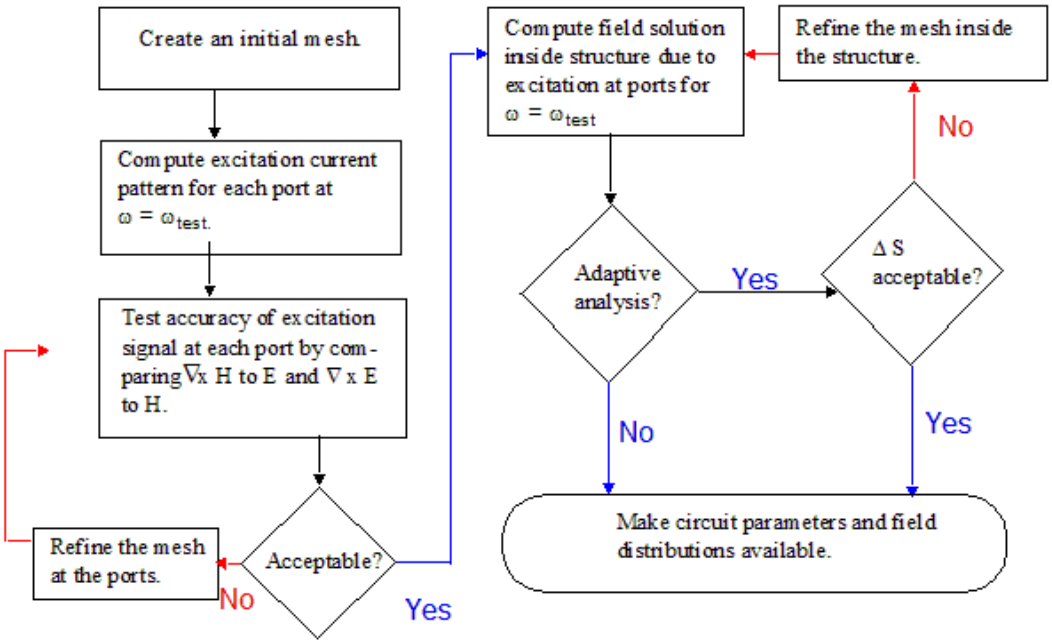
D. K. Sun, J. F. Lee and Z. J. Cendes, "Construction of nearly orthogonal Nedelec bases for rapid convergence with multilevel preconditioned solvers", *SIAM Journal on Scientific Computing*, Vol. 23, No. 4, pp. 1053-1076, 2001.

I. Bardi, G. Peng and Z.J. Cendes, "Improvements in Adaptive Mesh Refinement and Multi-level methods in High Frequency Electromagnetics", ACES Symposium, 2002.

Single Frequency Solution

A single frequency solution generates an adaptive or non-adaptive solution at a single frequency, the solution frequency specified in the **Solution Setup** dialog box, and is often the first step in performing a frequency sweep. An adaptive solution is one in which a finite element mesh is created and automatically refined in the areas of highest error — increasing the accuracy of succeeding adaptive solutions. The procedure for performing a single frequency solution is shown below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Frequency Sweeps in HFSS

Perform a frequency sweep when you want to generate a solution across a range of frequencies. You may choose one of the following sweep types:

Fast	Generates a unique full-field solution for each division within a frequency range. Best for models that will abruptly resonate or change operation in the frequency band. A Fast sweep will obtain an accurate representation of the behavior near the resonance.
Discrete	Generates field solutions at specific frequency points in a frequency range. Best when only a few frequency points are necessary to accurately represent the results in a frequency range.
Interpolating	Estimates a solution for an entire frequency range. Best when the frequency range is wide and the frequency response is smooth, or if the memory requirements of a Fast sweep exceed your resources.

Fast Frequency Sweeps

A Fast sweep generates a unique full-field solution for each division within a frequency range. Choose a Fast sweep if the model will abruptly resonate or change operation in the frequency band. A Fast sweep will obtain an accurate representation of the behavior near the resonance.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HFSS uses the center frequency of the frequency range to select an appropriate eigenvalue problem with which to generate a solution for the entire Fast sweep. It then uses an Adaptive Lanczos-Padé Sweep (ALPS)- based solver to extrapolate the field solution across the requested frequency range from the center frequency field solution.

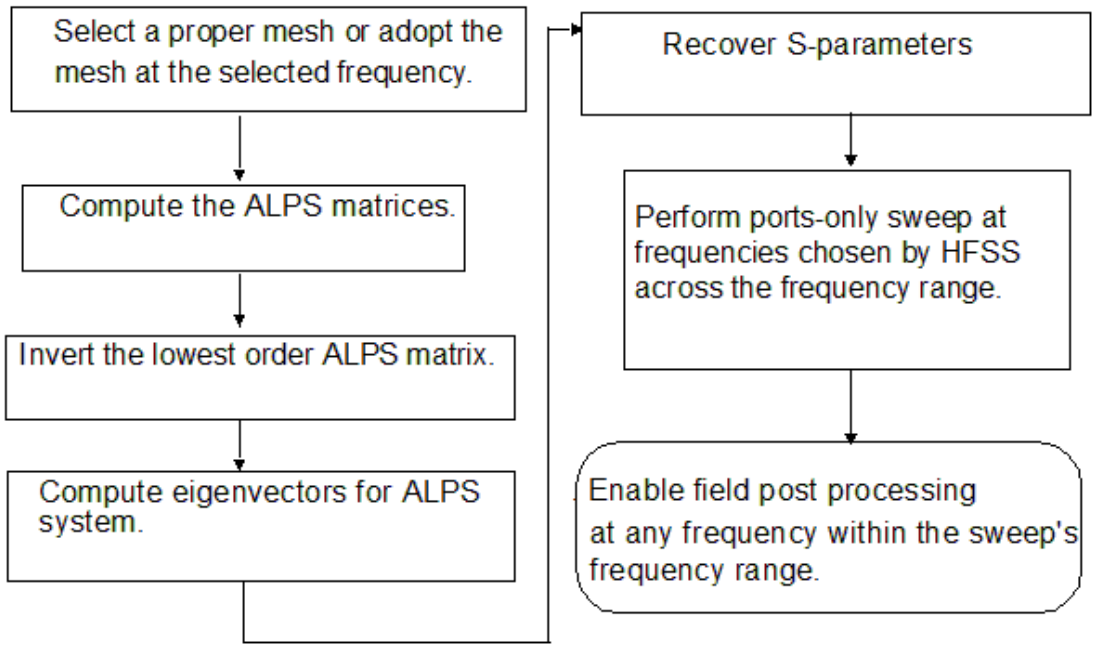
HFSS always uses the middle of the frequency range and the center frequency.

Be aware that HFSS uses the finite element mesh refined during an adaptive solution at the solution frequency or, if you did not request an adaptive solution, the initial mesh generated for the problem. It uses this mesh without further refinement. Also, the field solution at the center frequency is the most accurate. Depending upon the desired level of accuracy you require throughout the frequency range, you may wish to perform additional Fast sweeps at other center frequencies.

The full-field solution is saved only at the center frequency, while the S-parameters are saved for every frequency point; however, the Fast sweep allows the you to post process fields for any frequency entries to the sweep range.

The time and memory required for a Fast sweep may be significantly greater than the time and memory required for a single frequency solution.

The procedure for a Fast frequency sweep is shown below.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discrete Frequency Sweeps in HFSS

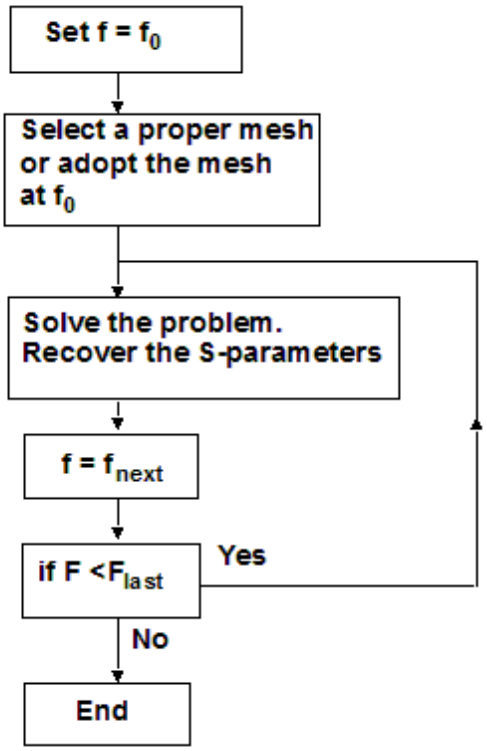
A Discrete sweep generates field solutions at specific frequency points in a frequency range. For example, if you specify a range of 1000 MHz to 2000 MHz, then a Step Size of 2.5, the result would be solutions at 1000, 1250, 1500, 1750, and 2000 MHz. By default, the field solution is only saved for the final frequency point computed, which would be at 2000 MHz in this case. Select the **Save Fields** option when setting up the points to solve if you want to save the field solution for a specific point. The S-parameters are saved for every frequency point. The more steps you request, the longer it takes to complete the frequency sweep.

Choose a Discrete sweep if only a few frequency points are necessary to accurately represent the results in a frequency range.

Be aware that HFSS uses the finite element mesh refined during an adaptive solution at the solution frequency or, if you did not request an adaptive solution, the initial mesh generated for the problem. It uses this mesh without further refinement. Because the mesh for the adaptive solution is optimized only for the solution frequency, it is possible that the accuracy of the results could vary at frequencies significantly far away from this frequency. If you wish to minimize the variance, you can opt to use the center of the frequency range as the solution frequency. Then, after inspecting the results, run additional solutions with the solution frequency set to the critical frequencies.

The procedure for a Discrete frequency sweep is shown below, where *n* equally spaced frequencies are included in the sweep.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Interpolating Frequency Sweeps in HFSS

An Interpolating sweep estimates a solution for an entire frequency range. HFSS chooses the frequency points at which to solve the field solution so that the entire interpolated solution lies within a specified error tolerance. The sweep is complete when the solution meets the error tolerance criterion or generates the maximum number of solutions. To view more information about the solution, increase the number of steps and perform the sweep again.

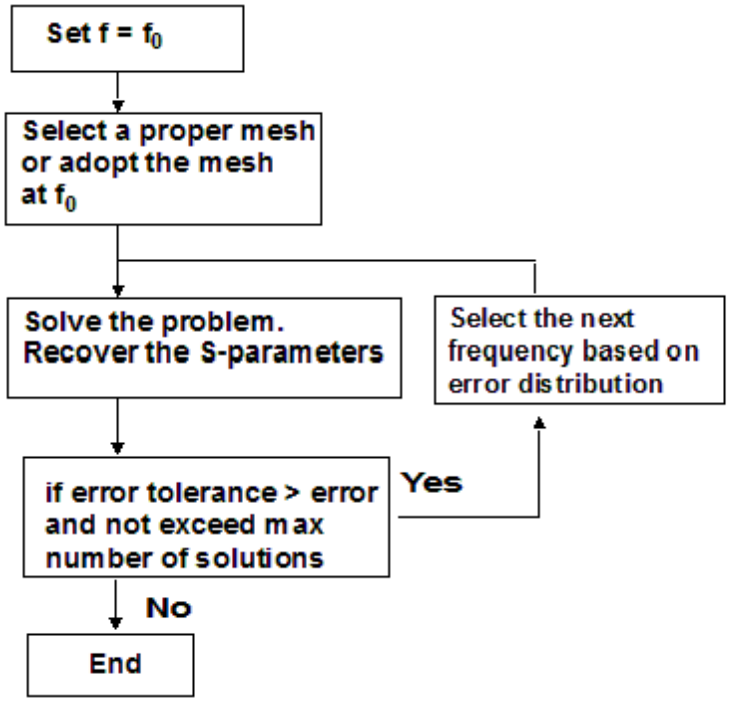
The field solution for each point is deleted so that a new field solution can be generated for the next point. The full-field solution is only saved for the final frequency point computed. The S-parameters are saved for every solved frequency point.

Choose an Interpolating sweep if the frequency range is wide and the frequency response is smooth with the exception of a few resonances, or if the memory requirements of a Fast sweep exceed your resources. An Interpolating sweep's time requirement may be much less than a Discrete sweep's because a solution for the entire frequency range is interpolated based on solutions for a minimal number of frequency points. The maximum time required for an Interpolating sweep is the time required for a single frequency solution multiplied by the maximum number of solutions.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Be aware that HFSS uses the finite element mesh refined during an adaptive solution at the solution frequency or, if you did not request an adaptive solution, the initial mesh generated for the problem. It uses this mesh without further refinement.

The procedure for an Interpolating sweep is shown below, where *n* frequencies determined by the system are included in the sweep.



Solution Types in HFSS

Solution Types	
HFSS	This enables the Driven Options Modal and Terminal. **HFSS** selection enables Mesh Fusion for FEM components. You can add padding to the mesh bounding volume. The finite array is disabled, and the Hybrid folder is removed from the project tree. The FEBI boundary becomes a regular boundary instead of hybrid region. When there are components set up for mesh fusion, the solution setup only allows direct solver and no derivatives.
HFSS with Hybrid	This enables the Driven Options Modal and

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and Arrays	Terminal. **HFSS with Hybrid and Arrays** selection enables mesh assembly for hybrid components. You cannot add padding to the mesh assembly bounding volume. The finite array is enabled, and the Hybrid folder is added to the project tree, along with the tab for Hybrid in the solution setup.
HFSS Transient	**Transient** - For calculating problems in the time domain. It employs a time-domain ("transient") solver. For **Transient** your choice of **Composite Excitation** or **Network Analysis** affects the options for the setup. If you select **Network Analysis** the setup includes an Input Signal tab for the simulation. Typical transient applications include, but are not limited to: • Simulations with pulsed excitations, such as ultra-wideband antennas, lightning strikes, electro-static discharge; • field visualization employing short-duration excitations; • time-domain reflectometry.
HFSS SBR+	This option simplifies design creation for SBR+ users. HFSS can use EMA3D shooting and bouncing ray (SBR) technology to calculate the far field from current sources and defined geometry via a one-way coupling. With this solution type, you do not need to specify explicit SBR+ Hybrid Regions. The Driven option for this solutions is Network Analysis only, with no Auto-Open Region. For details, see Design Flow for SBR+ Solution Type. You can also use parametric antennas in SBR+ solutions. For SBR+ solutions, you see an option in Initial Mesh Settings to **Allow tolerant meshing in SBR+ regions**. For HFSS SBR+solutions, you can also choose to import STL files as Lightweight Geometry.
HFSS Eigenmode	For calculating the eigenmodes, or resonances, of a structure. The Eigenmode solver finds the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	resonant frequencies of the structure and the fields at those resonant frequencies. Eigenmode designs cannot contain design parameters that depend on frequency, for example a frequency-dependent impedance boundary condition.
HFSS Characteristic Mode	This option is used for calculating the characteristic modes of a structure. The structure can be metal or dielectric. The solution reports the Number of Modes, the characteristic angle and current (amp/meter), the modal significance and quality factor, and the voltage per port based in edit sources weighting. The Selecting Characteristic Modes changes the Solution Setup criteria and dialog. You specify the minimum modal significance (default 0.02). Convergence is based on Max E rather than Max S (default (0.02). Only discrete sweeps are supported. Only the CMA solver is supported. Only lossless boundaries are allowed. Finite conductivity boundaries are allowed but are converted to lossless. The half-space boundary is not allowed.

Driven Modal

Choose the **Driven Modal** option for HFSS Driven when you want HFSS to calculate the modal-based S-parameters of passive, high-frequency structures such as microstrips, waveguides, and transmission lines. The S-matrix solutions will be expressed in terms of the incident and reflected powers of waveguide modes. **Network Analysis** is the default and functions as before.

Composite Excitation provides a method for solving fields in a large frequency domain problem.

Driven Terminal

Choose the **Driven Terminal** option for HFSS Driven when you want HFSS to calculate the terminal-based S-parameters of single and multi-conductor transmission line ports. The S-matrix solutions will be expressed in terms of voltages and currents on the terminals. **Network Analysis** is the default and functions as before.

Composite Excitation provides a method for solving fields in a large frequency domain problem.

Auto-Open Region

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For open region problems (typically antennas), you can choose **Auto-Open Region**. The option is available for HFSS Driven modal, terminal and transient solutions. This automatically creates an open region and a predefined Analysis setup for the project. You can select whether the region is Radiation, FE-BI, or PML. This simplifies the design process. If you do not choose **Auto-Open Region**, you must create an airbox and then assign a radiation boundary, either manually, or using the Create Open Region command. For more information on this Solution setting, see Using Auto-Open Region for the Solution Type for Antenna Designs.

Eigenmode Solutions

The Eigenmode solver can find the Eigenmodes of lossy as well as lossless structures, and can calculate the unloaded Q of a cavity. Q is the quality factor, and is a measure of how much energy is lost in the system. Unloaded Q is the energy lost due to lossy materials and boundary conditions. Because ports and other sources are restricted for eigenmode problems, the Q calculated does not include losses due to those sources.

The following restrictions apply to Eigenmode solution designs:

- No excitations may be defined.
- Radiation boundaries may not be defined.
- Frequency sweeps are not available.
- You may not view or plot the S-matrix data.
- Designs cannot include ferrite materials.

If a material or boundary condition, other than finite conductivity, varies with frequency, the eigenmode solution procedure will evaluate the frequency dependent property at the user specified minimum frequency and use that value during the S and T matrix assembly. This may limit the accuracy of the solved Eigenmodes depending on how significant the property varies from the minimum frequency to the eigen frequency.

Calculating the Resonant Frequency

Eigenmodes are the resonances of the structure. The eigenmode solver finds the resonant frequencies of the structure and the fields at those resonant frequencies. For a Driven Solution, HFSS solves the following matrix equation (for a lossless case):

$$Sx + k_o^2 Tx = b \quad (1)$$

where

- S and T are matrices that depend on the geometry, the materials, and the mesh.
- $\boldsymbol{x}$ is the electric field solution.
- k_o is the free-space wave number.
- b is the value of the source defined for the problem.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

However, in order to find the resonances of the structure, the eigenmode solver sets *b* to zero, and solves the equation

$$Sx + k_o^2 Tx = 0 \quad (2)$$

for sets of (*ko*,***x***), one *ko* for every ***x***. The variable *x* is still the electric field solution, and *ko* is the free space wave number corresponding to that mode. The resonant frequency of the eigenmode can be found using:

$$f = \frac{k_o c}{2\pi} \quad (3)$$

where

- *c* is the speed of light.

Calculating the Quality Factor

Q is the unloaded quality factor, and is a measure of how much energy is lost in the structure due to lossy materials. Because ports and other sources are restricted for Eigenmode solutions, the Q calculated does not include losses due to those sources.

HFSS uses the following equation to calculate the approximate quality factor:

$$Q = \left| \frac{Mag(freq)}{2 \cdot Im(freq)} \right| \quad (1)$$

The Fields Calculator can also be used to calculate Q. In general, the equation for Q is

$$Q = (2\pi)(freq)\frac{U}{P} \quad (2)$$

where:

- *U* is the total energy stored in the cavity.
- *P* is the power lost, from resistive losses, for example.

Q Factor of Resonant Cavities

Basic Definition

$$Q = \frac{\omega W}{P_{loss}} \qquad \omega = |\omega| \quad (1)$$

In complex case (see examples in textbooks)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Since P_{loss} comes form W

$$Q_{min} = \frac{1}{2} \quad (2)$$

Calculating from complex resonance Frequency

$$\omega = \omega_r + j\omega_i \quad (3)$$

Definition 1 (Correct)

$$Q = \frac{|\omega|}{2\omega_i} \quad (4)$$

Definition 2 (Incorrect)

$$Q = \frac{\omega_r}{2\omega_i} \quad (5)$$

Arguments for using Definition 1

- Collin always uses the right frequency
- Gardiole [1] always uses definition1.
- IEEE papers [2], [3], and [5] use Definition 1.

Example R,L,C Circuit

$$R + j\omega L + \frac{1}{j\omega C} = 0 \quad (6)$$

$$\omega_{res} = \frac{1}{2}\left(j\frac{R}{L} + \sqrt{\frac{4}{LC} - \left(\frac{R}{L}\right)^2}\right) \quad (7)$$

Definition 1 (Correct)

$$Q = \frac{\omega_o L}{R} \qquad \omega_o = \frac{1}{\sqrt{LC}} \quad (8)$$

Definition 2 (Incorrect)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$Q = \frac{\sqrt{\frac{4}{LC} - \left(\frac{R}{L}\right)^2}}{2\frac{R}{L}} \quad (9)$$

Example Cube Resonator

$$f_{res} = \frac{1}{2\pi} j\frac{\sigma}{2\varepsilon} + \sqrt{-\left(\frac{\sigma}{2\varepsilon}\right)^2 + \frac{1}{\mu\varepsilon}k^2} \quad (10)$$

Definition 1 (Correct)

$$Q = \frac{\omega_o \varepsilon}{\sigma} \qquad \omega_o = \frac{k}{\sqrt{\mu\varepsilon}} \quad (11)$$

Definition 2 (Incorrect)

$$Q = 0.5\frac{\sqrt{\frac{1}{\mu\varepsilon}k^2 - \left(\frac{\sigma}{2\varepsilon}\right)^2}}{\frac{\sigma}{2\varepsilon}} \quad (12)$$

- Both definitions give almost the same answer at low losses
- Definition 2 gives nonphysical values at high losses, so Definition 1 is the correct one

References for Eigenmode Solutions

[1] Gardiol, F.E, Introduction to Microwaves, Artech House, 1984, pp. 113-115.

[2] M. Tsuji, H. Shigesawa and K. Takiyama, " On the Complex Resonant Frequency of Open Dielectric Resonators, " IEEE Transactions on Microwave Theory and Techniques,Vol. MTT-31, No. 5, May 1983. pp. 392-396. Eq. (6)

[3] IEEE MTT Vol. 30, No. 11, November 1982. pp 1954.

[4] R.T Remsky, "Ansoft HFSS8 Solution of TEAM Workshop Problem 18 (Waveguide Loaded Cavity) Using Both Eigenmode and Driven Solutions Presented at COMPUMAG 2001, Evian, France. Ansoft, Intranet.

[5] C.R. I. Emson and A Ait-Sadi, "Results for TEAM Problem 18 - Waveguide With Iris Proceedings of the TEAM Workshop in the Sixth Round, March 20-21, 1996, Okayama, Japan.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[6] S. Groiss, I. Bardi, O. Biro, K. Preis and K.R. Richter, "Solution of the TEAM Benchmark Problem 18, Waveguide Loaded Cavity" Proceedings of the TEAM Workshop, July 14, 1995, Berlin, Germany.

[7] S. Groiss, I. Bardi, O. Biro, K. Preis and K.R. Richter, "Solution of the TEAM Benchmark Problem 19, Circular Cavity Resonator" Proceedings of the TEAM Workshop in the Sixth Round, March 20-21, 1996, Okayama, Japan.

[8] Bardi et al, "Solution of TEAM Benchmark Problem 18 - Waveguide Loaded Cavity," Proceedings of TEAM Workshop, Aix-les- Bains, France, July, 1994, pp. 3-5.

[9] Bardi, Zs. Badics, "Solution of TEAM Workshop Problem 18 by Maxwell Eminence,

Presented at TEAM Workshop meeting held after COMPUMAG'98, Rio, Brazil.

[10] I. Bardi, "Eigenmode and Q factor analysis of a waveguide loaded cylindrical cavity resonator" Ansoft, Intranet

[11] I Bardi, Z. Cendes, "Rigorous analysis of cavity resonators by the finite element method" Ansoft, Intranet

[12] R. E. Collin, "Foundations for Microwave Engineering ", Second Edition, McGraw Hill, Inc

Theory of Characteristic Modes

Characteristic modes or characteristic currents can be represented quantitatively as eigenfunctions of the following generalized eigenvalue problem:

$$XJ_n = \lambda_n RJ_n$$

where

λ_n = eigenvalue

J_n= eigenvector

R and X are the real and imaginary parts of the impedance operator.

The total power exiting a surface is:

$$P = \langle E, J \rangle$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Where E is the electric field on the surface and J is the surface current. We can use the electric field integral equation to relate these two quantities to obtain

$$P = \langle ZJ, J \rangle = \langle (R + jX)J, J \rangle$$

Where R and X are the real and imaginary parts of the EFIE impedance matrix, Z = R + jX. The real part of P is the power radiated and the imaginary part is the power stored in the near field. For antenna design, we want to maximize the power radiated, or equivalently, minimize the power stored in the near field. The solution to this minimization problem is the generalized eigenvalue problem given above.

Qualitatively, characteristic modes are equiphase current modes, which can be computed for metallic and dielectric structures of arbitrary shapes and sizes and are independent of any excitation. Characteristic modes not only allow modal analyses of conductors but they are also useful for antenna design and placement, and the properties of the characteristic modes provide insight and valuable information for designing antennas. In fact, these quantities provide a physical interpretation of the radiating characteristics of an antenna. Therefore, understanding them can be very useful in problems involving syntheses, analyses, and optimization of antennas and scatterers. These quantities, known as characteristic value and characteristic current, are obtained by solving the above generalized eigenvalue problem. Together they define a characteristic mode. In addition to characteristic value and characteristic current, HFSS also depicts these quantities in the form of characteristic angle and modal significance to provide a practical insight into the behavior of the modes.

Using the standard Galerkin discretization of the EFIE, the characteristic values and characteristic currents in a lossless system are purely real. When the characteristic value is 0, no power is stored in the near field--all the power is radiated. From the characteristic value, you can make the following inferences:

- when $\lambda_n = 0$, the mode is externally resonant.
- when $\lambda_n > 0$, the mode is inductive (the energy is stored in a magnetic field).
- when $\lambda_n < 0$, the mode is capacitive (the energy is stored in an electric field).

Since the characteristic values can range from -infinity to +infinity, it’s more convenient to represent this quantity in terms of characteristic angle.

Characteristic Angle

Characteristic angle represents the phase lag between the electric field and surface current on a conductor. Mathematically, characteristic angle is defined as:

$\alpha_n = 180° - \tan^{-1}\lambda_n$

Characteristic Angle can range from 90 deg to 270 deg leading to the following possibilities:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- when α_n= 180 deg, the mode is externally resonant.
- when 90 deg < α_n < 180 deg, the mode is inductive.
- when 180 deg < α_n < 270 deg, the mode is capacitive.

Modal Significance

Modal Significance, as the term indicates is a measure of how significant a mode is for solving the EFIE at a given frequency. It also determines how hard or how easy it is to excite a mode. Mathematically, Modal Significance is defined as:

$$MS = \frac{1}{|1+j\lambda_n|}$$

Since the characteristic current diagonalizes both the real and imaginary parts of the impedance matrix, we can define

v_n=1+ $j\lambda_n$

which is an eigenvalue of the EFIE impedance matrix, Z = R+ jX. The eigenvalue is significant since its magnitude provides information about how well the corresponding mode radiates. Using the impedance matrix, eigenvalues, as well as the characteristic currents, the EFIE is analytically solved and the modal excitation coefficients are obtained as follows--

$$c_n = \frac{\langle E_{tan}^{inc}, J_n \rangle}{1+j\lambda_n}$$

Thus, we write that the magnitude of the coefficient must be

$$|c_n| \leq MS * |\langle E_{tan}^{inc}, J_n \rangle| \Big|$$

where MS is the modal significance. The maximum value of modal significance is 1. This happens when the mode is externally resonant. Modal significance can decay to zero when the mode is not resonant.

Depending upon the electrical size of the antenna, when you perform a characteristic mode analysis, a large number of modes can be computed. From a practical standpoint, when an object is not electrically large, most of the modes are difficult to excite since their characteristic values are far from 0. While one can generate plots of the characteristic value (λ_n) as a function of frequency, it's often more convenient to plot a quantity like Modal Significance as a function of frequency. This is because one is interested mostly in the characteristic values close to 0, in other words, MS =1 where the mode is externally resonant. A graph depicting modal significance

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

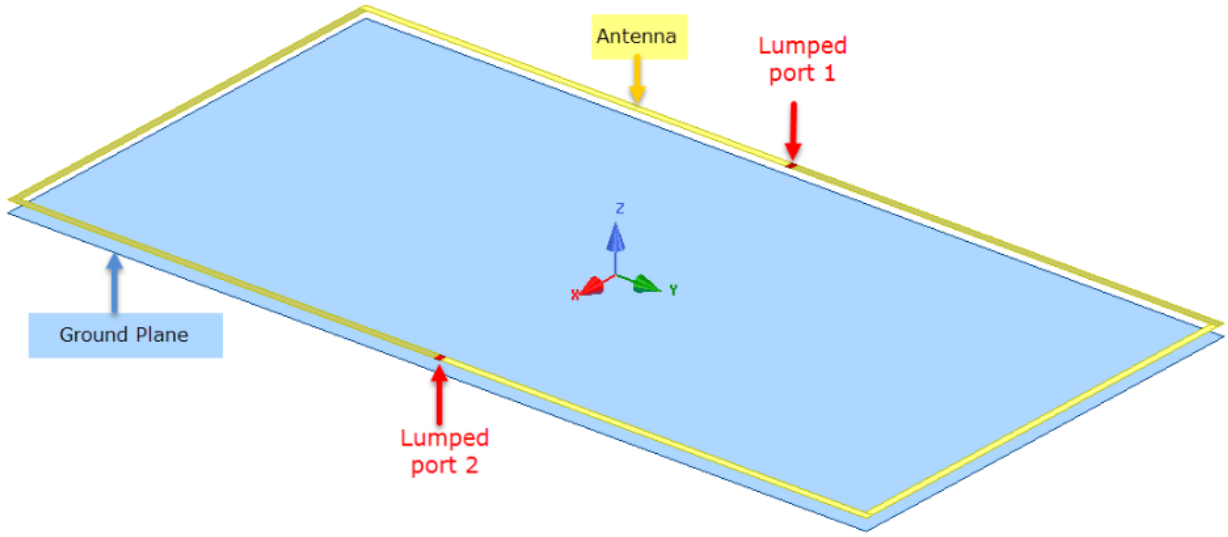
as a function of frequency can be very useful because such a graph lets you easily examine the importance of the modes and understand which of the modes contribute significantly to the radiated power (MS = 1), and the ones that are decaying (MS = 0).

Characteristic mode analyses give physical insight of the current pattern that can exist on an antenna and the platform upon which it operates. This insight can be valuable to a design engineer and help as a first step toward achieving correct antenna placement on electrically medium-sized structures.

The following designs are practical applications of characteristic mode analysis in HFSS.

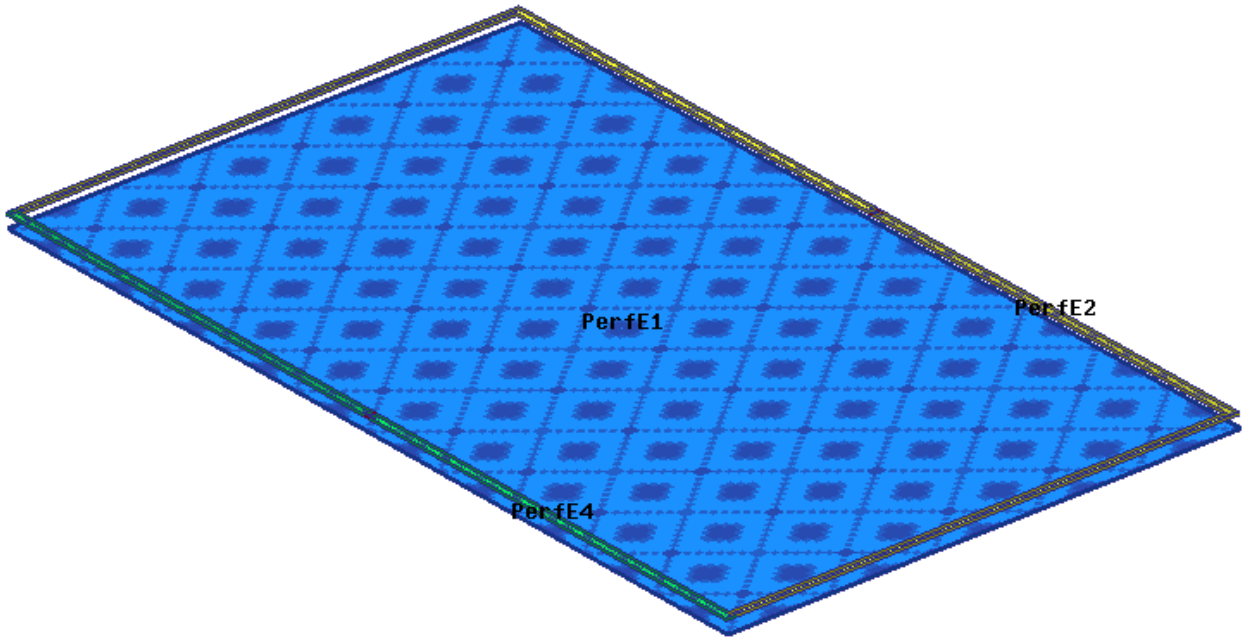
Loop Antenna of a Cell Phone Chassis

The following figure shows a cell phone chassis, which contains a simple plate representing its backplane. It also has a loop antenna.

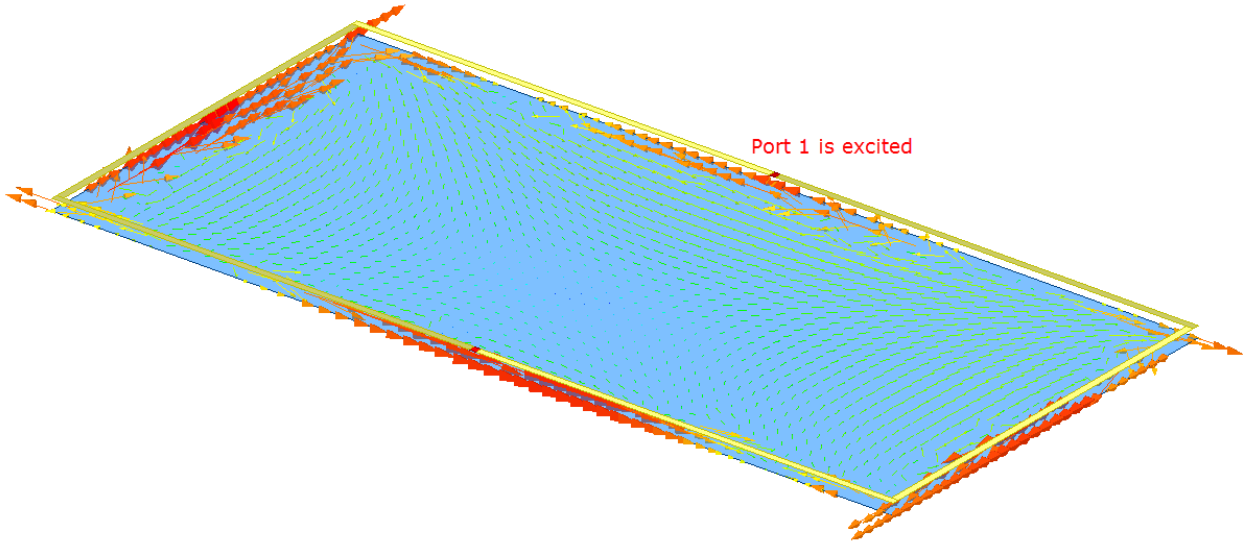


The boundary conditions are shown below. The plate and the antenna are assigned Perfect E boundaries. Also, the antenna modes are excited by assigning lumped ports. The simple plate acts as the antenna's ground plane, and its shape and size affect the performance of the antenna significantly. Therefore, if you perform a characteristic modal analysis of this design in HFSS, you can obtain critical information about the current modes and their pattern for this structure. Characteristic Mode Analysis solution type is chosen for this design. The design is analyzed at solution frequency of 1.6 GHz. Minimum Modal Significance is 0.02 so all modes above this threshold are calculated by HFSS. On the Edit Sources panel, you can configure the type of CMA Excitations as Port or Modes.

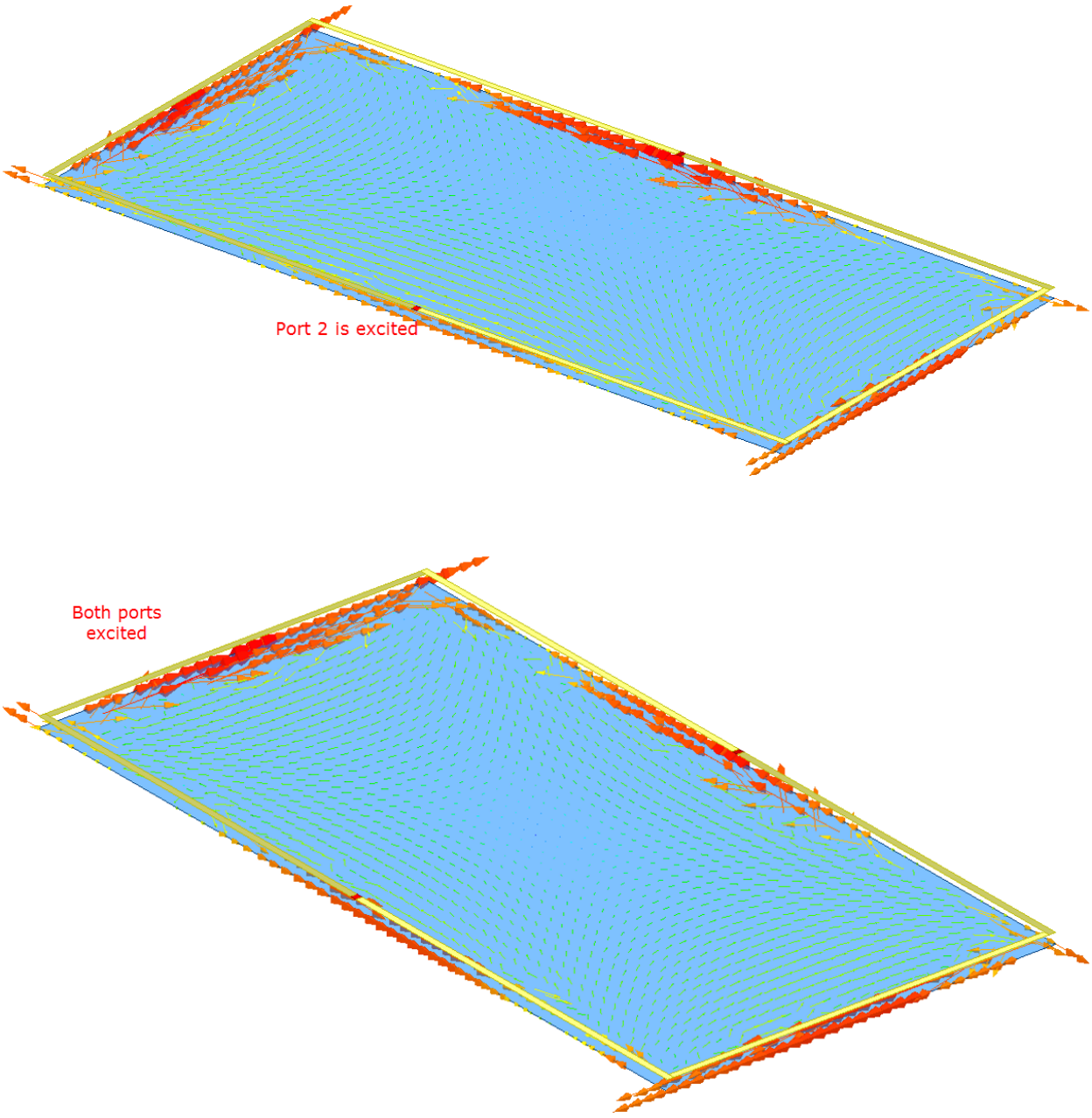
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Settings in the Edit Sources panel can help you understand the impact of the CMA excitations on the behavior of modes and their current patterns.

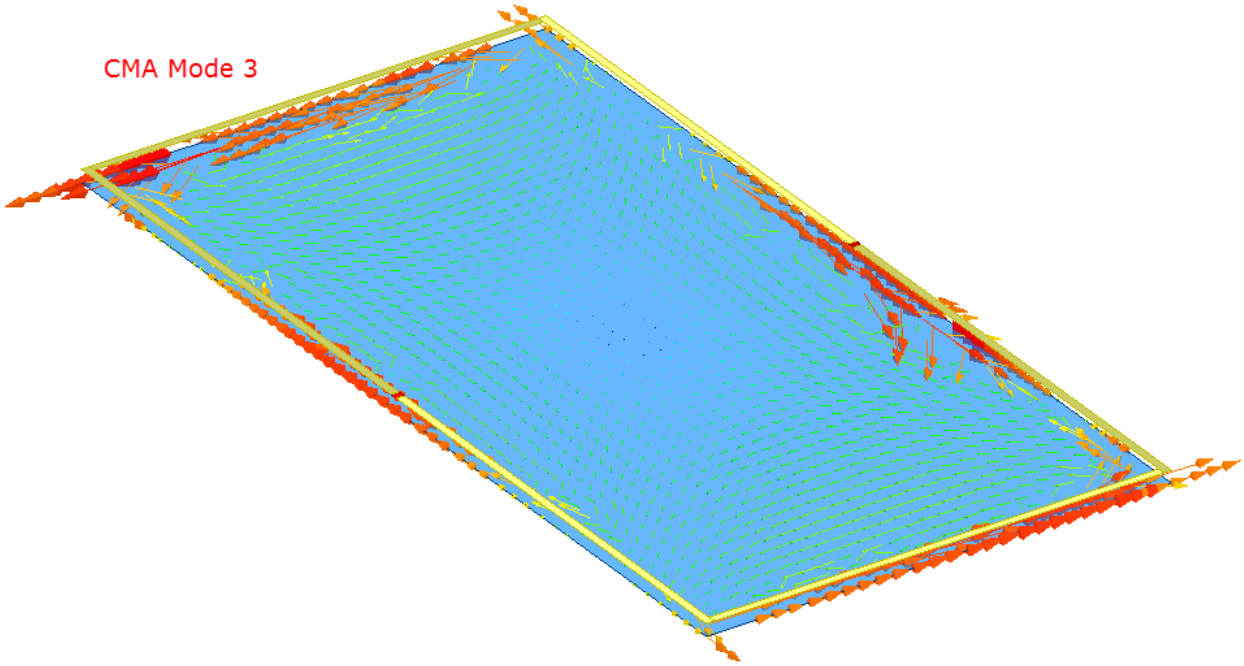


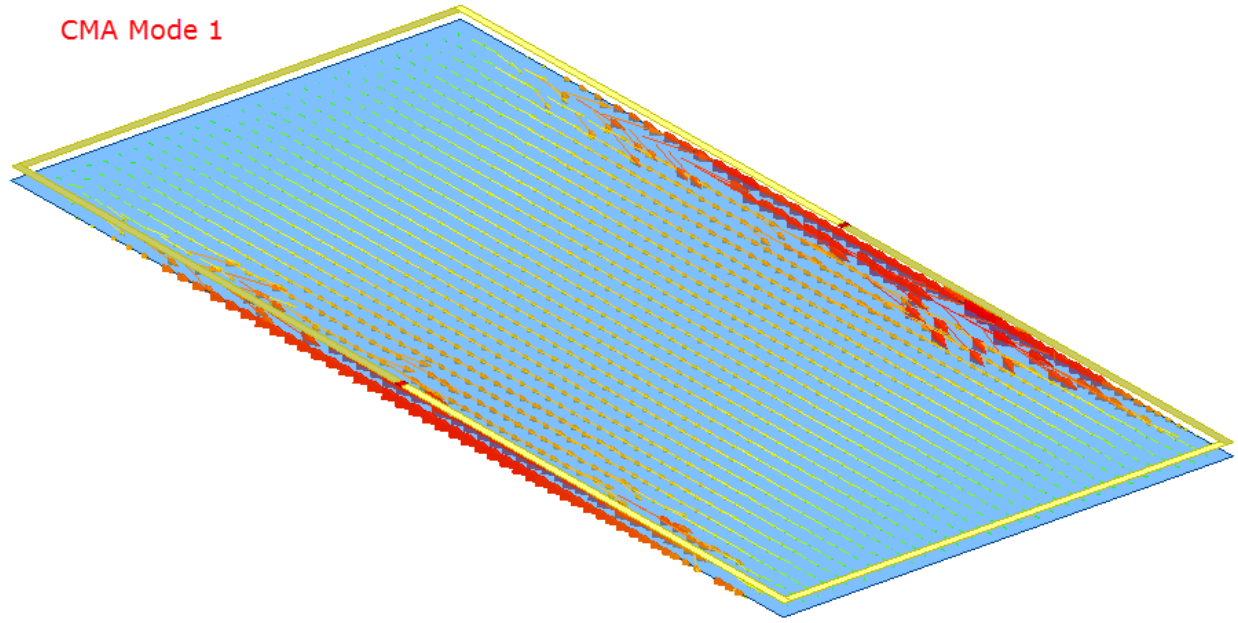
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



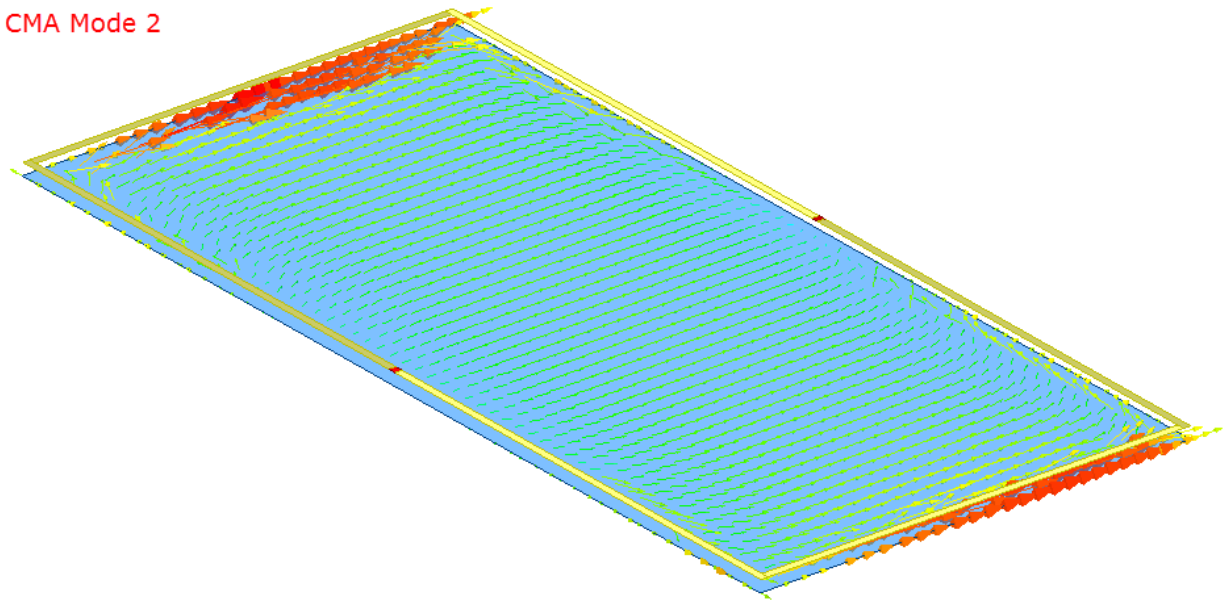
To illustrate, you can plot the characteristic currents in this design and understand how the mode behavior is affected by the port excitations. Depending upon how you define the CMA Port excitations, different modes are excited accordingly on the plane. For example, if you define lumped port 1 as the excitation for this structure, the current pattern mostly consist of mode 3. However, the other modes do have some contribution to the field pattern. Similarly, when lumped port 2 is excited the current pattern mostly consists of mode 3. If both ports are excited the pattern is identical to mode 3.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



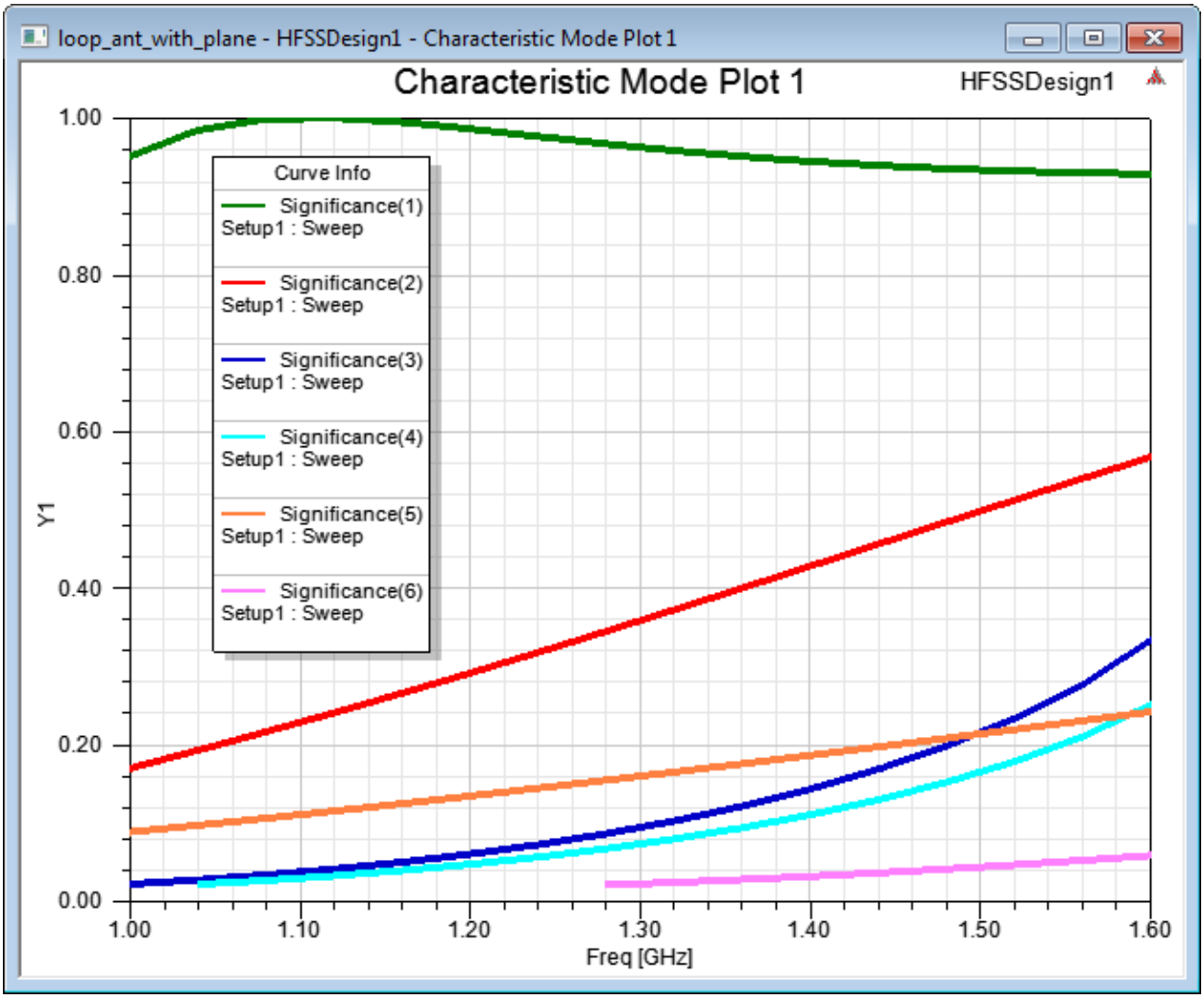


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The above images illustrate the mode currents when the CMA Excitations are configured to be modes 1, 2, or 3. Mode 3 is the dominant mode. Characteristic angles associated with the current modes can be plotted as a function of frequency.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once you have completed the modal analysis of your design, you can make modifications to your design as needed to excite the desired modes. This characteristic mode analysis example demonstrates how by viewing the current patterns you can excite only those modes of interest and maximize radiated power. Design modifications depending upon the modes you want to excite can improve antenna performance.

Field Solutions

During the iterative, adaptive solution process, the S-parameters typically stabilize before the full Electric field solution. Therefore, when you are interested in analyzing the field solution or quantities computed from the field solutions associated with a structure, it is advisable to use tighter convergence criteria.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In addition, for any given number of adaptive iterations, the magnetic field (H-field) is less accurate than the solution for the electric field (E-field) because the H-field is computed from the E-field using the relationship: .

$$\boldsymbol{H} = \frac{\nabla \times \boldsymbol{E}}{-j\omega\mu}$$

Field Overlay Plots

In HFSS, field overlays are representations of basic or derived field quantities on surfaces or objects. The objects on which you plot the fields may be pre-existing parts of the model geometry or they may be objects that you draw in post-processing mode.

If you select a surface, HFSS will plot the field quantities on the surface. If you select an object, HFSS will plot the field quantities within the volume of the object.

You can choose to create a scalar plot or a vector plot of the fields. A scalar plot can use several user selectable plot types to illustrate the magnitude of field quantities on surfaces or volumes. A vector plot uses arrows to illustrate the magnitudes of the x-, y-, and z-components of field quantities.

Field Quantities

The default field quantities that can be plotted, their definitions, and associated units are as follows (see the following note for issues in HFSS-IE for plotting magnetic and electric current):

Field Quantity	Definition	Units
CG Fields		
Q		
E		
Vector_E	The electric field, E(x,y,z,t).	V/m
Mag_E	The magnitude of the electric field, |E|(x,y,z,t).	V/m
DC R/L Fields		
Phidc	Voltage (electrostatic potential) from the DC solution, referenced to the sink terminal of the net.	V/m
VolumeJdc	DC current density vector in the conductor.	$Amps/m^2$
Mag_VolumeJdc	Magnitude of VolumeJdc.	$Amps/m^2$
Volume_Loss_DensityDc	The resistive (ohmic) power loss density from the DC solution, and is	W/m^3

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	defined as: $\frac{1}{\sigma}\left\|f_{\mathrm{dc}}\right\|^2$ Where conductivity of the metal.	
DC R/L Thin Conductor Fields		
JdcThinCond	DC surface current density vector in a thin conductor.	Amps/m
Mag_JdcThinCond	Magnitude of JdcThinCond.	V/m
PhidcThinCond	Electrostatic potential in a thin conductor.	V/m
AC R/L Fields		
J		
SurfaceJac	Surface current density vector from AC RL solution.	Amps/m
Mag_SurfaceJac	Magnitude of SurfaceJac vector.	V/m
H		
Mag_H	The magnitude of the magnetic field, |H|(x,y,z,t).	Amps/m
Vector_H	The magnetic field, H(x,y,z,t).	Amps/m

Field Quantity	Definition	Units
HFSS Fields		
E		
Mag_E	The magnitude of the electric field, |E|(x,y,z,t).	V/m
Complex Mag E	The complex magnitude of the electric field, |E|(x,y,z).	V/m
Vector_E	The electric field, E(x,y,z,t).	V/m
H		
Mag_H	The magnitude of the magnetic field, |H|(x,y,z,t).	Amps/m
Complex Mag H	The complex magnitude of the magnetic field, |H|(x,y,z).	Amps/m
Vector_H	The magnetic field, H(x,y,z,t).	Amps/m
J		
Mag Jsurf	The magnitude of the current density, |J|(x,y,z,t), on the surface.	Amps/m
Complex Mag Jsurf	The complex magnitude of the current density, |J|(x,y,z), on the surface.	Amps/m
Vector Jsurf	The current density, J(x,y,z), on the surface.	Amps/m
Mag Jvol	The magnitude of the current density, |J|(x,y,z,t), over the	Amps/m

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	volume.	^{2}
Complex Mag Jvol	The complex magnitude of the current density, |J|(x,y,z), over the volume.	Amps/m^2
Vector Jvol	The current density, J(x,y,z), over the volume.	Amps/m^2
Other		
Vector Real Poynting	The Poynting vector, defined as E x H*.	W/m^2
Surface_Loss_ Density	Resistive power loss density from the AC RL solution.	W/m
Volume_Loss_ Density	Resistive power loss volume from the AC RL solution.	W/m^3
Local SAR	The specific absorption rate.	W/kg
Average SAR	The average specific absorption rate.	W/kg
Certification SAR	The IEEE specific absorption rate certification number.	W/kg

Note: Plotting Magnetic current in HFSS-IE is relatively simple. It is M = Exn on an interface between two non-conducting materials. This is an equivalent current. Plotting Electric current in HFSS-IE involves three cases:

1. J = nxH on a surface of solid conductor (H is zero inside the conductor). This is a TRUE physical current.
2. J = nx(H1-H2) on a conducting sheet which is inside a region/background (or it is not an interface between two regions). This is a TRUE physical current.
3. J = nxH1 on a conducting boundary between two regions (it doesn't matter if the two regions have the same material). H1 is the field on the positive side of the surface. However, if region 2 is the background, we will display J = nxH2. This is an equivalent current.

Specifying the Phase Angle

Specifying the phase angle at which the field quantity is calculated enables you to compute the real part of the field's magnitude at different points in its cycle. These quantities can be represented in the form

$$A(x, y, z, t) = A(x, y, z)\cos(\omega t + \theta)$$

where

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- ω is the angular frequency specified during the solution.
- θ is the phase angle (the offset from a cosine wave that peaks).

Peak Versus RMS Phasors

This section concerns how field quantities are represented within HFSS. Some users will not need this information, such as those who wish to know port S-parameters or relative amplitudes of field solutions. Those that wish to find absolute field values, for example, will need to review the difference between the two types of field representation, peak and RMS.

HFSS solves in the frequency domain and obtains a phasor representation of the steady-state finite element field solution. Physical quantities such as the instantaneous (time domain) electric field are then obtained as derived quantities from the phasor representation.

If *Ex* is the x-component of a "peak" phasor quantity representing a time-harmonic electric field, the physical electric field x-component at time *t*, denoted $E_x(t)$, is computed from

$$\boldsymbol{E}_x(t) = \Re\left[E_x e^{j\omega t}\right] \quad (1)$$

where

- $\Re$ is the real part of a complex number or function.
- ω is angular frequency, $2\pi f$.
- *j* is the imaginary unit, $\sqrt{-1}$.
- *t* is the time.

On the other hand, if $\mathbf{E}_x$ is an "RMS" phasor, an additional factor of $\sqrt{2}$ is required as follows:

$$\boldsymbol{E}_x(t) = \Re\left[\sqrt{2}E_x e^{j\omega t}\right] \quad (2)$$

As a consequence of these equations, the peak physical field, max ($E_x(t)$) observed over a full time cycle is $max(\boldsymbol{E_x}(t)) = |E_x|$ for peak phasors and $max(\boldsymbol{E_x}(t)) = \sqrt{2}\,|E_x|$ for RMS phasors.

Additionally, given field phasors E and H, to compute the time-averaged power flow through a surface, the normal component of the real part of the complex Poynting vector is integrated over the surface. The correct form of the complex Poynting vector *S* depends on which phasor representation is used.

For peak phasors, $S = \frac{1}{2}E \times H^*$.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For RMS phasors, $E = E \times H^*$.

The conventions used by HFSS are as follows:

- Each propagating mode incident on a port contains 1 watt of time-averaged power.
- Circuit gap sources are specified in a peak sense. That is, if a voltage gap source magnitude is 5 volts, then the time domain circuit source behaves as $v(t) = 5\cos\omega t$. Likewise for a current gap source.
- Plane wave sources are specified in a peak sense. That is, if the plane wave magnitude is 5 V/m, then the plane wave incident field magnitude is $E(t) = 5\cos(\boldsymbol{k} \cdot \boldsymbol{r} + \omega t)$.
- Radiated power, as computed by the fields post processor, is a time-averaged quantity computed using the complex Poynting vector.
- Phasors in the Fields Calculator are peak phasors. The Poynting vector button in the calculator
  therefore implements the Poynting vector for peak phasors,

$$S = \frac{1}{2} E \times H^*$$

Calculations that compute either average or instantaneous time domain quantities must adhere to the peak phasor conventions.

Calculating the SAR

The specific absorption rate (SAR) is a measure of the amount of electromagnetic energy absorbed in a lossy dielectric material. The SAR is a basic scalar field quantity that can be plotted on surfaces or within objects in HFSS. The **HFSS>Fields>SAR Setting...** command lets you specify the SAR setting. You have a choice ofIEC/IEEE 62704-4 Draft and Gridless.

For the IEEE/IEC STD 62704-4 we subdivide the model space into a grid of voxels properly aligned along the global coordinate system. The density and local SAR values are then mapped over the grid of voxels. Afterwards the two-pass average SAR algorithm is applied to the voxels. Since this two pass algorithm is very time consuming we try to use parallel processing as much as possible. The algorithm is divided into two phases so that the results of the time consuming phase can be reused when fields change.

For more details, see IEC "Determining the Peak Spatial Average Specific Absorption Rate (SAR) in the Human Body from Wireless Communications Devices, 30 MHz - 6 GHz: Requirements for Using the Finite-Element Method for SAR Calculations, specifically involving Vehicle Mounted Antennas and Personal Wireless Devices."

For Gridless, HFSS uses the following equation to calculate the SAR: $\sigma * E^2/(2\rho)$.

where

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- σ = the material's conductivity. This is defined as: $\sigma_{bulk}+\omega\varepsilon_0\varepsilon_r tg\delta$
- ρ = the mass density of the dielectric material in mass/unit volume.

There are two types of SAR Field Overlay plots available in HFSS: local SAR, and average SAR. When calculating the local SAR, HFSS uses the equation above to calculate the SAR at each mesh point on an overlay plot. HFSS interpolates the values between the mesh points across the plot.

When plotting the average SAR, for each mesh point on the plot, HFSS reports the SAR averaged over a volume that surrounds that point. The volume is determined by the settings for the material's mass density and mass of the material surrounding each mesh point set in the **Specific Absorption Rate Setting** dialog box. When you use the Gridless method, the volume will not cross object boundaries. The IEEE/IEC method performs averaging over all tissues.

The Certification SAR quantity provided in the Fields Calculator is an efficient way to determine the peak spatial-average SAR. When computing the certification SAR using the Gridless method, HFSS assumes the peak spatial-average SAR is located on an object surface. HFSS therefore searches for the peak local SAR on the surfaces, and creates an averaging volume around that location, completely inside the object with an axis normal to the surface at the location of the peak value of the local SAR.

If the Average SAR method is IEC/IEEE 62704-4 Draft, then the IEC/IEEE 62704-4 Draft standard is followed. Voxel meshes are used, E fields, mass density and conductivity values are resampled onto the voxels first. Then the algorithm for IEC/IEEE 62704-1 is applied to calculate the psSAR. In order to reduce the computation time for the iterative resampling and SAR averaging, subregions with SAR hot spots are identified in a pre-scan. In these subregions, the psSAR is then calculated according to IEC/IEEE 62704-1. The maximum psSAR of all subregions is reported as the psSAR maximum. The psSAR is calculated on an initial voxel mesh according to the procedure defined in IEC/IEEE 62704-1. Then the computational domain shall be resampled with on a voxel mesh of half grid size. This procedure is repeated until the differences in psSAR from the previous iteration to the current iteration are less than 1%.

Average SAR Through Voxelization

To calculate the Average SAR, HFSS first subdivides the model space into a grid of voxels properly aligned along the global coordinate system. The density and local SAR values are then mapped over the grid of voxels. Afterwards, HFSS applies the two-pass average SAR algorithm to the voxels. Finally HFSS distributes the average SAR from the voxels back to the FEM mesh. Because this two pass algorithm is very time consuming, HFSS uses parallel processing as much as possible. The algorithm is divided into two phases so that the results of the time consuming phase can be cached and reused while fields change due to new combinations from edit sources.

HFSS progress messages are included here in parentheses and in red color.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Build the grid of voxels.
    a. If we have cached data, load the data. (2nd phase: Reading voxel data) Go to 3.
    b. Otherwise subdivide the model space into a grid of voxels and map density to voxels. See below for details on mapping. (1st phase: Mapping density to voxels...)
2. Collect information for all valid and unused voxels.
    a. Traverse all voxels to collect all valid voxels. For each valid voxel store the extends in one direction and the used ratio of the exterior voxels. See below for requirement for valid voxels. See Figures 1 and 2. (1st phase: 1st pass …)
    b. Traverse all valid voxels to collect all used voxels. (1st phase: 2nd pass ...)
    c. For the unused voxels find possible extends in the 6 directions, see Figure 3. For each unused voxel store an array of directions, extends and ratio of exteriors.
    d. Cache data from a and c.
3. Collect voxels needed for field plot and calculation (field geometry instance).
    a. Get overall elements for field geometry instance.
    b. Get all voxels that cover the nodes of these elements. (2nd phase: Collecting voxels containing elements)
    c. Classify these voxels as valid, used and unused. For each used voxel add all the valid voxels using it to the valid set.
    d. Collect all elements needed to get the data for all the voxels above. (2nd phase: Collecting involved elements)
    e. Map FEM local SAR to voxels. (2nd phase: Mapping data to voxels)
4. Calculate average SAR.
    a. Traverse all valid voxels. For each valid voxel collect average SAR and mark used voxels and propagate average SAR to used voxels. (2nd phase: 1st pass)
    b. Traverse all unused voxels. For each calculate the average SAR on possible directions and take the maximum. (2nd phase: 2nd pass)
5. Distribute average SAR to the FEM mesh from the voxels. For each element node take the SAR value from the voxel which contains this node. (2nd phase: Distributing fields)

This section describes how HFSS maps general data (mass density and conductivity) from FEM mesh to voxels. For each voxel:

1. Collect all intersecting tetrahedra and calculate data over these tetrahedra.
2. Cut each intersecting tetrahedron by this voxel. This gives a series of new smaller tetrahedra.
    a. For each new tetrahedron get data by interpolating over its parent (the original tetrahedron that it is cut from). Integrate the data to get the total over it.
    b. Add the contributions to get the total over the intersection of this voxel and this tetrahedron.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Now add the total from above for all intersecting tetrahedra. This gives the total over this voxel. Now take the average by dividing this total by the voxel volume.

A voxel is counted as tissue if its centroid lies in an FEM element with nonzero conductivity and the field value at this location is used for this voxel. The density at each voxel is the total tissue mass divided by voxel volume.

A voxel is valid if the following holds:

1. Expand a cubic volume centered at the voxel evenly in the six directions aligned along the global axis so that the enclosed tissue mass is within 0.0001% of the prescribed mass.
2. Less than or equal to 10% of the cube contains background material in volume.
3. None of the six faces is completely filled with background material.

Transient Solution Theory

An important advantage of the Finite-Element Method (FEM) is its ability to solve Maxwell's equations on an unstructured mesh [1]. In such a mesh, the elements have a range of sizes and orientations such that, rather than forming a regular grid, they conform to the geometry of the model, respecting all its details. The use of unstructured meshes, combined with automatic mesh adaptation in HFSS Frequency Domain, has been instrumental in delivering accurate results for complicated geometries.

So far, commercially available solvers based on time-domain methods have relied on the Finite Difference Time Domain method (FDTD) [2] or on the closely related Finite Integration Technique (FIT). The use of unstructured meshes for both schemes presents technical difficulties that have yet to be overcome, and thus both FDTD and FIT continue to rely on brick-based space partitioning.

Time-domain methods developed for unstructured meshes such as the Finite Volume Time Domain (FVTD) [3, 4] or the Finite Element Time Domain (FETD) [5] methods do not offer an acceptable trade-off between accuracy, speed and memory usage. However, recent advances in finite-element techniques have led to a new scheme based on the Discontinuous Galerkin [6] family of numerical methods. This scheme retains the flexibility, accuracy and reliability of unstructured-mesh finite-element methods but avoids the solution of a large matrix equation at each time step. Instead, in the Discontinuous Galerkin Time Domain (DGTD) method, each mesh element advances in time using its own time step in a synchronous manner. This results in a significant speed-up on unstructured meshes as the largest mesh elements typically take time steps that are two to five orders of magnitude larger than those of the smallest mesh elements.

This chapter provides an explanation of how the DGTD method has been implemented in HFSS Transient. as the Hybrid Solver. It also describes the Implicit Solver for efficient simulation of electrically small structures.

Discontinuous Galerkin Finite Element Method

Local Time Stepping

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Materials in Time Domain

Excitations in Time Domain

Analysis Setup in Time Domain

Target Residual for Transient Analysis Duration

Transient Implicit Solver

References for Time Domain

Discontinuous Galerkin Finite Element Method

The DGTD solver provides the time-domain solution of Maxwell's Equations.

$$\frac{\partial \vec{D}}{\partial t} + \vec{J} = curl(\vec{H}) \quad \frac{\partial \vec{B}}{\partial t} = -curl(\vec{E}) \tag{1}$$

$$div(\vec{D}) = \rho \qquad div(\vec{B}) = 0$$

The mathematical framework of the Discontinuous Galerkin family of numerical methods for Maxwell's equations has been presented by Jan Hesthaven and Tim Warburton [7, 8]. To apply the DG method, we choose basis and test functions local to each mesh element *T*, in a similar way to the Finite Element Method:

$$\int_T \frac{\partial \vec{D}}{\partial t} \cdot \vec{\varphi} = \int_T curl(\vec{H}) \cdot \vec{\varphi} - \int_T \vec{J} \cdot \vec{\varphi} = \int_T curl(\vec{\varphi}) \cdot \vec{H} - \int_T div(\vec{\varphi} \times \vec{H}) - \int_T \vec{J} \cdot \vec{\varphi} \tag{2}$$

$$\int_T \frac{\partial \vec{B}}{\partial t} \cdot \vec{\varphi} = \int_T -curl(\vec{E}) \cdot \vec{\varphi} - \int_T curl(\vec{\varphi}) \cdot \vec{E} + \int_T div(\vec{\varphi} \times \vec{E})$$

As is the case with the frequency-domain finite-element solver, the DGTD method is free of spurious solutions [9] and supports higher-order basis functions. In the time domain, the use of high orders minimizes dispersion errors occurring during the simulation of electrically large problems. The DGTD solver not only employs high-order elements but allows mixed element orders in the same mesh, known as hp-adaptivity, to optimize accuracy and solution times.

Following the same approach as in the Finite Volume Method, the divergence term containing volume integrals is transformed into surface integrals:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\int_T \frac{\partial \vec{D}}{\partial t} \cdot \vec{\phi} = \int_T curl(\vec{\phi}) \cdot \vec{H} + \int_{\partial T} (\vec{n} \times \vec{H}) \cdot \vec{\phi} - \int_T \vec{J} \cdot \vec{\phi} \quad (3)$$

$$\int_T \frac{\partial \vec{B}}{\partial t} \cdot \vec{\phi} = -\int_T curl(\vec{\phi}) \cdot \vec{E} - \int_{\partial T} (\vec{n} \times \vec{E}) \cdot \vec{\phi}$$

The tangential fields $\vec{n} \times \vec{E}$ and $\vec{n} \times \vec{H}$ are then evaluated as upwind fluxes at the surface of *T* by using a Riemann solver [10]. Separating the treatment inside the element from its surface provides a robust and flexible framework to take into account boundary conditions. This separation also enables various combinations of locally implicit and locally explicit time-integration schemes for instance to achieve an optimum trade-off between speed and memory, without compromising accuracy and stability.

Local Time Stepping

A prime benefit of satisfying Maxwell's equations in a way analogous to the Finite Volume Method is greatly reduced RAM usage. Since the DGTD solver avoids solving the large matrix equation that represents the entire computational domain at each time step, no large matrix needs to be stored.

One of the key features of the DGTD solver is an innovative local time-stepping scheme that advances in time each mesh element independently from its neighbors. This feature efficiently exploits the use of highly unstructured meshes and hp-adaptivity: in the DGTD solver, each mesh element has its own energy-based stability criterion [11], calculated from its size, material properties and basis-function type and order.

Figure 1 illustrates the process. This figure shows a cut-plane through an unstructured mesh model of a ground-penetrating radar. Each element of the mesh is colored according to the time-step values used to locally advance the fields in time. Large elements are colored yellow and use larger time-steps than the smaller elements colored green and or even smaller elements in blue. This results in fewer iterations for most mesh elements and reduced overall simulation times.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

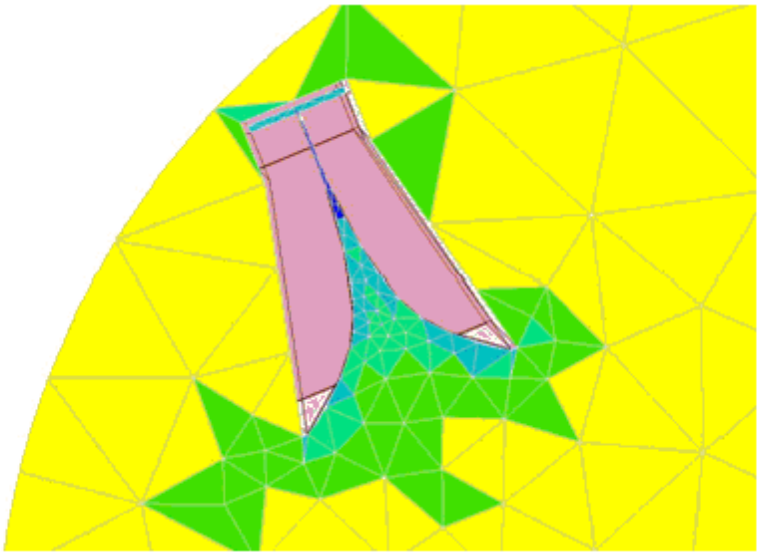


Fig. 1 Local time stepping

The time step of any mesh element is chosen such that at the largest time steps the entire simulation is "in sync." From a user point of view however, the local time-stepping scheme is transparent and all time-domain data are provided or rendered synchronously "on the fly" as needed. In contrast to FIT or FDTD, the DGTD solver doesn't advance in time uniformly using the smallest time step but uses an optimal range of time steps that minimize simulation times, ensures stability and minimizes the dispersion error.

One last important feature of DGTD solver is its affinity for parallel computations. This is a direct consequence of the local nature of its formulation that yields a spatially compact numerical scheme. Parallelism is naturally exploited with this scheme by partitioning the computational domain into sub-domains and solving each sub-domain concurrently on multi-core or many-core systems.

Materials in Time Domain

In any time-domain method, handling frequency-dependent material parameters is not straightforward. Two important cases, frequently encountered in electromagnetic simulations, are (1) non-perfectly conducting metals and (2) lossy dielectric materials.

In most high-frequency applications, one does not solve for fields inside non-perfectly conducting metals. Rather, the skin effect is expressed with a boundary condition on the surfaces of the metal. Since the skin impedance is frequency dependent, containing a factor Ö (frequency), a suitable way has to be found to approximate this in the time domain. In HFSS Transient, the Padé approximation [ ] is used. With this approximation one can expect accurate

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

results over a very wide band which includes the highest frequency of interest. Near DC, a modest reduction in accuracy is to be expected.

Lossy dielectric materials can be described by a bulk conductivity s or by a loss tangent tan(d). If the conductivity is the only loss mechanism, the relation between the two is tan(d) = s/(2p×frequency ×e_0e_r). In any case, the loss tangent of a material is frequency dependent, and a suitable way has to be found to approximate it in the time domain. In HFSS Transient, when a lossy dielectric is specified with a certain loss tangent at the highest frequency of interest, the Debye model is used to describe the material for all frequencies. The Debye model is more realistic than the assumption of a constant loss tangent.

Excitations in Time Domain

Excitations available in HFSS Transient are wave ports, lumped ports, voltage sources, current sources and incident waves. In the case of ports, the modal port solution is provided by the same 2D port solver as is used in HFSS Frequency Domain. If a lossy dielectric or a non-perfectly conducting metal touches the port, the port impedance will be frequency dependent. A frequency-dependent excitation poses a problem to any time-domain solver. HFSS Transient, as well as some competing time-domain codes, uses the mode belonging to the lossless case as an excitation in the port.

In order to deal with higher-order modes or with frequency-dependent (lossless) modes, HFSS Transient takes advantage of the trans-finite element method, which has always been an important technique in HFSS Frequency Domain. In HFSS Transient, elements touching the ports are set aside in an implicit time-stepping scheme that is close to the FETD method. Since this is a small fraction of the total number of elements, the penalty for solving the matrix equation is negligible, while the implicit method allows large time steps. The upshot is that higher-order modes and frequency-dependent (lossless) modes are handled accurately.

In a standard HFSS simulation a port excitation is the equivalent of an excited infinite transmission line connected to the port face. For a modal project this is easy to visualize but in a terminal project the user defines the input voltage. This raises the question of how this input voltage is applied to the port for terminal designs. While the following discussion is directed to transient simulations, it is also applicable to HFSS terminal designs where the Incident Voltage option in Edit Sources is used.

To derive the correct model for a terminal port, first consider the transmission line section shown in Figure 1. The port is at the point 0-0 and to the left is the equivalent infinitely long feed line.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

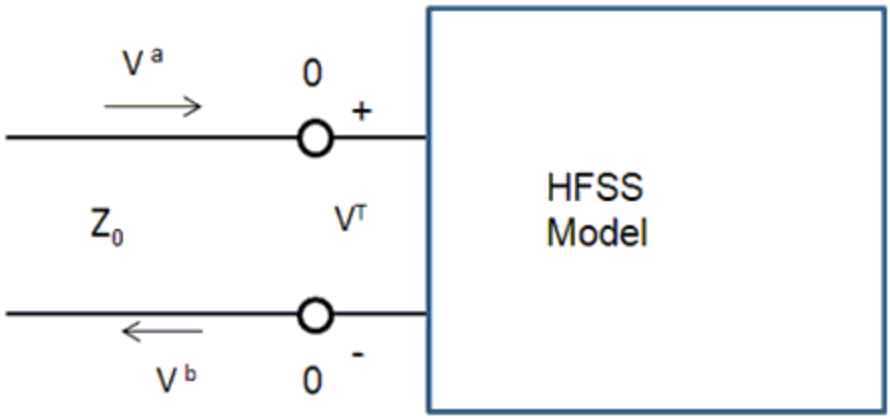


Figure 1

For this model then V^a is the incident voltage and V^b is the reflected voltage seen at the port. We know from the definition of voltage and current on a transmission line that for the total voltage, V^T, the following is true:

$$V^T = V^a + V^b \quad (1)$$

If the characteristic impedance of the line is equal to Z0 then we also know:

$$Z_0 I = V^a - V^b \quad (2)$$

If we add equations (1) and (2) we find that at point 0-0:

$$V^T + Z_0 I = 2V^a \quad (3)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If we apply Kicrchkoff's voltage law to the equivalent circuit shown in Figure 2 we see it yields the result shown in (3). This is the port model for a terminal port in HFSS.

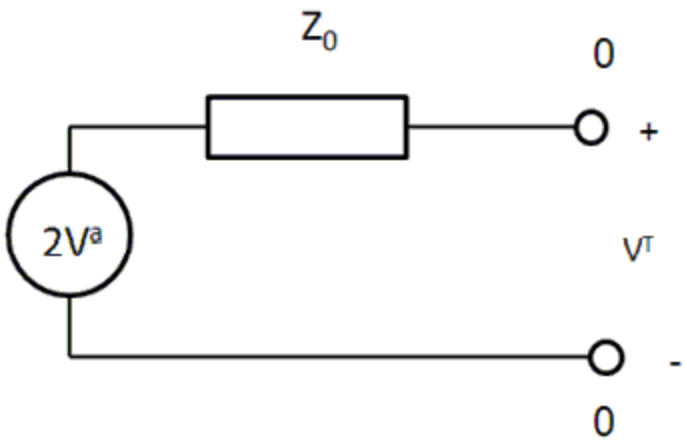


Figure 2

To be consistent then the same terminal port excitation is used in HFSS Transient. Therefore if you define an incident voltage signal in the transient setup that signal is applied to the 3D model as a voltage source of twice the user defined amplitude in series with the terminal impedance.

> **Note:**
>
> As a consequence of the above discussion, one needs to be careful when comparing results with a circuit tool or textbooks using the voltage source shown in Figure 2. If one volt is used for the voltage source in the circuit simulator, this should be compared with an HFSS transient simulation with edit sources set to 0.5 volt.

Before considering an example one more point needs to be made. In the reporter for HFSS Transient when plotting the voltage at a given port the options are Input( ) and Output( ). Input is the user defined incident voltage (Va in the Figure 2) and Output is the reflected voltage (Vb = VT - Va). We can see then that to view the total voltage at the port terminals one needs to create an output variable that is the sum of Input ( ) and Output( ). We will consider two examples to illustrate this.

Consider the HFSS Transient model of a simple coax shown in the Figure 3. It is an air filled 50Ω line that has a center section with a smaller inner radius that results in an impedance of 91.5 ohm in that part of the line. Each section of this air filled coax (the two 50 Ω and the one 91.5 ohm sections) has a length of 100mm. This length has a time delay of 1/3 nS. This was simulated in HFSS Transient using the incident voltage signal shown in the inset for Figure 3.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

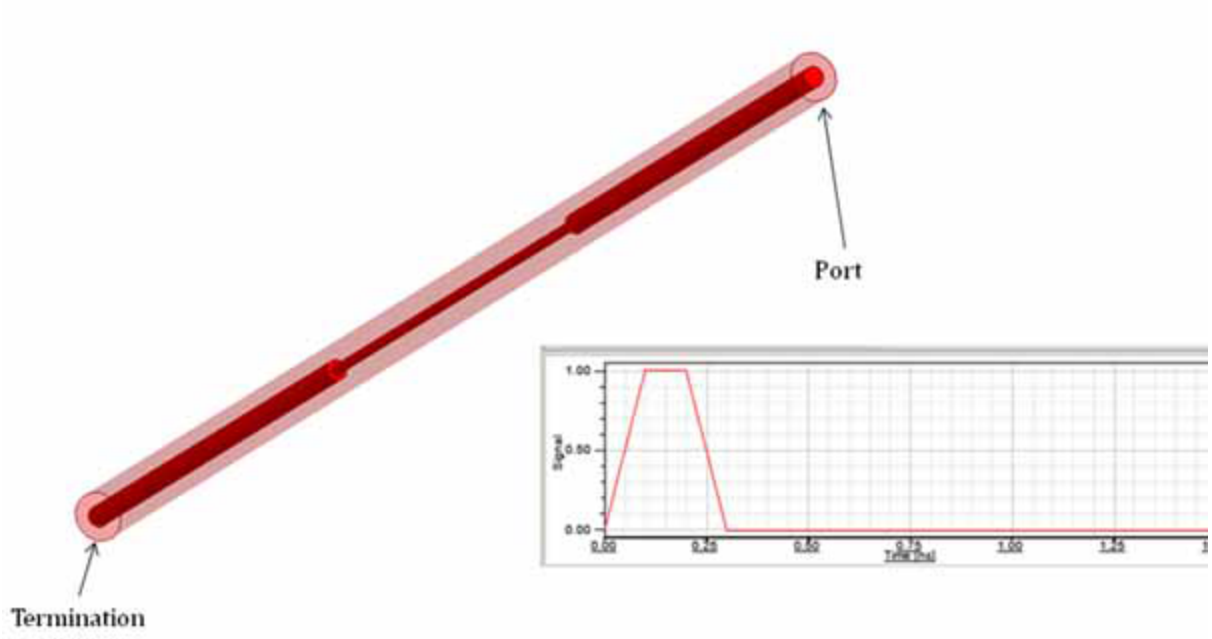


Figure 3

This model was simulated with a single 50 Ω terminal port on one end and a matched termination on the opposite end. The response seen at the port is shown below in Figure 4.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

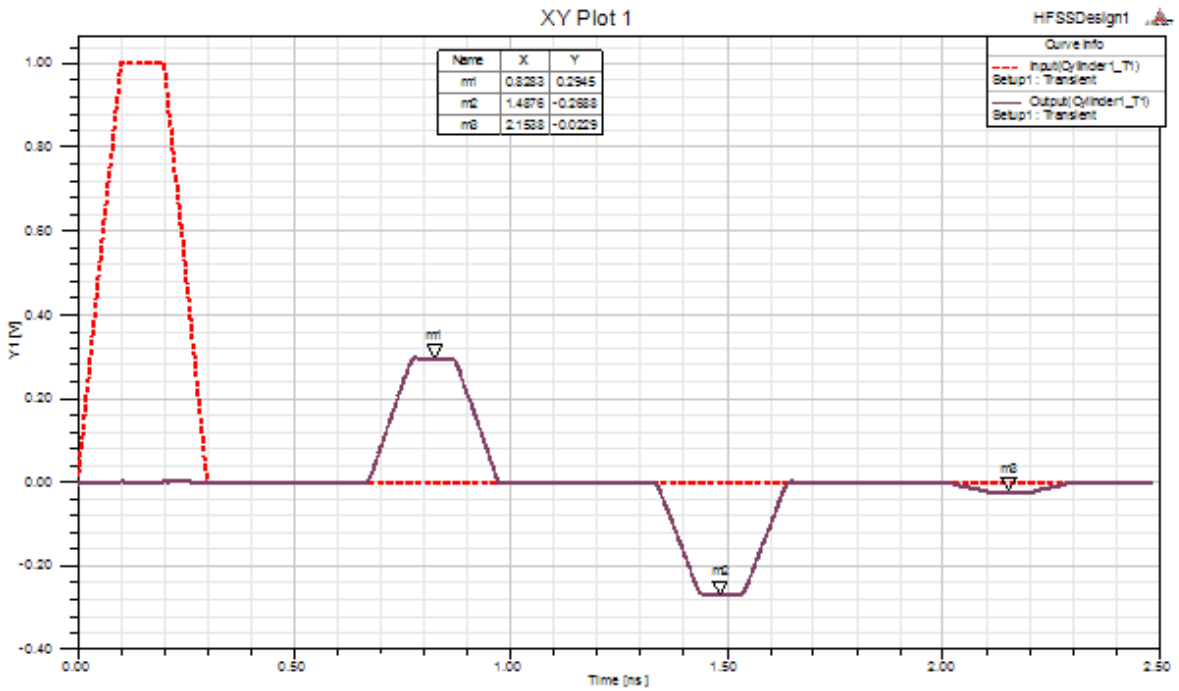


Figure 4

Figure 4 shows the Input and Output signals at the port terminal (Cylinder1_T1). Remember that Input( ) is the incident signal as shown in Figure 3. If one uses the equations for reflection and transmission to compute the amplitude of the signal at the 3 points shown in the figure these textbook equations would give 0.29, -0.27,-.02 respectively. We can see that the reflected voltage is accurately computed for the applied signal.

For an example where one might be more interested in the total voltage a second case was considered. The same HFSS Transient model was simulated with the same incident signal, but this time the port terminal impedance was set to 30 Ω. The HFSS Transient response for this simulation is shown in the Figure 5.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

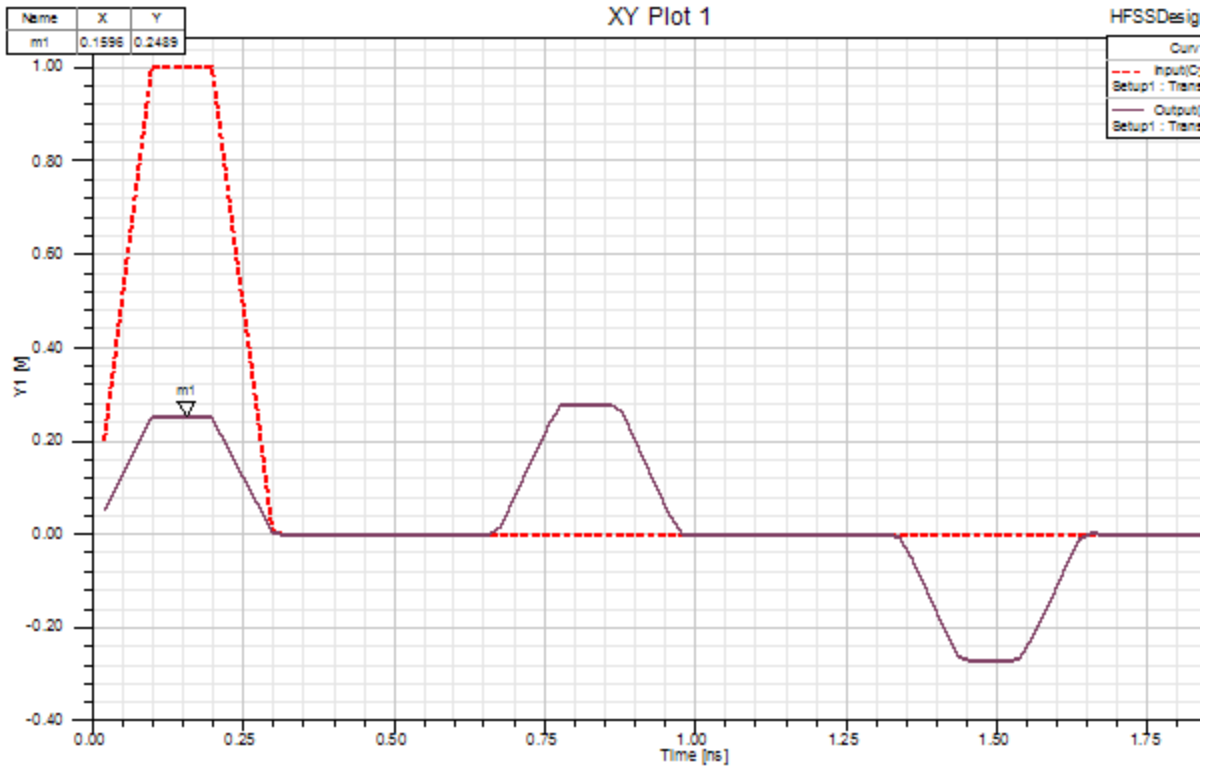


Figure 5

For the time between 0.1 - 0.2 nS the textbook response says that V^T should be 1.25V. Again that is the total voltage and if you add together the 2 curves shown in the plot you would get the correct response of 1.25V in that range.

Analysis Setup in Time Domain

Two classes of analyses exist: those focused on device characterization, and more general analyses. In the case of device characterization, one is interested in quantities like S-parameters and/or time-domain output at ports due to excitation of a specific port. This requires that excitations be "on" one at a time. For each active port and mode, a separate simulation is done. These simulations are independent and can be distributed over a network, to be executed in parallel. Each active port and mode will employ an excitation with the same time profile.

In the more general analysis type, all active ports and modes are "on" simultaneously. Each one can have an excitation with a different time profile. This type of analysis gives you more freedom, but S-parameters cannot be extracted.

In both classes of simulations, fields can be saved and visualized.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Target Residual for Transient Analysis Duration

The residual is defined to be the ratio of the global RMS value of the electric field at time *t* to the maximum RMS value in the time range zero to time *t*:

$$E_{rms}(t) = \sqrt{\frac{\sum_{i=1}^{N} \left|\vec{E}_i t\right|^2}{N}} \quad (1)$$

$$Residual(t) = \frac{E_{rms}(t)}{max\{E_{rms}(\tau): 0 \le \tau \le t\}} \quad (2)$$

where $\vec{E}_i$ is the electric field at the ith interpolation point in the mesh which contains a total of *N* interpolation points. Note that the number of interpolation points is typically much larger than the number of mesh vertices. The time duration of a simulation can be controlled by specifying a target residual in db20 located in the Duration tab of the solution setup.

Transient Implicit Solver

The implicit solver is based on the finite element time domain (FETD) method with an implicit time stepping [13-14]. The Implicit Solver is unconditional stable and is not limited by any Courant-Friedrichs-Lewy (CFL) type condition as the Hybrid Solver. Therefore the Implicit Solver can have much larger time step compared to the Hybrid Solver. To satisfy Nyquist sampling theory, the time step for the Implicit Solver is such that resolves the spectral content of the input signal (time-step = 1/(20*fmax)). On the other hand, the implicit solver solves a large sparse system matrix during time stepping. Therefore, each step of the implicit solver takes much longer than that of the hybrid solver. Both the memory usage and simulation time can increase rapidly as the problem size becomes larger. This renders the implicit solver less efficient for solving electrically large problems. Finally, the implicit solver does not support GPU acceleration. A warning message will be shown in the user interface when the GPU is enabled for an implicit simulation.

The hybrid solver is based on the explicit-implicit discontinuous Galerkin time domain method [15]. When the GPU is enabled, only the explicit part of the hybrid solver is running and being accelerated by GPUs. More information about the hybrid solver can be found in Transient Solution Theory.

References for Time Domain

[1] Z. Cendes, D.N. Shenton, and H. Shahnasser. "Magnetic field computation using Delaunay triangulation and complementary finite element method", IEEE Trans. on Magnetics, 19

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(6):2551, 1983.

[2] A. Taflove, "Application of the Finite-Difference Time-Domain Method to Sinusoidal Steady-State Electromagnetic-Penetration Problems, Electromagnetic Compatibility", IEEE Transactions on Volume EMC-22, Issue 3, Aug. 1980 Page(s):191 - 202

[3] P. Bonnet, X. Ferrières, B. Michielsen, and P. Klotz, "Time Domain Electromagnetics", S. M. Rao, Ed., Academic Press, 1997, ch. 9, pp. 307-367.

[4] D. Baumann, C. Fumeaux, and R. Vahldieck, "Field-Based Scattering-Matrix Extraction Scheme for the FVTD Method Exploiting a Flux-Splitting Algorithm," IEEE Trans. on Microwave Theory and Tech., vol. 53, no. 11, Nov. 2005.

[5] J-F. Lee, R. Lee, and A.C. Cangellaris, "Time-Domain Finite-Element Methods", IEEE Trans.Antennas Propag., vol. 45, pp. 430-442, Mar. 1997.

[6] B. Cockburn, G. Karniadakis, and C.-W Shu, "Discontinuous Galerkin Methods: Theory, Computation and Applications", Lecture Notes in Computational Science and Engineering (Springer-Verlag, New York, 2000), Vol. 11.

[7] J. Hesthaven,T. Warburton, "Nodal high-order methods on unstructured grids", Journal of Computational Physics, v.181 n.1, p.186-221, September 1 2002.

[8] J. Hesthaven and T. Warburton, "Nodal Discontinuous Galerkin Methods Algorithms, Analysis, and Applications", Springer series: Texts In Applied Mathematics, vol. 54, 2007.

[9] A. Buffa, I. Perugia, "Discontinuous Galerkin approximation of the Maxwell eigenproblem",SIAM J. Numer. Anal. 44, 2198, 2006.

[10] Leveque, Randall, "Finite Volume Methods for Hyperbolic Problems", Cambridge University Press, 2002.

[11] L. Fezoui, S. Lanteri, S. Lohrengel and S. Piperno, "Convergence and stability of a Discontinuous Galerkin time-domain method for the 3D heterogeneous Maxwell equations on unstructured meshes", ESAIM: Math. Model. and Numer. Anal., Vol. 39, No. 6, pp. 1149-1176, 2005.

[12] R.A. Chilton and R. Lee, "The discrete origin of FETD-Newmark late time instability, and a correction scheme," J. Comp. Physics, vol. 224, pp. 1293-1306, 2007.

[13] J.-M. Jin, The Finite Element Method in Electromagnetics, 2nd ed. Hoboken, NJ: Wiley, 2002.

[14] J.- M. Jin and D. J. Riley, Finite Element Analysis of Antennas and Arrays. Hoboken, NJ: Wiley, 2008.

[15] V. Dolean, H. Fahs, L. Fezoui, and S. Lanteri, "Locally implicit discontinuous Galerkin method for time domain electromagnetics," vol. 229, no. 2, Journal of Computational Physics, Jan. 2010.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Domain Decomposition Method

HFSS solves a linear matrix equation of the form:

$$\mathbf{Ax} = \mathbf{b} \quad (1)$$

where the excitation vector **b** is commonly called the right-hand-side or RHS. The number of RHS vectors depends on the total number of modes used to excite the problem.

HFSS provides three distinct solution methods. By default, a direct solver is used and is the preferred method for small to medium size problems and/or problems with a large number of modes. A major strength of the direct solver is the ability to provide a solution for a large number of modes with relatively small computational overhead for the multiple RHS. However, if the problem size is too large for the machine memory, the second option is to use the iterative solver which can handle larger problems but requires a complete solution for each RHS. The iterative solver modifies the matrix equation (1) by multiplying both sides with a pre-conditioner, M, in order to enable a faster solution:

$$\mathbf{MAx} = \mathbf{Mb} \quad (2)$$

Both the direct solver and the iterative solver work only with a single machine's shared memory. This limitation is addressed in HFSS with the introduction of an advanced solving technique called the Domain Decomposition Method (DDM) which enables the use of distributed memory across many different machines to solve very large problems.

- Basic DDM Theory
- Computational Memory
- Computational Time

Basic DDM Theory

DDM has emerged as a powerful and attractive technique due to its inherent parallelism which enables the use of distributed memory. DDM is based on a divide-and-conquer philosophy where instead of solving a large and complex problem directly, the original problem as defined by the mesh is partitioned into smaller, possibly repetitive, and easier to solve sub meshes or sub-domains. In this approach it is critical to enforce continuity of electromagnetic fields at the interfaces between adjacent sub-domains through some suitable boundary conditions.

To illustrate the basic idea of DDM, we solve (1) by decomposing the original problem into two domains. Subsequently, we have:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{bmatrix} \mathbf{A}_{11} & \mathbf{A}_{12} \\ \mathbf{A}_{21} & \mathbf{A}_{22} \end{bmatrix} \begin{bmatrix} \mathbf{x}_1 \\ \mathbf{x}_2 \end{bmatrix} = \begin{bmatrix} \mathbf{b}_1 \\ \mathbf{b}_2 \end{bmatrix} \quad (3)$$

where $\mathbf{A}_{ii}$, $\mathbf{x}_i$ and $\mathbf{b}_i$, i=1,2 are system matrix, solution vector and RHS for domain *i*, respectively and $\mathbf{A}_{12}$, $\mathbf{A}_{21}$ are the coupling matrices between the two domains. To solve in parallel sub-domain problems, one popular domain decomposition algorithm is of Jacobi type:

$$\begin{bmatrix} \mathbf{A}_{11} & \mathbf{A}_{12} \\ \mathbf{A}_{21} & \mathbf{A}_{22} \end{bmatrix} = \begin{bmatrix} \mathbf{A}_{11} & \mathbf{0} \\ \mathbf{0} & \mathbf{A}_{22} \end{bmatrix} + \begin{bmatrix} \mathbf{0} & \mathbf{A}_{12} \\ \mathbf{A}_{21} & \mathbf{0} \end{bmatrix}. \quad (4)$$

Using (4) and applying iteration, (3) can be solved as

$$\begin{bmatrix} \mathbf{A}_{11} & \mathbf{0} \\ \mathbf{0} & \mathbf{A}_{22} \end{bmatrix} \begin{bmatrix} \mathbf{x}_1 \\ \mathbf{x}_2 \end{bmatrix}^{(n)} = \begin{bmatrix} \mathbf{b}_1 \\ \mathbf{b}_2 \end{bmatrix} - \begin{bmatrix} \mathbf{0} & \mathbf{A}_{12} \\ \mathbf{A}_{21} & \mathbf{0} \end{bmatrix} \begin{bmatrix} \mathbf{x}_1 \\ \mathbf{x}_2 \end{bmatrix}^{(n-1)}. \quad (5)$$

Through some simple algebra, equation (5) leads to two coupled systems:

$$\mathbf{A}_{11}\mathbf{x}_1^{(n)} = \mathbf{b}_1 - \mathbf{A}_{12}\mathbf{x}_2^{(n-1)} \qquad \mathbf{A}_{22}\mathbf{x}_2^{(n)} = \mathbf{b}_2 - \mathbf{A}_{21}\mathbf{x}_1^{(n-1)} \quad (6)$$

By setting $\mathbf{x}_1^{(0)} = 0$ and $\mathbf{x}_2^{(0)} = 0$ and, we have initial guess $\mathbf{x}_i^{(1)} = \mathbf{A}_{ii}^{-1}\mathbf{b}_i$ which is the local solution of each sub-domain problem. These initial solutions are then refined through coupling matrices $\mathbf{A}_{12}$ and $\mathbf{A}_{21}$ until equilibrium is reached.

The approach (5) is known as a stationary Jacobi solution of (3) which unfortunately converges rather slowly. A more advanced approach is to apply as a preconditioner

leading to:

$$\begin{bmatrix} \mathbf{A}_{11} & \mathbf{0} \\ \mathbf{0} & \mathbf{A}_{22} \end{bmatrix}^{-1} \begin{bmatrix} \mathbf{A}_{11} & \mathbf{A}_{12} \\ \mathbf{A}_{21} & \mathbf{A}_{22} \end{bmatrix} \begin{bmatrix} \mathbf{x}_1 \\ \mathbf{x}_2 \end{bmatrix} = \begin{bmatrix} \mathbf{A}_{11} & \mathbf{0} \\ \mathbf{0} & \mathbf{A}_{22} \end{bmatrix}^{-1} \begin{bmatrix} \mathbf{b}_1 \\ \mathbf{b}_2 \end{bmatrix}, \quad (7)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{bmatrix} \mathbf{I} & \mathbf{A}_{11}^{-1}\mathbf{A}_{12} \\ \mathbf{A}_{22}^{-1}\mathbf{A}_{21} & \mathbf{I} \end{bmatrix} \begin{bmatrix} \mathbf{x}_1 \\ \mathbf{x}_2 \end{bmatrix} = \begin{bmatrix} \mathbf{A}_{11}^{-1}\mathbf{b}_1 \\ \mathbf{A}_{22}^{-1}\mathbf{b}_2 \end{bmatrix}. \quad (8)$$

Computational Memory

Using domain decomposition we need to solve (factorize) $\mathbf{A}_{11}$, $\mathbf{A}_{22}$ compared to solving **A** using the standard method. When a direct solver is used, DDM is inherently more memory efficient since a series of smaller matrices are needed to be solved compared to a large matrix. This is evident by assuming that domains are equally partitioned so that $N_1 = N_2 = N/2$ where N_1, N_2, and N are numbers of unknown for the sub-domains and the original problem respectively. Memory requirement for original problem typically scales as C_1N^a where C_1 is a constant and $\alpha > 1$. For an M domain decomposition solve, the total memory requirement is

$$MC_1\left(\frac{N}{M}\right)^{\alpha} = M^{1-\alpha}C_1N^{\alpha}$$

For example, if $\alpha = 1.5$, using 2-domains we save about 30% of total memory. Also keep in mind that the memory can be distributed across different machines enabling a problem to be solved which is not possible on a single machine.

Computational Time

A domain decomposition solution consists of three steps:

1. Mesh partitioning, a relatively fast process. HFSS will automatically partition the mesh into roughly equivalent sized mesh sub-domains to balance memory and solution times for the domains.
2. Matrix-assembly/solve where both **A** and **M** are solved in parallel
3. Domain iteration where we solve **MAx=Mb** via an iterative solver.

The solution time for solving the entire system (1) without DDM requires C_2N^{β} operations with a direct solver where $C_2$is a constant and $\beta > 1$ (and typically larger than α). For an M domain DDM solve, the factorization time of step 2 above is dominated by the most time consuming (i.e., largest) domains, given as:

$$C_2\left(\frac{N}{M}\right)^{\beta} = M^{-\beta}C_2N^{\beta}$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Therefore, given $\beta > 1$, this stage exhibits a super-linear speed-up. This type of speed-up is commonly referred to as "scalable".

For step 3 the major computation is on the matrix-vector multiplication (**MA**)**x** which can be parallelized. In each iteration, the master node produces a new solution vector **x** and then sends its appropriate components to each domain (cpu/node/machine). Each domain then sends back to the master the vector **y=MAx** which is used by the master to produce a new solution vector **x**. This process goes on until **x** converges. In this step, the cost of matrix-vector multiplication per iteration is also scalable with respect to number of domains. Therefore if the iteration number remains relatively unchanged as number of domains increases, the domain decomposition solver can exhibit overall scalable performance.

HFSS SBR+ Technical Overview

HFSS SBR+ is a shooting-and-bouncing ray (SBR) method and solver for efficient prediction of installed far-field antenna patterns, coupling (S-parameters) between antennas, and Radar Cross Section (RCS) in the presence of electrically large scattering geometries. These sections provide technical details for working with SBR+.

- Shooting and Bouncing Rays (SBR)
- Excitations for SBR+
- Geometries and Boundaries in SBR+
- Meshing Operations for SBR+
- Physical Theory of Diffraction (PTD) and Uniform Theory of Diffraction (UTD)
- GO Blockage Technical Note
- Creeping Waves
- Visual Ray Tracing (VRT)
- Current Conformance and Array Conformance in SBR+
- Ray Statistics for SBR+ Analysis

Shooting and Bouncing Rays (SBR)

SBR+ uses a high-frequency (asymptotic) ray-tracing approximation that efficiently computes scattering from electrically large geometries. SBR+ can be thought of as a hybridization of geometrical optics (GO) and physical optics (PO). Many GO rays are launched from a radiation source (e.g., Tx antenna) toward the visible surfaces of the scattering geometry. The launched GO rays are vector-field weighted by the source antenna pattern or current distribution and represent diverging volumetric ray tubes. When a ray hits a surface, the ray "paints" a PO current on the geometry according to the GO boundary condition. These currents are radiated to the analysis observation domain for determination of far-field radiation or antenna coupling.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Then, the ray is reflected and traced, possibly to a second bounce. Its vector fields are updated according to GO and the material properties of the reflecting surface. When the reflected ray hits a surface again, it paints a second-bounce current and progresses to third bounce and so on. The process continues until the ray escapes from the scene without hitting another surface or reaches a maximum bounce number. If a penetrable surface is encountered by the ray, then a transmission ray is also generated in addition to the reflection ray.

The SBR methodology is intended to predict scattering from large-scale geometry, such as an antenna installation environment or platform. In general, it is not suitable for detailed simulation of the antennas, themselves, excepting cases where the antenna proper includes electrically large scattering geometry (e.g., reflector antenna). For this reason, HFSS SBR+ simulation requires some electromagnetic description of the source that launches and weights the rays. Conceptually, this source description can come in one of several supported forms when using HFSS SBR+:

- FEBI and/or IE simulation of antenna, resulting in distributed current sources or a far-field pattern excitation
- Idealized antenna source models (e.g., dipoles, monopoles, slots, horns, etc.)
- Imported far-field antenna patterns
- Plane-wave source (RCS application)

The first three are antenna characterizations that can also serve as receivers in an SBR antenna-coupling simulation. In the case a plane-wave source for determining RCS, the rays are launched in parallel from a plane outside the scattering geometry, being given equal initial weight. In this case, the represented volumetric ray tubes are initially neither diverging nor converging.

SBR workflow

The algorithm implemented in HFSS SBR+ is summarized in the following steps:

1. SBR sources may be introduced through Parametric Antennas in the SBR+ Solution Type, Linked Source designs, or an antenna design (with FE-BI or Radiation boundaries) within the same Driven Modal or Driven Terminal solution type. Also, Incident Plane Wave may be used for Radar Cross Section (RCS).
2. Launch many rays from the source toward the scattering geometry. The rest of the steps are the same for each ray.
3. Assign each ray a vector field weight according to the source field distribution.
4. For each launched ray:
    a. If the ray escapes from scene without hitting any surface, ignore it.
    b. If the ray hits a surface, generate a reflected ray. The reflected ray propagates GO reflected fields modified to account for surface curvature at the hit point and the surface material interface.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. If the ray hits a penetrable surface, also generate a transmitted ray, heading in the same direction as the incident ray. The transmitted ray propagates GO transmitted fields modified to account for the surface material interface.

5. At the ray hit-point, compute the incident, reflected, and if penetrable transmitted, field based on material properties of the surface. Use these and the PO approximation to determine equivalent surface currents.
6. Radiate the equivalent currents to observation angles/points. If the observation point represents an Rx antenna, compute the coupling of the radiated fields into the receiver according to the Rx gain or current distribution.
7. Continue tracing the reflected and transmitted rays generated in step 3 and repeat from there. The process continues until either the ray escapes from scene or the maximum bounce limit is reached.

Step 6 is implemented with the following equation:

$$\bar{E}_s = \frac{1}{4\pi}\int_{s'} dS' \left[(\hat{n}' \cdot \bar{E})\nabla' \frac{e^{-jkR}}{R} + (\hat{n}' \times \bar{E})\nabla' \frac{e^{-jkR}}{R} - j\omega\mu_0(\hat{n}' \times \bar{H})\nabla' \frac{e^{-jkR}}{R}\right] \quad (1)$$

where $\bar{E}_s$ is the scattered electric field at the observation point, S' is the domain of the ray tube at the hit point and projected onto the surface, $\bar{E}$ and $\bar{H}$ are the total electric and magnetic fields at the surface when considering the incident, reflected ray and transmitted R is the distance from the source to the observer ($R = |\bar{r} - r'|$, which depends on the integration point within S'), r' is the position vector to the equivalent current density in the ray footprint, $\hat{n}'$ is the surface normal, $\bar{r}$ is the position vector to the observer, μ_0 is the free-space magnetic permeability. The equivalent currents mentioned in Step 5 are those obtained by the cross product or dot-product of $\hat{n}'$ with $\bar{E}$ or $\bar{H}$, as indicated in Eq. (1).

The above expression can be used to compute the scattered field contribution in near- or far-field observations. For far-field observation angles, the factor exp(*-jkR*)/*R* in Eq. (1) is replaced with a phase factor corresponding to the chosen phase reference point, yielding a result in units of volts instead of volts-per-meter.

For Tx/Rx antenna coupling, $\bar{E}_s$ is converted into the scattered field contribution to an S-parameter (i.e., coupling ratio) by computing received signal, *b*, at the Rx port using the Rx antenna gain or current distribution. The S-parameter is the ratio of the received signal to the incident signal, *a*, at the Tx port:

$S = b/a$.

When an Rx antenna is characterized to SBR in a far-field representation, the received signal *b* is given by

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$b = \frac{-j\lambda}{\sqrt{4\pi\eta_0}} \overline{E} \cdot \overline{g}_{Rx} \ , \qquad (2)$$

where $\overline{g}_{Rx}$ is the field-scale (complex), far-field, polarized (vector) gain of the Rx antenna. Here, gain differs from antenna directivity by incorporating the effects of mismatch between the Rx antenna and its port and also ohmic losses in the Rx antenna. $\overline{E}$ is the total electric field at the Rx antenna, and η_0 is the free-space impedance. The obtained quantity, b, is the complex amplitude of the received signal delivered to the port. Since antenna gain has angular dependence and cannot be reduced to single number (or vector) for a given antenna, a subtle point of Eq. (2) in this context is that it is actually a summation over all contributing “ray paths”. Each term in that summation has an associated electric-field contribution ($\overline{E}$), and the direction of the incoming ray path to the Rx antenna resolves the angular dependence in the Rx antenna gain to a concrete vector, allowing the dot-product to be evaluated. These contributions comprise the incident-field contribution from the Tx antenna and the scattered-field contributions (see Eq. (1)) from each ray footprint, thus yielding the total-field contribution to the received signal. “Ray paths” as used here should not be confused with SBR GO rays that, for the most part, travel nowhere near a given Rx antenna. Here, “ray path” merely means the vector from the Tx antenna (incident-field) or SBR ray footprint (scattered-field) to the Rx antenna.

For an Rx antenna described by a current distribution, Eq. (2) generalizes to a more accurate one where the incident and scattered electric- and magnetic-fields are multiplied by the amplitude vectors of the Rx antenna current distribution through a vector dot product. That is, the incident or scattered field is computed separately at each current "source" of the Rx antenna and the vector dot products are then coherently summed.

Although the current sources and gain pattern represent the Rx antenna as a radiator, they serve to also characterize the Rx antenna as a receiver according to the reciprocity theorem. For example, reciprocity is the basis for using Eq. (2) to determine the received signal of an Rx antenna characterized by its far-field radiation gain pattern.

Additional capabilities in SBR+

In HFSS SBR+, the baseline SBR solution described above can be improved with ray-based physics additions. Physical Theory of Diffraction (PTD) corrects surface currents near wedges. Uniform Theory of Diffraction (UTD) shoots additional diffracted rays from wedges. Creeping Waves shoots rays propagating along smoothly curved surfaces.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Excitations for SBR+

Antenna and SBR+ Hybrid Region in Driven Modal Solution Type

Within the Driven Modal and Driven Terminal solution types, HFSS can use SBR+ to compute the scattered field contribution from a platform geometry. More information can be found in Assigning SBR+ Hybrid Regions.

When an antenna is simulated via FEM, the antenna must be enclosed by a Radiation boundary or FE-BI Hybrid Region. Instructions may be found in Assigning Radiation Boundary and Assigning FE-BI Hybrid Region. First, the antenna is solved by HFSS in the absence of the SBR region. The bounding surface of the Radiation Boundary or FE-BI Hybrid Region is also used as the exterior Huygens surface whose field samples (equivalent currents) yield current sources or far-field radiation patterns to drive the SBR+ simulation or serve (via reciprocity) as a receiving antenna in an SBR+ simulation. When Near Field is selected for Field Type, currents are computed on the Huygens surface using

$$\boldsymbol{J} = \boldsymbol{n} \times \boldsymbol{H},$$

$$\boldsymbol{M} = -\boldsymbol{n} \times \boldsymbol{E},$$

where $\boldsymbol{J}$ and $\boldsymbol{M}$ are the equivalent electric and magnetic current sources, n is the outward normal of the Huygens surface, $\boldsymbol{H}$ and $\boldsymbol{E}$ are Electric Fields and Magnetic Fields computed by FEM solver.

A Perfect E Boundary may be assigned to some faces of the exterior Huygens surface, but at least one face must be assigned to Radiation Boundary or FE-BI region. When Perfect E is assigned on a face, no currents for SBR+ simulations are generated on that face. More instructions are found in Assign a Perfect E Boundary. Also, presence of Perfect E indicates that the antenna has a ground plane, and the antenna can be properly installed on a platform. When Current Conformance is set to Auto, current conformance is performed only on antennas where at least one surface is assigned to Perfect E.

When SBR and IE hybrid regions are assigned to different objects, the FEM with IE regions must first be solved together via two-way coupling. The instructions are found in Set Current Source Option. When two-way coupling is configured, current sources are also generated on the IE region to launch rays in an SBR+ simulation.

Antenna introduced via Linked Source

The antenna may be introduced via linked sources for SBR+ simulations. When a designed antenna is already analyzed, a linked source is a convenient way to introduce the antenna into SBR+ simulations without analyzing the antenna again. Instructions for configuring a linked source are found in Link from a Driven Modal or Driven Terminal Antenna to an SBR+ design for SBR+ solution type, and Far Field or Near Field representations for Driven Modal or Driven Terminal solution type.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When a near field linked source is introduced for SBR+ simulation, fields and current sources are also generated on the exterior Huygens surface.

A far-field linked source is converted to a pattern-weighted point-source for the SBR+ simulation. It is therefore handled with much greater efficiency by SBR+ than a near-field linked source, which presents SBR+ with a distribution of current sources, each of which launches its own ray set. *However, the far-field representation is only valid if the SBR+ scattering geometry is in far-field of the source*. More discussion on this crucial point is found in the later section, Near Field vs Far Field Source Representation. Moreover, the phase reference of the far-field representation must be manually specified to be as close as possible to the physical antenna phase center by selecting **Excitations>Set Far Field Phase Center...** in the source design of the antenna. More instructions are found in Link from Driven Modal or Driven Terminal Antenna to SBR+ Design.

Incident Plane Wave

A plane wave is a wave that propagates along a fixed direction where the electric and magnetic fields are in the transverse plane and perpendicular to each other. For SBR+, Incident Plane Wave is required to compute Radar Cross Section (RCS). More instructions are found in Incident Plane Wave.

The phase reference of the incident plane wave may be set as the Zero Phase Position in the Incident Wave panel. For monostatic RCS simulation, this serves as the phase reference for both illumination and observation. For bistatic RCS simulation, the Far Field Radiation Sphere Setup allows one to set the phase reference for observation, which can be set up via a relative coordinate system.

Parametric Antennas

The SBR+ Solution Type provides a suite of parametric antennas that are very convenient for system analyses. Parametric antennas are sources that either have no physical structure definition or have a highly idealized physical structure. No model parts are constructed or used in the simulation. Parametric antennas are intended to be very quick to define and place, copy and duplicate, and are thus intended for systems studies where the specific antenna structure is either unknown or where a simple, topologically representative antenna type is sufficient. The editable parameters depend on the antenna type. More detailed instructions are found in Parametric Antennas for SBR+ Solution Type.

Parametric Antennas create far-field pattern sources that can provide S-parameter outputs and composite far-field radiation patterns. Some Parametric antennas also provide a current-source representation option. In general, a current-source representation yields more accurate results than its far-field counterparts, especially when placed nearby platform scattering geometry within 2 or 3 wavelengths. This is because it provides a near-field description of the antenna that is physical everywhere, whereas the far-field description is only physical beyond a critical distance from the antenna where it’s far-field has converged. In particular, current-source

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

antennas with implicit ground-planes (e.g., Wire Monopole, Parametric Slot, Horizontal Dipole, and Cross-Dipole over Ground Plane) are intended to be placed directly on a highly conductive surface of the platform (SBR+ scattering region). In these situations, using the same antenna's far-field counterpart is not recommended. For example, if you place the far-field version of the Wire Monopole on a supporting platform that provides the grounding function to the wire monopole, the half-donut *far-field* pattern of the Wire Monopole will only interact with structures in the upper half-space of the placement surface. In contrast, when using the current-source representation of the Wire Monopole, the axial current sources radiate in all directions, and SBR can correctly predict the interaction with the placement structure directly underneath because of the higher-order fields imbedded in the current-source representation.

By File Antenna

User defined far-field data in .ffd format may be imported into the SBR+ solution type. The SBR+ solver will automatically interpolate the source fields from the file-based antenna in order to represent the antenna at all frequencies of the user defined frequency sweep in the solution setup. The file-based antenna must be defined with at least two frequencies, and the min/max frequencies of the simulation sweep should fall within the frequency band of the file-based antenna. In the model window, the antennas display as a scaled far-field pattern at the antenna origin. Instructions are found in Parametric Antennas for SBR+ Solution Type. The file formats for .ffd are found in Frequency Independent Far Fields or Frequency Dependent Far Fields.

Current Conformance

The instructions for activating current conformance are found in Set SBR+ Source Region. It is only available for antennas represented by Near Field and parametric antennas that allow an Antenna Representation set to Current Sources. This capability is primarily intended for antennas requiring a ground plane to operate (e.g., monopole, blade, microstrip patch, slot, etc.) and that will be placed on a flat or slightly curved metallic section of the platform that conceptually provides this grounding function. It is typically used to make small positional adjustments to current-source positions that will improve simulation accuracy. There are a couple of reasons why one should consider using the current conformance feature.

- The unadjusted sources may otherwise be too close or possibly even slightly underneath the platform placement surface due to small positioning errors in placing the antenna on the platform. In general, placing sources that are conceptually intended to be directly on the surface closer than λ/40 above the surface harms run time with little or no accuracy benefit, as in the case of horizontal magnetic sources representing a slot cut in the placement surface.
- The platform placement surface is slightly curved, the shape of the actual antenna will conform to this curvature, yet the antenna might not exhibit this intended curvature (it is flat).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These situations are fairly common, so the current conformance feature is a valuable capability. When the antenna structure is to be placed directly on the platform, such that it touches the platform, conformance is often beneficial.

Current Reduction

For SBR+ simulations where the Tx and/or Rx antennas are defined by an equivalent currents on a large Huygens surface (e.g., FEBI sources), the current-source reduction feature can significantly improve run time by removing a large number of very weak sources. The instructions for setting up the current reduction are given in Set SBR Current Source Option.

When antennas are represented by current sources, run time generally grows with the number of current sources. When applied to both the Tx and Rx antenna in an antenna coupling simulation, this effect is multiplicative, not additive. Antennas are sometimes represented by hundreds or thousands of current sources, and this can result in excessive SBR+ run times. The SBR+ solver will dynamically aggregate current sources when illuminating far-away surfaces to mitigate this run-time impact. While quite effective, it only applies to Tx antennas, not Rx antennas. At the same time, many of the current sources may be very weak, with the dominant ones just a small fraction of the total. One way to reduce run time without significant compromise of accuracy is to thin the current sources according to their relative powers, and this is the function of the Current Reduction option. Current reduction (optional) and dynamic aggregation (always on) can operate simultaneously, and current reduction has the further advantage of also speeding up computations involving Rx antennas.

Near Field vs Far Field Source Representation

Representing an antenna by its near-fields on a Huygens surface is always more accurate than representing it by a far-field pattern. This is because the current distribution produced on the Huygens surface captures both the near-field and far-field radiation fields of the antenna, whereas the far-field pattern sacrifices near-field information. By reciprocity, the same is true when an antenna is used as a receiver in antenna coupling simulations.

If there are surfaces in the SBR+ scattering geometry in the near-field of the antenna, then it ***is not suitable*** *to represent the antenna in SBR+ by its far-field antenna pattern*. Thus, when representing an antenna directly mounted on a grounding surface, such as a fuselage, the antenna should always be specified according to its current distribution, not its far-field pattern. Otherwise, one cannot expect accurate results, especially in the shadow region under the mounting surface.

As a practical matter, installed antennas always require some sort of mounting structure, and since the part of this structure attaching to the antenna must be in its near-field, using the far-field antenna pattern representation might never seem justified. *This would be the wrong conclusion.* For example, consider the case of a reflector antenna with a feed horn at its focal point. Suppose we wish to model this case in SBR+, where the horn is the source antenna and the reflector is the SBR+ scattering geometry described by the CAD model. The feed horn is

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

attached to a mounting plate that is supported by thin rods extending from the outer rim of the reflector and converging on the sides of the mounting plate. As a very good approximation, we can model this by specifying in SBR+ the far-field pattern of the feed horn. Now, technically speaking, the sections of supporting rods near the mounting plate are in the near-field of the horn. However, depending on the details, these are not significant scatterers in the context of determining the radiation pattern of the reflector.

We may even choose to include the supporting rods in the CAD model, recognizing that the scattering from the rod sections near the horn is not accurately captured since they are illuminated by a far-field pattern that has not yet had sufficient radius to form. However, since scattering from these nearby portions of the support rods is small compared to the overall scattering of the reflector, this problem can possibly be ignored.

Another example would be an input antenna mounted on the front of a wing pod. If the antenna is self-contained and predominantly radiates in the forward direction (such as a cavity-backed spiral), a reasonable approach is to model this antenna according to its forward half-space antenna pattern (or perhaps an even larger conical section), setting the fields in the backward half-space (or conical section) to zero. That way, one does not illuminate the mounting pod with far-field data that are not relevant to the mounting conditions and that have, at any rate, not yet had sufficient distance to be valid. Alternatively, we may remove the mounting pod from the CAD model.

With a current-source representation, run time grows in proportion to the number of current sources in the Huygens surface, although this behavior is substantially mitigated by the *SBR+*dynamic aggregation feature (described below). This is because a separate set of rays must be launched from each current source and then processed. Thus, the run time for a Tx antenna described by a single current source is roughly the same as that for a Tx antenna (placed in the same environment) represented by a far-field antenna pattern. However, if the antenna is instead represented by N current sources, the run time will be up to N times as long. For this reason, if accurate results can be obtained using the far-field pattern representation, and run time is a consideration, this is the approach that should actually be used.

There is an important caveat to the above point about run time. Many antennas need to be characterized by a large number of current sources, numbering in the hundreds or thousands. With a brute force simulation, the SBR+ run times could then be excessive. However, SBR+ invokes an important feature known as dynamic zoning that greatly mitigates the run-time impact of having a large number of current sources with negligible loss of accuracy.

Given the above guidance, it is important to understand what is meant here by the complementary terms, "near-field" and "far-field". The far-field region of an antenna extends from some minimum distance R_{min} to infinity, where R is the radius measured from the center of the antenna. That is, we require $R > R_{min}$ for an observer at R to be in the far-field of the antenna. For a transmitting antenna, R_{min} is the distance beyond which the radiated fields decay as $1/R$ and are transverse (perpendicular) to the observation direction. If one of these conditions is not satisfied, then one is in the near-field of the antenna.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Owing to the continuous nature of fields, there is no precise way to quantify R_{min} for a particular antenna. That is, the far-field conditions identified above are only precisely satisfied in the limit as R goes to infinity. However, there are well established formulas for specifying R_{min} beyond which these conditions are satisfied to an excellent approximation. Specifically, we require

$$R > \frac{2D^2}{\lambda}$$

and

$$R > 3\lambda$$

The first criterion is the parallax far-field criterion. Here, D is the largest linear dimension of the antenna. It is generally relevant for large aperture antennas and antenna arrays. The second criterion is based on the distance beyond which only the propagating field components remain and field components associated with stored energy and charge contributions have decayed to negligible levels. Here, R must be measured conservatively (i.e., measured between the observation point and the nearest point on the antenna). The second criterion is more relevant to small, single element antennas, such as a dipole or monopole. Regardless, if either of these criteria is violated, then one is in the near-field of the antenna, whether it is serving as a Tx or Rx antenna. If there are *significant* scattering surfaces in the CAD model within either of these distances, then a current-source distribution representation should be used.

Geometry, Boundaries, and Dielectric Region Support in SBR+

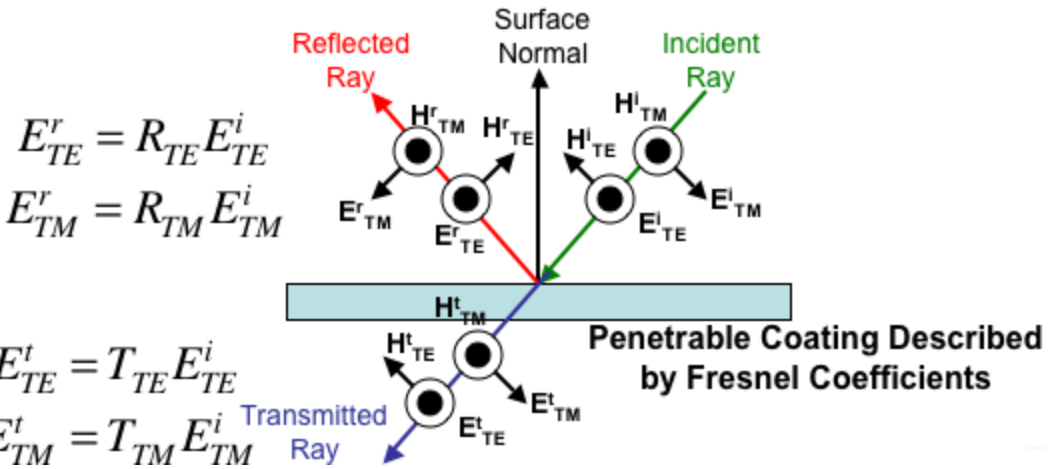
Boundaries Concept in SBR+

In SBR+ simulations, all geometries other than dielectric regions (discussed in Assigning a Dielectric or Conducting Material to Volumetric Region) are introduced as surfaces based on Fresnel reflection and transmission coefficients. Tables of these coefficients are either developed internally or they can be read from a user-provided file.

When a ray hits a surface, its Geometrical Optics (GO) ray fields are decomposed into surface-local transverse-electric (TE) and transverse-magnetic (TM) polarizations. Then, based on the boundary assigned to the surface and the incidence angle relative to the surface normal, the Fresnel TE and TM coefficients are obtained from the table. These are applied to the incident field to determine the GO fields of the reflected ray. If the boundary is penetrable, then Fresnel transmission coefficients are also developed from the same table and applied to determine the fields of the transmitted ray. Because the equivalent currents are developed from the total GO

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

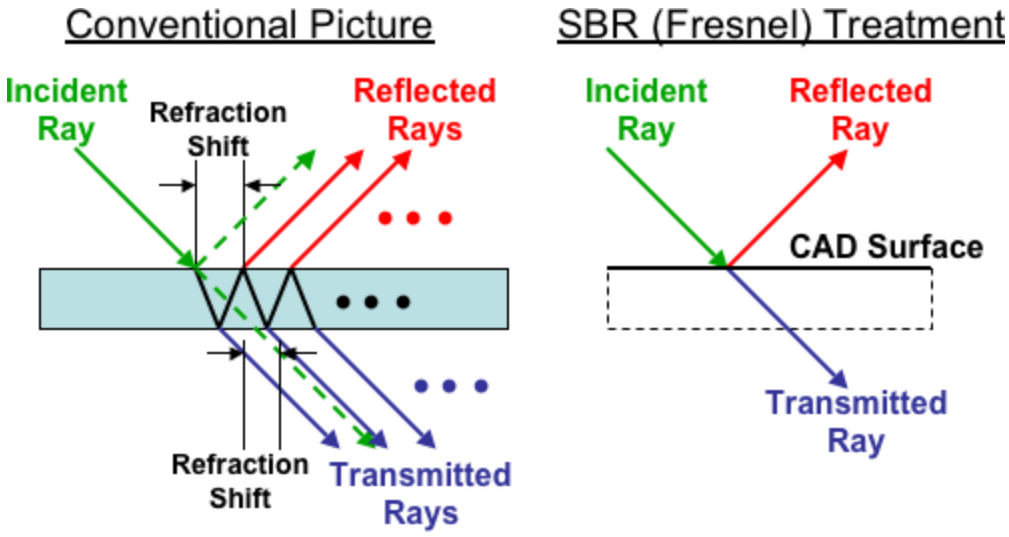
field at the surface, the Fresnel coefficients not only influence fields of the GO reflected and transmitted rays but also the scattered field results.



The Fresnel coefficients are defined assuming an infinitely extended plane wave and a layered planar surface that is laterally uniform and unbounded. Since, these assumptions are never satisfied - the scattering geometry is finite, rarely flat, and the boundaries change from region to region - the question arises about the suitability of Fresnel coefficients. The same objection can be made of the Physical Optics (PO) approximation applied to metallic surfaces, yet PO is a well-established approximation. This approach to handling layers is essentially a generalization of traditional PO. Provided the surface does not curve too rapidly and spans a moderate number of wavelengths, using Fresnel coefficients to determine the GO ray fields and the equivalent currents that produce the scattered field is a good approximation of the actual fields that would be measured.

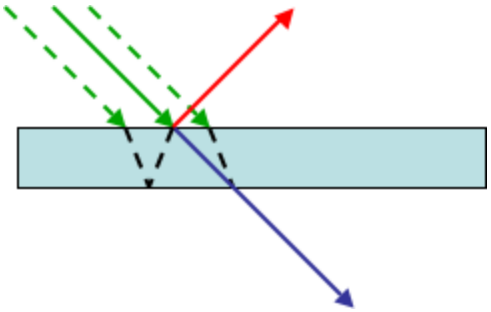
The question often arises as to whether this boundary model includes the effects of refraction, especially if the dielectric layers are electrically thick. Consider a single dielectric layer. The common picture from ray optics is that a ray enters a dielectric layer from free-space, bends due to the change in refractive index, travels through the layer, and then bends back upon leaving the layer. Hence, the exiting ray travels in the same direction as the entering ray, but it is shifted over. The reflected ray is also shifted. Moreover, there are many reflecting and transmitting rays escaping due to internal reflection within the layer. This complex picture, shown on the left in the figure below, is in sharp contrast to the simple diagram on the right that shows the rays as they are actually traced in SBR. Note that the finite thickness layer is only manifested in the reflection and transmission coefficients applied to the GO rays, not in the thin-surface CAD model itself.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In SBR, there is no spatial shift in the transmitted or reflected ray, and multiple exiting rays are not generated due to internal reflections within the layer. Again, the question arises as to whether the implementation is missing something important. The answer is that nothing is missing. The Fresnel coefficients entirely capture the impact of refraction and internal reflections, but not in the way we are used to thinking about it.

To see why, it is important to understand that while rays are traced individually, an individual ray with no neighbors has little physical meaning. In SBR+, and in GO more generally, each ray is a sampling of a larger wavefront. Individual, isolated rays do not really refract because individual, isolated rays do not even exist. The more complete picture for SBR is shown in the figure below. To preserve the refraction picture as commonly understood, one can think of the reflected or transmitted ray as continuing not the original incident ray but one of its neighbors. Likewise, the multiply reflected and transmitted rays are just continuations of other ray neighbors. The entire effect is built into the magnitudes and phases of the Fresnel coefficients. Moreover, if the simulation is applied to very thick materials and conducted over a wide frequency band, with the incident field Fourier-transformed into a narrow time-domain pulse, then the Fourier-transformed scattered fields will manifest multiple internal reflections that occur in the dielectric layer.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Perfect E Boundary

A Perfect E boundary represents a Perfect Electric Conductor (PEC) coated surface for SBR+ simulations. The configuration instructions are found in Assigning Perfect E Boundaries. In SBR+ simulations, Perfect E is a reasonable assignment for surfaces that are highly conductive at the simulation frequency, such as Copper, Aluminum, and many other high-conductivity metals at radio or microwave frequencies. Alternatively, users may consider to use a Finite Conductivity boundary in for improved accuracy.

When a ray hits a Perfect E boundary, the reflected ray field has the same magnitude as the incident ray field. The total field at the reflecting surface (incident plus reflected ray fields) has zero electric field tangent to the surface and double the incident electric field normal to surface. The opposite is true of the magnetic field (i.e., no normal component and double tangential components). Based on these constraints, and following the polarization conventions for TE and TM illustrated earlier, one can show that R_{TE} = -1 and R_{TM} = +1 at all incidence angles for a Perfect E boundary.

When an antenna excitation is introduced in SBR+ simulations via Huygens box, Perfect E may be assigned on its faces. When Perfect E is assigned on a face, no currents for SBR+ simulations are generated on that face. Also, presence of Perfect E indicates that the antenna has a ground plane, and the antenna can be properly installed on a platform. When Current Conformance is set to Auto, current conformance is performed only on Huygens surface at least a surface assigned to Perfect E.

Perfect H Boundary

A Perfect H boundary represents Perfect Magnetic Conductor (PMC) coated surface for SBR+ simulations. The configuration instructions are found in Assigning Perfect H Boundaries. Unlike Perfect E, Perfect H does not approximate a typical condition at a physical boundary, so it's use is much more specialized.

When a ray hits a Perfect H boundary, the reflected ray field has the same magnitude as the incident ray field. The total field at the reflecting surface (incident plus reflected ray fields) has zero magnetic field tangent to the surface and double the incident magnetic field normal to surface. The opposite is true of the electric field (i.e., no normal component and double tangential components). Based on these constraints, and following the polarization conventions for TE and TM illustrated earlier, one can show that R_{TE} = +1 and R_{TM} = -1 at all incidence angles for a Perfect H boundary.

Finite Conductivity Boundary

The finite conductivity boundary is valid only if a material is classified as a PEC-like conductor. See section of PEC-like State of a Boundary below. Transmitted rays are not generated. Further instructions are found in Assigning Finite Conductivity Boundaries.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Layered Impedance Boundary

A Layered Impedance Boundary is used to model a stack of uniform layers with dielectric or conductive materials. Further instructions are found in Assigning Layered Impedance Boundaries.

The top surface of the top layer is conformal with the mesh surface, and the implied thickness of the layer is built up under the surface with respect to the incident ray. Thus, when a ray hits the boundary from above, the layers are built up below the boundary. Likewise, when a ray hits the boundary from below, the layers are built up (in the opposite direction) above the boundary.

While this is an inconsistent structural arrangement, it is convenient from a modeling perspective. The implications for users are that some thought must be given to how best to place the boundary surface in the CAD model, and this depends on which side of the surface will be predominantly illuminated by rays.

For example, consider a platform with a radome that is to be modeled with one or more dielectric layers (and no backing material). Physically, the radome has an interior surface and an exterior surface. When one is interested to predict the radiation characteristics of an antenna located behind the radome, then it is best to define the radome surfaces in the CAD model according to its interior surface. On the other hand, when one is interested to model the scattering contribution of the radome due to some source outside the radome, then it is best to define the radome surface in the CAD model according to its exterior surface. In situations where the radome is expected to be significantly illuminated by rays from both sides, the best approach is to compromise by placing the radome CAD model surface half-way between the actual radome's interior and exterior surfaces.

Layered Impedance Boundary may be configured as one- or two-sided. When configured as one-sided, the finite-thickness layers stack on top of a backing material with infinite thickness (e.g., half space). One-sided Layered Impedance is not penetrable and transmitted rays are not generated. On the other hand, when configured as two-sided, the surface is penetrable and transmitted rays are generated (see diagram above), weighted by the Fresnel transmission coefficients derived from the material and thickness properties of the layers.

Impedance Boundaries

An Impedance Boundary (IB) represents a surface with resistive and/or reactive properties. The behavior of the field at the surface and the losses generated by the currents flowing inside the surface are computed using analytical formulae. SBR+ does not actually simulate any fields inside the surface, and transmitted rays are not generated. The instructions are found in Assign Impedance Boundaries.

For an IB characterized by a surface impedance Z_S, the boundary condition at the surface is $E_{tan} = Z_S H_{tan}$, where E_{tan} and H_{tan} are the surface-tangent components of the total (incident +

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

reflected) electric and magnetic field, respectively. Angular and frequency dependent Fresnel reflection coefficients are internally developed by the SBR+ solver from this relationship.

Fresnel (SBR+) Boundaries

Fresnel (SBR+) boundary introduces Perfect Absorber Boundary and imported Reflection – Transmission Table for SBR+ simulations. The instructions are found in Assigning (SBR+) Perfect Absorber Boundaries and Assigning (SBR+) Fresnel Table Boundaries.

Perfect Absorber

Any ray hitting a face with Perfect Absorber gets terminated, thus creating no further reflections or transmissions. *However, this does not imply that there is no radiation from surface currents on perfectly absorbing surfaces*. The perfect absorber still generates very strong forward scattering. This scattering is associated with the incident GO ray fields at the surface. These ray fields are converted into equivalent currents at the surface that are radiated to observation angles/points in any direction (including the backward direction); the scattered field tends to be strongest in the forward direction relative to the incident ray. At the same time, since the reflected ray has zero field strength (and therefore no reflected ray is traced), the equivalent surface currents associated with the reflected ray are also zero, so scattering in the backward half-space (relative to the incident ray) is weak.

Since we properly expect an absorbing surface to cast a shadow behind it, the strong forward scattering of the Perfect Absorber may seem unintuitive. The intuitive expectation is recovered by considering the impact on the total field rather than the scattered field. For example, consider the simplified case an antenna illuminating flat plate configured as Perfect Absorber. Behind the plate, the incident field from the antenna is strong because, by definition, *the incident field is the radiation of the source in the absence of any scatterers*. During SBR ray-tracing, the antenna also illuminates the perfectly absorbing plate with rays, creating a strong incident field on the plate surface. This results in a strong scattered field behind the plate relative to antenna. However, in this region behind the plate, the scattered field is 180^{o} out-of-phase relative to the incident field of the antenna. Hence, in the “shadow” region behind the plate, the incident and scattered field destructively interfere, leaving a very weak total field. Moreover, the non-zero total field in this shadow region corresponds to fields diffracting around the edges of the perfectly absorbing plate.

Reflection-Transmission Table (.rttbl)

A Reflection – Transmission Table directly specifies Fresnel coefficients of the boundary through a reflection/transmission coefficients table file. Fresnel coefficients are based on the assumption of plane-wave incidence and reflection and represent the ratio of the reflected field to the incident field. Ray bundles can represent complex wavefronts, and each ray represents a sample of that wavefront that is assumed to be locally planar. Provided the wavefront is not too irregular, and the layers are not too thick compared to their lateral extent, the Fresnel coefficients

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

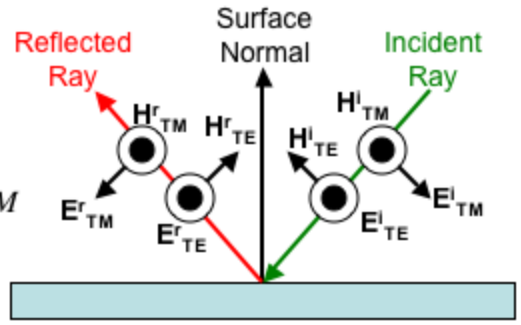
applied in conjunction with individual rays provide a good approximation of the reflection of an arbitrary wavefront off a surface.

There are two types of the tables supported by SBR+.

- reflection coefficients
- reflection + transmission coefficients

Fresnel coefficients depend on the polarization of the incident plane wave. There are two unique, orthogonal polarizations: transverse electric (TE) and transverse magnetic (TM). SBR+ adopts the convention given in the diagrams below. In this convention, PEC metal has a TE reflection coefficient of -1 and a TM reflection coefficient of +1. In general, reflection coefficients are complex quantities that depend on incident angle and frequency.

$$E^r_{TE} = R_{TE} E^i_{TE}$$
$$E^r_{TM} = R_{TM} E^i_{TM}$$



Reflective Coating Described by Fresnel Reflection Coefficients

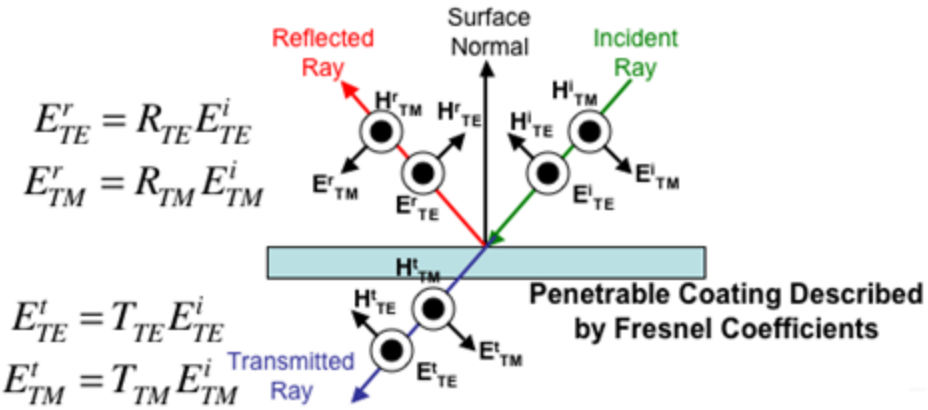
When developing tables of Fresnel coefficients for import into HFSS SBR+, it is vital to observe these polarization conventions. If the tables are generated by existing software that exports Fresnel coefficient data, then when converting the exported data into the file format expected by HFSS, special care should be taken in noting the polarization convention of that software and making sign corrections as applicable.

When a ray hits a surface, the SBR+ solver determines its incidence angle relative to the surface normal. The incident ray tube fields are decomposed into TE and TM components. SBR+ then looks up the TE and TM reflection coefficients for that angle, whether internally generated or provided by an external table as described here. These are applied to the respective incident ray field components to determine the TE and TM components of the reflected ray field.

On the other hand, penetrable boundaries, as are encountered in the case of canopies and radomes, should be described by both reflection and transmission coefficients. When creating *a Layered Impedance Boundary in SBR+*, if one specifies *Two sided*, then the solver will treat this as a penetrable and internally generate both reflection and transmission coefficients. Likewise, arbitrary boundaries that are penetrable can be entered as a table of reflection and transmission in SBR+.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When a ray hits a penetrable boundary, it generates both a reflected ray and a transmitted ray, as shown in the diagram below, which also extends the polarization convention of the previous diagram. In the diagram below, it is intentional that the transmitted ray is shown as starting from the top of the surface, rather than the bottom. In SBR+, CAD models represent infinitely thin surfaces. The effect of layer thickness is entirely captured in the reflection and transmission coefficients. *Hence, in creating a reflection + transmission table input for SBR+, it is critical to do so such that the phase reference of the transmission coefficient is the surface nearest the incident ray*. That way, the transmitted ray will include the correct phase delay given that it is launched from exactly the same point as where the incident ray hit the surface.



PEC-like State of a Boundary

The PTD, UTD, and Creeping Wave enhancements in SBR+ are developed from formulations that assume perfectly conducting boundaries. However, their results remain valid for highly conducting boundaries provided they are sufficiently “PEC-like”. In addition to the Perfect E boundary, HFSS SBR+ permits deployment of these enhancements on Layered Impedance and Finite Conductivity boundaries provided their behavior is PEC-like, as determined from details of their configuration.

In the case of a Layered Impedance boundary, whether it is considered PEC-like by SBR+ depends upon the simulation frequency and the material properties of its exposed layer (the first layer in the stack.) The following table illustrates the frequency dependence for several typical metals. For example, if the exposed boundary layer is Copper and 30-μm thick, then the entire Layered Impedance boundary stack will be considered PEC-like between f_{min} = 192 MHz and f_{max}= 735 GHz. The 192-MHz lower bound is not explicitly shown in the table but is consistent with the Minimum Thickness columns for 10 MHz and 1 GHz. For simulation frequency sweeps residing within this band, wedges with this top-layer will be considered PEC-like and therefore eligible for deployment of PTD and UTD. However, if the frequency sweep partially or completely falls outside of this band, then the same wedge will ineligible. To prevent discontinuous sweep results, HFSS SBR+ will not switch on PTD, UTD, or Creeping Wave mid-sweep if the sweep crosses one of these frequency boundaries.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material	Conductivity [S/m]	Dielectric Constant	Rel. Magnetic Permeability	Minimum Thickness [um] @ 10 MHz	Minimum Thickness [um] @ 1 GHz	Minimum Thickness [um] @ 100 GHz	Maximum Frequency [GHz]
Silver	6.10E+07	1	0.99998	128.04	12.80	1.28	772.589
Copper	5.80E+07	1	0.999991	131.31	13.13	1.31	734.585
Gold	4.10E+07	1	0.99996	156.18	15.62	1.56	519.292
Stainless Steel	1.10E+06	1	1	953.46	95.35	NA	13.932
Nickel	1.45E+07	1	600	10.72	1.07	NA	0.306
Iron	1.03E+07	1	4000	4.93	NA	NA	0.033

For a top-layer coating to be PEC-like, several conditions must be met, and the above table is developed from these conditions. The top layer must have free-space electric permittivity ($\varepsilon_r = 1$) and lossless magnetic permeability ($\mathrm{Im}\{\mu_r\} = 0$). In addition, the dielectric layer must possess the very high conductivity (σ) of a metal. Furthermore, the thickness of the top-layer coating must be greater than 2π skin depths $\delta = 1/\sqrt{(\pi f \sigma \mu)}$ as computed at the lower bound of the frequency sweep. Increasing the layer thickness or its conductivity lowers the minimum frequency (f_{min}) at which the layer exhibits good PEC properties. Because of this skin depth requirement, additional layers do not change the PEC-like status of a Layered Impedance boundary. Likewise, whether the Layered Impedance boundary is One-sided or Two-sided makes no difference.

The maximum frequency f_{max} at which the coating is deemed PEC-like increases with the top layer's conductivity σ and decreases with its relative magnetic permeability μ_r. In HFSS SBR+, f_{max} is computed as

$$f_{max} = (0.1\ \mathrm{Ohm}^2)\ \sigma/(2\pi\mu_0\mu_r)$$

Assigning a Dielectric or Conducting Material to a Volumetric Region

For a discussion, see Assigning a Dielectric or Conducting Material to Volumetric Region.

Meshing Operations for SBR+

Surface Curvature Information, Curvilinear Meshing, and Curvature Extraction

When faceted formats such as STL are imported into HFSS, each triangle of the faceted model is flat. Faceted models lack intrinsic curvature information. In order to support methodology features of SBR+ that depend on surface curvature information, such as the ray-tube divergence correction and creeping wave enhancement, local curvature is automatically synthesized

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(estimated) during SBR+ simulation according to spatial orientation relationships among the immediate neighbors of each flat facet, a process known as curvature extraction.

Unless disabled by the user, curvature extraction is applied to all faceted surfaces. Smooth curvature will be extracted under the assumption of smooth curved surfaces extending across the facet edges. For this extraction process to work well, clean imported STL models satisfying the following criteria are required:

- Two adjacent facets share two common vertices across an edge.
- Highly curved surfaces must be meshed with small faces to eliminate large changes in surface slope between neighboring facets.

Since 2019 R2, Curvilinear Meshing in SBR+ is supported for a) curved-surface geometries built within HFSS and b) imported geometries that retain explicit surface curvature information, such as SAT and IGES format models. With curvilinear meshing, accurate curvature information from the objects is directly used by the SBR+ solver. With curvature extraction, an algorithm is deployed to recover or infer information that was either lost at some point prior to synthesis and import of the faceted model, or that was never available to begin with, so it is necessarily an approximation. Curvilinear meshing is therefore preferred over curvature extraction in those situations where it is available for use with SBR+.

Since 2019 R2, explicitly flat surfaces can be specified to the SBR+ solution by selecting Disable Curvature Extraction for Faceted Surfaces. Users may disable curvature extraction for specific flat surfaces and introduce polygonal surfaces with flat curvatures. When SBR rays reflect off explicitly flat surfaces, the divergence/convergence rate of their associated ray-tube is not modified as it would be with a curved reflecting surface. Also, the creeping wave rays will not propagate over explicitly flat surfaces, as the creeping wave formulation assumes surfaces have finite (non-zero) convex curvature.

Ray-tube Divergence Correction

In SBR+, the one-dimensional rays traced on the CAD model actually represent volumetric ray tubes whose cross-sections are precisely maintained through multiple reflections and transmissions. The 1-D rays shown in the visual ray tracer reside in the centers of these ray tubes.

When rays are launched from a point and represent a spherical wavefront, as with rays launched from antenna sources, their cross-sectional area grows in proportion to R^2, where R is the propagation distance. Correspondingly, GO fields of the ray tubes decrease as $1/R$. This is known as ray-tube divergence. When a ray (tube) reflects off of a flat surface, the rate of ray-tube divergence is unchanged, where R is taken as the total path distance of the ray when starting from the Tx antenna.

Reflection off a curved surface is more complicated. When a ray tube reflects off a convex surface, its rate of divergence increases. When reflecting off a concave surface, its rate of divergence decreases. If the surface has enough concavity, the ray tube will actually start to

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

converge and pass through one or two caustics (focus points) before diverging again. In this process, the shape of the ray tube changes as it propagates. With CAD models described by curved surfaces, SBR+ accounts for this effect precisely, considering both the non-spherical curvature of the incident wavefront (if the ray already reflected off another curved surface) and also localized curvature conditions of the CAD model at the ray hit point. It is important to capture this effect, as it influences the distribution and strength of currents painted on surfaces after the reflection, and these are the currents that determine the scattered field.

In the case of an incident plane-wave source, as when using SBR+ to compute RCS, a set of rays is launched in parallel toward the scattering geometry. In consequence, the associated ray-tubes initially have zero rate of divergence, and this condition will be preserved after the rays reflect off flat surfaces. However, upon encountering curved surfaces of the scattering geometry, the reflecting ray tubes will start to converge or diverge, depending on details of the surface curvature and the angle of incidence relative to the surface perpendicular at the ray hit point.

Dynamic meshing, Tolerant Meshing, Lightweight geometry

Unlike FEM or IE, the ray-tracing methodology of SBR+ does not require sub-wavelength meshing of the scattering geometry. Rather, it is sufficient that the mesh accurately represents the shape of the scattering geometry. For large CAD models when interactive 3D visualization manipulations, geometry meshing, or post-meshing SBR+ simulation becomes too time consuming, one may consider using the relaxed import and meshing options described below.

Note that all of these relaxed meshing options skip the automated process of identifying and eliminating overlapping (coplanar) surfaces. When overlapping surfaces are passed to the SBR+ solver, this can confuse its ray-tracing engine, leading to unexpected and inaccurate results. For example, coplanar surfaces create ambiguity as to which surface was actually hit first, and this can lead to trapped rays bouncing between them. Therefore, when using these relaxed import/meshing options, there is greater responsibility upon the user to avoid the introduction of overlapping surfaces.

Lightweight geometry

Importing large STL files into Ansys Electronics Desktop can be time consuming. STL files contain triangle surface meshes. Importing them to the modeler creates one face entity for each STL triangle and one edge entity for each triangle edge. This can result in a large number of faces and edges, leading to sluggish interactive behavior with 3D visualization. The lightweight geometry import option for STL models bypasses this overhead and associated meshing operations, passing the original STL mesh directly to the SBR+ simulation engine. This results in more responsive UI operations and faster SBR+ simulations.

Dynamic meshing

Dynamic Meshing eliminates mesh generation during SBR+ simulations, which can significantly reduce simulation time. When importing STL models with the Lightweight Geometry option,

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dynamic Meshing is automatically engaged (and it cannot be switched off). However, even when Lightweight Geometry has not been engaged, Dynamic Meshing can be switched on in the Initial Mesh Settings dialog in order to receive the benefits of bypassing potentially expensive meshing operations prior to engaging the SBR+ solver.

Tolerant Meshing

If the design contains an SBR+ Region, the Initial Mesh Settings window will include a Tolerant Meshing option. This is because the SBR+ ray-tracing solver has relaxed surface meshing requirements relative to other HFSS solvers. If an imported scattering geometry for the SBR+ region contains anomalies that make it too time-consuming or impossible to use Auto (default) meshing, the tolerant meshing option should be attempted, as it will often result in a successful meshing of the geometry in cases where Auto does not. On the other hand, if the imported geometry is clean and Auto meshing works well, Tolerant Meshing can sometimes result in an excessive number of surface mesh triangles that increase SBR+ solution time. The best meshing option for SBR+ is very dependent on the quality of the imported geometry, so users need to consider both approaches and choose the one that works best for their particular scattering geometry.

HFSS SBR+ Technical Overview

Physical Theory of Diffraction (PTD) and Uniform Theory of Diffraction (UTD)

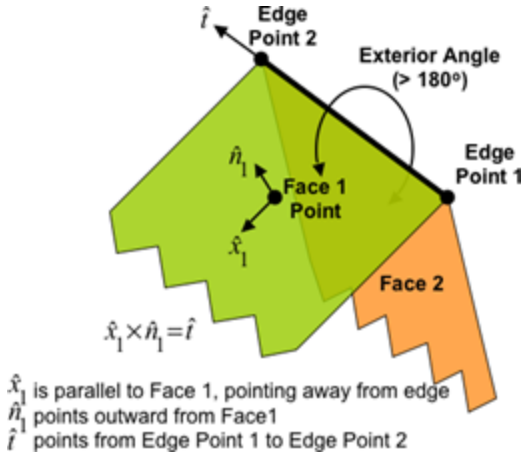
For SBR+ simulations, the Physical Theory of Diffraction (PTD) and Uniform Theory of Diffraction (UTD) physics enhancements can account for additional phenomenology not predicted by ordinary SBR due to truncation of uniform Physical Optics (PO) currents at sharp angular discontinuities (“wedges”) on metallic surfaces and blockage of SBR’s Geometrical Optics (GO) rays. PTD is a numeric correction to the scattered fields radiated by incorrect PO currents near wedges. UTD launches bundles of edge-diffraction rays from directly illuminated portions of each wedge along the Keller cone. Once launched, the UTD rays behave exactly like regular SBR rays, propagating according to GO and painting PO currents at each bounce that contribute to the scattered field. The UTD rays often illuminate portions of the SBR scattering geometry that are never reached by SBR GO rays launched directly from the field source.

The PTD and UTD wedge features are only deployed for metallic wedges with line-of-sight visibility from the source (Tx) location. If either adjacent surface of the wedge is non-PEC and not within the tolerance for PEC-like, or if the *entire* edge segment is invisible to the source, the wedge will be ignored in the SBR+ simulation and for visual ray tracing (VRT).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SBR+ Wedges

SBR+ wedges are required to enable PTD/UTD corrections. Wedges are extracted from metal surfaces if their exterior *angle* is larger than the **Maximum Wedge Angle** specified in SBR+ Wedge Settings or Wedge Settings. The default Maximum Wedge Angle is 225 degrees. Also, toggle on *Include Sheet Edges* to extract knife edges from open-ended sheets. When many wedges are extracted, PTD/UTD computations may be time consuming. The SBR+ wedges can be selectively filtered out using **Apply Source Distance filter** or **Apply Box Filter**, and PTD/UTD corrections are then applied only to the wedges that remain.



Physical Theory of Diffraction (PTD)

When SBR+ wedges are extracted from metal surfaces, SBR+ can apply a PTD wedge correction to improve the accuracy of the baseline SBR solution. Recall that SBR uses GO to extend PO to multiple bounces. The first-bounce solution of SBR is therefore traditional PO, with ray tracing used to determine which surfaces are lit. A limitation of this approximation is that it does not account for the distortion of surface currents in the vicinity of discontinuities in the geometry, such as sharp edges.

For example, consider a metal plate of moderate size in wavelengths. In the middle region, PO does a good job of approximating the currents. However, near the plate edges, PO assumes uniform currents up to the edge boundary. In reality, approaching the edge, current components perpendicular to the edges smoothly drop to zero while components parallel to the edges become very large. This localized PO inaccuracy is of little importance near specular scattering regions (i.e., near GO inter-bounce or exit ray paths in the near- or far-field) since the field results are dominated by the influence of the large area of currents over the plate interior where PO is accurate. However, the currents near the plate edges have a strong influence on diffraction in the shadow region and other regions far removed from specular scattering angles. Hence, while PO (and SBR) alone predicts diffraction in the shadow regions, it cannot be expected to consistently yield accurate levels of diffraction.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PTD theory was originally developed to address this very problem. The PTD wedge treatment is an asymptotic, coherently additive correction to PO that is designed to account for localized non-uniformity of currents near metal wedges and edges. It is somewhat analogous to the UTD correction to GO in this sense, except that the PTD correction to PO is milder than the UTD correction to GO; GO yields zero fields in the shadow region while PO always yields finite fields and already partially accounts for diffraction.

PTD was originally formulated as a correction term in the scattered fields rather than as a correction to the currents near edges that give rise to these correction fields. From the perspective of practical computation, the methodology was subsequently improved by casting PTD in terms of equivalent (artificial) edge currents running along the edges. These are sometimes referred to as incremental length diffraction coefficients (ILDCs) in the literature. One integrates these 1-D edge-axial currents along the actual path of the platform edges. For curved wedges, the equivalent currents are only based on local conditions that vary along the edge according to the changing incident and scattered field angles relative to the wedge geometry. From the standpoint of practical computation, the key benefit of using ILDCs is that they allow PTD to be applied to complex geometrical shapes encoded in 3-D CAD models. The implementation of the PTD wedge correction in SBR+ is based on this approach of equivalent, filamentary edge currents.

Users should be aware of a several limitations of the PTD wedge correction in SBR+. First, it can only be applied to metallic (PEC or PEC-like) wedges. There is no PTD correction formulation for dielectric wedges. SBR+ takes steps to ensure that the correction is not applied to wedges with one or two non-metal faces; the PTD correction is automatically skipped during the simulation when encountering wedges with non-conductive boundaries. Second, the correction is only applied to wedges that are directly illuminated by the Tx antenna, not edges indirectly illuminated via GO multi-bounce. Third, the PTD correction is only valid for field observation points on the order of a wavelength or more from the edge scattering point. This is because the correction is formulated in terms of 1-D equivalent (artificial) currents instead of a distribution of 2-D physical surface currents in the vicinity of the edge. This limitation has no impact on far-field results, but it can matter for near-field results when the observation points and Rx antennas get too close to a wedge in the CAD model. In fact, left to itself, the PTD correction based on equivalent currents becomes singular approaching the edge. To prevent this from happening, SBR+ automatically tapers the correction field generated by the equivalent currents at small distances from the edge. For example, at 1.0, 0.5, 0.2, and 0.1 wavelengths distance from a given edge, only 98%, 90%, 60%, and 28% of the correction is applied.

When a wedge consists of two flat faces meeting to from a common edge, the edge has finite axial extent. However, the faces are assumed by the PTD formulation to extend away from the edge without ending. This assumption limits accuracy when a source, wedge face, and observation angle/point are aligned.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Uniform Theory of Diffraction (UTD)

When SBR+ wedges are extracted from metal surfaces, the traditional GO rays can be augmented by another class of rays that are created by diffraction at these wedges. These rays are called UTD rays in SBR+, since they are governed by the uniform theory of diffraction at metallic wedges. UTD, formulated by Kouyoumjian and Pathak in 1974, is an extension of Keller's geometrical theory of diffraction (GTD). The coefficients of the diffracted rays in Keller's GTD exhibited singularities at the incident and reflection shadow boundaries. These singularities were removed by UTD, yielding a description of diffracted rays that is uniformly valid at all angles.

The UTD rays are an augmentation of GO rays, much like the PTD Wedge Correction is an augmentation of PO radiation. Without the UTD rays, only the regions of the CAD model that have a line-of-sight (LOS) or multi-bounce GO path from the transmitter are illuminated. This direct and indirect illumination is effected by the GO rays launched from the transmitter. With the addition of UTD rays, portions of the CAD model that do not have a LOS or multi-bounce GO paths to the transmitter can also be illuminated, improving the ray coverage on the CAD model. This type of indirect illumination is achieved when rays leaving the transmitter diffract at wedges (which are required to be in direct LOS of the transmitter, as they are for PTD wedge correction) in accordance with the uniform theory of diffraction. The resulting UTD rays reach previously shadowed points on the CAD model. It should be noted that the UTD illumination mechanism is not limited to shadow regions: UTD rays can also hit portions of the CAD model that are in the LOS of the transmitter.

The existence of a UTD ray is not guaranteed for every point on the CAD model. The defining property of a UTD ray is the minimum distance principle, as it is for a GO ray. An equivalent expression of this property is that a UTD ray leaving a wedge lies on the Keller cone at the diffraction point. The angle formed between the incident ray and the edge line at the diffraction point defines the characteristic half-angle of the Keller cone, whose cone axis is aligned with the edge. There may be portions of the CAD model that do not intersect any of the Keller cones defined by the transmitter and the existing wedges. There may also be blockage effects preventing the propagation of UTD rays to some regions. Conversely, some points on the CAD model may be illuminated by multiple UTD rays if there are valid UTD paths to those points through multiple wedges.

Once a UTD ray is launched from a wedge, it behaves in exactly the same way as other GO rays in the SBR algorithm. It gets reflected, transmitted, and absorbed at subsequent bounces, and either reaches the maximum number of bounces or escapes to infinity. However, the initial state of the UTD ray upon leaving the wedge depends heavily on the diffraction geometry.

Creeping Waves

Traditional SBR is limited by its reliance on straight-line geometrical optics (GO) ray tracing for the electromagnetic interaction with the scattering geometry. The Creeping Wave (CW) option is

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

an additive physics enhancement to SBR (and part of HFSS SBR+) that addresses this limitation. The CW scattering mechanism complements both the field-transport and the current radiation aspects of SBR.

In a similar way as SBR rays, CW rays paint currents on the scattering geometry that are then radiated to field observers and receiving antennas. The main difference is that creeping rays are traced on the scattering surface in geodesic paths rather than through space in straight-line paths. As with SBR's GO rays, the CW rays are never used to directly transport energy to field observers, and they comply with Fermat's principle of least time.

The currents painted by SBR on the CAD geometry via the physical optics (PO) approximation are abruptly truncated at the so-called shadow boundary, the mathematical border between directly illuminated and shadowed surface regions. This non-physical truncation of the surface current leads both to a false diffraction effect (which is particularly a problem in Radar Cross Section (RCS) predictions) and to missing progressive surface diffraction beyond the shadow boundary (which is particularly a problem in cosite antenna coupling predictions). The introduction of CW rays provides a current-based correction to the SBR-PO currents that addresses both problems by painting currents beyond the shadow boundary, smoothly blending SBR and CW currents across the shadow boundary.

Creeping wave propagation follows a minimum path distance rule (i.e., Fermat's least-time principle), resulting in a geodesic path on a curved surface. Creeping waves decay quickly in the shadow region as they propagate since they progressively shed energy to space along their direction of travel. Even if creeping rays can indefinitely propagate on an uninterrupted smooth surface, such as a sphere, the currents they paint on a surface are important only up to a certain travel distance. Ray Decay Threshold [dB] controls how far the creeping rays are allowed to propagate by terminating each ray once its power density decays below a threshold, defined as dB down from its power density at the shadow boundary.

The SBR+ Creeping Wave implementation is entirely consistent with its baseline SBR solution as a coherently additive correction component to the scattered field. That is, both can concurrently contribute to the scattered field, and in a manner that improves accuracy over the baseline SBR solution alone. In areas of the lit region where SBR first-bounce rays and CW rays would both paint currents, care is taken in the implementation to prevent double-counting of scattering mechanisms.

For RCS simulations, the shadow boundary is identified via SBR first-bounce rays from plane wave incidence. The CW rays are then launched into the shadow region from many points along the shadow boundary, initially advancing in the same the direction of the incident plane wave. CW currents are painted as the CW rays advance along geodesic paths that adhere to the curved surface. The formulation for first-bounce SBR currents is also modified so that surface currents across the shadow boundary transition smoothly between the lit and shadow regions. This has the important benefit of eliminating a spurious diffraction from the shadow boundary that arises in the SBR-only solution due to abrupt truncation of PO currents there.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For simulations involving transmitting antennas, CW analysis can only be performed from either 1) a near-field link to an HFSS source such as a FEBI or IE region, or 2) one of the current source based parametric antennas, which are available in the SBR+ solution type. In either case, the antenna must also have an infinite PEC ground plane, and part of the antenna must be located close (distance less than λ/2) to the nearest PEC or PEC-like surface.

The latter requirement is why only near-field sources are supported. With transmitting antennas, creeping waves launch from the closest point on the surface below each current source.

The creeping wave in SBR+ is limited to specific conditions:

- The creeping wave rays will only be launched and continue propagating on metal (PEC-like) surfaces
- The creeping wave will only advance on convex surfaces

The implementation does not require that the scattering geometry to be globally convex, such as a sphere or ellipsoid. However, the creeping-wave rays will terminate upon encountering local regions of surface concavity or saddle-shaped curvature (convex along one principal direction, concave along the other). This a requirement of the implemented CW formulation, but it can also be understood in terms of the of the minimum-path-distance rule. Where the surface becomes concave, the shortest path between the surface crests bounding the concavity no longer resides on the surface but, instead, is a straight line through space. When a local concavity is small and should be ignored, a relaxed value of Irregular Surface Tolerance allows CW rays to propagate over it.

Irregular Surface Tolerance is a single control for a collection of tolerances that determine creeping rays' sensitivities to local surface irregularities that theoretically should block their further propagation. Moving the slider toward Relaxed allows creeping rays to propagate through small surface irregularities that may arise through defects in the input geometry or due to limitations of curvature extraction accuracy. However, setting this tolerance too relaxed may allow creeping waves to pass through concave surfaces, saddle surfaces, or wedges, which are not supported by creeping wave physics. Therefore, users are strongly encouraged to perform a creeping-wave visual ray trace (VRT) to determine an acceptable setting for this tolerance, which will differ from model to model. The intention is that creeping rays only propagate over the convex portions of surfaces, and some user judgment is required in correctly setting the Irregular Surface Tolerance for each situation.

When a CW ray is shot, each CW ray stops propagating when it triggers a termination condition. The CW ray termination type for VRT with Creeping Wave may be found as render and filter parameters appearing in Properties of the parent Visual Ray Trace CW folder. Use the Render tab to set the color for each termination type. The configurable filters help in identifying issues for a specific termination type. Termination types include:

- **Natural Decay**: Decayed normally according to Ray Decay Threshold [dB]
- **Curvature Not Supported**: Encountered explicitly flat surface or where curvature could not be extracted

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Non-PEC Surface**: Encountered non-metallic surface (creeping wave formulation is limited to PEC-like surfaces)
- **Non-Smooth Crossing**: Encountered excessive slope discontinuity upon attempted crossing to next surface patch
- **Saddle Point**: Encountered section of surface with non-negligible saddle curvature (creeping wave formulation is limited to convex surfaces)
- **Double-Concave Point**: Encountered section of surface with non-negligible concave curvature (creeping wave formulation is limited to convex surfaces)
- **Unknown Creeping-Ray Error**: Creeping-ray algorithm encountered an unexpected condition
- **Geodesic Hit an Edge**: Encountered a surface edge (i.e., no continuing patch detected)
- **Geodesic Width Too Large**: Creeping ray footprint grew too wide
- **Unknown Geodesic Error**: Geodesic ray-trace algorithm encountered an unexpected condition
- **Natural Decay**: Decayed normally according to Ray Decay Threshold [dB]
- **Curvature Not Supported**: Encountered a surface where curvature could not be extracted
- **Non-Metallic Surface**: Encountered non-metallic surface (creeping wave formulation is limited to PEC-like surfaces)
- **Non-Smooth Crossing**: Encountered excessive slope discontinuity upon attempted crossing to next surface patch
- **Saddle Point**: Encountered section of surface with non-negligible saddle curvature (creeping wave formulation is limited to convex surfaces)
- **Double-Concave Point**: Encountered section of surface with non-negligible concave curvature (creeping wave formulation is limited to convex surfaces)
- **Unknown Creeping-Ray Error**: Creeping-ray algorithm encountered an unexpected condition
- **Geodesic Hit an Edge**: Encountered a surface edge (i.e., no continuing patch detected)
- **Geodesic Width Too Large**: Creeping ray footprint grew too wide
- **Geodesic Hit an Invalid Surface**: Encountered finding an incorrect neighboring surface.
- **Geodesic Blocked by Surface**: Creeping ray is blocked by another surface
- **Unknown Geodesic Error**: Geodesic ray-trace algorithm encountered an unexpected condition
- **Rough Surface**: Encountered section of surface where rough surface is defined

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GO Blockage Technical Note

In SBR+, rays are used to transport fields from the source to the scattering geometry (first bounce) and between different surfaces of the scattering geometry (multi-bounce). However, rays are not used to propagate incident or scattered fields to field observers, whether far-field, near-field, or receiving antennas. Rays are only used to paint equivalent currents on the scattering geometry. Once these currents are determined, SBR+ rays serve no further purpose and are discarded. The painted currents are then radiated in free space to all field observers to establish the scattered field contribution. The scattered field coherently combined with the incident field from the source, also propagated in free space, yields the total (i.e., measurable) field at all defined field observers.

Since propagating the source incident field and radiating the scattered fields to field observers is performed in free space, these operations of the SBR+ algorithm are not mediated by rays. This means that SBR+ applies a physical optics (PO) blockage model: total-field shadows cast by the scattering geometry relative to the field source emerge through destructive interference of the incident and scattered fields. PO blockage not only occurs in connection with the incident field from the source, but also within the scattered field itself. Scattered fields from the previous bounce are diminished behind a blocking surface via destructive interference when the reflecting or transmitting rays from the previous bounce hit that surface and the equivalent currents painted by the incident ray fields at the current bounce are radiated as part of the scattered field contribution.

Because the PO blockage model depends on destructive interference to recover the total-field shadow cast by an obstruction, the PO blockage model requires accurate currents over the extent of the obstruction. This can be hard to achieve in the Geometrical Optics (GO) approximation inherent in a ray propagation model. For example, the obstruction may not be well illuminated by direct rays from the field source or indirect multi-bounce rays after an earlier reflection or transmission. Even small inaccuracies in the painted equivalent currents can spoil the coherent cancellation mechanism needed to yield a robust shadow. An alternative is to invoke the SBR+ GO blockage model. In this model, the scattering contribution of a painted equivalent current to a field observer is included after passing a line-of-sight (LoS) check. This check is performed using the same geometric ray tracing engine that deploys the SBR+ rays. The GO blockage model can be more accurate than the default PO blockage formulation.

In the GO blockage model, it is ambiguous whether a field observer on the opposite side of a thin surface relative to the reflected/transmitted ray, but otherwise having line-of-sight path to the hit point, is blocked. The SBR+ GO blockage model provides an option to resolve this ambiguity. If Surface Self-Block is switched on (true), then any observation points in the opposite half-space of the hit surface with respect to the reflected/transmitted ray will be blocked. If Self-Block is switched off (false), then only a different surface (or different part of the same surface if the surface is curved) can cause blockage.

When GO blockage is used, blockage of the incident field from the field source by directly illuminated surfaces of the scattering geometry is computed by the PO cancellation method, but

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

scattered fields arising from interacting with the geometry are added subject to a line-of-sight blockage check to the field observers. When GO blockage is enabled, PTD, UTD and Creeping Wave radiation are also subject to a line-of-sight blockage check. It is recommended to use the GO blockage feature for SBR-only simulations, as the GO blockage effects for other diffraction mechanisms might not produce accurate results.

The GO blockage model has a non-trivial computational cost relative to the default PO blockage model. This is because many LoS test rays must be shot: one from each ray hit point to each field observer. However, when the GPU is enabled for the analysis, this additional cost is substantially mitigated since the ray tracing operations of these LoS blockage checks are performed very efficiently on the GPU.

Visual Ray Tracing (VRT)

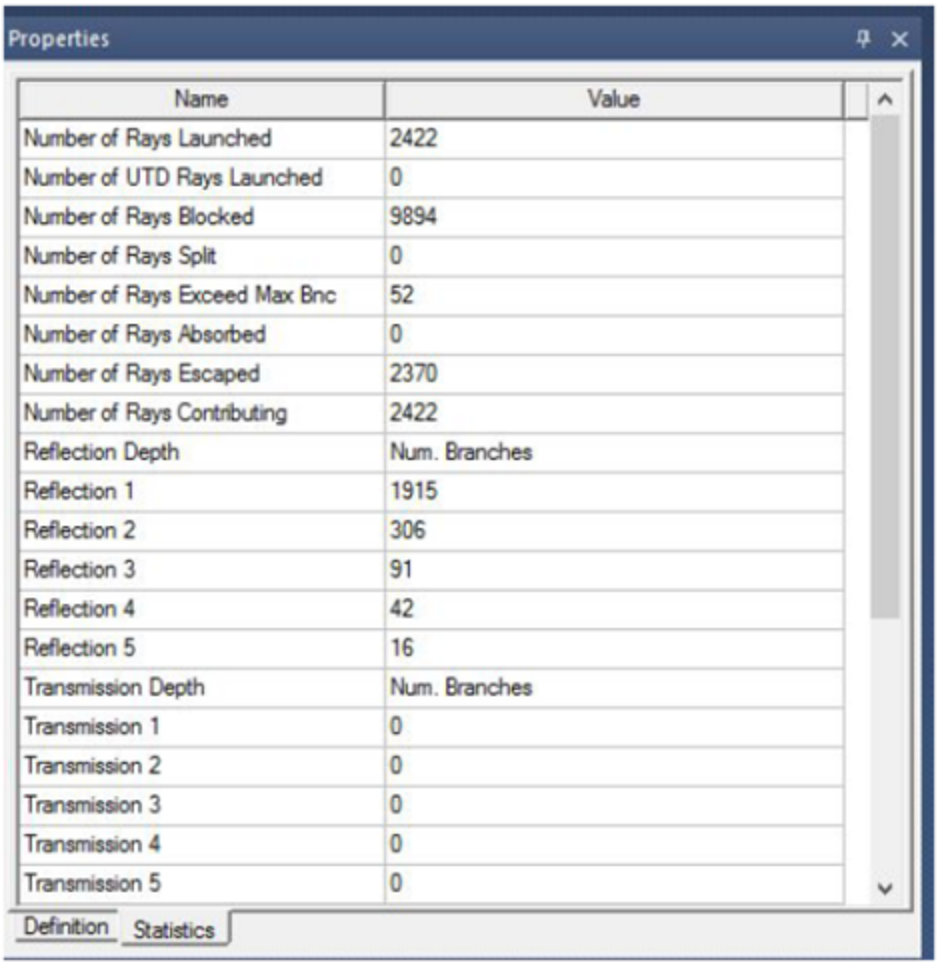
Visual Ray Trace (VRT) is a powerful feature available in HFSS for visualizing the ray geometry and interactions for an SBR+ simulation. Since SBR+ relies on “painting” physical optics (PO) currents on the geometry, VRT can provide valuable insight for analyzing these PO contributions from the direct illumination first-bounce rays and the multi-bounce GO rays. VRT can selectively display:

- ray hit (bounce) points on the scattering geometry,
- ray paths (lines) between each bounce, and
- triangular ray “footprints” that are used to integrate the PO currents.

The VRT feature includes a number of rendering controls and options to filter whole ray tracks or individual ray bounces, visualize ray scattering directions, and color ray point/lines/footprints according to different criteria.

When SBR rays are shot in VRT, SBR ray statistics are collated and displayed within the Statistics tab of the Properties window for the SBR Plot node of the Visual Ray Trace SBR plot folder:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

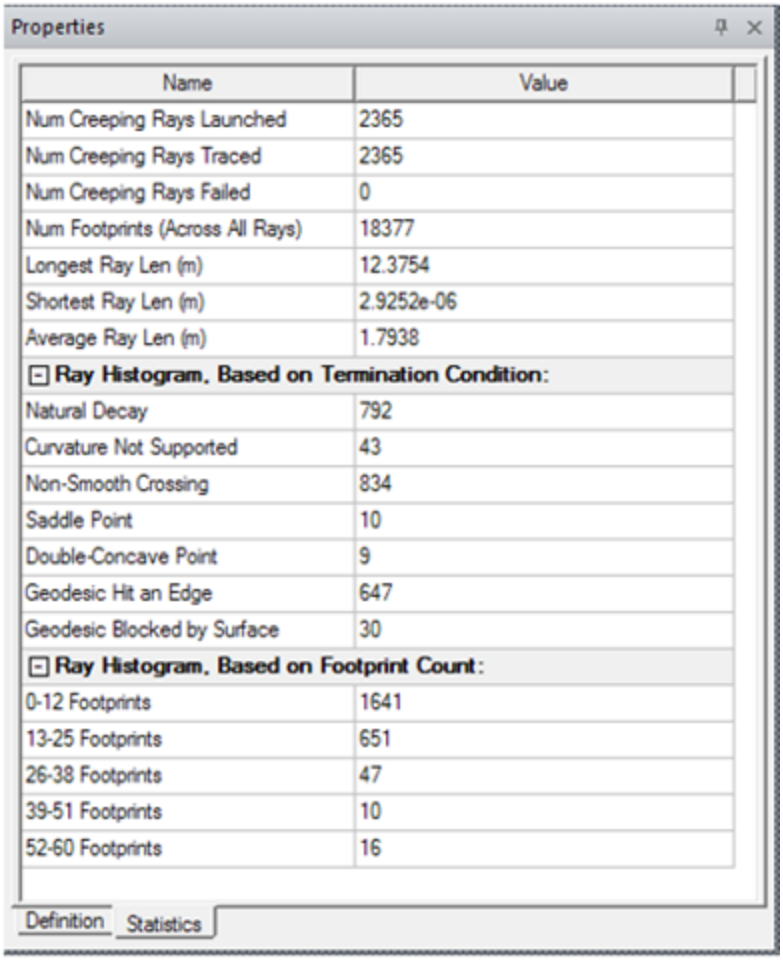


- Number of Rays Launched: Number of GO (non-UTD) rays illuminating the CAD model from the antenna or source, not including those blocked by other surfaces
- Number of UTD Rays Launched: Number of UTD rays illuminating the CAD model, not including those blocked by other surfaces
- Number of Rays Blocked: Number of potentially launched rays whose line-of-sight path was blocked by part of the CAD model
- Number of Rays Split: Number of rays that hit a penetrable surface (e.g., 2-sided Layered Impedance Boundary). These rays are split to form both a transmitted and reflected ray
- Number of Rays Exceed Max Bnc: Number of rays that would have continued propagating through the model, but were terminated because they reached the Max Number of Bounces limit set in the plot Definition
- Number of Rays Absorbed: Number of rays that hit a surface with a Perfect Absorber boundary. Ray track branches terminate on such surfaces.
- Number of Rays Escaped: Number of ray track branches whose final ray escapes the scene

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Number of Rays Contributing: Number of ray track branches that would contribute to the scattered field result in a corresponding simulation
- Reflection <N> under Reflection Depth: Number of branches that terminate/escape after <N> reflections
- Transmission <N> under Transmission Depth: Number of branches that terminate/escape after <N> transmissions

When Creeping Wave (CW) rays are shot in VRT, CW ray statistics are collated and similarly displayed:



- Num Creeping Rays Launched: Total number of creeping rays launched
- Num Creeping Ray Traced: Number of launched creeping rays that were successfully traced
- Num Creeping Ray Failed: Number of launched creeping rays that failed to trace
- Num Footprints (Across All Rays): Total number of ray footprints generated across all traced creeping rays

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Longest Ray Len (m): Length of longest creeping ray (in meters)
- Shortest Ray Len (m): Length of shortest creeping ray (in meters)
- Average Ray Len (m): Average length among all creeping rays (in meters)
- Ray Histogram Based on Termination Condition: Breakout of number of rays terminated due to each termination type (e.g., Natural Decay, Non-smooth Crossing)
- Ray Histogram Based on Footprint Count: Number of creeping rays with a footprint count residing within a particular min/max range of footprint counts

Current Conformance and Array Conformance in SBR+

Activate Current Conformance to shift the heights of the current sources to better accommodate to the underlying platform structure with which they will be interacting. This capability is primarily intended for antennas requiring a ground plane and that will be placed on a flat or slightly curved metallic section of the platform that will provide this grounding function. It is typically used to make slight adjustments that will improve simulation accuracy and speed. There are a couple of reasons why one should consider using the adjust/conform antenna feature.

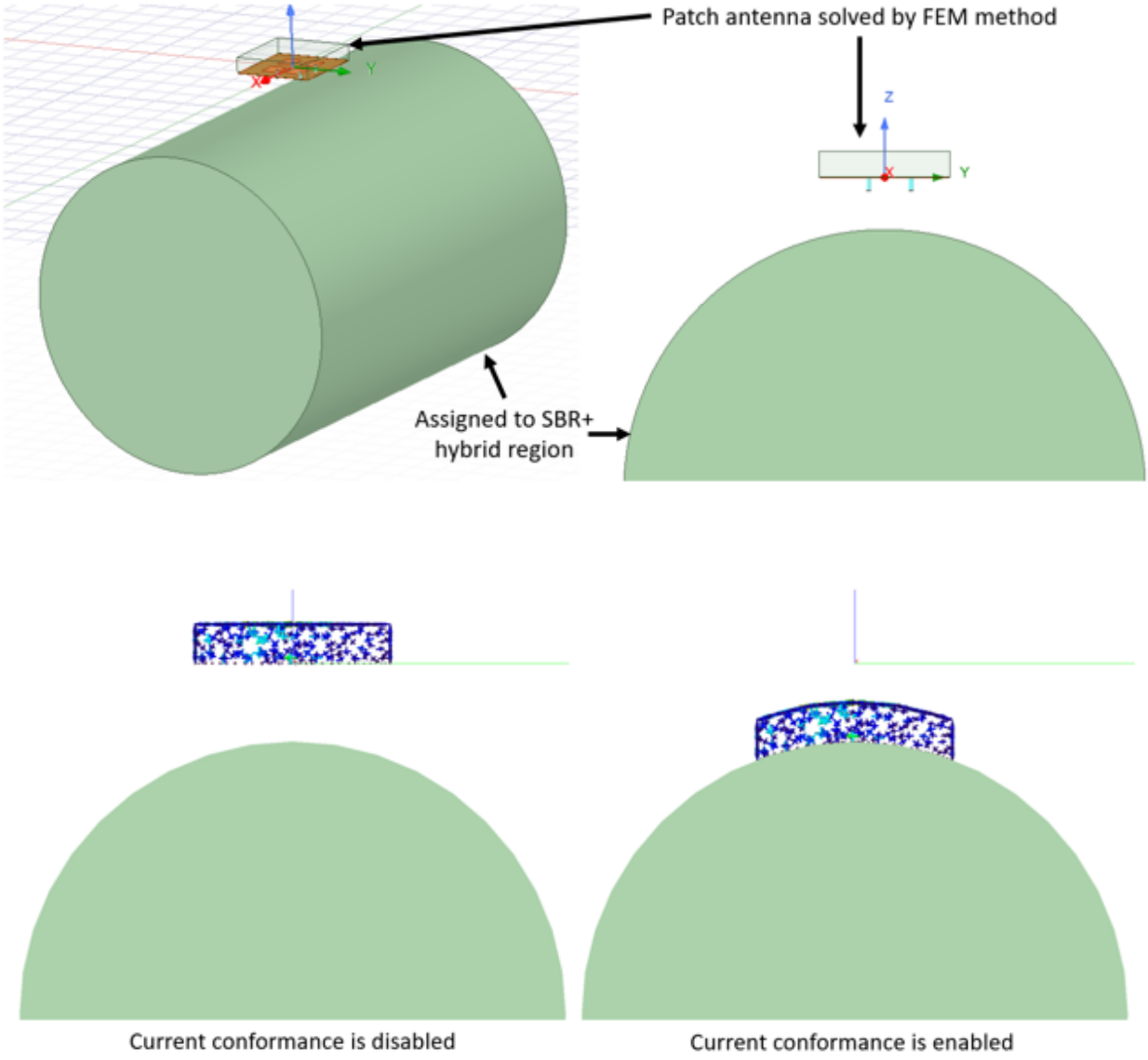
- The unadjusted sources may otherwise be too close or possibly even slightly underneath the platform placement surface. In general, placing sources that are conceptually intended to be directly on the surface closer than λ/20 above the surface harms run time with little or no accuracy benefit, as in the case of horizontal magnetic sources representing a slot cut in the placement surface.
- The platform placement surface is slightly curved, the shape of the actual antenna will conform to this curvature, yet the antenna might not exhibit this intended curvature (it's flat)

These situations are common, so the Current Conformance feature is an important capability. When the antenna structure is to be placed directly on the platform, such that it touches the platform, a small source adjustment will often be beneficial.

Example of Current Conformance

The following driven terminal hybrid design has a patch antenna surrounded by a FEBI region with a Perfect E infinite ground plane, and a PEC cylinder assigned to an SBR+ hybrid region. The fields on the FEBI surface are converted to current sources in the SBR+ simulation. The patch antenna is placed too high relative to the cylinder, as evidenced by the gap between its ground plane and the cylinder. If Current conformance is enabled, the current sources will be shifted parallel to the antenna’s z-axis toward the placement surface, such that their heights above the curved PEC placement surface will match their original heights above the ground plane .

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Conformance in SBR+ Parametric Array

Parametric array conformance is processed in two stages, array element conformance, and current conformance if the element antenna is based on current sources.

Array conformance (Conform to Geometry) for Parametric Array

Array conformance may be selected from **Conform to Geometry** which is found in first page of **Create Parametric Array: General** dialog. Select **Perpendicular to Plane** or **Along Normal** to shift the heights of the elements to better accommodate the underlying platform structure with which they will be interacting. This capability is primarily intended to conform an antenna array

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

that will be placed on a flat or slightly curved metallic section of the platform. See Create and Use Parametric Array in SBR+.

Element Offset specifies the offset as a 3-D Cartesian vector. The elements will be shifted by the fixed offset specified in the local coordinate system of the parametric array. The z-directed offset is also maintained after conformance above the underlying platform structure when **Along Normal** or **Perpendicular to Plane** are chosen. Pay attention to sources of element antenna that are not too close to the platform surface. Placing sources directly on the platform surface closer than λ/20 above the surface harms run time with little or no accuracy benefit. Use **Element Offset** to shift the heights of the elements such that the sources are λ/20 away from platform, in particular, when the element antenna is far-field source. On the other hand, **Current Conformance** is recommended when element antenna is current source or Near-field link which is explained in later section.

The array conformance is applied after the **Element Offset** is applied to each array element in antenna local coordinates. There are three options:

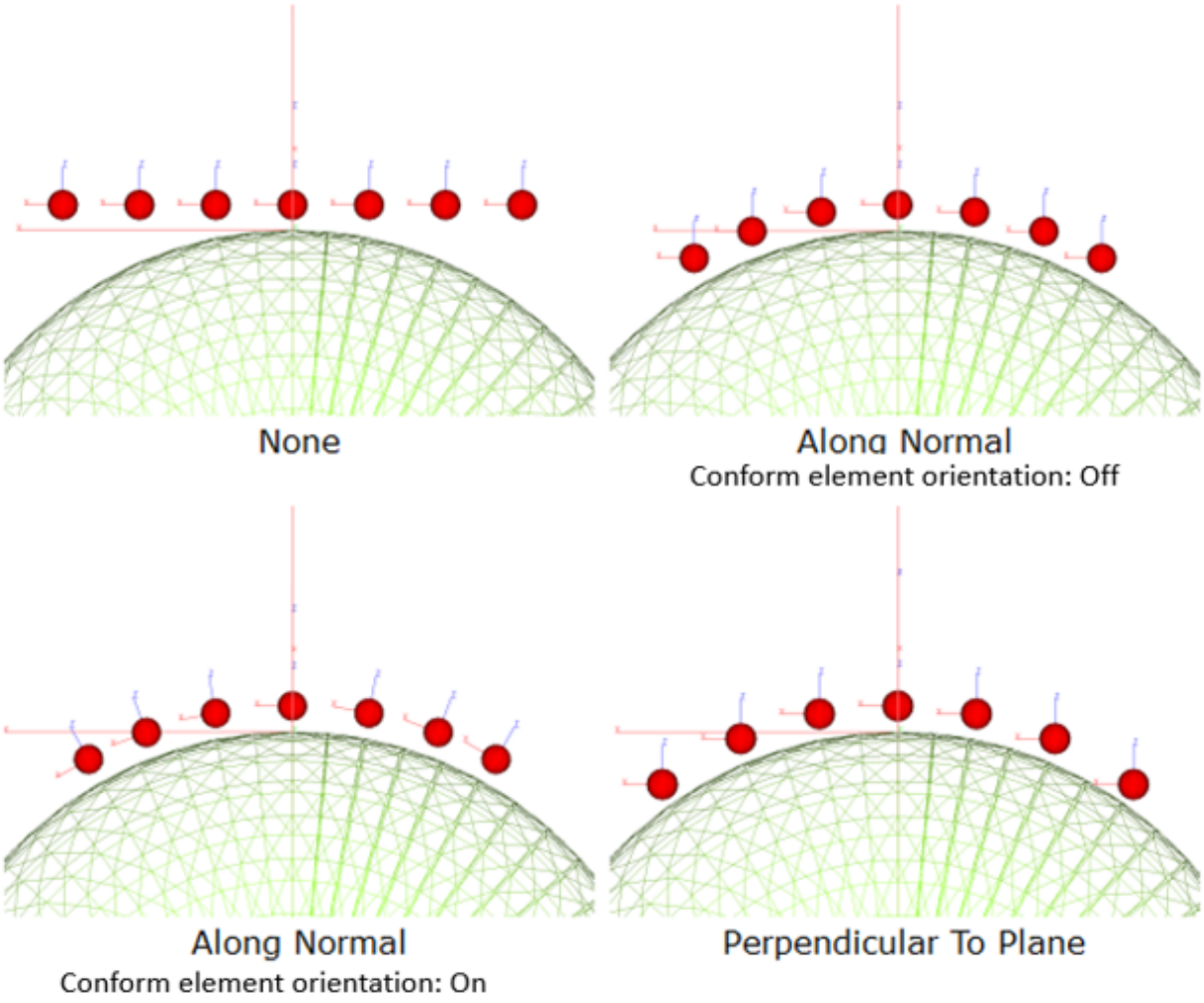
None - No further adjustment after **Element Offset** is applied.

Along Normal: Element positions are further modified according to the position and perpendicular of the platform surface point *Pn* closest to each array element. If **Conform Element Orientation** is also switched on, then the element orientations are similarly modified according to the surface perpendicular at *Pn*.

Perpendicular to Plane: Element positions are further modified according to the position of the platform surface point *Pu* directly under each array element.

The behavior of these three options is summarized in the illustration below.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The concept behind array conformance is to shift the array element positions so that their height with respect to the array platform on the XY plane is maintained with respect to the SBR CAD platform placement surface. If the platform placement surface is uniformly flat over the extents of the array elements, the elements will be shifted downward to maintain their heights above flat platform surface. If the platform curves away, then the elements will be shifted downward to maintain their heights above the actual curved surface.

One intended use of array conformance is to simplify the modeling task for antennas conforming to the platform surface. Many antennas are not engineered with a particular platform in mind, and their shape is fixed independent of the shape of the placement surface. However, there are applications requiring the antenna to be shaped according to the intended placement surface. For example, this occurs with conformal arrays placed on a fuselage, wing, or some other curved surface.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In these situations, one would ideally capture this curvature in the geometry definition of the array antenna. However, it may not be convenient to do so, as it entails greater effort in developing the model.

The difference between Array Conformance **Along Normal** and **Perpendicular to Plane** is the determination of the surface points *Pn* and *Pu* relative to which the array elements are repositioned, and also the direction in which these adjustments are applied.

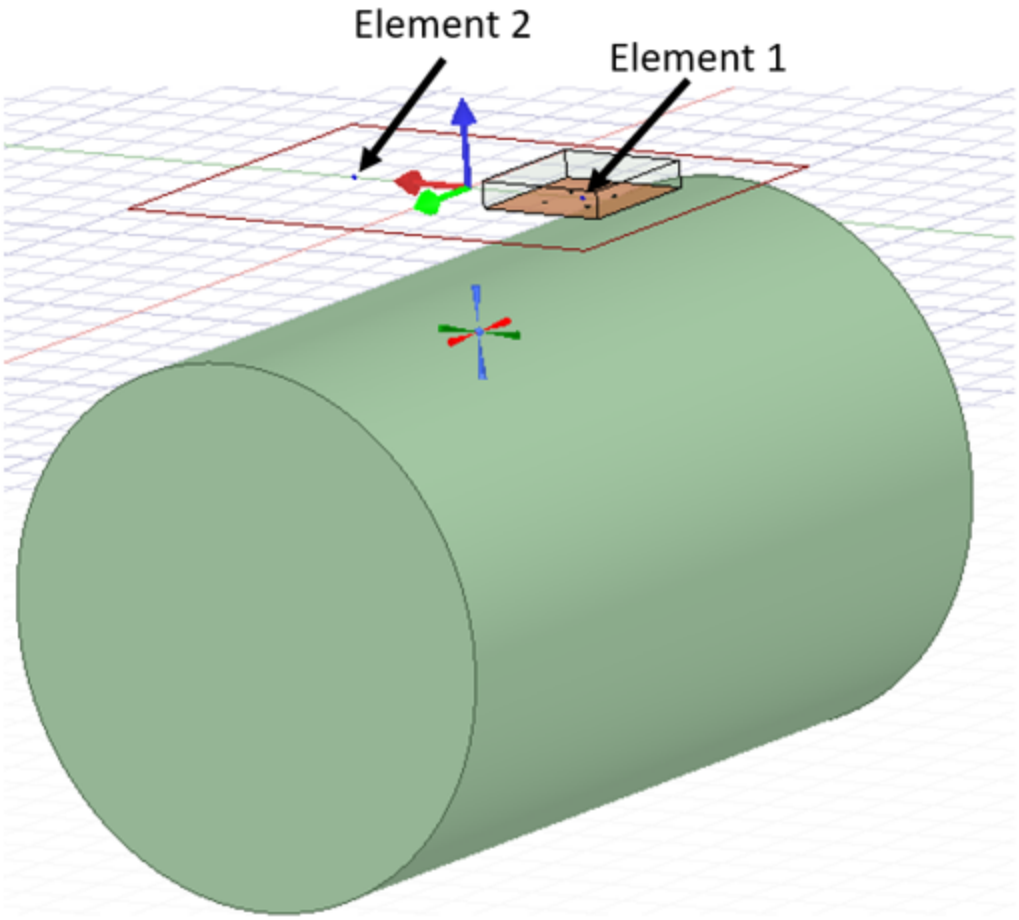
If Array Conformance is **Along Normal**, then *Pn* is the surface point nearest the n^{th} array element (after application of **Element Offset**). The array element is then repositioned at distance *h* from *Pn* along the surface perpendicular at that point.

If **Conform Element Orientation** is switched on (True), then each array element is also rotated to maintain the same orientation relative to the surface tangent plane at each *Pn* as it had relative to the Reference XY-Plane of the array coordinate. Specifically, if the rotation matrix *R* would rotate the Reference Plane to the tangent plane at *Pn*, then the same rotation matrix *R* is applied to the orientation of the n^{th} array element. It is only available when **Conform to Geometry** is set to **Along Normal**.

If Array conformance is **Perpendicular to Plane**, the surface point *Pu* is determined by extending a line from each array element (after application of **Element Offset**) toward the Reference XY-Plane and perpendicular to the same. The first platform surface point intersecting this line is designated as the surface point *Pu* "under" the array element. The array element is then repositioned along this line to maintain the same distance relative to *Pu* as its original height *h*. For this adjustment option, the array elements are never re-oriented; only their positions are shifted.

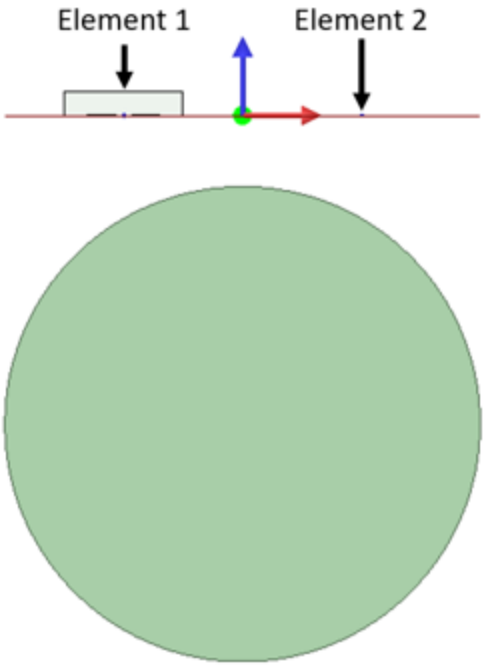
An example of two element linear array of a patch antenna is shown below. Each element is assigned the same FEBI box source. The first of these two elements is indicated by the element-positioned FEBI box. The second element is depicted as a simple point, but it is important to understand that it also has the same FEBI box centered on it during simulation. In general, with Parametric Array rendering in Desktop, the FEBI box is only drawn for the first element of a parametric ray. This is done to reduce clutter in the visualization and keep the focus on the arrangement of array elements.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

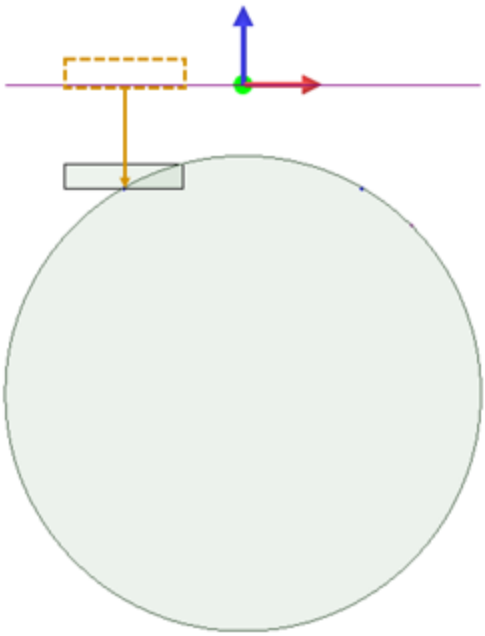


When the parametric array is conformed with Array Conformance “None”, all elements are located on XY-plane of the antenna coordinate. On the other hand, when **Along Normal** or **Perpendicular to Plane** are selected, elements and element antennas are moved toward the underlying platform structure. Also, when **Along Normal** and **Conform Element Orientation** are selected, elements are rotated such that the rotated to maintain the same orientation relative to the surface tangent plane as it had relative to the Reference XY-Plane of the array coordinate.

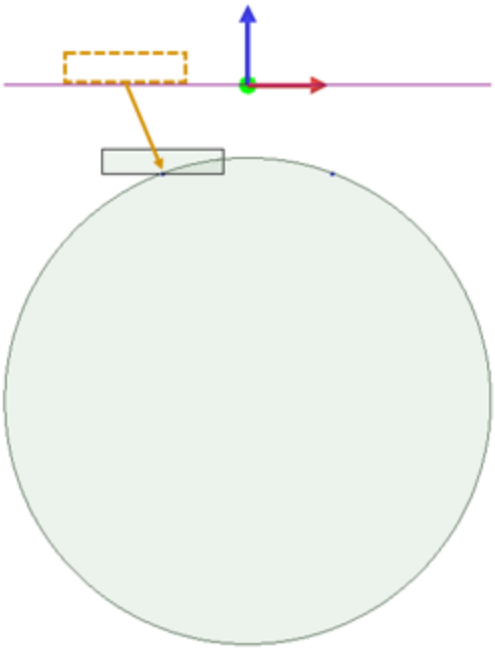
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



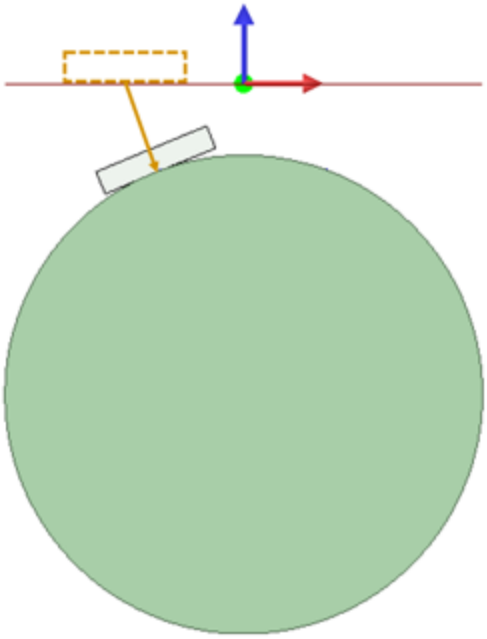
Array Conformance: None



Array Conformance: Perpendicular to Plane



Array Conformance: Along Normal

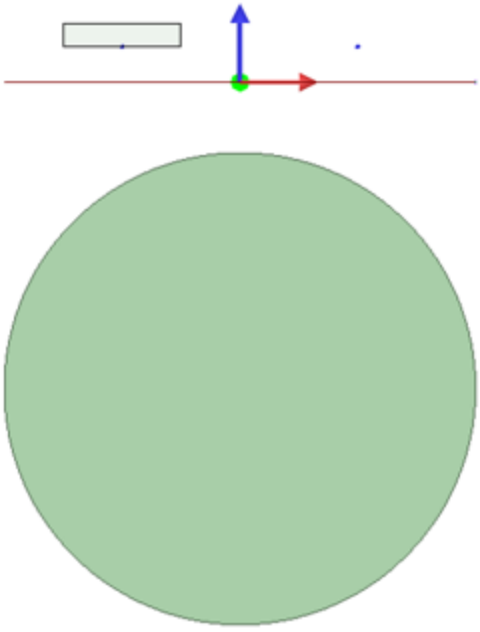


Array Conformance: Along Normal with Conform Element Orientation

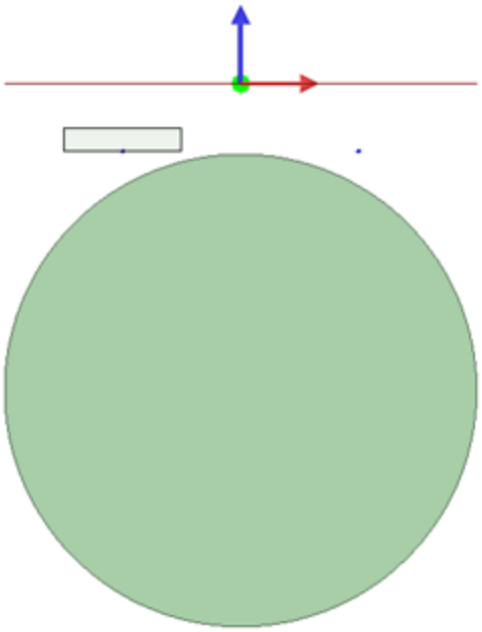
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Elements of the array might intersect the placement surface after array conformance. One way to avoid this problem, is to adjust the element offset Z used as the height of the elements above the underlying platform structure. However, this should only be done if it makes physical sense given the actual array being simulated. The alternative resolution, and often the superior one, is to use **Current Conformance**, which is explained in next section.

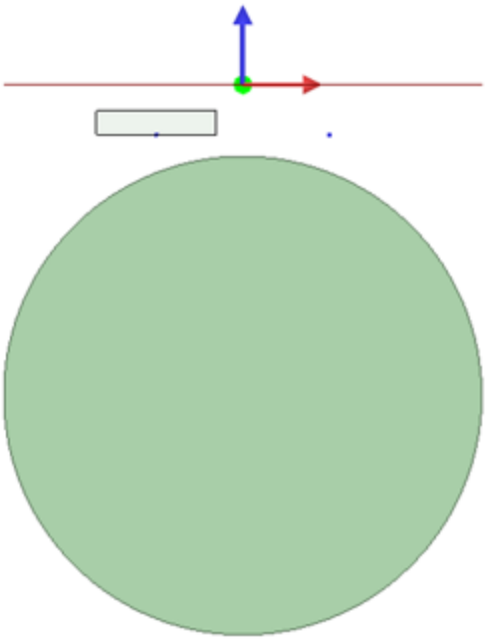
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



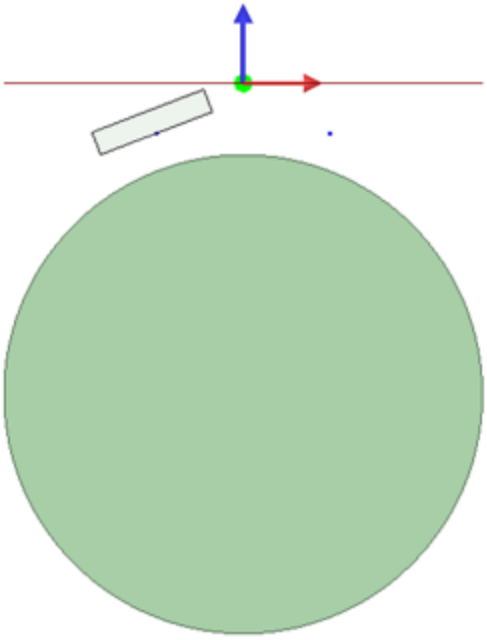
Array Conformance: None



Array Conformance: Perpendicular to Plane



Array Conformance: Along Normal



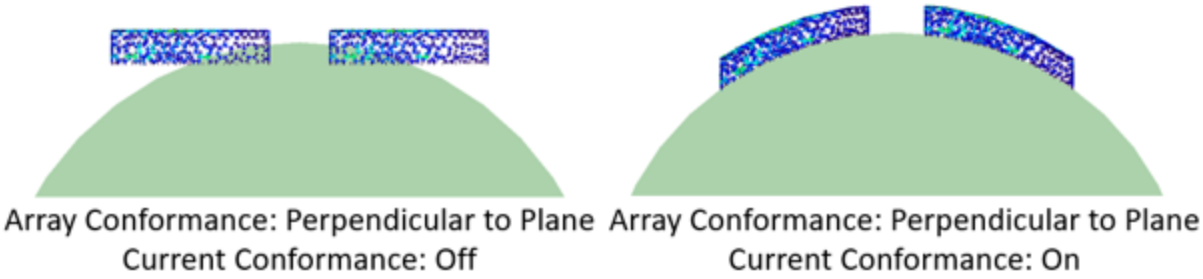
Array Conformance: Along Normal with Conform Element Orientation

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

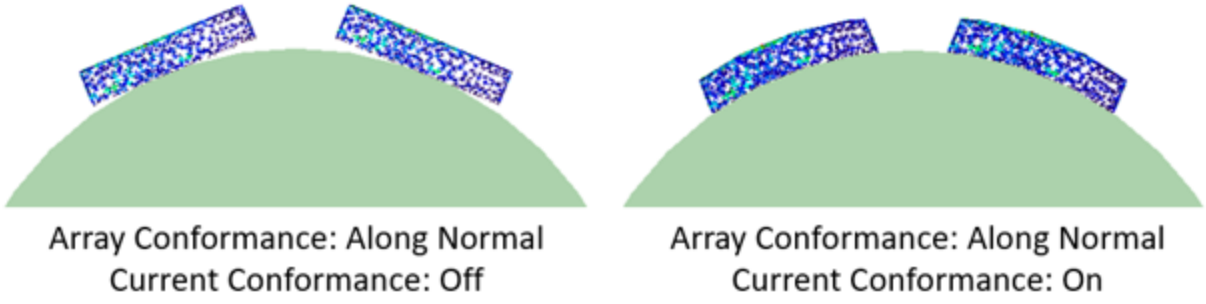
Current Conformance for Parametric Array

In addition to Array conformance, Current Conformance is recommended for Parametric Arrays whose elements are parametric antennas with current sources or Near-Field linked antenna.

In previous example, Array Conformance **Perpendicular to Plane** without offset resulted in array elements intersecting the underlying platform structure. When current conformance is off, some current sources intersect the platform structure or end up under the platform structure. When current conformance is enabled, the element current sources are conformed to the platform structure.



In general, it is recommended to enable current conformance, even in cases where no intersection occurs between the array elements and the platform. This feature helps in detecting if any current source results in being positioned below the platform after conformance, in which case the simulation is aborted with an error message.

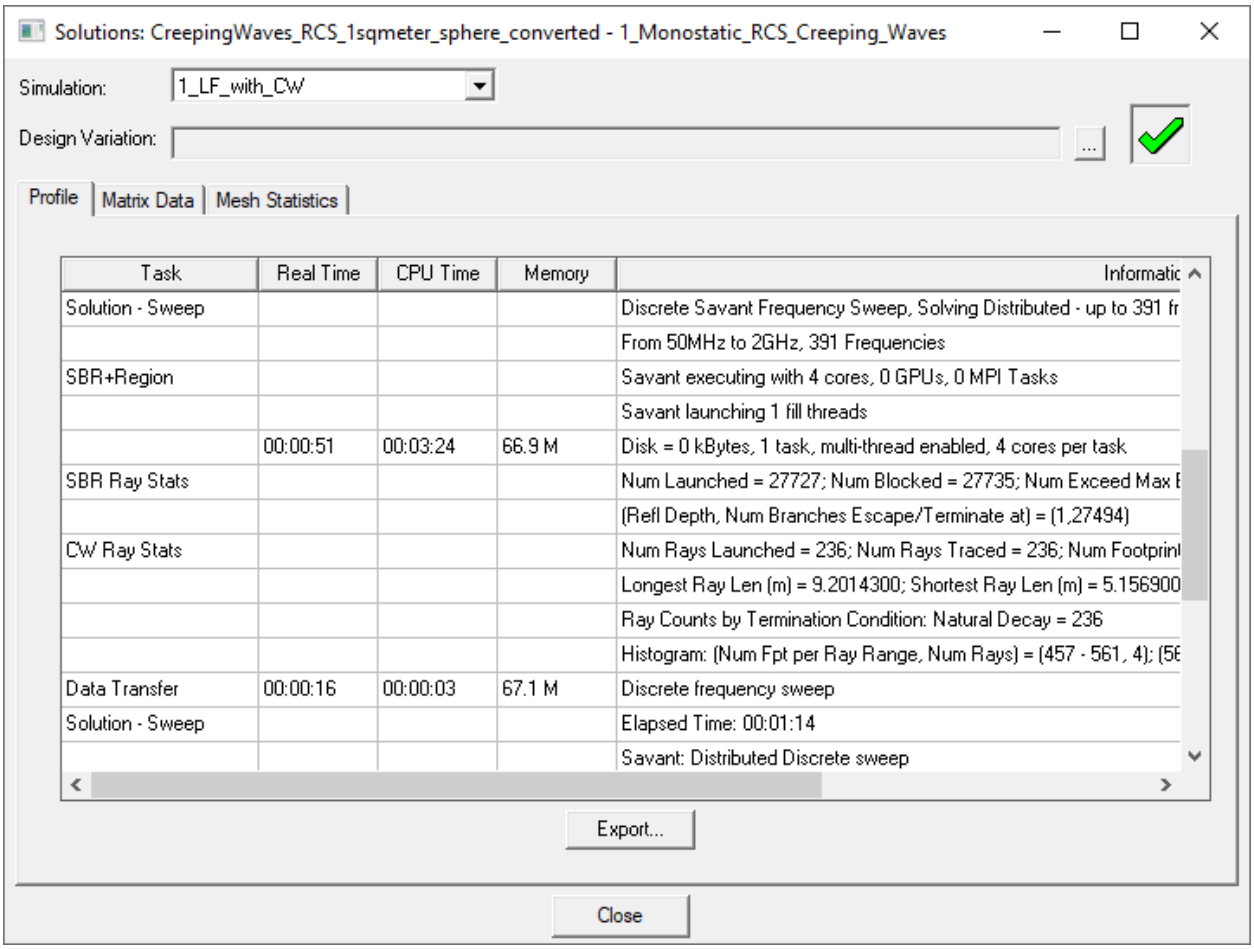


Ray Statistics for SBR+ Analysis

Ray statistics summarize the ray tracing performed during the SBR+ simulation. Ray statistics are primarily useful for understanding the role of multi-bounce, UTD, creeping-wave, and other geometrical-optics mechanisms in the simulation. They are also useful for understanding the run time of SBR+ simulations, especially in the context of penetrable surfaces that lead to many split rays.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ray statistics are found in the profile after an SBR+ run. Ray statistics are reported cumulatively for all Tx antennas in the simulation. Ray statistics appear under two different task names, with “SBR Ray Stats” reserved for SBR+ and "CW Ray Stats” reserved for Creeping Wave.



In the following, SBR and creeping-wave ray statistics are explained separately in more detail.

SBR Ray Stats

An SBR ray track is a sequence of connected ray segments (bounces). When a ray hits a surface, a reflected ray is generated, extending the track. If the surface is penetrable on a two-sided Layered Impedance Boundary or a face on dielectric solid object, a transmitted ray is also generated. This is called a split, leading to a new branch of the original ray track. On models with no penetrable surfaces, each ray track has only a single branch. On models with penetrable surfaces, each track may consist of 1 - N branches. If UTD rays are enabled for the simulation, the number of UTD rays are also listed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Num Launched:** Number of (non-UTD) rays illuminating the CAD model from the antenna, not including those blocked by other surfaces
- **Num Launched (UTD):** Number of UTD rays illuminating the CAD model, not including those blocked by other surfaces
- **Num Blocked:** Number of potentially launched rays whose line-of-sight path was blocked by part of the CAD model
- **Num Split:** Number of rays that hit a penetrable surface. These rays are split to form both a transmitted and reflected ray
- **Num Exceed Max Bnc:** Number of rays that would have continued propagating through the model, but were terminated because they reached the Max Number of Bounces limit set in the project
- **Num Absorbed:** Number of rays that hit a surface coated with a perfect absorber. Ray track branches terminate on such surfaces.
- **Num Escaped:** Number of ray track branches whose final ray escapes the scene
- **Num Contributed:** Number of ray track branches that contributed to the scattered field
- **Num Invalid Interface:** Number of rays that encounter volume interface inconsistency.

SBR ray histograms on the SBR ray track are also reported. The histograms are reported as pairs of number, with the first number being the Reflection or Transmission Depth followed by the number of branches that are terminated or escaped at that depth. For example,

```
(Refl Depth, Num Branches Escape/Terminate at) = (1,18353)
```

means 18353 branches are terminated or escaped after 1 reflection.

CW Ray Stats

If Creeping Rays are enabled, creeping rays are launched and traced on curved surfaces. Like SBR rays, creeping rays have footprints that cover finite areas on the curved surface. If the creeping rays are launched densely enough, they collectively cover the surface on which they propagate without overlap. The creeping-wave ray statistics provide information on the number and distribution of footprints across all creeping rays. More details can be found in Creeping Wave.

Typically, all these rays are traced successfully, and they contribute to the scattered field. If a ray encounters a critical error because of an unhandled contingency in the formulation, it gets discarded and its contribution to the scattered field is not computed. The number of these rays is given in the Num Failed.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Num Launched:** Total number of creeping rays launched during simulation
- **Num Traced:** Number of creeping rays successfully traced and radiated
- **Num Failed:** Number of creeping rays failed and discarded without being radiated
- **Num Footprints (Across All Rays):** Total number of footprints across all creeping rays
- **Longest Ray Len (m):** Length of longest creeping ray (in meters)
- **Shortest Ray Len (m):** Length of shortest creeping ray (in meters)
- **Average Ray Len (m):** Average length among all creeping rays (in meters)

Creeping-wave ray statistics also provide a break-down of the ray counts with respect to how the rays are terminated.

- **Natural Decay:** Decayed normally according to Ray Decay Threshold [dB]
- **Curvature Not Supported:** Encountered a surface where curvature could not be extracted
- **Non-Metallic Surface:** Encountered non-metallic surface (creeping wave formulation is limited to PEC-like surfaces)
- **Non-Smooth Crossing:** Encountered excessive slope discontinuity upon attempted crossing to next surface patch
- **Saddle Point:** Encountered section of surface with non-negligible saddle curvature (creeping wave formulation is limited to convex surfaces)
- **Double-Concave Point:** Encountered section of surface with non-negligible concave curvature (creeping wave formulation is limited to convex surfaces)
- **Unknown Creeping-Ray Error:** Creeping-ray algorithm encountered an unexpected condition
- **Geodesic Hit an Edge:** Encountered a surface edge (i.e., no continuing patch detected)
- **Geodesic Width Too Large:** Creeping ray footprint grew too wide
- **Geodesic Hit an Invalid Surface:** Encountered finding an incorrect neighboring surface.
- **Geodesic Blocked by Surface:** Creeping ray is blocked by another surface
- **Unknown Geodesic Error:** Geodesic ray-trace algorithm encountered an unexpected condition
- **Rough Surface:** Encountered section of surface where rough surface is defined

Occasionally creeping rays cannot be launched because of problems related to the source.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Far-Field Antenna:** Far-field antennas do not support CW
- **Antenna Too Far From Platform:** The entire antenna is too far from the platform
- **No Ground Plane:** The antenna has no ground plane
- **No Closest Point:** No closest point could be found on the surface
- **Closest Point Failure:** The vector connecting the closest point to the source is not perpendicular to the surface
- **Current Source Too Close to Platform:** The current source is too close to the platform
- **Current Source between Original and Fitted Surface:** When an original (flat) surface was fitted with a curved surface based on its curvature information, the current source ended up between these two surfaces.
- **Curvature Not Supported:** The closest point is on a surface that does not have curvature information
- **Closest Point Not Metal:** The closest point is not located on a metallic surface
- **Closest Point On Concave Surface:** The closest point is on a surface with concave curvature
- **Point On Saddle Surface:** The closest point is on a surface with saddle curvature
- **Closest Point On Rough Surface:** The closest point is located on a surface that has a surface roughness model defined

Similarly to SBR ray statistics, CW ray statistics include a footprint histogram,

```
(Num Fpt per Ray Range, Num Rays) = (39 - 51, 285)
```

means 285 creeping rays are terminated in a range between 39 and 51 footprint counts.

Integral Equation Method Used in for IE Regions

The most commonly used electromagnetic integral equation is the Electric Field Integral Equation, or more commonly referred to by its acronym, EFIE. Consider a perfect conductor that is illuminated by a plane wave or any other source. On the surface of a perfect conductor the total tangential electric field must be zero. The total electric field comprises the incident electric field and the scattered electric field; hence, on the conducting surface.

$$\hat{n} \times (\vec{E}^{inc} + \vec{E}^{scat}) = 0 \quad (1)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The above relation only specifies the scattered field on the surface of the conductor. To fully understand the system, we need to be able to calculate the scattered field everywhere; one way to do this is to calculate the induced current which generates the scattered electric field.

The field generated by a current is

$$\bar{E}(\vec{r}) = \frac{1}{j\omega\varepsilon}\left(\nabla\nabla\bullet + k^2\right)\int J(\vec{r}')\,G\left(\left|\vec{r}-\vec{r}'\right|\right)d\vec{r}' \quad (2)$$

where G is the Green's function defined as

$$G(r) = \frac{1}{4\pi}\frac{e^{-jkr}}{r} \quad (3)$$

and J is the surface current density. Combining the relationship between current and scattered field with the boundary condition yields the EFIE

$$G(r) = \frac{1}{4\pi}\frac{e^{-jkr}}{r} \quad (4)$$

$$\hat{n}\times\vec{E}^{inc} = \hat{n}\times\left[\frac{1}{j\omega\varepsilon}(\nabla\nabla\bullet + k^2)\int J(\vec{r}')G(|\vec{r}-\vec{r}'|)d\vec{r}'\right] \quad (5)$$

The unknown current density is within the integral operator; these types of equations are known as integral equations. Also note the integral is a convolution. Electromagnetic integral equations are convolutions, and this allows us to make an analogy to system theory. The Green's function is the transfer function, or the impulse response of the system, and the incident electric field is the output. Our task is to find the input, or the current density, from the output knowing the transfer function.

Solving the EFIE is typically done through a finite element technique. We divide the surface of the conductor into many, many triangles, and on each triangle we assume a current distribution. This converts EFIE into a matrix equation, and in the electromagnetics literature this technique is called the method of moments (MoM).

The EFIE is just one of many different integral equations used in electromagnetics. We can vary the boundary conditions we enforce, or the method we use to specify the field everywhere. For example, if on the surface of the conductor, we enforce the continuity of the magnetic field, following a similar procedure yields the magnetic field integral equation (HFIE). All of them are similar in form to the EFIE and are solved in a similar manner. We use EFIE and CFIE formulations in our IE solver for standalone IE region applications. We don't use MFIE alone. However, CFIE is a linear combination of EFIE and MFIE. We use EFIE on open surfaces and small closed surfaces, and use CFIE on large closed surfaces.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

References:

1. Roger Harrington, *Time Harmonic Electromagnetic Fields*, McGraw-Hill, New York, NY 1961.
2. Andrew Peterson, Scott Ray, Raj Mittra, *Computational Methods for Electromagnetics*, IEEE Press, New York, NY 1998.

S-Parameters

One of the most important features of HFSS is the capability to categorize an electromagnetic structure in a compact manner using the concept of S-parameters. S-parameters are available for projects that contain ports and this document focuses on the case when the S-parameters are related to the eigen modes of the ports. The use of S-parameters for terminals are described in Modes to Terminals Conversion.

At each port, the modal representations of the electric and magnetic fields assuming *K* modes are:

$$\mathbf{E} = \sum_{m=1}^{K} (a_m + b_m)\mathbf{e}_m \quad (1)$$

$$\mathbf{H} = \sum_{m=1}^{K} (a_m - b_m)\mathbf{h}_m \quad (2)$$

where a_m and b_m are unitless complex amplitudes of the incident and reflected modal fields, respectively. Given a particular electromagnetic structure, it can be categorized in terms of the incident and reflected/transmitted modal amplitudes using the *NxN* generalized S-matrix, *S*:

$$\mathbf{b} = S\mathbf{a} \quad (3)$$

where **a** and **b** are unitless complex modal coefficient vectors. The size of these vectors, *N*, is the total number of modes obtained by adding up the number of the modes on all ports.

An entry of the S-matrix, S_{ij}, specifies the multiplication factor to be applied to the incident modal amplitude a_j to get the reflected/transmitted modal amplitude b_i given that all the other incident modes are turned off. This can be described mathematically as:

$$S_{ij} = \left.\frac{b_i}{a_j}\right|_{a_k = 0,\ k \neq j} \quad (4)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HFSS automatically provides an S-matrix when a project contains ports. The following links contain information on various topics related to S-matrices.

- Post Processed S-Matrices
- Passivity
- Alternative Matrix Characterizations

Post Processed S-Matrices

HFSS automatically computes the generalized S-matrix during the solve time when a project contains ports. The S-matrix can be modified after solution time by applying post processing transformations. The following post processing transformations are supported by HFSS:

- Renormalized S-Matrices in HFSS
- De-embedded S-Matrices in HFSS

Renormalized S-Matrices in HFSS

Before a structure's generalized S-matrix can be used in a high frequency circuit simulator to compute the reflection and transmission of signals, it must be normalized to the appropriate impedance. For example, if a generalized S-matrix has been normalized to 50 ohms, it can be used to compute reflection and transmission directly from signals that are normalized to 50 ohms.

To renormalize a generalized S-matrix to a specific impedance, HFSS first calculates a unique impedance matrix Z, associated with the structure defined as follows:

$$Z = \sqrt{Z_0}\,(I - S)^{-1}\left(I + S\right)\sqrt{Z_0} \qquad (1)$$

where

- S is the n x n generalized S-matrix.
- I is an n x n identity matrix.
- Z0is a diagonal matrix having the characteristic impedance (*Z0*) of each port as a diagonal value.

The renormalized S-matrix is then calculated from the unique impedance matrix using this relationship:

$$S_\Omega = \sqrt{Y_\Omega}\left(Z - Z_\Omega\right)\left(Z + Z_\Omega\right)^{-1}\sqrt{Z_\Omega} \qquad (2)$$

where

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Z is the structure's unique impedance matrix.
- $Z\Omega$ and $Y\Omega$ are diagonal matrices with the desired impedance and admittance as diagonal values. For example, if the matrix is being renormalized to 50 ohms, then $Z\Omega$ would have diagonal values of 50.

Visualize the generalized S-matrix as an S-matrix that has been renormalized to the characteristic impedances of the structure. Therefore, if a diagonal matrix containing the characteristic impedances of the structure is used as $Z\Omega$ in the above equation, the result would be the generalized S-matrix again.

HFSS needs to calculate the characteristic impedance of each port in order to compute a renormalized S-matrix.

The S-matrices initially calculated by HFSS are generalized S-matrices that have been normalized to the impedances of each port; however, you can compute S-matrices that are normalized to specific impedances, such as 50 ohms.

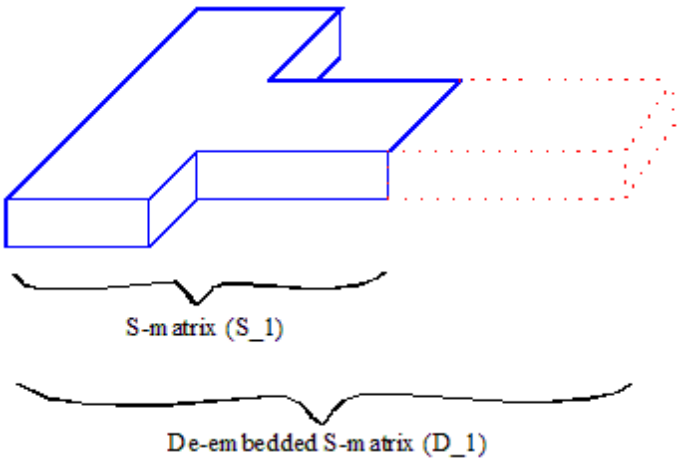
To convert a generalized modal S-matrix to a renormalized modal S-matrix, HFSS first needs to compute the characteristic impedance at each port. There are several ways to compute characteristic impedance. Two methods — the Zpv and Zvi methods — require an impedance, or *integration*, line.

HFSS will always calculate Zpi impedance, the impedance calculation using power and current, which are well-defined for a port because they are computed over the area of the port. Zpv and Zvi are not calculated by default. This is because v is computed by integrating along a user-defined integration line. If Zpi is very small (less than 10^-14) HFSS uses Zpv (if it is non-zero). In this case, a warning says that the design has too few conductors touching the lumped port. To renormalize the solution to a Zpv or Zvi characteristic impedance, you must define an integration line.

De-embedded S-Matrices in HFSS

De-embedding allows you to shift the reference plane of the S-matrix for any port by adding (or subtracting) a length of lossless transmission line to (or from) a port. The figure shown below illustrates such a case.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The S-matrix, S_1, describes the behavior of the structure that was modeled.

The de-embedded S-matrix, D_1, describes the behavior of a structure created by subtracting or adding a length of lossless transmission line to the structure that was modeled. It is computed by phase rotating the S-matrix inward toward or outward away from the structure by the specified length at each port.

To shift the reference plane of the structure, the S-matrix of the modified structure can be calculated using the following relationship:

$$[S'] = [e^{\gamma l}][S][e^{\gamma l}]$$

- where:

  $[e^{\gamma l}]$ is a diagonal matrix with the following entries:

$$\begin{bmatrix} e^{\gamma_1 l_1} & 0 & 0 \\ 0 & e^{\gamma_2 l_2} & 0 \\ 0 & 0 & e^{\gamma_3 l_3} \end{bmatrix}$$

- $\gamma = \alpha + j\beta$ is the complex propagation constant, where α is the attenuation constant of the current and β is the phase constant associated with the uniform transmission line at port i.
- l_i is the length of the reference plane shift. A positive value indicates a shift toward the structure by a length l.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternative Matrix Characterizations

- Calculating the Z-Matrix
- Calculating the Y-Matrix

Calculating the Z-Matrix

The impedance matrix, *Z*, is calculated from the S-matrix as follows:

$$Z = \sqrt{Z_0}(I-S)^{-1}(I+S)\sqrt{Z_0} \quad (1)$$

where

- *S* is the *n x n* generalized S-matrix.
- *I* is an *n x n* identity matrix.
- Z_0 is a diagonal matrix having the characteristic impedance (Z_0) of each port as a diagonal value.

Under the **Matrix Data** tab of the **Display Items Dialog**, the Z-matrix can be displayed as magnitude/ phase, real/ imaginary, magnitude, phase, real, or imaginary.

Calculating the Y-Matrix

The admittance matrix, *Y*, is simply the inverse of the impedance matrix, *Z*.

Under the **Matrix Data** tab of the **Solution Data** dialog box, the Y-matrix can be displayed as magnitude/ phase, real/ imaginary, magnitude, phase, real, or imaginary.

Passivity

Passive devices can only dissipate or temporarily store energy, but never generate it. The mathematical definition of passivity is based upon the following condition:

Q = I - conjugate(transpose(S)) * S must be a positive semidefinite matrix.

where:

- S is the S-parameter matrix
- I is an identity matrix.

A positive semidefinite matrix has only non-negative eigenvalues. The passivity test computes the eigenvalues of the matrix Q above at each frequency in the sweep. If any of the eigenvalues is negative, and larger (in magnitude) than the specified passivity tolerance, then a violation of passivity is reported to the user. The default value for passivity tolerance is .0001 tolerance above 1.0 magnitude.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Radiated Fields

When HFSS calculates radiation fields, the values of the fields over the radiation surface are used to compute the fields in the space surrounding the device. This space is typically split into two regions — the near-field region and the far-field region. The near-field region is the region closest to the source. In general, the electric field **E**(x,y,z) external to the region bounded by a closed surface may be written as

$$\boldsymbol{E}(x,y,z) = \int_s (\langle j\omega\mu_0 \boldsymbol{H}_{tan}\rangle G + \langle \boldsymbol{E}_{tan} \times \nabla G\rangle + \langle \boldsymbol{E}_{normal} \nabla G\rangle) ds \quad (1)$$

where

- *s* represents the radiation boundary surfaces.
- *j* is the imaginary unit, $\sqrt{-1}$.
- ω is the angular frequency, $2\pi f$.
- μ_0 is the relative permeability of the free space, $4\pi\times10^{-7}$ Wb/Am.
- $\boldsymbol{H}_{tan}$ is the component of the magnetic field that is tangential to the surface.
- $\boldsymbol{E}_{normal}$ is the component of the electric field that is normal to the surface.
- $\boldsymbol{E}_{tan}$ is the component of the electric field that is tangential to the surface.
- *G* is the free space Green's function, given by

$$G = \frac{e^{-jk_0|\boldsymbol{r}-\boldsymbol{r}'|\sqrt{\mu_r\varepsilon_r}}}{|\boldsymbol{r}-\boldsymbol{r}'|} \quad (2)$$

where

- k_0 is the free space wave number,
  $k_o = \omega\sqrt{\mu_o\varepsilon_o} = \omega/c$
- ***r*** and $\boldsymbol{r}'$ represent field points and source points on the surface, respectively.
- ε_0 is the permittivity of free space, $1/(c^2\mu_0)$
- ε_r is the relative permittivity of a dielectric.
- μ_r is the relative permeability of a dielectric.

This *r* dependence is characteristic of a spherical wave, a key feature of far fields. The far field is a spherical TEM wave with the following equation:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\boldsymbol{E} = \eta_o \sqrt{\frac{\mu_r}{\varepsilon_r}} \boldsymbol{H} \times \hat{\boldsymbol{r}} \quad (3)$$

where η_0 is the intrinsic impedance of free space, $\eta_o = \sqrt{\frac{\mu_o}{\varepsilon_o}}$.

You can select the material in which the objects are embedded (the default is vacuum). In that case, the Green's function and the intrinsic impedance of that material will be used, so the far field will be correct. See Global Material Environment.

When calculating the near fields, HFSS uses the general expressions given in (**eq. 1**). You must specify the radial coordinate *r*. Because it can be used to compute fields at an arbitrary radius from the radiating structure, this command can be useful in EMC applications.

> **Note:**
>
> If HFSS calculates the near fields in a problem containing an incident wave, the radius at which the fields are calculated is very important. If the radius is within the solution region, then the fields calculated are either the total fields or the scattered fields depending upon which is selected. If the radius is outside the solution region, then the fields calculated are only the scattered fields.

When calculating the far fields, the previously discussed far-field approximations are used, and the result is valid only for field points in the far-field region.

> **Warning:**
>
> A radiation or PML boundary or Floquet port must have been defined in the design for HFSS to calculate radiated fields.

Spherical Cross-Sections

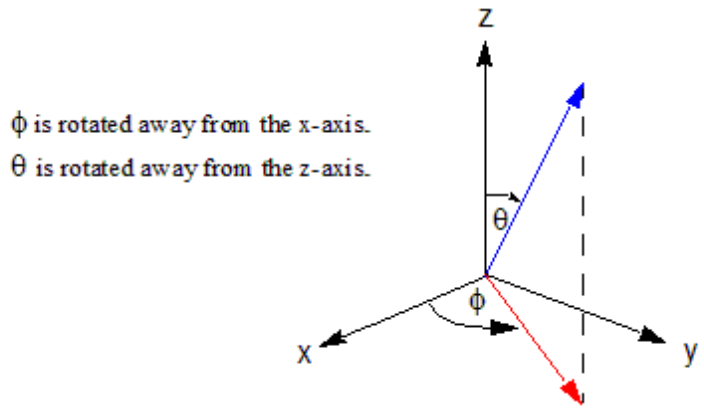
When you set up a spherical surface over which to analyze near or far fields, you specify a range and step size for phi and theta, or for far fields, also Azimuth over Elevation. These indicate the spherical direction in which you want to evaluate the radiated fields. For every value of phi there is a corresponding range of values for theta, and vice versa. This creates a spherical grid. Each grid point indicates a unique direction along a line that extends from the center of the sphere through the grid point. The radiated field is evaluated in this direction. The number of grid points is determined by the step size for phi and theta.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The sphere can be defined according to any defined coordinate system and before or after a solution has been generated.

> **Note:**
>
> For parts of the sphere outside of the model region, near-field approximation is calculated. However, if parts of the sphere are inside the model region, the model fields are used to compute interpolated values. A section of the sphere is considered to overlap the model if it lies in the enlarged model region after accounting for symmetry planes.

The relationship between phi and theta is shown below.



Antenna pattern measurements are calculated in a spherical coordinate system using the following basis vectors:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\hat{r} = \begin{bmatrix} \cos(\emptyset)\sin(\theta) \\ \sin(\emptyset)\sin(\theta) \\ \cos(\theta) \end{bmatrix}$$

$$\hat{\theta} = \begin{bmatrix} \cos(\emptyset)\sin(\theta) \\ \sin(\emptyset)\sin(\theta) \\ -\sin(\theta) \end{bmatrix}$$

$$\hat{\emptyset} = \begin{bmatrix} -\sin(\emptyset) \\ cos(\theta)] \\ 0 \end{bmatrix}$$

When HFSS evaluates the radiated fields, it needs at least two directions along which to plot the fields. Therefore, if the step size for phi is zero, then the step size for theta must be greater than zero, and vice versa. This ensures that the fields are plotted in at least two directions.

When setting up the sphere, phi and theta angles must be specified between -360 degrees (**deg**) and 360 degrees (**deg**), or the equivalents in radians (**rad**). If **deg** nor **rad** is specified, HFSS assumes the value to be in degrees.

Following are additional guidelines for specifying **Phi** in the **Near Field Radiation Sphere Setup** window or the **Far Field Radiation Sphere Setup** window:

Start	The point where the rotation of phi begins. The **Start** value must be equal to or greater than one.
Stop	The point where the rotation of phi ends. The **Stop** value must be greater than the **Start** value and less than 360. If the **Stop** value is equal to the **Start** value, then HFSS assumes that only one angle should be used and the **Step Size** value will be ignored.
Step Size	The number of degrees or radians (spherical grid points) between the sweep of phi. For example, to divide a sweep from 0° to 180° into 10° increments, you would enter **10**. Entering zero for the **Step Size** causes the sweep to consist of one point, the start value. If the **Step Size** value is zero, then HFSS assumes that only one angle should be used.

Following are additional guidelines for specifying **Theta**:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Start	The point where the rotation of theta begins. The **Start** value must be greater than -90 degrees, or the equivalent in radians.
Stop	The point where the rotation of theta ends. In classic definition the **Stop** value must be greater than the **Start** value and less than or equal to 180 degrees, or the equivalent in radians. In HFSS, you can input 360 degrees. If the **Stop** value is equal to the **Start** value, HFSS assumes that only one angle should be used and the **Step Size** value will be ignored.
Step Size	The number of degrees or radians (spherical grid points) between the sweep of theta. For example, to divide a sweep from -60 degrees to 60 degrees into 10-degree increments, you would enter **10deg**. Entering zero for the number of steps causes the sweep to consist of one point, the **Start** value. If the **Step Size** value is zero, then HFSS assumes that only one angle should be used.

For a discussion of using Azimuth and Elevation, see Setting up a Far Field Infinite Sphere.

Maximum Near-Field Data

The parameters listed in the **Max Field Data** window remain the same regardless of the geometry over which they were calculated. However, the coordinates displayed change depending on the geometry.

On a sphere, the coordinates — phi and theta — of the maximum value are listed under **Phi** and **Theta**. The values are given in volts per meter. Along a line, the coordinates — x, y, and z — of the maximum values are listed under **X**, **Y**, and **Z**. The values are given in volts per meter, and the coordinates are given in meters.

The following parameters are listed:

Total	The maximum of the total E-field.
X	The maximum E-field in the x-direction.
Y	The maximum E-field in the y-direction.
Z	The maximum E-field in the z-direction.
Phi	The maximum E-field in the ϕ-direction.
Theta	The maximum E-field in the θ-direction.
LHCP	The maximum left-hand circularly polarized component, which is equal to $\left\lvert\frac{1}{\sqrt{2}}(E_\theta - jE_\phi)\right\rvert$.
RHCP	The maximum right-hand circularly polarized component, which is equal to

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	$\left\|\frac{1}{\sqrt{2}}(E_\theta + jE_\phi)\right\|$.
Ludwig 3/X dominant	The maximum of the dominant component, *Vmain*, for an x-polarized aperture using Ludwig's third definition of cross polarization. This is equal to $\|E_\theta cos\phi - E_\phi sin\phi\|$.
Ludwig 3/Y dominant	The maximum of the dominant component, *Vmain*, for a y-polarized aperture using Ludwig's third definition of cross polarization. This is equal to $\|E_\theta sin\phi + E_\phi cos\phi\|$.

Maximum Far-Field Data

When HFSS calculates antenna parameters, the following maximum field data is calculated:

Total	The maximum of the total rE-field.
X	The maximum rE-field in the x-direction.
Y	The maximum rE-field in the y-direction.
Z	The maximum rE-field in the z-direction.
Phi	If you specified Phi-Theta coordinate system in the Far Field Sphere, the maximum rE-field in the ϕ-direction.
Theta	If specified Phi-Theta coordinate system in the Far Field Sphere, the maximum rE-field in the θ-direction.
Azimuth	If you specified Azimuth over Elevation or Elevation over Azimuth in the Far Field Sphere, the maximum rE for Azimuth.
Elevation	If you specified Azimuth over Elevation or Elevation over Azimuth in the Far Field Sphere, the maximum rE for Elevation.
LHCP	The maximum left-hand circularly polarized component, which is equal to $\left\|\frac{1}{\sqrt{2}}(E_\theta - jE_\phi)\right\|$.
RHCP	The maximum right-hand circularly polarized component, which is equal to $\left\|\frac{1}{\sqrt{2}}(E_\theta + jE_\phi)\right\|$.
Ludwig 3/X dominant	The maximum of the dominant component, *Vmain*, for an x-polarized aperture using Ludwig's third definition of cross polarization. This is equal to $\|E_\theta cos\phi - E_\phi sin\phi\|$.
Ludwig 3/Y dominant	The maximum of the dominant component, *Vmain*, for a y-polarized aperture using Ludwig's third definition of cross polarization. This is equal to $\|E_\theta sin\phi + E_\phi cos\phi\|$.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When calculating the maximum far field values, the distance *r* is factored out of the E-field. Therefore, the units for the maximum field data values are given in volts.

Array Factor Calculation

HFSS enables you to compute antenna array radiation patterns and antenna parameters for designs that have analyzed a single array element. You can define array geometry and excitation. HFSS models the array radiation pattern by applying an "array factor" to the single element's pattern.

Two array geometry types are supported. The "regular uniform array" geometry defines a finite 2D array of uniformly spaced, equal-amplitude elements. This is a natural specification after analyzing a single-unit cell of an infinite array. The regular array type may be scanned to a user-specified direction. Scan direction can be specified in terms of spherical coordinate angles in the radiation coordinate system. The regular array geometry type also allows scan specification in terms of differential phase shifts between elements.

The "custom array" geometry allows for greater flexibility. It defines an arbitrary array of identical elements distributed in 3D space with individual user-specified complex weights.

Cautionary Note for Array Factor Use

The field factorization (eq. 1) and consequent use of an array factor are useful tools for analyzing the radiated fields of antenna arrays; however, the analysis can yield incorrect results if used improperly. An HFSS single array element solution does not generally take into account the effects of the element's hypothetical neighbors.

For closely spaced array elements, these proximity effects (mutual coupling) may be significant. Consequently the patterns of the array elements vary with their position in the array and may depart significantly from the isolated element pattern. In such cases, the primary assumption in the use of the array factor is violated and the results will be inaccurate.

Note in particular that the array power expressions (eq. 13) and (eq. 14) neglect mutual coupling between elements of the finite array. Unless mutual coupling effects are negligible or have been implicitly included in the single element solution, the normalizations (eq. 13) and (eq. 14) gain and directivity are incorrect.

Theory of the Array Factor Calculation

The composite far-field pattern, $\boldsymbol{E}_{array}(\phi, \theta)$ from an array of *N* identical radiating sources, each with far-field pattern $\boldsymbol{E}_{element}(\phi, \theta)$, may be factored into the form

$$\boldsymbol{E}_{array}(\phi,\theta) = AF(\phi,\theta)\boldsymbol{E}_{element}(\phi,\theta) \quad (1)$$

where the "array factor" AF (ϕ, θ) is defined as

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$AF(\phi,\theta) = \sum_{n=1}^{N} W_n e^{jk\mathbf{r}_n \cdot \hat{\mathbf{r}}} \quad (2)$$

and where

- (ϕ, θ) are the field-point spherical angles.
- W_n is the complex weight assigned to element *n*.
- *j* is $\sqrt{-1}$.
- *k* is $2\pi/\lambda$., whereλ is the wave length in the global material environment.
- $\mathbf{r}_n$ is the position vector of element *n*, <*xn*, *yn*, *zn*>.
- $\hat{\mathbf{r}}$ is the pattern angle unit vector, <sinθcosϕ, sinθsinϕ, cosθ>.

The complex weights W_n in (eq. 2) may be written in terms of a (real) voltage amplitude A_n and (real) phase Ψ_n as:

$$W_n = A_n e^{j\Psi n} \quad (3)$$

To scan a regular array in the direction (ϕ_0, θ_0), the element phases Ψ_n are set to

$$\Psi_n = -k\mathbf{r}_n \cdot \hat{\mathbf{r}}_o \quad (4)$$

where

$$\hat{\mathbf{r}}_0 = <\sin\theta_0\cos\phi_0,\ \sin\theta_0\sin\phi_0,\ \cos\theta_0> \quad (5)$$

is the scan-angle unit vector.

Regular Uniform Arrays

Let us define a uniform array as an array with unity amplitude weights for all elements ($A_n = 1$ for all *n*). For the case in which a uniform array is scanned to direction $\hat{\mathbf{r}}_0$, the array factor (eq. 2) becomes

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$AF(\phi,\theta) = \sum_{n=1}^{N} e^{jk\mathbf{r}_n \cdot (\hat{\mathbf{r}} - \hat{\mathbf{r}}_0)} \quad (6)$$

For a "regular" uniform array with element spacing defined by lattice vectors ***u*** and ***v***, the element position vectors $\mathbf{r}_n$ may be written in the doubly indexed form

$$\mathbf{r}_{mn} = (m-1)\mathbf{u} + (n-1)\mathbf{v} \quad (7)$$

with *m* = 1, 2, ..., *Nu* and *n* = 1, 2, ..., *Nv*. The total number of elements in the array is given by *N* = *NuNv*.

The array factor (eq. 6) for the *Nu*x*Nv* array becomes

$$AF(\phi,\theta) = \sum_{m=1}^{N_u}\sum_{n=1}^{N_v} e^{jk\mathbf{r}_{mn} \cdot (\hat{\mathbf{r}} - \hat{\mathbf{r}}_0)} \quad (8)$$

Scan Specification for Regular Uniform Arrays

$\hat{\mathbf{r}}_0$

To develop this alternate scanning phase description, (eq. 7) is used to rewrite the expression (eq. 4) in doubly indexed form as follows:

$$\Psi_{mn} = -k\mathbf{r}_{mn} \cdot \hat{\mathbf{r}}_0 = -k(m-1)\mathbf{u} \cdot \hat{\mathbf{r}}_0 - k(n-1)\mathbf{v} \cdot \hat{\mathbf{r}}_0 \quad (9)$$

(Let us define ψ_u as the differential phase between adjacent elements in the **u** direction. Similarly, let us define ψ_v as the differential phase between adjacent elements in the **v** direction. Then

$$\Psi_u \equiv \Psi_{m+1,n} - \Psi_{m,n} = -k\mathbf{u} \cdot \hat{\mathbf{r}}_0 \quad (10)$$

and

$$\Psi_v \equiv \Psi_{m,n+1} - \Psi_{m,n} = -k\mathbf{v} \cdot \hat{\mathbf{r}}_0 \quad (11)$$

The scanning phase (eq. 4) may now be rewritten in terms of ψ_u and ψ_v as

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\psi_{mn} = (m-1)\psi_u + (n-1)\psi_v \quad (12)$$

Thus in the case of a regular uniform array, the angle pair (ψ_u, ψ_V) may act as a substitute scan definition for the more general (ϕ_0, θ_0).

Custom Arrays

Once you have imported the array factor information from a text file, HFSS uses (eq. 8) to compute the array factor. When a custom array is defined, no scan direction is set and the array factor phase weights are those specified on an element-by-element basis in the geometry file.

The text file must have the following format. Coordinate values are assumed to be model units, and phase and amplitude are in SI units (radians and volts respectively):

```
N
x_1 y_1 z_1 A_1 P_1
x_2 y_2 z_2 A_2 P_2
...
 ...
 x_N y_N z_N A_N P_N
```

where

- x_1 is the x-coordinate position of the first element.
- y_1 is the y-coordinate position of the first element.
- z_1 is the y-coordinate position of the first element.
- A_1 is the amplitude weight of the first element. Amplitude references voltage.
- P_1 is the phase weight for the first element. Phase references radians.

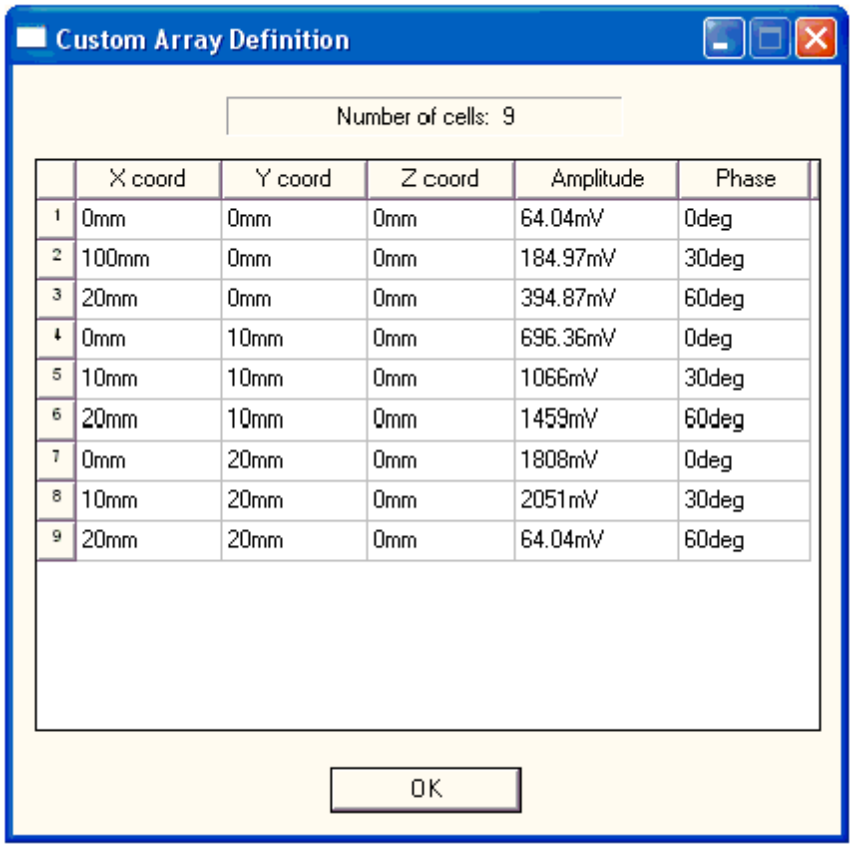
Following is an example of a square 3 x 3 custom array geometry defined in a text file. The array elements are uniformly weighted and separated from one another in the x- and y-directions by 0.6729 user units.

```
! first line is a count of cells
9

! then [x,y,z] position of each cell
! and amplitude and phase
! default units for position are model units
! default units for amplitude is Volts
! default units for phase is radians

0 0 0 64.04mV 0
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
10cm 0 0 184.97mv 30deg
20 0 0 394.87mv 60deg
0 10 0 696.36mv 0
10 10 0 1.066 30deg
20 10 0 1.459 60deg
0 20 0 1.808 0
10 20 0 2.051 30deg
20 20 0 64.04mv 60deg
```

The information will appear as follows in the **Custom Array Definition** window:



Power Normalizations

When the array factor feature is in use, the power normalization used to compute realized gain, gain, and directivity are modified as follows.

Let $P^{inc}_{element}$, $P^{acc}_{element}$, and $P^{rad}_{element}$ denote the incident, accepted, and radiated power of the single array element. When the array factor feature is invoked for an array of N elements, the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

array incident power, array accepted power, and the array radiated power will be computed simply as the sums of the corresponding element powers. Symbolically

$$P_{array}^{inc} = N \bullet P_{element}^{inc} \quad (13)$$

$$P_{array}^{acc} = N \bullet P_{element}^{acc} \quad (14)$$

$$P_{array}^{rad} = N \bullet P_{element}^{rad} \quad (15)$$

Caution: The above power array formulas are applicable to uniform excitation of large planar arrays analyzed using a unit cell with linked boundaries. For array elements that have been analyzed in isolation, only the incident power formula is rigorous, and the accepted and radiated power normalizations are generally not accurate.

Antenna Parameters

Generally, when dealing with radiated fields, you are also interested in the antenna properties of the radiated bodies. You can right click on an Antenna Parameters Overlay to open a Properties dialog that lets you select a set of the available parameters.

HFSS 3D Layout can calculate the following antenna properties:

- Maximum intensity (Max U)
- Peak directivity
- Peak gain
- Peak realized gain
- Radiated power
- Accepted power
- Incident power
- System Gain (see Scaling Sources of HFSS)
- System Power (see Scaling Sources for HFSS)
- Radiation efficiency
- Total efficiency
- System efficiency
- Front to Back Ratio
- Decay Factor

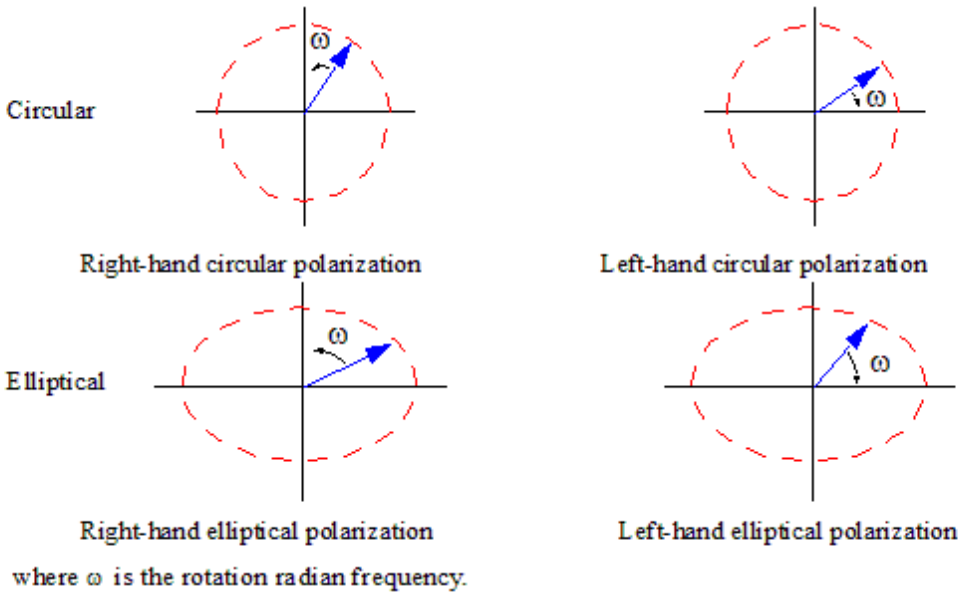
> **Warning:**
>
> A radiation or PML boundary must have been defined in the design for HFSS to calculate radiated fields.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Polarization of the Electric Field

At each aspect angle in the far field of a radiating source, the electric and magnetic field vectors lie in a fixed plane. Over time, the instantaneous electric field vector traces out a figure or shape in this plane. This figure defines the polarization state of the field.

In general, this figure is an ellipse and is called the *polarization ellipse*. The wave is said to be *elliptically polarized* when the instantaneous electric field traces out an ellipse. As a special case, the polarization ellipse may be a circle, in which case the wave is *circularly polarized*. Elliptical and circular polarization have two different states, *left* and *right*, distinguished by the sense of rotation of the electric field vector. Some of these figures, or states, are shown below. In each case the direction of propagation is off the screen.



A second special case occurs when the polarization ellipse degenerates to a straight line. In this case the wave is *linearly polarized*.

To completely describe the polarization state of a radiated field, two independent components are required. HFSS supports three types of descriptions:

- Spherical polar
- Ludwig-3
- Circular

Spherical Polar

$$E = (E_\theta, E_\phi)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The polarization ratio for a predominantly ϕ-polarized antenna is equal to

$$\left|\frac{E_\phi}{E_\theta}\right| \quad (1)$$

The polarization ratio for a predominantly θ-polarized antenna is equal to

$$\left|\frac{E_\theta}{E_\phi}\right| \quad (2)$$

Ludwig-3 Polarization

Arthur C. Ludwig wrote a classic paper [Ref. 1]on the definition of cross polarization. In particular, his third definition is often used since it describes the field components that are typically measured on a far-field antenna test range. Using his definition, the radiated field may be written as $E = E_x, E_y$ where

$$E_x = E_\theta \cos\phi - E_\phi \sin\phi \quad (1)$$

$$E_y = E_\theta \sin\phi + E_\phi \cos\phi \quad (2)$$

and phi is the usual azimuthal angle in the reference spherical coordinate system.

[1] Arthur C. Ludwig, The Definition of Cross Polarization, IEEE Transactions on Antennas and Propagation, vol. AP-21 num. 1, pp. 116 -119, Jan. 1973.

Circular Polarization

For antennas designed to receive or transmit circularly polarized fields, a meaningful description is in terms of pure left and right circular states. In this description, the field may be written as $E = E_R, E_L$ where

$$E_R = \frac{1}{\sqrt{2}}(E_\theta + jE_\phi) \quad (1)$$

$$E_L = \frac{1}{\sqrt{2}}(E_\theta - jE_\phi) \quad (2)$$

Axial Ratio

Axial ratio is defined as the ratio of the major to the minor axis of the polarization ellipse.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. E_x and E_y are orthogonal complex-valued field components. If either is zero, HFSS treats the field as linearly polarized. However, if neither E_x and E_y is zero:
2. Compute circular components E_- and E_+ from:

   $E_- = E_x - jE_y$ (1)

   $E_+ = E_x + jE_y$ (2)

3. If $E_+ = 0$ or $E_- = 0$ HFSS understands the field as perfectly circular, the axial ratio is 1. Otherwise, for the elliptical polarization case, HFSS determines the tilt angle τ from:

   $\text{phase}(E_-/E_+) = 2\tau$ (3)

4. Rotate the original data to coincide with the axes of the polarization ellipse.

   $$E'_x = E_x \cos\tau - E_y \sin\tau \quad (4)$$

   $$E'_y = E_x \sin\tau + E_y \cos\tau \quad (5)$$

5. The Axial Ratio AR is given by:

   $$AR = \left|\frac{E'_y}{E'_x}\right| \quad (6)$$

   Because the above definition does not discriminate between major and minor ellipse axes, to enforce the convention that $AR \leq 1$, it is necessary to check this condition and if necessary invert the value obtained.

Polarization Ratio

The IEEE defines the (complex) polarization ratio as, "For a given field vector at a point in space, the (magnitude of the) ratio of the complex amplitudes of two specified orthogonally polarized field vectors into which the given field vector has been resolved." [Ref. 2] HFSS computes the following six polarization ratios at each selected aspect angle:

$$Circular/LHCP = \left|\frac{E_L}{E_R}\right| \quad (1)$$

$$Circular/RHCP = \left|\frac{E_R}{E_L}\right| \quad (2)$$

$$Spherical/Phi = \left|\frac{E_\phi}{E_\theta}\right| \quad (3)$$

$$Spherical/Theta = \left|\frac{E_\theta}{E_\phi}\right| \quad (4)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$Ludwig \qquad 3/X = \left|\frac{E_x}{E_y}\right| \quad (5)$$

$$Ludwig \qquad 3/Y = \left|\frac{E_y}{E_x}\right| \quad (6)$$

[2] IEEE Standard Definitions of Terms for Antennas, IEEE Transactions on Antennas and Propagation, vol. AP-31 num. 6, Nov. 1983.

Front to Back Ratio

Front-to-back-ratio is one value for the entire far field setup (and is an unambiguous single quantity for a given setup) and is maximum rETotal (equivalently directivity) as the front value, then the rETotal from the opposite direction and use the ratio of front to back values, without log.

Max U

The radiation intensity, *U*, is the power radiated from an antenna per unit solid angle. HFSS calculates the radiation intensity in the direction in which it has the maximum value. The maximum intensity of the radiation is measured in watts per steradian and is calculated by

$$U(\theta,\phi) = \frac{1}{2}\frac{|E|^2}{\eta_0}r^2 \quad (1)$$

where

- $U(\theta,\phi)$ is the radiation intensity in watts per steradian.
- $|E|$ is the magnitude of the E-field.
- η_0 is the intrinsic impedance of free space — 376.7 ohms.
- r is the distance from the antenna, in meters.

Peak Directivity

Directivity is defined as the ratio of an antenna's radiation intensity in a given direction to the radiation intensity averaged over all directions. Peak directivity, in turn, is the maximum directivity over all the user-specified directions of the far-field infinite sphere.

Directivity is a dimensionless quantity represented by
where

$$directivity = \frac{4\pi U}{P_{rad}} \quad (1)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- U is the radiation intensity in watts per steradian in the direction specified.
- P_{rad} is the radiated power in watts.

> **Note:**
>
> The peak directivity displayed in the **Antenna Parameters** window is the directivity in the direction of maximum radiation intensity, U_{max}.

- For a lossless antenna, the directivity will be equal to the gain. However, if the antenna has inherent losses, the directivity is related to the gain by the radiation efficiency of the antenna.

Peak Gain

Gain is four pi times the ratio of an antenna's radiation intensity in a given direction to the total power accepted by the antenna. Peak gain, in turn, is the maximum gain over all the user-specified directions of the far-field infinite sphere.

The following equation is used to calculate gain in HFSS:

$$gain = 4\pi\frac{U}{P_{acc}} \quad (1)$$

where

- U is the radiation intensity in watts per steradian in the direction specified.
- P_{acc} is the accepted power in watts entering the antenna.

Gain can be confused with directivity, since they are equivalent for lossless antennas. Gain is related to directivity by the radiation efficiency of the antenna. If the radiation efficiency is 100%, they are equal.

> **Note:**
>
> Because the gain is calculated from the input signal at the port, a port must be defined for this quantity to be displayed.

Peak Realized Gain

Realized gain is four pi times the ratio of an antenna's radiation intensity in a given direction to the total power incident upon the antenna port(s). Peak realized gain, in turn, is the maximum realized gain over all the user-specified directions of the far-field infinite sphere. Time-average power of the incident wave is referenced for realized gain. This quantity can be small for the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

case when the imaginary part of port impedance is much larger than real part. In fact, it can be significantly smaller than accepted power which means that realized gain can be larger than gain.

The following equation is used to calculate realized gain in HFSS:

$$\text{realized gain} = 4\pi \frac{U}{P_{incident}} \quad (1)$$

where

- U is the radiation intensity in watts per steradian in the direction specified.
- $P_{incident}$ is the incident power in watts.

> **Note:**
>
> Because the gain is calculated from the input signal at the port, a port must be defined for this quantity to be displayed.

Radiated Power

Radiated power is the amount of time-averaged power (in watts) exiting a radiating antenna structure through a radiation boundary.

For a general radiating structure in HFSS, radiated power is computed as

$$P_{rad} = \Re \int_s \boldsymbol{E} \times \boldsymbol{H}^* \cdot d\boldsymbol{s} \quad (1)$$

where

- P_{rad} is the radiated power in watts.
- $\Re$ is the real part of a complex number.
- s represents the radiation boundary surfaces.
- E is the electric field.
- H^* is the conjugate of **the magnetic field**.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- *ds* is the local radiation-boundary unit normal directed out of the 3D model.

> **Note:**
>
> The accuracy of the computed radiated power depends on the accuracy of ***E*** and ***H***. In some cases it is possible that the computed radiated power may deviate slightly from the actual radiated power.

The accuracy of the computed radiated power depends on the accuracy of E and H on the absorbing boundary. In some cases it is possible that the computed radiated power may deviate slightly from the actual radiated power. To increase the accuracy of the radiated power, seed the mesh on the absorbing boundary.

As a default behavior for SBR+ design and SBR+ hybrid design, an additional SBR+ power loss is subtracted from the total radiated power. For each source *i*, the SBR+ power loss $L_i$over all other sources *j* is computed as

$$L_i = 0.5 \sum_j \Re(b_j P_i b_j^*), b_j = S_{ji} a_i$$

where

- $\Re$ is the real part of a complex number
- S_{ji} is the S parameter from SBR+. When source *i* and *j* come from the same component instance, S_{ji} includes only scatter component. Otherwise,S_{ji} includes both scatter and incident components.
- a_i is the scaling factors specified in Edit Source.
- b_j^* is the conjugate of b_j.
- P_i is port characteristic matrix

Accepted Power

The accepted power is the time-averaged power (in watts) entering a radiating antenna structure through one or more ports.

Fundamentally in HFSS, accepted power is computed as

$$P_{acc} = Re \int_A \boldsymbol{E} \times \boldsymbol{H}^* \cdot d\boldsymbol{S} \quad (1)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where

- P_{acc} is the accepted power in watts.
- *Re* is the real part of a complex number.
- *A* is the union of all port boundaries in the model, including Wave and Lumped Ports, but excluding Floquet ports.
- *E* is the electric field.
- H^* is the conjugate of the magnetic field.
- *dS* is the local port-boundary unit normal directed into the 3D HFSS model.

Example: Consider the simple case of a single port antenna with zero imaginary port impedance modeled in a modal project. Then the general expression for accepted power reduces to

$$P_{acc} = |a|^2(1 - |s_{11}|^2) \quad (2)$$

where

- $|a|$ is the complex modal excitation specified in the **Edit Sources** window.
- s_{11} is the single-entry generalized scattering matrix.

> **Note:**
>
> Because the accepted power is calculated from the input signal at the port, a port must be defined for this quantity to be displayed.

Thus when $|s_{11}|$ approaches 1 (total reflection), P_{acc} approaches zero. When $|s_{11}|$ approaches 0 (perfect match), $P_{acc} = |a|^2$, the total power carried by the incident mode.

In the multiport case the physical interpretation of accepted power can sometimes be confusing. By the fundamental definition, energy that leaves the antenna structure through a port reduces accepted power. This, in a multiport antenna, energy cross coupled from an excited port to a different port reduces accepted power. Therefore in such cases, even if an excited port is well-matched, P_{acc} may be small.

In the Edit Sources window of a terminal project, you specify total voltage V instead of model incident power. Using the underlying model S-matrix, HFSS converts the terminal excitation to total fields at the ports, and then applies the general definition.

Incident Power

Incident power for an antenna structure is defined as the time-averaged power incident on the port(s) of the antenna:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$P_{inc} = Re\int_A \boldsymbol{E_{inc}} \times \boldsymbol{H^*_{inc}} \cdot \boldsymbol{dS} \quad (1)$$

where

- P_{inc} is the incident power in watts.
- *Re* is the real part of a complex number.
- *A* is the union of all port boundaries in the model, including Wave and Lumped Ports, but excluding Floquet ports.
- $\boldsymbol{E}_{inc}$ is the incident electric field.
- $\boldsymbol{H}^*_{inc}$ is the conjugate of the incident magnetic field.
- ***dS*** is the local port-boundary unit normal directed into the 3D HFSS model.

In the Edit Sources window of a modal project, you specify the power as the magnitude of the Poynting vector along with the a phase specification for the complex modal excitation. The magnitude of the Poynting vector is computed from:

$$Q_{inc} = Mag\int_A \boldsymbol{E_{inc}} \times \boldsymbol{H_{inc}} \cdot \boldsymbol{dS} \quad (2)$$

For a lossless port the incident power is the same as what is specified in edit sources. However, for lossy ports this is not true and therefore the incident power can be different from what is specified in edit sources.

In the **Edit Sources** window of a terminal project, you specify total voltage V instead of the modal incident power. Using the underlying modal S-matrix, HFSS converts the terminal excitation to total fields at the ports, and then applies the general definition. As a result of this procedure, the incident power in a terminal project depends on both the user-specified stimulation as well as the model itself.

Radiation Efficiency

The radiation efficiency is the ratio of the radiated power to the accepted power given by

$$e = \frac{P_{rad}}{P_{acc}} \quad (1)$$

where

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- P_{rad} is the radiated power in watts.
- P_{acc} is the accepted power in watts.

> **Note:**
>
> Because the radiation efficiency is calculated from the accepted power, a port must be defined for radiation efficiency to be displayed.

Total Efficiency

The total efficiency is Radiated Power over Incident Power.

System Efficiency

The system efficiency is Radiated Power over System Power.

Decay Factor

When Global Environment Material is lossy, strictly speaking, the far field is zero, since the Green's function is:

$$|g| = \frac{1}{r}e^{-kr} = \frac{1}{r}e^{-(\alpha + j\beta)r} \quad (1)$$

where

$$k = j\omega\mu(\sigma + j\omega\varepsilon) = \alpha + j\beta \quad (2)$$

Against that fact, HFSS calculates the far field using the imaginary part of vector ***k***.

You can then modify the magnitude of the field pattern by using the real part (α) of the propagation constant, which is displayed as "Decay Factor).

Ansys Near Field Description File Format Specification

The purpose of the Ansys Near Field Description (AND) File is to facilitate the auto-generation of HFSS design for any Near Field Data the user may wish to use for an HFSS simulation. These Near Field Data files could be in a variety of different formats and files, so the AND Format was designed to be as flexible as possible to accommodate them.

Overview

- Recognized by the ".and" extension
- 'NearFieldHeader' Block - Provides format, field type, and geometry information.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 'NearFieldData' Block - Links the user's near field data files to particular solve frequencies.

Blocks are defined with opening ($begin 'block_name') and closing ($end 'block_name') block tags that contain either key/value pairs, or function/argument calls described below.

```
$begin 'BlockName'
  key=value
  func(arg1, arg2)
$end 'BlockName'
```

NearFieldHeader Block

Every AND File requires a NearFieldHeader block shown below. This header is used to provide information to HFSS about the Near Field Data that may not be present in the Data Files themselves. Depending on the Near Field Data Format, some of this information may be redundant and merely used for verification purposes.

Key	Value Format	Description
type	*'format type*'	[Required] Format Type of the of the User's Near Field Data files.
fields	*'field type*'	[Optional] Field Type of the User's Near Field Data. Defaults to "EH" when not present.
geometry	*'primitive type*'	[Required] Primitive describing Near Field Source Geometry.
center	'10mm, 10mm, 10mm'	[Based on Geometry] Center of geometric primitive.
size	'10mm, 10mm, 10mm' '20mm, 20mm	[Based on Geometry] xSize, ySize, zSize of primitive. Only two dimensions (XY, YZ or XZ) required for Plane Geometry.
axis	'0, 0, 1'	[Based on Geometry] Primary axis of primitive.
normal	'0, 1, 0'	[Based on Geometry] Normal axis of primitive.
radius	'20mm'	[Based on Geometry] Radius
height	'20mm'	[Based on Geometry] Height
fsweep	'1GHz, 2GHz, … 5.6 GHz'	[Required] The frequency band of the analysis.
fcoords	*'fcoords type'*	[Optional] Coordinate System used to describe data points

Valid Dimension Units:

Standard SI and Imperial Length Units are supported in Geometry Dimensions including:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

'nm', 'mm', 'meter', 'cm', 'ft', 'in', 'mil', 'uin'

Standard SI Frequency Units are supported in 'fsweep' including:

'Hz, 'kHz, 'MHz, 'GHz, 'THz

Supported '*format type*':

- "nfd" - HFSS External Near Field Data File generated by exporting Fields.

Supported '*field type*':

- "EH" - Both E and H incident fields are provided. (Default)
- "E" - Only E Fields are provided.
- "H" - Only H Fields are provided.
- "JM" - Both J and M Tangential Currents are provided.
- "J" - Only J Current is provided.
- "M" - Only M Current is provided.

Supported '*primitive type*':

- ""sphere" - Spherical Geometry requiring center, radius.
- ""box" - Cube Geometry requiring center, size. Must be axis aligned.
- ""cylinder" - Cylinder Geometry requiring center, axis, radius, height. Must be axis aligned.
- ""plane" - Pseudo-Plane Geometry requiring center, normal, size. Must be axis aligned. Plane data is really treated as a Rectangle so a two dimensional size must be specified.

Supported '*fcoords type*':

- "cartesian" - Data Points are described using cartesian coordinates.
- "spherical" - Data Points are formatted using spherical coordinates.
- "cylindrical" - Data Points are formatted using cylindrical coordinates.

The Coordinate System Type is optional and will default to the corresponding system for that primitive type

("spherical" for Spheres, "cylindrical" for Cylinders, "cartesian" for Box/Plane).

Example NearFieldHeader Blocks

Box

```
$begin 'NearFieldHeader'
  type='nfd'
  fcoords='cartesian'
  fields='JM'
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  geometry='box'
  center='0mm,0mm,0mm'
  size='20mm,20mm,20mm'
  fsweep='1GHz, 3GHz'
$end 'NearFieldHeader '
```

Sphere

```
$begin 'NearFieldHeader'
  type='nfd'
  fcoords='spherical'
  fields='EH'
  geometry='sphere'
  center='0mm,0mm,0mm'
  radius='150mm'
  fsweep='1GHz, 3GHz'
$end 'NearFieldHeader'
```

Plane

```
$begin 'NearFieldHeader'
  type='nfd'
  fields='E'
  geometry='plane'
  center='0mm,0mm,0mm'
  size='20mm,20mm'
  normal='0,1,0'
  fsweep='1GHz, 3GHz'
$end 'NearFieldHeader '
```

Cylinder

```
$begin 'NearFieldHeader'
  type='nfd'
  geometry='cylinder'
  center='0mm,0mm,0mm'
  axis='0,0,1'
  radius='20mm'
  height='258mm'
  fsweep='1GHz, 3GHz'
$end 'NearFieldHeader'
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NearFieldData Block

The NearFieldData Block is used to specify which external data files to use for which frequencies. External Near Field Data can be spread across multiple files and multiple frequencies. This block provides a means of organizing these data files.

To bind a single Near Field Data file to a particular frequency, use:

```
FreqData("1GHz", "path/to/data.nfd")
```

To split Near Field Data files (one contains E Fields, one contains H Fields), use:

```
FreqData("1GHz", "path/to/efields.nfd", "path/to/hfields.nfd")
```

To bind a default Near Field Data file to all frequencies not explicitly defined, use:

```
FreqData("0", "path/to/data.nfd")
```

Example NearFieldData Blocks

```
$begin 'NearFieldData'
  FreqData("0", "exportfields.nfd")
$end 'NearFieldData'
$begin 'NearFieldData'
  FreqData("1GHz", "exportfields.nfd")
  FreqData("2GHz", "exportfields2.nfd")
  FreqData("3GHz", "exportfields3.nfd")
  FreqData("4GHz", "exportfields4.nfd")
  FreqData("5GHz", "exportfields5.nfd")
$end 'NearFieldData'

$begin 'NearFieldData'
  FreqData("0", "efields0.nfd", "hfields0.nfd")
  FreqData("2GHz", "efields1.nfd", "hfields1.nfd")
  FreqData("4GHz", "efields2.nfd", "hfields2.nfd")
$end 'NearFieldData'
```

Example .and Files

```
$begin 'NearFieldHeader'
  type='nfd'
  geometry='box'
  center='0mm,0mm,0mm'
  size='20mm,20mm,20mm'
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  fsweep='1GHz, 3GHz'
  $end 'NearFieldHeader'
  $begin 'NearFieldData'
  FreqData("0", "exportfields.nfd")
$end 'NearFieldData'

$begin 'NearFieldHeader'
  type='nfd'
  fcoords='cartesian'
  geometry='cylinder'
  center='0mm,0mm,0mm'
  axis='0,0,1'
  radius='20mm'
  height='258mm'
  fsweep='1GHz, 2GHz, 3GHz, 4GHz, 5GHz'
$end 'NearFieldHeader'
$begin 'NearFieldData'
  FreqData("1GHz", "exportfields.nfd")
  FreqData("2GHz", "exportfields2.nfd")
  FreqData("3GHz", "exportfields3.nfd")
  FreqData("4GHz", "exportfields4.nfd")
  FreqData("5GHz", "exportfields5.nfd")
$end 'NearFieldData'

$begin 'NearFieldHeader'
  type='nfd'
  fcoords='cartesian'
  geometry='plane'
  center='0mm,0mm,0mm'
  normal='0,1,0'
  size="100mm,100mm"
  fsweep='1GHz, 2GHz, 3GHz, 4GHz'
  $end 'NearFieldHeader'
  $begin 'NearFieldData'
  FreqData("0", "efields0.nfd", "hfields0.nfd")
  FreqData("2GHz", "efields1.nfd", "hfields1.nfd")
  FreqData("4GHz", "efields2.nfd", "hfields2.nfd")
```


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
$end 'NearFieldData'
```

Emission Test

The Emission Test in HFSS is intended for EMC analysis. EMC regulations specify the maximum radiated emissions as a function of frequency. The Emission Test in HFSS produces a plot that can be compared with these regulations. Assuming that the HFSS simulation contained a frequency sweep in which all the fields were saved (a fast frequency sweep is very suited for this), the Emission Test does the following:

1. It steps through the frequencies of your sweep;
2. At each frequency, it determines the maximum value of |E| on a sphere around the radiating device, regardless of direction;
3. It plots this $|E_{max}|$ as a function of frequency. Optionally, $|E_{max}|$ is scaled to account for the spectral density of a pseudo-random digital signal.

Geometry, S-parameters and antenna pattern

Let's start with a small model that solves quickly. Figure 1 shows an open-ended rectangular waveguide of which two side walls are perfect-H walls. Symmetry has been exploited twice, so we end up with a quarter of the geometry.

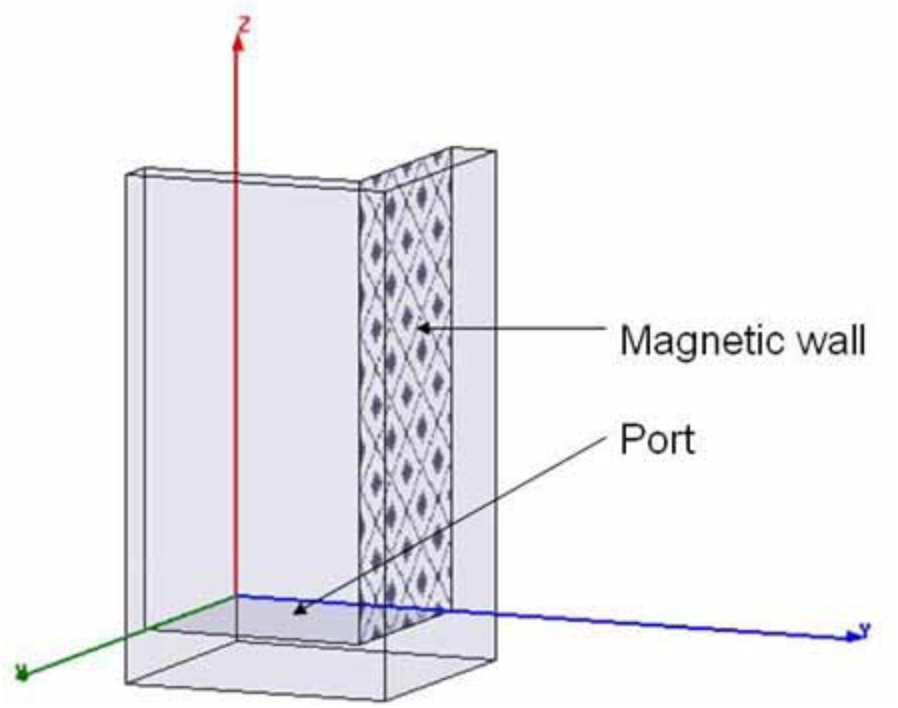


Fig. 1 The geometry: waveguide without cut-off

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Around the waveguide is a larger volume of air with a radiation boundary. The model has been solved from DC to 12 GHz. Figure 2 shows the S parameters.

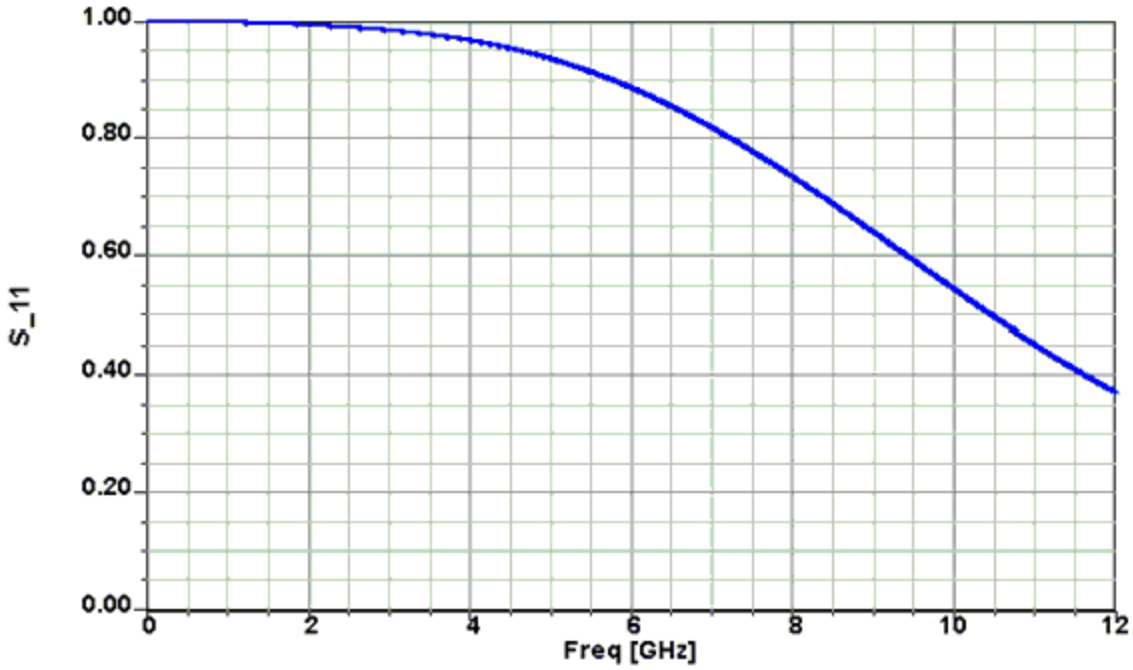


Fig. 2 S11 of the open-ended waveguide with two magnetic walls

Figure 3 shows a plot of the electric field at 12 GHz on a sphere with a radius of one meter. In this case, this is already way in the far field, so we are really dealing with radiated fields. Note that the electric field has a maximum of 28.8 V/m.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

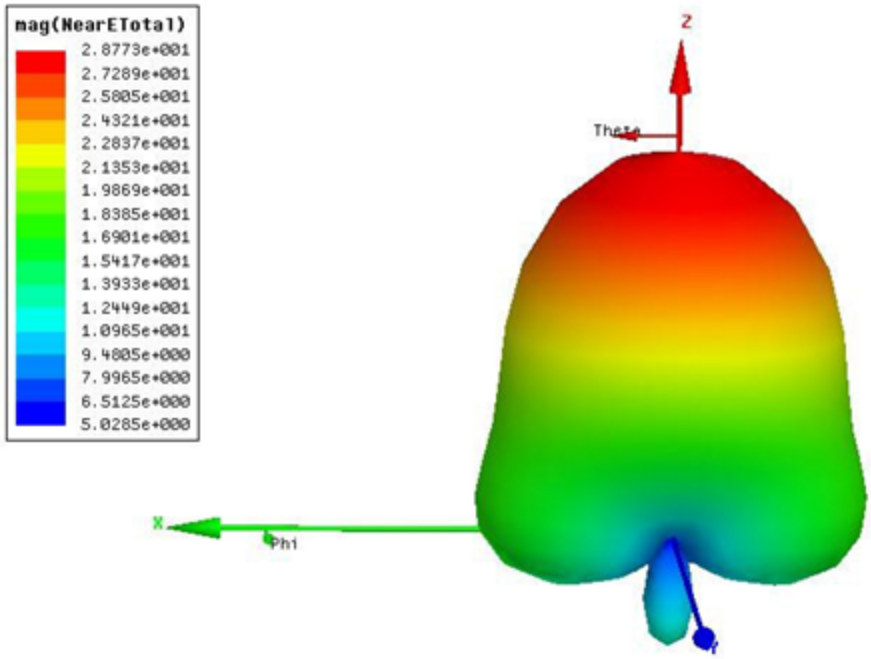


Fig. 3 Radiated electric field on a sphere with radius one meter

The average value in figure 3 is roughly 14 V/m. This corresponds to a power flux of $0.5|\underline{\mathbf{E}}\times\underline{\mathbf{H}}| = 0.5|E|^2/Z = 0.5\times196/377$ W/m^2 = 0.26 W/m^2. Over the sphere, area 4π m^2, this gives a power of 3.3 W. This value makes sense: the power into a quarter model is 1 W, so the total

power is 4 W, of which a small fraction is reflected back to the source.

Emission Test: Set Up and Results

To perform an Emission Test, right-click Results and select Create Report. For Report Type, select Emission Test (Fig. 4). It is not necessary to define a Radiation Near Field Setup in advance - the software will do this automatically for you.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

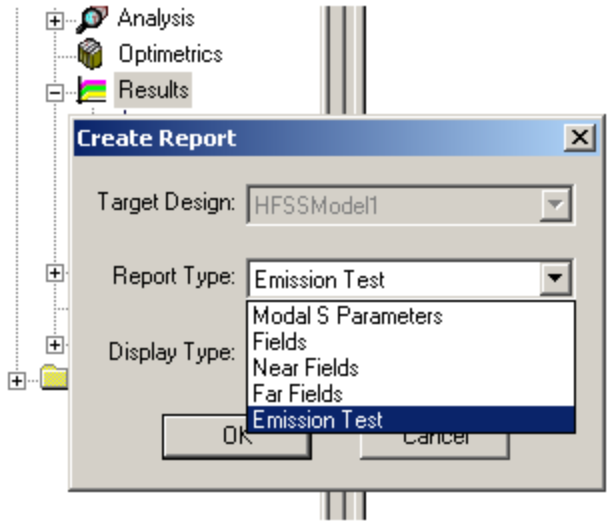


Fig. 4 Preparing an Emission Test

The simplest form of the Emission Test does not take the actual spectrum of your digital signal into account. This is the case for the first four spheres under Quantity (Fig. 5). For every frequency, the source excitation is the same power or voltage (depending on your model and your settings under Fields/Edit Sources), and the maximum |E| on the sphere is reported (Fig. 6).

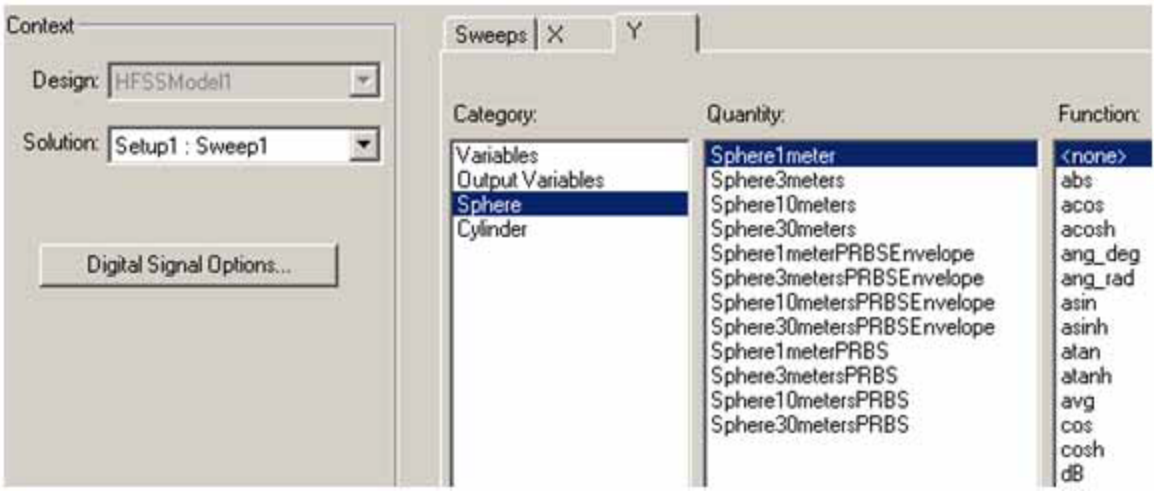


Fig. 5 Selecting an Emission Test at one meter distance from the origin, without scaling for the spectrum of the actual signal

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fig. 6 Result of the Emission Test of Fig. 5

In principle, the data in Fig. 6 can be compared with EMC regulations. However, in practical situations one wants to scale these data by taking the spectrum of the actual digital signal into account. To do this, specify your rise time and hold time under Digital Signal Options (Fig. 5) and select one of the spheres with PRBS in the name under Quantity (Fig. 5). PBRS stands for Pseudo-Random Binary Sequence.

In this example, we look at a digital signal with a rise time of 200 ps and a hold time of 800 ps (Fig. 7). We plot the maximum |E| on a one-meter sphere around the origin with two weighting functions: the exact one, which has many zeroes, and "envelope", which shows the envelope. Figure 8 shows the result.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

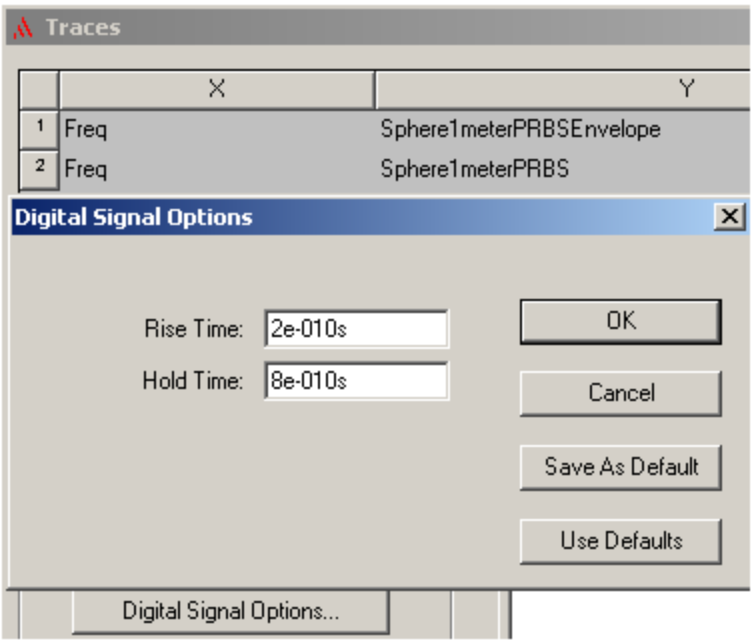


Fig. 7 Setting up an Emission Test with scaling for the spectrum of the digital signal

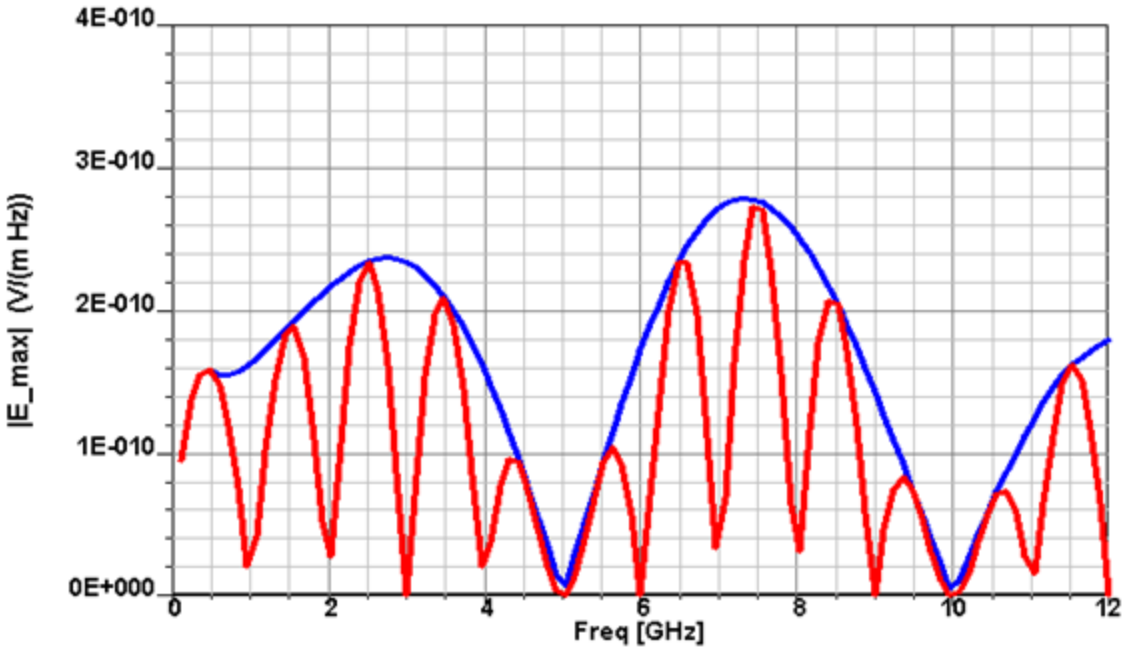


Fig. 8 Emission Test for a pseudo-random digital signal.
Red: exact scaling; blue: envelope scaling.

Scaling factor

The red line in figure 8 is the result of multiplying figure 6 by

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$2(T+T_r)\frac{\sin(\pi T_r f)\sin(\pi(T+T_r)f)}{(\pi T_r f)(\pi(T+T_r)f)} \quad (1)$$

where T is the hold time, T_r is the rise time, and *f* is the frequency. If figure 6 corresponds to a situation where the source produces an AC signal with an amplitude V_0 (peak-to-peak swing $2V_0$) at every frequency, then figure 8 corresponds to a situation where the source produces a PRBS with levels $\pm V_0$ (a swing of $2V_0$) and the given hold and rise times. Let's make a sanity check at 7.5 GHz. For our digital signal, at this frequency, the factor becomes

$$E-9)\frac{\sin(1.5\pi)\sin(7.5\pi)}{(1.5\pi)(7.5\pi)}s = \frac{2E-9}{11.25\pi^2}s = (1.8E-11)Hz^{-1} \quad (2)$$

Indeed, |Emax| in Fig. 6 is 16 V/m at 7.5 GHz, and in Fig. 8 the value is 280 pV/(m Hz).

Comparison with regulations

How does this level relate to EMC regulations? First of all, one has to take into account that EMC regulations specify a bandwidth of the measurement equipment of at least 100 kHz. The level of 2.8E-10 V/(m Hz) would register 28 µV/m when measured with equipment of which the bandwidth is 100 kHz (or a little over 100 kHz with roll-off at the edges). Typical MIL-STD EMC limits are on the order of several hundred µV/m at 1m. Hence, this simple, admittedly unrealistic example stays well within the limits. One reason is that its considerable power is spread out over a very broad range of frequencies. A regular clock pulse transmitted by this device would have its power concentrated in narrow bands and would certainly violate regulations. Another reason why the spectral density is low is that 7.5 GHz is well beyond the knee frequency of 0.35/Tr = 1.8 GHz.

Why is the envelope function provided?

Note that the red line in Fig. 8 has many zeroes in the frequency spectrum. There are two reasons for providing the envelope function.

1. If you would produce a frequency sweep and/or a plot with a small number of points, many points would be likely to lie in the valleys and you might not distinguish the peaks anymore. This would give you a false impression.
2. The red line is based on the assumption of bits with perfect trapezoidal shapes. In reality, your signal would not have been perfectly trapezoidal when it was generated, and may have suffered some additional distortions due to interactions in your device. This reduces the value of the fine structure in the spectrum.

Result for a regular clock pulse

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The spectrum of a repetitive clock signal is very different from the spectrum of a pseudo- random binary sequence: it consists of narrow spikes. The Emission Test in Fig. 6 can be used to produce the radiated spectrum of a clock pulse. For example, if we have a clock pulse with zero rise time and f_c=1GHz, we would take the points at 1 GHz, 3 GHz, 5 GHz, 7 GHz, etc., and multiply those fields by $2/\pi$, $2/(3\pi)$, $2/(5\pi)$, $2/(7\pi)$, etc. It would look like the following figure.

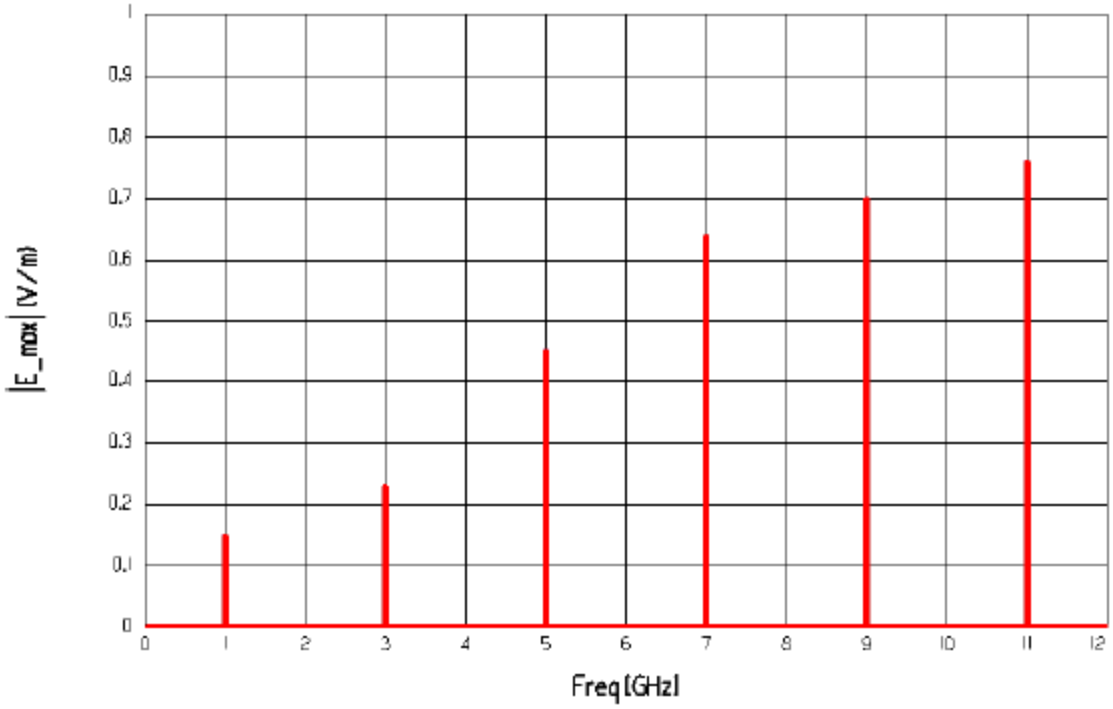


Fig. 9 Emission Test for a clock pulse with a clock frequency of 1 GHz

Whereas the PBRS satisfied strict military standards, the clock pulse violates them by several orders of magnitude. This is because the spectrum of a clock pulse is concentrated in narrow high spikes.

Mathematical Background: Spectra of Digital Signals

In the remainder of this note we derive the spectra of different digital signals of interest. These signals can be divided into two categories: the first is deterministic signals and the second is stochastic (random) signals.

Deterministic Signals

To characterize deterministic signals, we employ the Fourier transform. Let the input signal be $x(t)$, typically measured in Volts. We seek the Fourier transform $X(j\omega)$, typically measured in Volts/Hz. This is given by

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$X(j\omega) = \int_{-\infty}^{\infty} x(t)e^{-j\omega t}dt \quad (1)$$

Here $\omega=2\pi f$ is the angular frequency (radians/sec) and *f* is the frequency in Hz. Also, $j^2 = -1$. There is also an inverse Fourier transform defined by

$$x(t) = \frac{1}{2\pi}\int_{-\infty}^{\infty} X(j\omega)e^{j\omega t}d\omega \quad (2)$$

First let's consider a simple digital pulse with zero rise time. We will then extend the result to a pulse with finite rise and fall time, and then to the case of periodically repeating pulses.

Zero Rise Time Pulse

The input signal is held at the value 1 Volt for T seconds, and is zero at all other times. It looks like this:

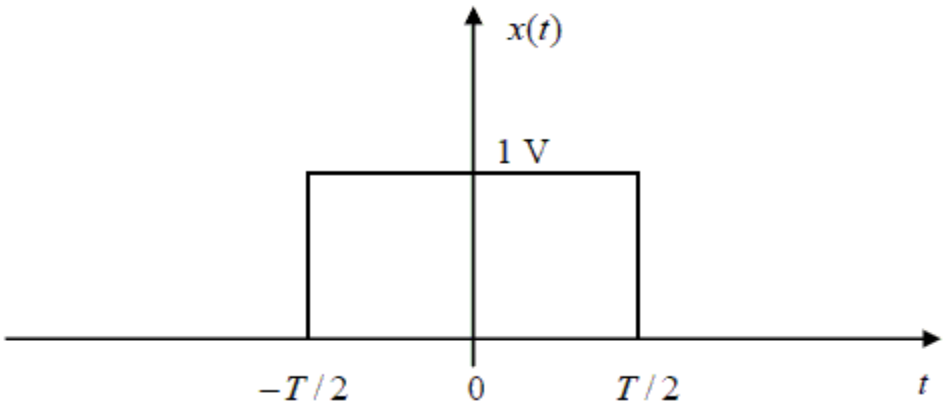


Fig. 10 A zero-rise time pulse

Applying the Fourier transform (1), we get

$$X(j\omega) = \int_{-T/2}^{T/2} 1 \cdot e^{-j\omega t}dt = \frac{e^{-j\omega T/2} - e^{j\omega T/2}}{-j\omega} = \frac{2\sin(\omega T/2)}{\omega} \quad (3)$$

Converting this to frequency units of Hertz yields

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$X(f) = T\frac{\sin(\pi fT)}{\pi fT} \quad (4)$$

The Fourier transform for the zero-rise time signal looks like this:

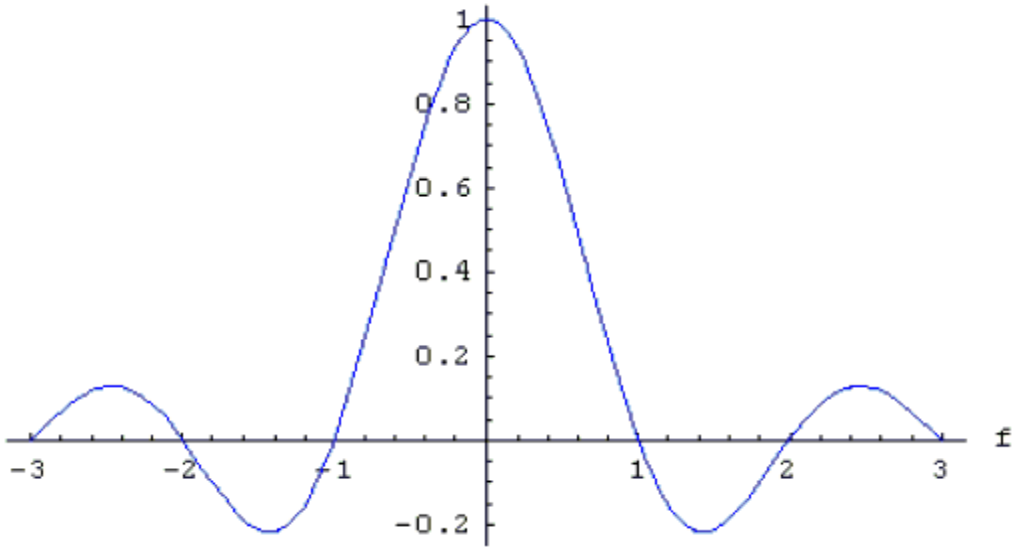
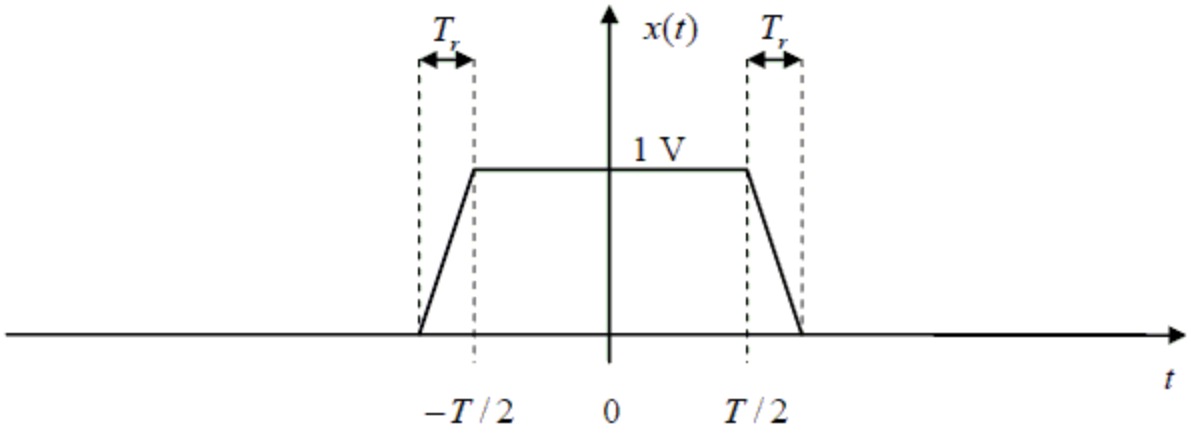


Fig. 11 Fourier transform of the zero-rise time pulse on a normalized scale

Notice that the maximum value of the function in (3) is *T*, attained at *f* = 0 . Evaluation of this function will require special care in the limit as *f*g0 to avoid dividing by zero.

Finite Rise Time Pulse

In this case, the input signal has the appearance shown in the following figure. The maximum value is 1 Volt. The total width of the pulse is $T + 2T_r$.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fig. 12 A finite rise-time pulse

The waveform may be expressed analytically as

(5)

$$x(t)=\begin{cases} 0 & , \quad t<-T/2-T_r \\ \dfrac{T/2+T_r+t}{T_r} & , \quad -T/2-T_r\le t<-T/2 \\ 1 & , \quad -T/2\le t<T/2 \\ \dfrac{T/2+T_r-t}{T_r} & , \quad T/2\le t<T/2+T_r \\ 0 & , \quad t\ge T/2+T_r \end{cases}$$

Now the Fourier transform becomes

(6)

$$X(j\omega)=\frac{1}{T_r}\int_{-T/2-T_r}^{-T/2}(\frac{T}{2}+T_r+t)e^{-j\omega t}dt+\int_{-T/2}^{T/2}e^{-j\omega t}dt+\frac{1}{T_r}\int_{T/2}^{T/2+T_r}(\frac{T}{2}+T_r-t)e^{-j\omega t}dt$$

After some manipulation this can be simplified to

(7)

$$X(j\omega)=\frac{4\sin\left(\frac{\omega T_r}{2}\right)\cdot\sin\left(\frac{\omega(T+T_r)}{2}\right)}{\omega^2 T_r}$$

If we convert this to frequency in Hertz, we have

(8)

$$X(f)=(T+T_r)\frac{\sin(\pi f T_r)\cdot\sin[\pi f(T+T_r)]}{(\pi f T_r)\pi f(T+T_r)}$$

The maximum value of this function is $T + T_r$, approached in the low-frequency limit. A normalized plot of this frequency response for $T = 0.6$, $T_r = 0.4$ is given below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

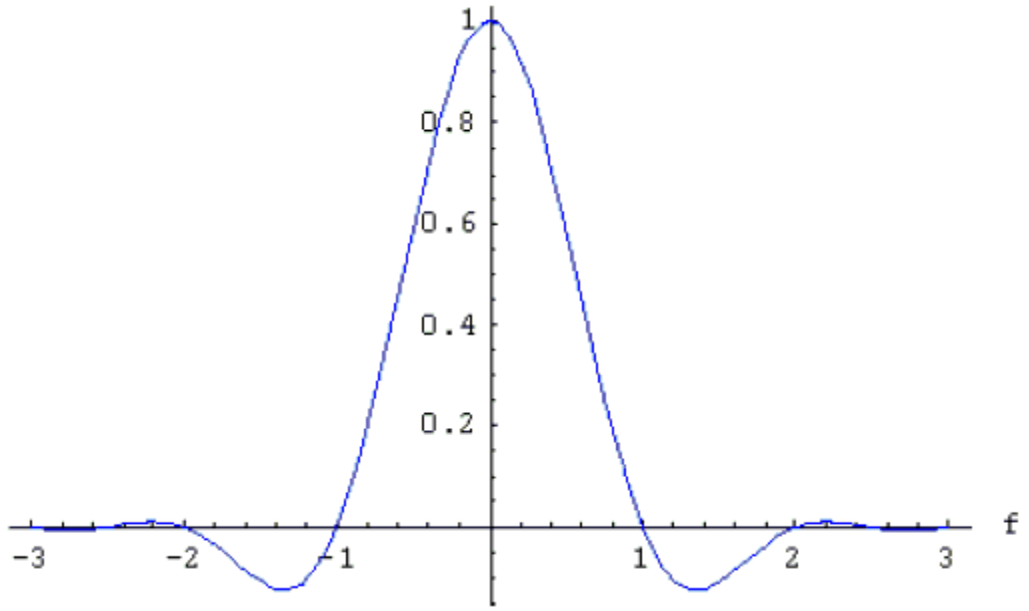


Fig. 13 Frequency response for a finite rise-time signal with T=0.6, T_r = 0.4

Notice that figures 11 and 13 look fairly similar. At higher frequencies, the amplitude of the frequency response falls off more quickly for the signal with finite rise time.

Periodically Repeating Pulse

Now we consider a digital pulse that repeats regularly with a period T_0. This signal is shown in figure 14 below.

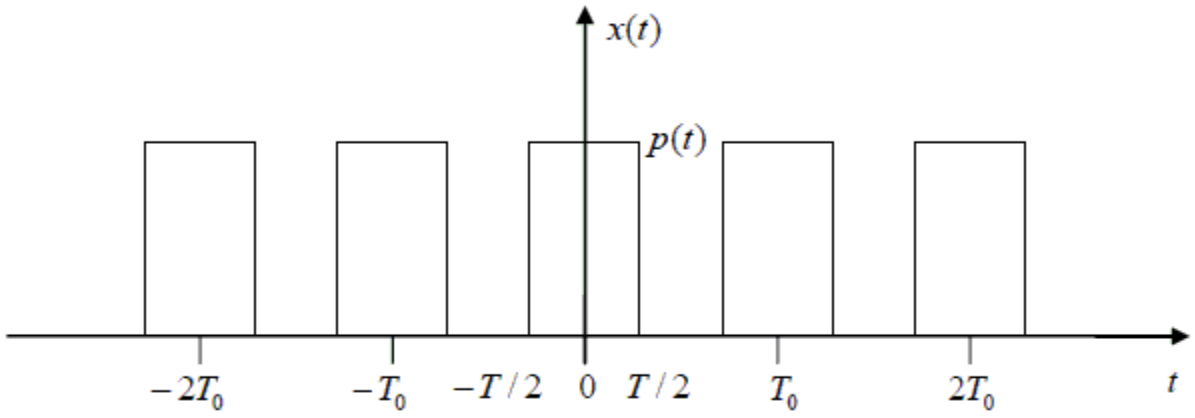


Fig. 14 Example of a periodically repeating digital pulse.
In this example, the basic pulse shape is a zero-rise time pulse.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The basic pulse shape for a single period is given by the function $p(t)$, which is nonzero only for $-T_0/2 < t \leq T_0/2$. The complete periodic signal can be expressed as an infinite summation of delayed pulses:

$$x(t) = \sum_{n=-\infty}^{\infty} p(t - nT_0) \tag{9}$$

An alternative representation for a periodic signal is its *Fourier* series

$$x(t) = \sum_{k=-\infty}^{\infty} a_k e^{-jk\omega_0 t} \tag{10}$$

Here $\omega_0 = 2\pi/T_0$ is the fundamental frequency of the periodic signal, and the complex Fourier series coefficients a_k can be computed with the following integral:

$$a_k = \frac{1}{T_0} \int_{-T_0/2}^{T_0/2} p(t) e^{-jk\omega_0 t} dt \tag{11}$$

The similarities of this integral to the Fourier transform in equation (1) are obvious. In the case of the Fourier series, the integral is carried out over a single period, while the Fourier transform integral in (1) is taken over all time. Another difference is that the Fourier series requires only the discrete set of frequencies $k\omega_0$, whereas the Fourier transform integral deals with a continuous spectrum of frequencies from.

$$-\infty < \omega < \infty$$

Another difference is the leading factor of $1/T_0$ in the Fourier series integral. This leads to a coefficient a_k with units of Volts, not Volts/Hz.

Let us compute the Fourier transform $P(j\omega)$ of the basic pulse shape function $p(t)$. Thanks to the time-limited nature of $p(t)$, this is:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$P(j\omega) = \int_{-\infty}^{\infty} p(t)e^{-j\omega t}dt = \int_{-T_0/2}^{T_0/2} p(t)e^{-j\omega t}dt \quad (12)$$

Comparing with equation (11), we see that

$$a_k = \frac{1}{T_0}P(jk\omega_0) \quad (13)$$

Thus we can rewrite the Fourier series (10) for the periodic signal as

$$x(t) = \sum_{k=-\infty}^{\infty} \frac{1}{T_0}P(jk\omega_0)e^{-j\omega t}dt \quad (14)$$

It is easy to show (by using the inverse Fourier transform in (2)) that the Fourier transform of the complex exponential signal $e^{jk\omega_0 t}$ is a Dirac delta function, $2\pi\delta(\omega\text{-}k\omega_0$. Using this fact, we can now find the Fourier transform of the repeating pulse signal in (14):

$$X(j\omega) = \sum_{k=-\infty}^{\infty} \frac{2\pi}{T_0}P(jk\omega_0)\delta(\omega - k\omega_0) = P(j\omega)\sum_{k=-\infty}^{\infty} \frac{2\pi}{T_0}\delta(\omega - k\omega_0) \quad (15)$$

If we convert this to frequency in Hertz, we get

$$X(f) = \frac{1}{T_0}P(f)\sum_{k=-\infty}^{\infty} \delta(f - kf_0) \quad (16)$$

Here $f_0=1/T_0$ is the fundamental frequency in Hz.

Thus we have shown that the Fourier transform of a repeating pulse is an infinite series of delta functions, scaled by $1/T_0$ times the Fourier transform of the basic pulse shape. Such a signal is impossible to plot, because the delta function is an infinitely high, infinitely narrow spike. It is, however, possible to plot the closely related Fourier series coefficients. For a zero-rise time pulse with $T = 1$ and a period $T_0 = 4$ we get the following plot:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

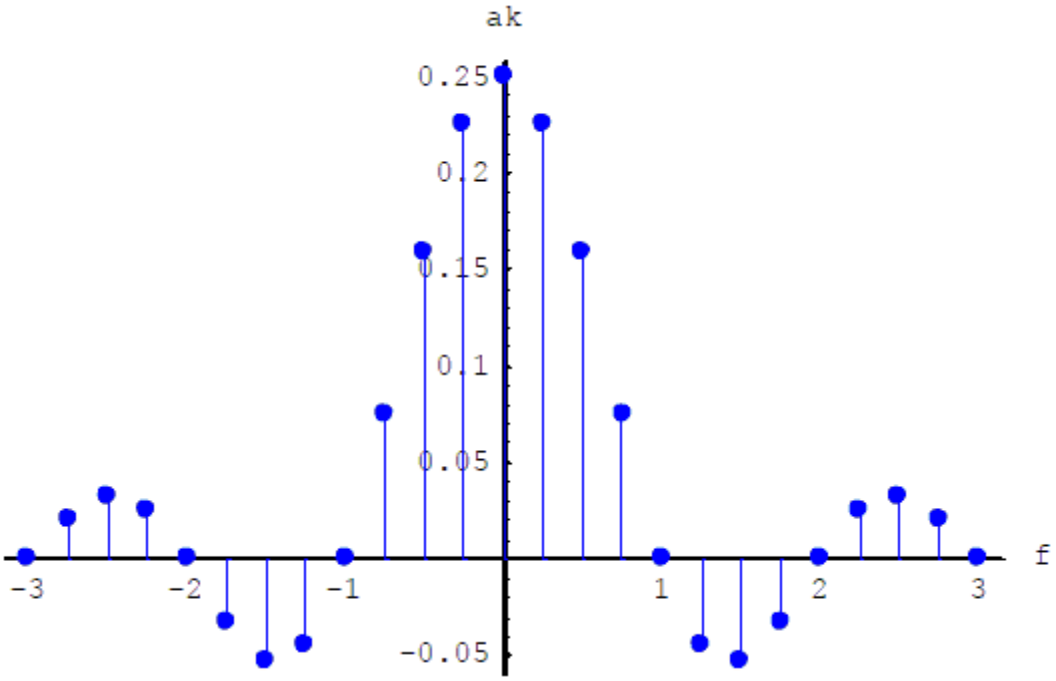


Fig. 15 The Fourier coefficients for a periodic repeating zero rise-time pulse Compare this with figure 11 for a single pulse.

Special case: clock pulse with $T_0=T_{clock}=1/f_{clock}$ and $T+T_r=T_0/2$

In this case we have from (16)

$$X(f) = \frac{1}{T_{clock}} P(f) \sum_{k=-\infty}^{\infty} \delta(f - kf_{clock})$$

and we substitute from (8)

$$P(f) = \left(\frac{T_{clock}}{2}\right) \frac{\sin(\pi f T_r) \cdot \sin[(\pi f T_{clock})/2]}{(\pi f T_r)(\pi f T_{clock}/?)}$$

This gives

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$X(f) = \frac{1}{2}\delta(f) + \sum_{k=1}^{\infty} \frac{\sin(\pi f T_r)}{(\pi f T_r)} \frac{\sin(\pi f T_{clock})}{(\pi f T_{clock})} \delta(f - k f_{clock})$$

The first term on the right-hand side, 1/2δ(*f*) , is the DC term which comes from k=0.

Indeed, in this case the average value of the signal is ½.

In the summation, by combining the positive and negative values of k, we implicitly limited the observations to positive frequencies and inserted a factor of two. See the practical note later in this document.

The product $\sin(\pi f T_{\text{clock}} / 2)\ \delta(f - k f_{clock})$ is zero for even values of k and is $\pm\delta(f - k f_{clock})$ for odd values of k. We therefore find

$$|X(f)| = \frac{1}{2}\delta(f) + \sum_{k = 1, 3, 5, 7, \ldots} \frac{\left|\sin(\pi f T_r)\right|}{(\pi f T_r)} \frac{1}{k\pi/2} \delta(f - k f_{clock})$$

The unit is V/Hz if the clock signal is a voltage. Thus, the spectrum consists of a DC term of 0.5 V plus AC terms of

$$\frac{2}{k\pi} \frac{\sin(\pi f T_r)}{\pi f T_r} \text{Volt}$$

at frequencies f=kfclock, k=1,3,5,7,…

If the rise time is zero, the sinc function is one and we simply have AC terms of at frequencies f=kfclock, k=1,3,5,7,… This is confirmed by Eric Bogatin in [1].

[1] Eric Bogatin, *Signal Integrity - Simplified*, ISBN 0-13-066946-6, published by Prentice Hall, Chapter 2: Time and Frequency Domains.

Using the Envelope Function for Radiated Field Computations

In any single-input linear time-invariant system, the Fourier transform *Y*(*j*ω) of the output is related to that of the input signal by a transfer function *H*(*j*ω):

$$Y(j\omega) = H(j\omega)X(j\omega) \quad (17)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If there are *N* inputs, with Fourier transforms $X_1(j\omega)$, $X_2(j\omega)$,...$X_N(j\omega)$, output is characterized by *N* different transfer functions:

$$Y(j\omega) = \sum_{k=1}^{N} H_k(j\omega)X_k(j\omega) \tag{18}$$

The input signals in an HFSS project are typically incident modal waves or terminal voltages. The output signals could be reflected waves (if we are dealing with S-parameters) or the vector components of near or far electromagnetic fields. HFSS computes the transfer functions from the inputs to the outputs of interest.

The Edit Sources function in the fields postprocessor takes care of forming the following linear combination:

$$Y(j\omega) = \sum_{k=1}^{N} H_k(j\omega)XC_k \tag{19}$$

where the C_k are complex-valued constants (specified by their magnitude and phase.)

If all of the input signals are deterministic digital signals *with the same shape* (aside from a constant scaling factor C_k), then we can write $X_k(j\omega)=C_kX_0(j\omega)$ where $X_0(j\omega)$ ισ the Fourier transform of the common signal shape. In this case, equation (18) reduces to

$$Y(j\omega) = X_0(j\omega)\sum_{k=1}^{N} H_k(j\omega)C_k \tag{20}$$

Therefore we can simply multiply the Edit Sources result in equation (19) by the Fourier transform $X_0(j\omega)$ of the (common) digital signal shape. If, at some future point, we want to consider inputs that carry *different* digital signals, then modifications would be needed to Edit Sources to implement equation (18) in its full generality.

If we are dealing only with the magnitude of the response $Y(j\omega)$, then we have

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$|Y(j\omega)| = X_0(j\omega)\left|\sum_{k=1}^{N} H_k(j\omega)C_k\right| \quad (21)$$

A Note about the Signal Scale Factors

Strictly speaking, only real-valued (positive or negative) signal scale factors C_k should be allowed when we are dealing with digital signals. If an arbitrary complex number is used, then the resulting scaled time-domain signal will not be real-valued. This is non- physical.

It would be more reasonable to interpret the phase as being the phase lag (modulo 2π) *at a certain frequency*; then an equivalent delay T_d could be computed. This could be used to compute a frequency dependent scale factor $\exp(-j\omega T_d)$ for the input signal.

A Practical Note about Measurement Equipment

Most frequency domain measurement equipment (spectrum analyzers and vector network analyzers) will display signal amplitude, not the Fourier transform amplitude. If the input signal is a cosine function with amplitude 1, its Fourier transform is

$$x(t) = \cos(2\pi f_0 t) = \frac{1}{2}e^{j2\pi f_0 t} + \frac{1}{2}e^{-j2\pi f_0 t} \rightarrow X(f) = \frac{1}{2}\delta(f - f_0) + \frac{1}{2}\delta(f + f_0) \quad (22)$$

This would suggest that a spike of amplitude ½ should appear at the frequency f_0. Most measurement equipment is calibrated with exactly this sort of reference signal, but usually the persons performing the calibration adjust the equipment to display a value of 1 as the output. Therefore, if one wishes to duplicate what is shown on the measurement equipment, the Fourier transform of the output signal should be multiplied by a factor of 2.

Stochastic Signals: Random Bit Streams

Now we consider random bit streams. The signal is a continuous function of time, and consists of pulses with a fixed basic pulse shape $p(t)$. Each delayed pulse is modulated by a random variable $\mathbf{x}_n$. We represent the complete random bit stream as

$$\mathbf{x}(t) = \sum_{n=-\infty}^{\infty} \mathbf{x}_n p(t - nT_0) \quad (23)$$

For a digital random process, the discrete random variables $\mathbf{x}_n$ can take on only two distinct real values, representing a logic 1 or 0. We make some assumptions about the statistics of these

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

discrete random variables. First, we assume that the mean (expected) value of $\mathbf{x}_n$ is independent of time:

$$E\{\mathbf{x}_n\} = E\{\mathbf{x}_m\} \qquad \text{for all } m, n \quad (24)$$

Here $E(.)$ denotes the expected value of a random variable. Secondly, we assume that the discrete *autocorrelation* function of the bit values $E\{\mathbf{x}_m\mathbf{x}_n\}$ depends only on the time difference m-n, and not upon the absolute values of m and n :

$$E\{\mathbf{x}_m\mathbf{x}_n\} = R_{m-n} \quad (25)$$

To use the formal language of random processes, we are assuming that the discrete random sequence $\{\mathbf{x}_n\}$ is *wide-sense stationary*.

For a real-valued random process $\mathbf{x}(t)$ that is a function of a continuous time variable t, we define the *autocorrelation function* $R_{xx}(t, \tau)$ as

$$R_{xx}(t, \tau) = E\{\mathbf{x}(t)\mathbf{x}(t+\tau)\} \quad (26)$$

In this note, we are dealing with random processes of the form shown in equation (23). Such a process is *cyclostationary*. In this case, the autocorrelation is a periodic function of time:

$$R_{xx}(t + nT_0, \tau) = R_{xx}(t, \tau) \quad (27)$$

For a cyclostationary signal, we can introduce a time-averaged autocorrelation, where the time averaging is over a single period. In this case the autocorrelation depends only upon the time shift τ:

$$R_{xx}(\tau) = \frac{1}{T_0}\int_{-T_0/2}^{T_0/2} E\{\mathbf{x}(t)\mathbf{x}(t+\tau)\}\,dt \quad (28)$$

The *power spectral density* (or PSD) of a random process is the Fourier transform of its autocorrelation function:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$S_{xx}(j\omega) = \int_{-\infty}^{\infty} R_{xx}(\tau) e^{-j\omega\tau} d\tau \quad (29)$$

Typically we use the power spectral density to characterize random processes in the frequency domain.

Deterministic Signals with Finite Energy

The autocorrelation and PSD are also defined for deterministic signals, but the definition of the autocorrelation is somewhat different. For a real-valued deterministic signal $x(t)$ with finite energy (i.e., non-periodic), we define the autocorrelation as

$$R_{xx}(\tau) = \int_{-\infty}^{\infty} x(t)x(t+\tau)dt \quad (30)$$

The PSD is still defined as the Fourier transform of the autocorrelation. We can show that the PSD is the magnitude squared of the Fourier transform X j of the deterministic signal:

$$S_{xx}(j\omega) = |X(j\omega)|^2 \quad (31)$$

Deterministic Periodic Signals

For a deterministic signal that is periodic and real-valued, the autocorrelation is defined using an integral over a single period:

$$R_{xx}(\tau) = \frac{1}{T_0} \int_{-T_0/2}^{T_0/2} x(t)x(t+\tau)dt \quad (32)$$

(Notice the similarity to equation (28), which applies to cyclostationary random signals.) If $x(t)$ has the Fourier series coefficients a_k, then we can show that its autocorrelation function is also periodic, and has a Fourier series with the coefficients $|a_k|^2$.

Here again the PSD is the Fourier transform of the autocorrelation. Because the autocorrelation is periodic, the PSD is a set of impulses in frequency:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$S_{xx}(j\omega) = \sum_{k=-\infty}^{\infty} 2\pi |a_k|^2 \delta(\omega - k\omega_0) \tag{33}$$

or in units of Hz,

$$S_{xx}(f) = |a_k|^2 \delta(f - kf_0) \tag{34}$$

Finding the Power Spectrum of a Random Bit Stream

We are now ready to attempt to find the power spectral density (PSD) of the random bit stream in equation (23). The procedure will be to compute its autocorrelation $R_{xx}(\tau)$ using equation (28), and then to take the Fourier transform of this to get the PSD.

Substituting (23) into (28), we find that the autocorrelation is

$$R_{xx}(\tau) = \frac{1}{T_0} \int_{-T_0/2}^{T_0/2} E\{\mathbf{x}(t)\mathbf{x}(t+\tau)\}\,dt \tag{35}$$

$$= \sum_{m=-\infty}^{\infty} \sum_{n=-\infty}^{\infty} E\{x_m x_n\} \frac{1}{T_0} \int_{-T_0/2}^{T_0/2} p(t - mT_0)p(t + \tau - nT_o)\,dt$$

$$= \sum_{m=-\infty}^{\infty} \sum_{n=-\infty}^{\infty} R_{n-m} \frac{1}{T_0} \int_{-T_0/2}^{T_0/2} p(t - mT_0)p(t + \tau - nT_o)\,dt$$

Here again we are using $R_k = E\{\mathbf{x}_n \mathbf{x}_{n+k}$ to denote the autocorrelation of the discrete random bit values xn . Now let us define $k = n - m$ and eliminate n from the above expression. We rewrite the summation as

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$R_{xx}(\tau) = \sum_{k=-\infty}^{\infty} R_k \sum_{m=-\infty}^{\infty} \frac{1}{T_0} \int_{T_0/2}^{T_0/2} p(t - mT_0)p(t + \tau - (m+k)T_0)dt \tag{36}$$

$$= \sum_{k=-\infty}^{\infty} R_k \sum_{m=-\infty}^{\infty} \frac{1}{T_0} \int_{T_0/2}^{T_0/2} p(u)p(u + \tau - kT_0)dt$$

where we have introduced $u = t - mT_0$. Notice now that the integrals being summed up over non-overlapping intervals of length T_0, when taken together, actually constitute a single integral over the entire time axis. Thus we can write

$$R_{xx}(\tau) = \sum_{k=-\infty}^{\infty} R_k \frac{1}{T_0} \int_{-\infty}^{\infty} p(u)p(u + \tau - kT_0)dt \tag{37}$$

$$= \frac{1}{T_0} \sum_{k=-\infty}^{\infty} R_k R_{pp}(\tau - kT_0)$$

where $R_{pp}(\tau)$ is the autocorrelation of the basic pulse shape $p(t)$:

$$R_{pp}(\tau) = \int_{-\infty}^{\infty} p(t)p(t + \tau)dt \tag{38}$$

To get the PSD of the random bit stream, we now need to take the Fourier transform of the autocorrelation in (37). Making use of standard Fourier transform properties, this is

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$S_{xx}(j\omega) = \frac{1}{T_0}|P(j\omega)|^2 \sum_{k=-\infty}^{\infty} R_k e^{-jk\omega T_0} \quad (39)$$

Here $P(j\omega)$ is the Fourier transform of the basic pulse shape; thus $|P(j\omega)|^2$ is the PSD of the basic pulse.

The power spectrum expression in (39) is fairly general. We can now consider some important special cases.

A Random Bit Stream with Bipolar Modulation

In this case, the random variables $\mathbf{x}_n$ take on the values +1 or -1 with equal probability (1/2), and we assume that the values of any two different bits are independent of one another. The signal is illustrated below in figure 16.

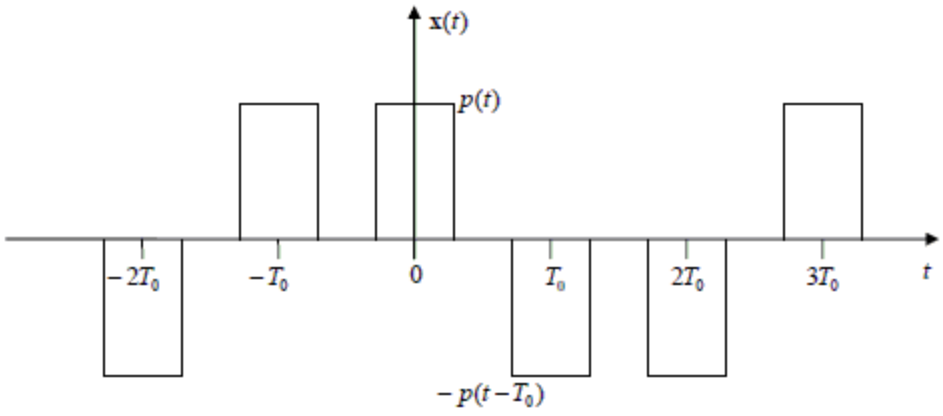


Fig. 16 A sample of an ideal random digital bit stream with bipolar modulation It is modulated by a sequence of random variables that assume the values +1 or -1 in each interval.

Thanks to the assumed statistics, we have

$$R_k = E\{\mathbf{x}_n \mathbf{x}_{n+k}\} = \begin{cases} 1, & k = 0 \\ 0, & k \neq 0 \end{cases} \quad (40)$$

Therefore (39) immediately simplifies to

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$S_{xx}(j\omega) = \frac{1}{T_0}|P(j\omega)|^2 \quad (41)$$

This derivation does not depend upon the particular pulse shape $p(t)$. Therefore it applies equally well to zero rise time pulses and to finite rise time pulses. An illustration for a zero-rise time pulse ($T = 1$) and a pulse period $T_0 = 4$ is shown below in figure 17:

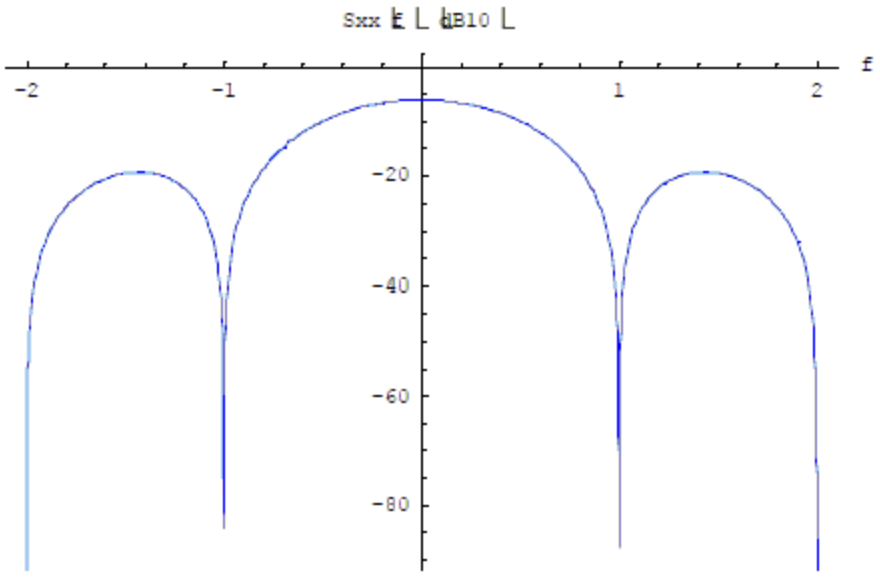


Fig. 17 Power spectral density for a random bit stream with bipolar modulation and a pulse shape corresponding to a zero rise time pulse. A logarithmic dB10 scale has been used for clarity.

Special case with bipolar modulation

Let's consider the special case where $T+2T_r=T_0$. This means that the signal x(t) in figure 16 never stays at zero for any length of time. From (41) we have

$$S_{xx}(j\omega) = \frac{1}{T_0}|P(j\omega)|^2$$

and from (8) we have

$$P(f) = (T + T_r)\frac{\sin(\pi f T_r) \cdot \sin[\pi f(T + T_r)]}{(\pi f T_r)\pi f(T + T_r)}$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This leads to

$$S_{xx}(f) = \frac{(T+T_r)^2}{T+2T_r}\frac{\sin^2(\pi f T_r)}{(\pi f T_r)^2}\frac{\sin^2[\pi f(T+T_r)]}{(\pi f(T+T_r))^2}$$

For zero rise time, this equation simplifies considerably:

$$S_{xx}(f) = T\frac{\sin^2(\pi f T)}{(\pi f T)^2}$$

The field- or voltage based spectral density is simply given by (8)

$$P(f) = (T+T_r)\frac{\sin(\pi f T_r)\cdot\sin[\pi f(T+T_r)]}{(\pi f T_r)\pi f(T+T_r)}$$

which for zero rise time simplifies to

$$P(f) = T\frac{\sin(\pi f T)}{\pi f T}$$

To see that P(f) indeed accounts correctly for the field- or voltage based spectral density, consider the following situation. The signal x(t) is a voltage that is applied to a one-Ohm resistor. If we have a sequence of N bits, -1V or 1V, with zero rise time, and with a total duration of NT seconds, how much **energy** would this signal dissipate in the 1-Ω resistor?

The energy deposited in the resistor will be

(VI in the frequency domain) × number_of_bits.

Both V and I will now be given by P(f), one being in V/Hz and the other in A/Hz.

$$T^2\frac{\sin^2(\pi f T)}{(\pi f T)^2}\times N$$

This results in a power dissipation (energy/time = energy/(NT)) of

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$T\frac{\sin^2(\pi fT)}{(\pi fT)^2} \qquad W/(Hz)$$

in which we recognize the expression for $S_{xx}(f)$.

At this stage, both positive and negative frequencies are allowed. Because of the practical note elsewhere in this document we can limit ourselves to positive frequencies, add a factor of two, and state that the voltage- or field based spectral density is given by

$$P(f) = 2(T+T_r)\frac{\sin(\pi fT_r)\cdot\sin[\pi f(T+T_r)]}{(\pi fT_r)(\pi f(T+T_r))} \qquad f>0$$

A Random Bit Stream with Unipolar Modulation

Now we will consider the case where the modulating random process $\mathbf{x}_n$ takes on the values +1 or 0 (with equal probability of 1/2) in each interval. We refer to this as unipolar modulation. The signal is illustrated below:

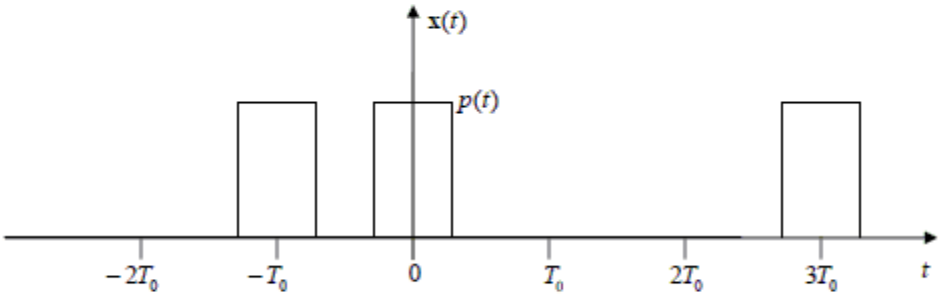


Fig. 18 Random bit stream with unipolar modulation

Again we assume that the values of two different bits are independent. Here the autocorrelation function values are different. For $m = n$, the two possible values of $\mathbf{x}_m\mathbf{x}_n$ are 1 and 0, and each occurs with probability ½. For mm n, the possible values of $\mathbf{x}_m\mathbf{x}_n$ are also 1 and 0, but because the two random variables are independent, a value of 1 occurs only ¼ of the time. Thus we have

$$R_k = E\{\mathbf{x}_n\mathbf{x}_{n+k}\} = \begin{cases}(1/2), & k = 0\\ (1/4), & k \neq 0\end{cases} \qquad (42)$$

Inserting this into (39) gives

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(43)

$$S_{xx}(j\omega) = \frac{1}{T_0}|P(j\omega)|^2\left[\frac{1}{2} + \sum_{\substack{k=-\infty \\ k\neq 0}}^{\infty} e^{-jk\omega T_0}\right]$$

$$= \frac{1}{T_0}|P(j\omega)|^2\left[\frac{1}{4} + \frac{1}{4}\sum_{k=-\infty}^{\infty} e^{-jk\omega T_0}\right]$$

The infinite sum can be recognized as a Fourier series with the coefficients a_k =1. Thus it represents a periodic signal. As we show in the appendix, the signal with this particular set of Fourier series coefficients is an infinite train of impulses. Replacing the Fourier series with the periodic impulse train, we obtain

(44)

$$S_{xx}(j\omega) = \frac{1}{T_0}|P(j\omega)|^2\left\{\frac{1}{4} + \frac{1}{4}\sum_{k=-\infty}^{\infty}\frac{2\pi}{T_0}\delta(\omega - k\varpi_0)\right\}$$

If we prefer to work in frequency units of Hz, this becomes

(45)

$$S_{xx}(f) = \frac{1}{4T_0}|P(f)|^2\left[1 + \frac{1}{T_0}\sum_{n=-\infty}^{\infty}\delta(f - nf_0)\right]$$

Once again, the derivation applies to any pulse shape, with zero or finite rise time. The above result for a unipolar bit stream is distinctly different from the result for a bipolar stream, thanks to the train of impulses represented by the second term. The power spectrum contains a continuous part, exactly ¼ as large as the spectrum of a bipolar signal, plus a set of narrow spikes at discrete frequencies (multiples of 1/ T_0). It can be shown that one half of the total signal power is in the discrete part of the spectrum, and one half is in the continuous part.

Obtaining a plot of this function is difficult, because it contains both infinite impulses and a finite continuous part. If we assume that the frequency response is run through a detector with finite frequency sensitivity Δf, then the apparent amplitude of the spikes will be finite and scaled by the factor $1/\Delta f$. The amplitude of the continuous part of the spectrum will be almost unaffected.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Taking, for example, $T_0 = 4$, a zero rise time pulse shape with $T = 1$, and $\Delta f = 10^{-4}$, we arrive at the following plot:

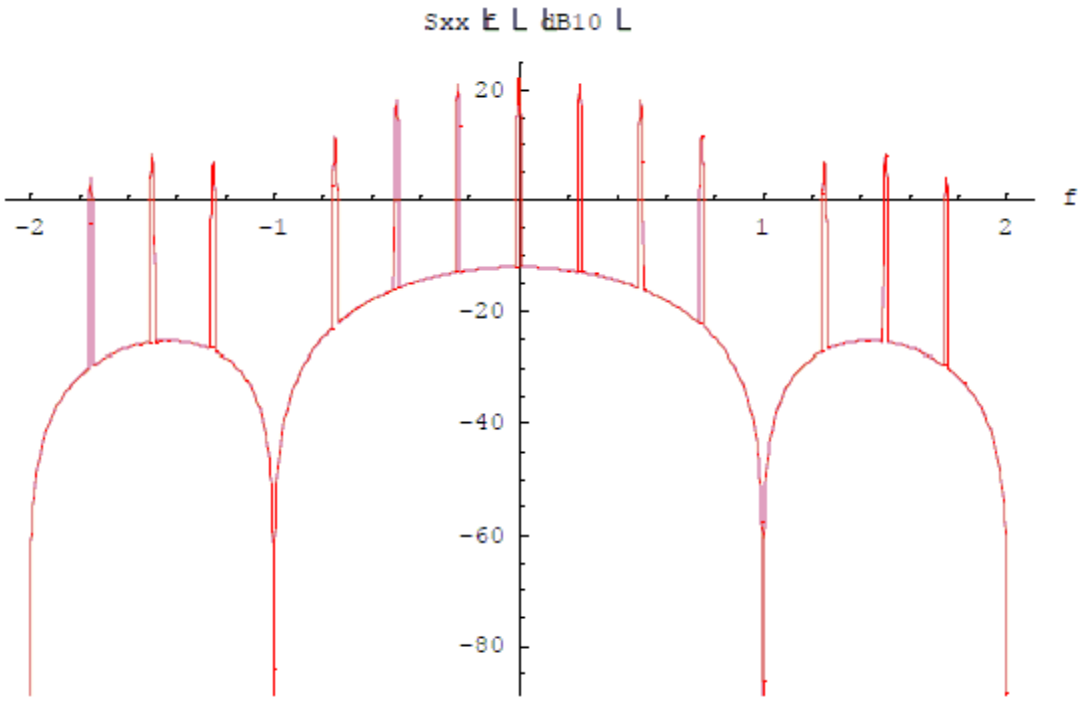


Fig. 19 Power spectrum for the random bit stream with unipolar modulation, showing the periodic spikes. A dB10 logarithmic scale has been used for clarity.

Special case with unipolar modulation

Let's consider the special case where T+Tr=T0. This means that the random signal spends equal amounts of time at level zero and at level one. We repeat (45) for convenience:

$$S_{xx}(f) = \frac{1}{4T_0}|P(f)|^2\left[1 + \frac{1}{T_0}\sum_{n=-\infty}^{\infty}\delta(f - nf_0)\right]$$

In this equation P(f) is given by (8)

$$P(f) = (T + T_r)\frac{\sin(\pi f T_r)\cdot \sin[\pi f(T + T_r)]}{(\pi f T_r)\pi f(T + T_r)}$$

which can now be written as

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$P(f) = T_0 \frac{\sin(\pi f T_r) \cdot \sin[\pi f T_0]}{(\pi f T_r)(\pi f T_0)}$$

This leads to

$$S_{xx}(f) = \frac{T_0}{4} \frac{\sin^2(\pi f T_r)}{(\pi f T_r)^2} \frac{\sin^2(\pi f T_0)}{(\pi f T_0)^2} \left[1 + \frac{1}{T_0} \delta(f) + \frac{1}{T_0} \sum_{n \neq 0} \delta(f - n f_0) \right]$$

Note that, for *n*m0,

$$\frac{\sin(n \pi f_0 T_0)}{(n \pi f_0 T_0)} = 0$$

so the summation with delta functions over nonzero values of *n* disappears. What remains is

$$S_{xx}(f) = \frac{1}{4} \delta(f) \frac{T_0}{4} \frac{\sin^2(\pi f T_r)}{(\pi f T_r)^2} \frac{\sin^2(\pi f T_0)}{(\pi f T_0)^2}$$

This can be recognized as a DC term of 0.25 W plus a spectrum for all nonzero frequencies. The DC term of 0.25 W makes sense since the average value of the signal x(t) is 0.5 V in this case.

The field- or voltage based spectral density is simply given by (8)

$$P(f) = \frac{T_0}{2} \frac{\sin(\pi f T_r) \cdot \sin[\pi f T_0]}{(\pi f T_r)(\pi f T_0)}$$

which for zero rise time simplifies to

$$P(f) = \frac{T_0}{2} \frac{\sin(\pi f T_0)}{\pi f T_0}$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To see that P(f) indeed accounts correctly for the field- or voltage based spectral density, consider the following situation. The signal x(t) is a voltage that is applied to a one-Ohm resistor. If we have a sequence of N bits, 0V or 1V, with zero rise time, and with a total duration of NT seconds, how much energy would this signal dissipate in a 1-Ω resistor?

We can expect a DC level of 0.5V with a DC current of 0.5A, plus bits with the given spectrum for an additional voltage swing of ±0.5V with a corresponding additional current of ±0.5A. If there are N bits for a total duration of the stream of NT seconds, the energy deposited in the resistor will be

DCterm × time + ACterm × number_of_bits

$$\frac{1}{4}NT + \frac{1}{4}T^2\frac{\sin^2(\pi fT)}{(\pi fT)^2} \times N$$

This results in a power dissipation (energy/time = energy/(NT)) of 0.25 W for the DC term plus a power dissipation of

$$\frac{T}{4}\frac{\sin^2(\pi fT)}{(\pi fT)^2} \qquad W/(Hz)$$

to account for the pseudo-random voltage swing of ±0.5 V around the DC level. Here, we recognize the result we had earlier.

This expression corresponds to the expression given for power spectral density by Clayton Paul [2].

[2] Clayton R. Paul, "Introduction to Electromagnetic Compatibility", ISBN 0-471- 54927-4, published by John Wiley and Sons, 1992, section 7.6: Signal Spectra, Representation of Random Signals.

Finally, because of the practical observation elsewhere in this note, we can limit ourselves to non-negative frequencies and include a factor of two:

$$P(f) = T_0\frac{\sin(\pi fT_r)\cdot\sin[\pi fT_0]}{(\pi fT_r)(\pi fT_0)} \qquad f>0$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using the Power Spectrum for Radiated Field Computations

Suppose we have a single-input linear time-invariant system, and let the input be a random process with power spectral density $S_{xx}(j\omega)$. Let the transfer function of the linear system be $H(j\omega)$. Then we can show that the power spectrum $S_{yy}(j\omega)$ of the output is related to that of the input signal as follows:

$$S_{yy}(j\omega) = |H(j\omega)|^2 S_{xx}(j\omega) \tag{46}$$

If there are N inputs to the system, then it becomes considerably more challenging to compute the power spectrum of the output. In general, the input signals may be correlated to one another. Then the output power spectrum is given by

$$S_{yy}(j\omega) = \sum_{m=1}^{N} \sum_{n=1}^{N} H_m^*(j\omega) S_{x_m x_n}(j\omega) H_n(j\omega) \tag{47}$$

Here x^* denotes the complex conjugate of x, and $S_{x_m x_n}(j\omega)$ is the *cross-power spectral density* of the two random signals $\mathbf{x}_m(t)$ and $\mathbf{x}_n(t)$. This is the Fourier transform of the *cross-correlation* function $R_{x_m x_n}$. For our cyclostationary, real-valued random digital signals the cross-correlation is defined as

$$R_{x_m x_n}(\tau) = \frac{1}{T_0} \int_{-T_0/2}^{T_0/2} E\{\mathbf{x}_m(t)\mathbf{x}_n(t+\tau)\}\, dt \tag{48}$$

Another way of expressing (47) is as a matrix inner product:

$$S_{yy}(j\omega) = \begin{bmatrix} H_1(j\omega) & \cdots & H_N(j\omega) \end{bmatrix}^* \begin{bmatrix} S_{x_1x_2}(j\omega) & \cdots & S_{x_1x_N}(j\omega) \\ \vdots & \ddots & \vdots \\ S_{x_Nx_1}(j\omega) & \cdots & S_{x_Nx_N}(j\omega) \end{bmatrix} \begin{bmatrix} H_1(j\omega) \\ \vdots \\ H_N(j\omega) \end{bmatrix} \tag{49}$$

Bipolar Modulation (Statistically Independent Signals)

If the input signals are all *uncorrelated* then (by definition) $R_{x_m x_n}(\tau)=0$ and $S_{x_m x_n}(j\omega)=0$ for all $m \neq n$. Then the expression for the output power spectrum will simplify to

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(50)

$$S_{yy}(j\omega) = \sum_{n=1}^{N} |H_n(j\omega)|^2 S_{x_n x_n}(j\omega) \qquad \text{(uncorrolated input signals)}$$

Equivalently, we could say that the power spectrum matrix in equation (49) becomes diagonal in this case.

The input signals will be uncorrelated if they are statistically independent of one another and we are using bipolar modulation. This is a result of the fact that the expected value of the product of any two bits from different signals is 0.

Unipolar Modulation (Statistically Independent Signals)

If we use unipolar modulation, then the input signals will be correlated, even if the bit patterns for two random digital bit streams are statistically independent of one another. For unipolar modulation, we can show that a cross-correlation between independent bit streams of $R^k = 1/4$ (for all k) will result. This will produce the cross power spectral density $Sx_m x_n(j\omega)$.

(51)

$$S_{x_m x_n}(j\omega) = \frac{1}{4T_0}|P(j\omega)|^2 \left[\frac{2\pi}{T_0}\sum_{n=-\infty}^{\infty} \delta(\omega - n\omega_0)\right]$$

The derivation of this result is very similar to that for equation (44). Here we have assumed that the two bit streams have the same basic pulse shape. With this result in hand, the power spectrum of the output signal can be computed using equation (47). We can simplify the expression to

(52)

$$S_{yy}(f) = \frac{1}{4T_0}|P(f)|^2 \left[A + \frac{B}{T_0}\sum_{n=-\infty}^{\infty} \delta(f - nf_0)\right]$$

In the above expression

(53)

$$A = \sum_{n=1}^{N} |H_n(j\omega)|^2 \qquad \text{and}$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$B = A + 2\sum_{m=1}^{N}\sum_{n=m+1}^{N} \mathrm{Re}[H_m^*(j\omega)H_n(j\omega)]$$

Inputs that Are Not Independent: Differential Signals

Differential signals represent an important special case where two input signals are not at all independent of one another. In the case of bipolar modulation, we can simply write one member of the pair as the negative of the other. The cross-correlation between bits will be -1 for $k = 0$ and 0 for k m 0. Therefore the cross-power spectral density for the pair will be nonzero. We can show that it will be equal to the negative of equation (41).

Suppose that the two signals of the differential pair are the only inputs to the system and bipolar modulation is used. Then we can use equation (49) to show

(54)

$$S_{yy}(j\omega) = \frac{1}{T_0}|P(j\omega)|^2[H_1(j\omega)H_2(j\omega)]^*\begin{bmatrix}1 & -1\\ -1 & 1\end{bmatrix}\begin{bmatrix}H_1(j\omega)\\ H_2(j\varpi)\end{bmatrix}$$

$$= \frac{1}{T_0}|P(j\omega)|^2\left\{|H_1(j\omega)|^2 - 2\mathrm{Re}[H_1^*(j\omega)H_2(j\omega)] + |H_2(j\omega)|^2\right\}$$

For a typical differential pair, the two transfer functions will be nearly equal, and so this expression shows that the output power will be substantially reduced when compared to single-ended signaling.

If we use unipolar modulation for our differential pair, then a bit of value $\mathbf{x}_m$ (0 or 1) in one signal becomes 1- $\mathbf{x}_m$ (1 or 0, respectively) in the other signal. For two bits in the same period (k =0) the product is always zero, so $R_k = 0$ for k =0. For two bits in different periods the expected value is ¼ (just as it was for independent unipolar signals), so R_k = 1/ 4 for km0. Plugging this into (48) and simplifying will result in

(55)

$$S_{x_1x_2}(j\omega) = \frac{1}{T_0}|P(j\omega)|^2\left\{-\frac{1}{4} + \frac{1}{4}\sum_{k=-\infty}^{\infty}\frac{2\pi}{T_0}\delta(\omega - k\omega_0)\right\}$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thus the impulse train part of the cross power spectrum is unchanged (as compared to equation (51) for independent unipolar signals), but now a continuous component has been added. The continuous component of the cross power spectrum is the negative of the continuous component of a self-power spectrum (compare equation (44)).

If we now suppose that the signals of the differential pair are the only inputs to the system, then the output power spectrum has the same form as equation (52), but now the coefficients will be

$$A = \left|H_1(j\omega)\right|^2 - 2\mathrm{Re}[H_1{}^*(j\omega)H_2(j\omega)] + \left|H_2(j\omega)\right|^2 \tag{56}$$

$$B = \left|H_1(j\omega)\right|^2 + 2\mathrm{Re}[H_1{}^*(j\omega)H_2(j\omega)] + \left|H_2(j\omega)\right|^2 \tag{57}$$

Conclusions and Discussion

This note has dealt with the output responses of linear systems with deterministic (finite energy as well as periodic) and stochastic (random) inputs.

When the signals are deterministic, and have the same time-domain waveforms (apart from constant scaling factors) it is fairly straightforward to consider multiple inputs and compute the Fourier transform of the output signal. The Fourier transform could be a continuous function of frequency (for finite energy signals) or a set of sharp lines (for periodic signals.)

If the signals are random, the process of combining multiple inputs is more complex. In addition to the power spectrum for each input, we must also know the cross power spectrum for every pair of inputs. The cross power spectrum depends on whether the inputs are statistically independent or not. We saw that differential signals are an important special case where signals are strongly dependent on one another. Despite all of these complications, it is still possible to compute the output signal power, and the procedure is sufficiently mechanical for it to be reduced to a computer program.

In this note we have focused our attention on random binary-valued logic signals. These are the sorts of signals we expect to see on the internal buses of a digital system. There are other random signal types (not considered here) that could be of interest. One class that is worthy of mention is digital communication signals, such as PAM, CPM, CPFSK, QPSK, etc.

Appendix A: Handy Delta Function Facts

Definition of the Dirac Delta Function

The Dirac delta function $\delta(x)$ is defined by the following relations:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. $\delta(x) = 0 \quad \text{for} \quad x \neq 0$
2. $\delta(x) \quad \text{is undefined (infinite) at} \quad x = 0$
3. $$\int_{-\infty}^{\infty} g(x)\delta(x)dx = g(0)$$

Delayed Delta Functions

Using Property 3 above, and a change of variables, we can show that if the delta function is shifted:

$$\int_{-\infty}^{\infty} g(x)\delta(x - x_0)dx = g(0)$$

Time Reversal of a Delta Function

$$\int_{-\infty}^{\infty} g(x)\delta(-x)dx = g(0) = \int_{-\infty}^{\infty} g(-u)\delta(u)d(-u) = \int_{-\infty}^{\infty} g(-u)\delta(u)du = g(0)$$

Thus a time reversal has no effect on the basic properties of the delta function.

Products of Delta Functions with Regular Functions

It is easy to see that

$$\int_{-\infty}^{\infty} g(x)\delta(x - x_0)dx = g(0) = \int_{-\infty}^{\infty} g(x_0)\delta(x)dx$$

Thus we are free to replace $g(x)\delta(x-x_0)$ by $g(x_0)\,\delta(x-x_0)$, or vice versa.

Scaling the Argument for Delta Functions

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This often occurs when converting from angular frequency (radians/s) to Hertz. Let A be a positive, constant scale factor and define $u = Ax$. Then we can show

$$\int_{-\infty}^{\infty} g(x)\delta(x-x_0)dx = \frac{1}{A}\int_{-\infty}^{\infty} g\left(\frac{u}{A}\right)\delta(u-Ax_0)du = \frac{1}{A}g(x_0)$$

Therefore, we can replace

$$\delta(A(x-x_0)) = \frac{1}{A}\delta(x-x_0)$$

When $A = 2\pi$, we have the common case of converting from ω to f.

The above discussion can also be used to explain the units associated with a delta function. The definition of the delta function requires that its argument should be dimensionless. Nonetheless we often write expression like $\delta(t - t_0)$ where the argument is a variable (for example, time) with physical dimensions (seconds in this case.) To reconcile this with the definition, we must introduce a scaling factor. Suppose we introduce a dimensionless time variable τ such that $t = (1\text{s})\tau$. Then the scaling factor A = 1s (1 second) and we have

$$\delta(t-t_0) = \delta\left((1s)\left(\tau - \frac{t_0}{1\text{s}}\right)\right) = \frac{1}{1\text{s}}\delta\left(\tau - \frac{t_0}{1\text{s}}\right)$$

Thus a delta function implicitly carries units of 1/[units of its argument.]

Fourier Transform of a Delta Function in Time

Using the definition of the Fourier transform and the definition of the delta function, we get

$$\int_{-\infty}^{\infty} \delta(t-t_0)e^{-\omega}dt = e^{j\omega_0 t}$$

Inverse Fourier Transform of a Delta Function in Frequency

Using the definition of the inverse Fourier transform, we have

$$\frac{1}{2\pi}\int_{-\infty}^{\infty} 2\pi\delta(\omega-\omega_0)e^{j\omega t}dw = e^{j\omega_0 t}$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fourier Transform of an Infinite Train of Impulses

Suppose we have an infinitely long "train" of time-domain impulses:

$$x(t) = \sum_{n=-\infty}^{\infty} \delta(t - nT_0)$$

This is a periodic signal. Its Fourier coefficients are

$$a_k = \frac{1}{T_0} \int_{-T_0/2}^{T_0/2} \delta(t) e^{-j\omega t} dt = \frac{1}{T_0}$$

Therefore we can represent the signal as its Fourier series

$$x(t) = \sum_{k=-\infty}^{\infty} \frac{1}{T_0} e^{jk\omega_0 t}$$

The Fourier transform is then easily found from one of the above relationships:

$$X(j\omega) = \sum_{k=-\infty}^{\infty} \frac{2\pi}{T_0} \delta(\omega - k\omega_0)$$

where $\omega_0 = 2\pi / T_0$. If we prefer to measure frequency in units of Hz, this becomes

$$X(f) = \sum_{k=-\infty}^{\infty} \frac{1}{T_0} \delta(f - kf_0)$$

where $f_0 = 1/T_0$. Thus we have shown that the Fourier transform of an infinite impulse train is another infinite impulse train.

The same thing works for a frequency-domain train of impulses. We have

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$X(j\omega) = \sum_{k=-\infty}^{\infty} \delta(\omega - k\omega_0) = \sum_{n=-\infty}^{\infty} a_n e^{-jn\omega T_0}$$

The Fourier series coefficients are

$$a_n = \frac{1}{\omega_0} \int_{-\omega_0/2}^{\omega_0/2} \delta(\omega) e^{jn\omega T_0} d\omega = \frac{1}{\omega_0} = \frac{T_0}{2\pi}$$

Thus

$$X(j\omega) = \sum_{k=-\infty}^{\infty} \delta(\omega - k\omega_0) = \sum_{n=-\infty}^{\infty} \frac{T_0}{2\pi} e^{-jn\omega T_0}$$

This can be used to prove the useful fact that

$$\sum_{n=-\infty}^{\infty} e^{-jn\omega T_0} = \sum_{k=-\infty}^{\infty} \frac{2\pi}{T_0} \delta(\omega - k\omega_0)$$

Calculating Finite Thickness Impedance

The **Assign DC Thickness** option on the **HFSS** menu is enabled if at least one object contains a good conducting isotropic material (such as copper), and the **Solve Inside property** is not selected. If the object meets these conditions, you can assign a DC thickness. If the thickness of the layer is finite, the surface impedance is calculated as:

$$Z = R + jX \quad (1)$$

$$R = \frac{1}{\sigma\delta} \frac{sh(2v) + \sin(2v)}{ch(2v) - \cos(2v)} \quad (2)$$

$$X = \frac{1}{\sigma\delta} \frac{sh(2v) - \sin(2v)}{ch(2v) - \cos(2v)} \quad (3)$$

where

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\delta = \sqrt{\frac{2}{\omega \mu \sigma}} \quad (4)$$

$$\nu = \frac{h}{\delta} \quad (5)$$

where *h* is the layer thickness.

Similar surface impedance is assigned to surfaces of 3D objects of good conductors, which are of NoSolveInside and Thickness for DC Resistance is set.

Modes to Terminals Conversion

This section describes the conversion of modal S-parameters, computed using HFSS's full-wave field solver, into terminal or voltage-based pseudo-S-parameters used in circuit theory.

The S-matrix solutions in HFSS are expressed in terms of incident and reflected complex amplitudes of waveguide modes. This description is mathematically and physically rigorous, but for the case of multi-conductor lines where multiples modes are propagating simultaneously it is convenient to convert the modes to physical quantities directly related to conductors such as currents. Examples of such situations include coupled transmission lines on printed circuit boards, multi-conductor cables, and many common types of electrical connectors. For these types of structures, which generally support multiple, quasi-transverse electromagnetic (TEM) modes of propagation, it is difficult to excite and measure a single mode. In any practical, or laboratory-like, measurement situation:

- measurements (e.g., voltage) contain contributions from several modes
- the applied stimuli excite several modes simultaneously

Terminal support was added to HFSS to enable the simulation of terminal currents and voltages directly, without requiring engineers to determine the needed linear combinations of waveguide modes.

Background

Recall from electromagnetic field theory that the solutions for the transverse components of the electric and magnetic fields in a waveguide with a uniform cross-section can be written as

$$\vec{E}_t = \sum_n a_n \vec{e}_n(x,y) e^{-\gamma z} + \sum_n b_n \vec{e}_n(x,y) e^{\gamma z} \quad (1)$$

$$\vec{H}_t = \sum_n a_n \vec{h}_n(x,y) e^{-\gamma z} - \sum_n b_n \vec{h}_n(x,y) e^{\gamma z} \quad (2)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Here the a_n and b_n are the intensities of the forward and backward traveling modal waves $\vec{e}_n$ and $\vec{h}_n$ are the transverse electric and magnetic field patterns. Note that the a_n and b_n are considered to be dimensionless quantities here, while $\vec{e}_n$ and $\vec{h}_n$ have the normal units for electric and magnetic fields. For this discussion's purpose, assume that each port is defined at z = 0; this permits the removal of exponential terms in the above equations.

Terminal Voltages and Currents

It is possible to define a set of voltages for a port by establishing a number of different integration paths across the port. If a port supports *N* quasi-TEM modes of propagation, one can set up *N* different open contours of integration, C_1, C_2, ... C_N, and define *N* different voltages, v_1, v_2, ... v_N, according to the integral formulas

Alternatively, you can define a set of currents **{**i_k**}**using a set of closed contours of integration **{**D_k**}**:

$$v_k = -\int_{C_k} \vec{E}_t \cdot \vec{dl} \quad (3)$$

$$i_k = \oint_{D_k} \vec{H}_t \cdot \vec{dl} \quad (4)$$

These closed contour paths are set when you define the terminals on a port. HFSS does not require you to set up the voltage integration paths separately. The software automatically infers the voltage paths from the chosen current contours using a power conservation relationship.

Each definition produces a set of relationships between the modal intensities and the voltages or currents. Since the total transverse fields can be expanded as a linear combination of modes, we can rewrite the previous equations as

$$v_k = -\sum_n a_n \int_{C_k} \vec{e}_n(x,y) \cdot \vec{dl} - \sum_n b_n \int_{C_k} \vec{e}_n(x,y) \cdot \vec{dl} \quad (5)$$

$$i_k = \sum_n a_n \int_{D_k} \vec{h}_n(x,y) \cdot \vec{dl} - \sum_n b_n \int_{D_k} \vec{h}_n(x,y) \cdot \vec{dl} \quad (6)$$

To simplify, introduce the following terms:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$t_{kn} = -\int_{C_k} \vec{e}_n(x,y) \cdot \vec{dl} \quad (7)$$

$$u_{kn} = \int_{D_k} \vec{h}_n(x,y) \cdot \vec{dl}. \quad (8)$$

Now v_k and i_k become

$$v_k = \sum_n a_n t_{kn} + \sum_n b_n t_{kn} \quad (9)$$

$$i_k = \sum_n a_n u_{kn} - \sum_n b_n u_{kn} \quad (10)$$

Since there are *N* possible voltage definitions and *N* quasi-TEM modes, there are *N*x*N* matrices that relate the terminal voltages and currents to the following modal intensities:

$$[i_k] = [u_{kn}]([a_n] - [b_n]) \quad (11)$$

$$[v_k] = [t_{kn}]([a_n] + [b_n]) \quad (12)$$

Now introduce the matrices $T = [t_{kn}]$ and $U = [u_{kn}]$ to represent these transformations. Note that the units of *T* are volts, while the units of *U* are amperes.

In summary, the **3D Post Processor** could be used to load different modal field patterns and manually perform the integrals defined previously over each terminal contour line. This would be a tedious and error-prone process, which HFSS's modes-to-terminals conversion feature eliminates.

Calculating the W-Elements

W-elements are a distributed model for transmission lines used in certain HSPICE-compatible circuit simulators. The W-element models in HFSS are computed from the 2D port solutions.

$$Z = R + j\omega L = \gamma * Z_0 \quad (1)$$

$$Y = G + j\omega C = \gamma * \text{inverse}(Z_0) \quad (2)$$

Extensions to the multi conductor case are possible but are beyond the scope of this discussion.

There are two flavors of W-element export in HFSS: Tabular and RLGC. (You see them in the Equivalent Circuit Export Options panel.) The more basic "RLGC" format fits the computed RLGC data to the model equations repeated below. For RLGC format RG parameters are frequency dependent such that HSPICE computes them as

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

R(f) = Ro + Rs * sqrt(f) (3)

G(f) = Go + Gd * f (4)

where Ro is the low frequency resistance and Go is the low frequency conductance. Rs and Gd represent the high-frequency asymptotes due to skin effect and loss tangent, respectively. L and C are assumed to be fixed over the frequency band.

Thus the "RLGC" format uses four matrices (Ro, Rs, Go, Gd) to model frequency dependence for R(f) and G(f), and it neglects the frequency dependence of L and C.

In contrast the "Tabular" format is not fitting the data to this simplified model: it is printing out the actual RLGC values at each frequency as computed by HFSS from the port solution.

Under some conditions, during W-Element export, HFSS may issue a warning that "One or more of the diagonal terms in the W-element <designated> matrix for some frequency are negative for a <specified port>." There may also be warnings if there are insufficient low-frequency points for the W- element data.

This signifies a non-physical result. A negative diagonal entry in the RLGC matrices indicates that the model is non-passive at that frequency (i.e., can produce energy, which is probably not realistic for a metal interconnect).

This can be caused by a number of things:

- Interpolating sweeps.
- Non-causal dielectric material models, such as the traditional constant loss tangent, constant permittivity dielectric model. The Djordjevic or Debye models in HFSS would be the preferred ones to use here.
- Allowing radiation boundaries on the ports.
- Solving to a very low frequency (one so low that the field solution from the full-wave solver contains significant numerical errors.) For frequencies under 10 MHz, you must have at least 10 sampling points per decade.

Terminal-Based Models for Circuit Analysis

It is possible to use the transformations developed above to compute terminal-based admittance (Y), impedance (Z) and pseudo-S-matrices (S_p). First consider the admittance case. A relationship of the form $i = Yv$ is needed that gives the vector of terminal currents $i = [i_k]$ as a function of the vector of terminal voltages $v = [v_k]$. It is known that

$$v = T(a + b) \quad (1)$$

and

$$i = U(a - b) \quad (2)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

It is also known that the incident and scattered modal amplitudes are related by $b = Sa$, where S is the modal S-matrix computed by HFSS. Therefore:

$$v = T(a + Sa) = T(I + S)a \quad (3)$$

$$i = U(a - Sa) = U(I - S)a \quad (4)$$

Here I denotes an identity matrix of the same size as S. i can then be solved for in terms of v by eliminating the incident wave variable a.

$$a = (I + S)^{-1} T^{-1} v \quad (5)$$

$$i = U(I - S)(I + S)^{-1} T^{-1} v \quad (6)$$

The terminal-based admittance matrix Y is identified from the above expression as:

$$Y = U(I - S)(I + S)^{-1} T^{-1} \quad (7)$$

A similar relationship can be developed for the terminal-based impedance matrix:

$$Z = T(I + S)(I - S)^{-1} U^{-1} \quad (8)$$

It is also possible to convert the terminal-based admittance and impedance matrices into a terminal-based "pseudo-S-matrix" S_p. To do this, a reference impedance matrix must be specified. Then standard formulas are used to convert the terminal impedance matrix Z into the terminal S-matrix

$$S_p = Z_{\text{ref}}^{1/2} (Z + Z_{\text{ref}})^{-1} (Z - Z_{\text{ref}}) Z_{\text{ref}}^{-1/2} \quad (9)$$

Z_{ref} can be either a user defined diagonal reference impedance matrix or the terminal characteristic impedance matrix, Zo, which is the matched case for terminal-based models.

The terminal S-matrix S_p relates the intensities of the incident and reflected pseudo-waves at the terminals.

$$\beta = S_p \alpha \quad (10)$$

These pseudo-waves are defined by:

$$\alpha = \frac{1}{2} Z_{ref}^{-1/2} (v + Z_{ref} i) \quad (11)$$

$$\beta = \frac{1}{2} Z_{ref}^{-1/2} (v - Z_{ref} i) \quad (12)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the units of α and β are watts$^{1/2}$. The terminal voltages and currents can also be written in terms of the pseudo-waves.

$$v = Z^{1/2}(\alpha + \beta) \quad (13)$$

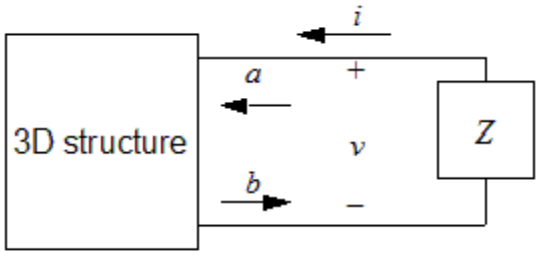
$$i = Z^{-1/2}(\alpha - \beta) \quad (14)$$

Unlike true waveguide modes, the pseudo-waves α and β have no associated propagation constant. The pseudo-waves represent linear combinations of several modes, which may all have differing propagation constants. HFSS is still capable of performing de-embedding on the terminal-based S-matrix, but this is accomplished by first de-embedding the modal S-matrix and then performing the transformation back to a terminal-based S-matrix.

For the normalization of terminal voltages in the Fields Post Processor, see **Scaling a Source's Magnitude and Phase**.

Terminal Characteristic Impedance Matrix

Consider the situation illustrated below. A 3D structure with one multi-mode waveguide port is loaded by an *N*-port matrix impedance *Z*. The structure contains internal sources, which generate outgoing waves that exit through the waveguide port and strike the impedance *Z*. If *Z* does not match the impedance of the waveguide in some sense, a reflection will occur from this load and will return to the 3D structure, where it is interpreted as an incident wave.



An optimal choice of *Z* will prevent any reflections from the load returning as an incident wave *a*. Note that the circuit equations at the load are $v = -Zi$ (the minus sign is due to the sense of the current *i*). By replacing the voltages and currents with their modal expansions, the voltage becomes

$$v = T(a + b) = -ZU(a - b) \quad (1)$$

Rearranging this to isolate *a* and *b*, it is determined that:

$$(T + ZU)a + (T - ZU)b = 0 \quad (2)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Now notice that if we select $T = ZU$; the incident wave a vanishes. Corresponding to this condition is an "optimal" choice Z_0 for the impedance Z:

$$Z_0 = TU^{-1} \quad (3)$$

Z_0 is the terminal characteristic impedance matrix for the multi-mode waveguide port. This value of impedance will completely absorb any linear combination of modal waves leaving the port. As such, it should be of interest to circuit designers wishing to control reflections. In the important, special case of a lossless waveguide, it can be shown that $Z_0 = TT^T$ is a real-valued, symmetric impedance matrix. It is then easy to synthesize a network of resistors with the specified matrix impedance.

Overview of the Technical Approach for Derivatives in HFSS

HFSS can compute the derivatives of S-parameters and related matrix quantities with respect to design parameters you have defined. The variables of differentiation could be geometric parameters, such as the length of a waveguide, or material parameters, such as the relative permittivity of a dielectric.

To make use of this feature, you must first create design or project variables and use them in the definition of an object or a material. You must also enable derivative calculation in the solution setup. After HFSS solves for the nominal value of an output quantity, for example S_{12}, it will also compute its derivative $\partial S_{12}/\partial p$ with respect to the parameter you defined.

There are several possible applications of derivatives. One use is to estimate how much the quantity of interest will change if the parameter is varied: $\Delta S_{12} \cong ((\partial S_{12})/\partial p)\Delta p$. Such an estimate can provide guidance on how to best choose a parameter value, or can help to decide which parameters require tighter control of tolerance. Another possible use is to compute the derivatives with respect to several design parameters to see which parameters are more effective in controlling the quantity of interest.

The basic approach in HFSS can be described as follows. For the nominal solution of a problem, HFSS forms a finite element matrix A and computes a solution vector x (containing fields and S-parameters) for a given excitation vector b. If the problem is parameterized, then the entries of the matrix depend on the design parameter p. The nominal problem is therefore a linear system of the form

$$A(p)x(p)=b$$

Now if we are interested in the derivatives of the solution quantities with respect to p, we can differentiate the original equation:

$$\partial A/\partial p \; x(p)+A(p)\; \partial x/\partial p=0$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

We have assumed here that the excitation vector b is independent of the design parameter. The derivative of the solution vector is then obtained by solving

$$A(p)\,\partial x/\partial p = -\partial A/\partial p\; x(p)$$

In practice, many problems in HFSS are analyzed with a direct matrix solver which computes an *LU* factorization of the matrix *A*. Once the *LU* factors have been obtained, it is very efficient to solve additional right-hand side vectors, such as the one produced in the derivative calculation above. We already have the solution $x(p)$ for the nominal problem, so to form the right-hand side vector the only additional requirement is the derivatives of the matrix entries with respect to p. In HFSS this is done analytically for the most part. Thus a matrix assembly algorithm has been crafted that fills $\partial A/\partial p$ in a manner similar to how *A* itself is filled.

It is worth mentioning that the derivatives of the matrix entries $\partial A/\partial p$ are computed analytically for the most part in HFSS. Therefore the computed derivatives of the output quantities are nearly exact. This should be contrasted with the alternative approach where the design parameter is perturbed from p to $p+\Delta p$ and a new solution $x+\Delta x$ is calculated by solving a new finite element system $A(p+\Delta p)$. That approach is computationally more expensive, because a new finite element system must be formed and factorized, and it is less accurate because $\Delta x/\Delta p$ is not a true partial derivative.

Some technical details on the implementation of derivatives in HFSS may be found in: "Sensitivity Analysis of S-parameters Including Port Variations Using the Transfinite Element Method", L. Vardapetyan, J. Manges, Z. Cendes, 2008 *IEEE International Microwave Symposium*, Atlanta, Georgia, June 2008.

Units for Calculated Derivatives

Note: The derivatives calculated by HFSS are dependent on the units of the design parameters. For example, if the design parameter p is originally defined with units of millimeters, then the value of the derivative gives the (linearized) change in the output quantity when p is varied by one unit (1 mm). If the units of p are changed to an equivalent value in micrometers, and the derivative is recomputed, then it will appear to drop in magnitude by a factor of 1000 because it now represents how much the output changes when p is varied by one unit (now 1 micrometer.) Therefore, when reporting derivative values it is necessary to record the units of the output quantity as well as the units for the variable of differentiation.

Best Practices for Accurate and Causal Broad Band Frequency Sweeps

This section presents practices applied in the Finite Element Method in order to extract accurate and causal broad band frequency sweeps when simulating Printed Circuit Boards (PCBs) and interconnects. The importance and treatment of metal traces of finite thickness, accurate DC

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

point, causal material models, causal surface roughness models and deembedding parasitic effects will be discussed and demonstrated by simulating real life problems.

Introduction to Causality Issues for Simulations

Signal integrity (SI) engineers are challenged to efficiently design and verify high-speed PCBs and interconnects by simulation. When working in the multi-GHz frequency range, simulation typically requires full-wave electromagnetic analysis. The Finite Element method (FEM) is the most common way to rigorously and accurately simulate 3D structures in a broad frequency range and extract the S parameters, which will then be used in a transient SPICE simulation as black boxes to predict the signals in the overall system. A critical consideration for these is the causality behavior of the resulting S-parameter models. Causality means that an output signal cannot start to change before the input signal changes. A necessary and sufficient condition for the causality is that the frequency domain transfer functions (S parameters) have to satisfy the Kramers - Kronig relations [1]. This technical note presents the best practices for getting accurate and causal solutions and not how to enforce causality of existing broad band S parameters. The Kramers - Kronig relationship will just be applied as a causality checker.

Physical Parameters Compared to Simulation

There might be two sources of the discrepancies between the physical parameters and the results of the simulation: one is the improper mathematical model used for the FEM calculations and the other is the numerical solution itself. Both discrepancies can result in causality violation. The paper focuses on the proper mathematical models. It is supposed that the adaptive mesh refinement has converged for the finite element solution, so the mesh is fine enough to approximate the exact solution well.

Modeling Issue: Solve Inside and Surface Impedance Boundaries

One of the most important modeling issues is how the solution should be treated in metals when the frequency range is broad. It is well known that the skin depth varies between infinity and very small value. A large skin depth typically necessitates solving the field inside the metal objects. This solve inside option is impractical for very small skin depths (high frequency), because a large mesh is needed to model the very strong decay of the field. A widely used method is to apply the infinite half space skin impedance as a Surface Impedance Boundary Condition (SIBC), replacing the solution inside the metal objects. It is accurate for high frequencies, MHz to GHz for typical interconnect and PCB geometries, but very inaccurate/unphysical for low frequencies when the finite thickness of the object approaches the skin depth δ, because the real part of the impedance goes to zero with decreasing the frequency instead of going to an appropriate finite DC value. An obvious option would be to combine two methods; apply solve inside at low frequencies and use SIBC at high frequencies. However problems would arise in the transition region, strongly violating causality by introducing jumps or non-physical smoothing. The paper presents a method, which uses the analytical impedances of the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

conductors with finite thickness in the whole frequency region. The method doesn't violate the causality, since the analytical formulas satisfy the Maxwell's equations and the DC point can also be approximated with arbitrary accuracy.

Modeling Material Properties

A common dielectric material model is to assume constant properties across the frequency range. However this is not a physical model of the loss mechanism in dielectrics and violates the causality of the results. Fitting a Debye or Djordjevic - Sarkar material model [3] to the input data will provide a causal model definition.

Modeling Surface Roughness

Another issue is how to take the loss mechanism of surface roughness into account. Two methods for obtaining causal surface roughness models will be presented and discussed in the paper.

When extracting S parameters of the system, ports are inserted into the simulation model. Ports won't introduce any error when they model perfect transmission line terminations, which is possible in FEM codes, unlike Method of Moment (MoM) codes, where so called port calibration is always necessary. In geometries where a transmission line port is not possible, the FEM method allows for non-transmission line terminations (e.g., lumped gap ports). Such a situation most often occurs when there is not even room for a complete transmission port in the design. The user is still interested in extracting S parameters as if the model was terminated by perfect transmission line wave ports. A method will be presented to deembed the parasitic effects introduced by using lumped gap ports.

If all the above mentioned techniques are used in a FEM extraction and the adaptive meshing procedures have converged within a reasonably small tolerance, the S parameters will be accurate and causal. Real-life problems will be presented to demonstrate this using Ansys HFSS, where all the techniques are implemented.

Next

Surface Impedance Boundary Condition (SIBC) for Metal Traces of Finite Thickness

Surface Impedance Boundary Condition (SIBC) for Metal Traces of Finite Thickness

As previously noted, it is not recommended to solve inside of conductive objects in the whole frequency range. The task is to apply equivalent SIBC on the surface which result in the right circuit impedance in the whole frequency range. The derivations will be detailed for internal objects only.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

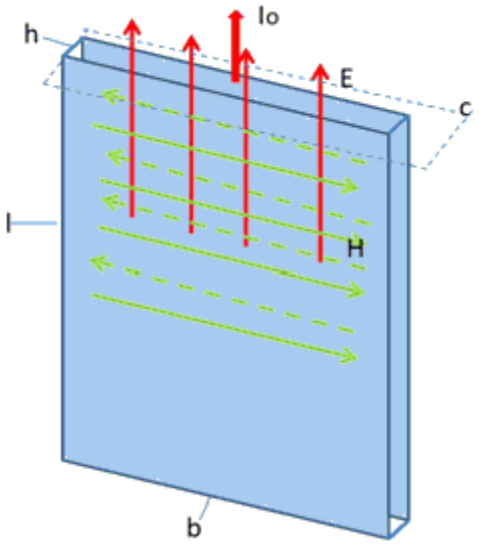


Figure 1: A conductor of finite thickness

Using the notations of Figure. 1 and applying Ampere's law, we get:

$$\oint_c H dc = I_o \qquad 2Hb = I_o \quad (1)$$

Integrating the Poynting vector over both sides of the object:

$$P = 2EHlb = I_o^2 Z_{circuit} \quad (2)$$

Then we get:

$$Z_{field} = \frac{E}{H} = 2Z_{circuit}\frac{b}{l} \quad (3)$$

Using the circuit impedance analytical formula from [5], the real and imaginary parts of the field impedance for the SIBC are:

$$R_{field} = \frac{1}{\sigma\delta c}\frac{sh(v) + \sin(v)}{h(v) - \cos(v)} \quad (4)$$

$$X_{field} = \frac{1}{\sigma\delta c}\frac{sh(v) - \sin(v)}{h(v) - \cos(v)} \quad (5)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Here $v=h/\delta$ and δ is the skin depth. The formulas satisfy the asymptotic conditions, namely at low frequencies and

$$R_{circuit_DC} = \frac{1}{\sigma h} \text{ at low frequencies and } R_{field_high_freq} = X_{field_high_freq} = \frac{1}{\sigma\delta} \quad (6)$$

at high frequencies. For outer surfaces, where the SIBC is applied to the outer boundary of the computational domain,

$$Z_{field} = \frac{E}{H} = Z_{circuit}\frac{b}{1} \quad (7)$$

$$R_{field} = \frac{1}{\sigma\delta}\frac{sh(2v)+\sin(2v)}{ch(2v)-\cos(2v)} \quad (8)$$

$$X_{field} = \frac{1}{\sigma\delta}\frac{sh(2v)-\sin(2v)}{ch(2v)-\cos(2v)} \quad (9)$$

Using SIBC, the solution, including the extrapolated DC point, will be more accurate because the DC behavior is modeled accurately. Also the causality is not violated because the analytical formulas satisfy the Maxwell's equations. Experimental data shows that FEM with low frequency treatment can cope with structures when the ratio of the average mesh size and the wave length is 10-10. Assuming a PCB with 1um average mesh size. It allows us to solve down to 300 Hz.

Next

Using Causal Material Models

Using Causal Material Models

Using causal dielectric material models is one of the most important considerations to extract causal S parameter models. Causal material model means that the real and imaginary parts of the permittivity should satisfy the causality relationship (causal magnetic materials are not discussed here).

The most general material model is the multipole Debye model [3]:

$$\varepsilon_r(\omega) = \varepsilon'_\infty + \sum_{i=1}^{N}\frac{\Delta\varepsilon_i}{1+j\frac{\omega}{\omega_1}} - j\frac{\sigma}{\omega\varepsilon_o} \quad (10)$$

It is applicable when we have information about the permittivity and the loss tangent of the material in the frequency range, such as when measured material data are available. Single pole

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Debye model is also a good approximation when just polarization losses are present and the polarization frequency is known:

$$\varepsilon_r(\omega) = \varepsilon_{r_optical} + \frac{\varepsilon_{r_static} - \varepsilon_{r_optical}}{1 + j\frac{\omega}{\omega_{pol}}} \quad (11)$$

In other cases, especially when the permittivity and loss tangent are known at only one or two frequencies, the broad band Djordjevic - Sarkar (D-S) material model can be used:

$$\varepsilon_r(\omega) = \varepsilon_{r_optical} + \frac{\varepsilon_{r_static} - \varepsilon_{r_optical}}{m_2 - m_1}\frac{\ln\frac{\omega_2 + j\omega}{\omega_1 + j\omega}}{\ln 10} - j\frac{\sigma}{\omega\varepsilon_o} \quad 12$$

where

$$\omega_1 = 10^{m_1} \qquad \omega_2 = 10^{m_2}$$

are the limits of the frequency range. The D-S model can also be viewed as a multipole Debye model with an infinite continuum of poles.

Using causal material models leads to frequency dependent complex materials, which is standard in most finite element codes.

Next

Surface Roughness Modeling

Surface Roughness Modeling

Surface roughness plays an important role in accurately modeling radio frequency components (RFC), such as resonators and signal integrity projects (e.g., microstrips, printed circuit boards (PCBs)). The increase of the surface resistance due to surface roughness results in lower quality factor (Q) in resonators and higher insertion losses for microstrips and PCBs. The scale of the roughness is different in the different application areas so different surface roughness models are used. The common theme in the different models is that an effective frequency dependent conductivity is defined depending on the type and magnitude of the roughness, which shows the increase of the surface resistance compared to the smooth surface resistance.

For HFSS and HFSS 3D Layout designs, Groiss [6] or Huray [2] surface roughness models may be employed, the latter being a complex-value-based causal modification of the original real-

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

value-based model. For SIwave and 2D Xtractor designs, the original Hammerstad-Jensen [4] or modified Huray [2] model may be employed.

Impedance at a rough surface is higher than the impedance at a smooth surface by a factor $\boldsymbol{fact_R}$ as shown in the general equation below:

$$Z_{rough} = Z_{smooth} * fact_R \quad (13)$$

where Z_{rough} is the impedance at the rough surface and Z_{smooth} is the impedance at the smooth surface.

For the **Groiss** model, the surface roughness factor is given by the equation:

$$fact_R = 1 + e^{-\left(\frac{\delta}{2s}\right)^{1.6}} \quad (14)$$

where δ is the skin depth and s is the surface roughness.

> **Note:**
>
> The Groiss surface roughness model is not causal, and therefore, it is only suitable for frequency-domain computations, not for the time-domain.
>
> Additionally, you can see, from equation (14), that the maximal roughness factor is 2. So this model is only applicable to RFC problems when the surfaces are highly polished to avoid lower Q. A surface roughness model of higher losses is necessary to model the fabrication effects in PCBs, where the roughness of the deposited metal is considerably higher.

The alternative modified **Huray** [2] surface roughness model, available in the Ansys Electronics Desktop software, is a complex-value-based causal model that is suitable for both frequency-domain and time-domain computations. It also overcomes the factor of 2 limitation of the Groiss model, as it supports rougher surfaces with factors greater than 2:

Refer to the Technical Note: Huray Surface Roughness Model for the equations and definitions that are applicable to this model.

The Huray roughness factor adjusts the surface impedance in the same manner as shown in equation (13) except that the roughness factor variable is referred to as $\boldsymbol{H_c}$ instead of $\boldsymbol{fact_R}$. (15)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All SIBCs depend on the material conductivity too, because the skin depth varies with conductivity.

Next

Deembedding Parasitic Port Effects

Deembedding Parasitic Port Effects

Figure 4 shows a strip line that is fed by internal lumped ports instead of perfect wave ports. The FEM simulation was performed using these excitations and resulted in the entire scattering matrix, *S**. The goal is to calculate scattering matrix *S* (scattering matrix of the DUT), as if the structure were fed by perfect virtual ports. It can be done if S matrices of the transition or error boxes are known ($S^{(1)}$and $S^{(2)}$ in Fig.2). When **b* =S*a*** is known and **b=Sa** is sought, we get:

$$\mathbf{S} = [\langle \mathbf{C}_{bb} \rangle + \mathbf{S}^* \langle \mathbf{C}_{ab} \rangle]^{-1} [\mathbf{S}^* \langle \mathbf{C}_{aa} \rangle - \langle \mathbf{C}_{ba} \rangle] \tag{16}$$

where

$$\langle \mathbf{C}_{aa} \rangle, \langle \mathbf{C}_{ab} \rangle, \langle \mathbf{C}_{ba} \rangle, \text{and } \langle \mathbf{C}_{bb} \rangle$$

are diagonal matrices:

$$\mathbf{C}^i_{aa} = \frac{1}{S_{21}^{(i)}}; \quad \mathbf{C}^i_{ab} = \frac{S_{22}^{(i)}}{S_{21}^{(i)}}; \quad \mathbf{C}^i_{ba} = \frac{S_{11}^{(i)}}{S_{21}^{(i)}}; \quad \mathbf{C}^i_{bb} = S_{12}^{(i)} - \frac{S_{11}^{(i)} S_{22}^{(i)}}{S_{21}^{(i)}} \tag{17}$$

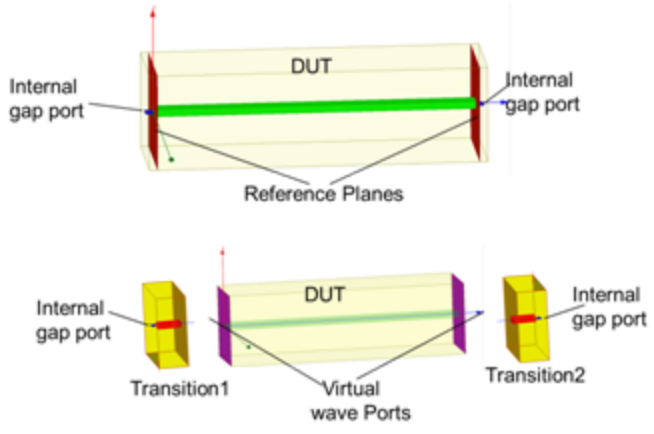


Figure 4: Strip line (DUT) fed by two internal lumped ports

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using Circuit Ports Does Not Change the Deembedding Analysis

Figure 4 shows a strip line, which is fed by internal lumped ports instead of perfect wave ports. Using circuit ports would not change the deembedding analysis.

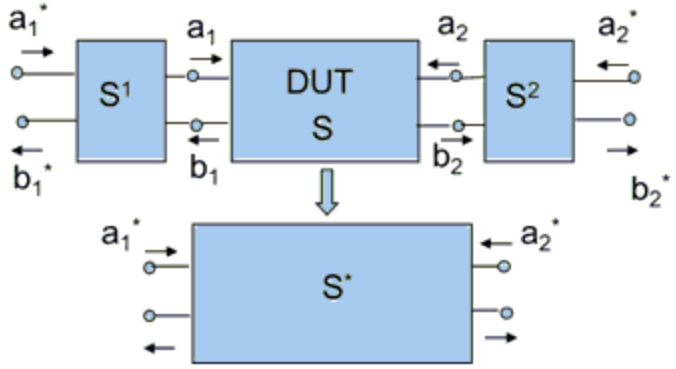


Figure 5: Black boxes of a strip line fed by two internal lumped ports

Usually, the device under test (DUT) is very large and complex. It is not possible to create perfect wave ports inside the complex geometry. However, it is not necessary to do that, when using the proposed method. The transition boxes can be independently created and solved as very small, 3D projects with two ports. In that case, the method is exact, within computational errors. In simple cases, *S* matrices of the transition boxes can be estimated, (for example, by a series impedance). The simplest case is when a lumped gap port is represented by a "parasitic" inductance. Applying experimental approximations for the calculation of the partial inductance of a lumped gap port ([7], [8]):

(18)

$$L = \frac{\mu_0 len}{2\pi}\left[\ln\left(u+\sqrt{u^2+1}\right)+u\ln\left(\frac{1}{u}+\sqrt{\frac{1}{u^2}+1}\right)+\frac{1}{3}\left(u^2+\frac{1}{u}-\frac{(u^2+1)^{3/2}}{u}\right)\right]$$

Where:

Input: w = width of the lumped port [m]

len = length of the lumped port[m]

u = len/w

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

t = 0.0

Output:

L[H]

*From C.R.Paul, *Inductance. Loop and partial*, Wiley, 2010

And

$$Z_s = R_s + jX_s \qquad X_s = \omega L 10^{-9} \quad (6)$$

Currently, $R_s = 0$

$$\mathbf{Y} = -\begin{bmatrix} \frac{1}{Z_s} & -\frac{1}{Z_s} \\ -\frac{1}{Z_s} & \frac{1}{Z_s} \end{bmatrix} \quad (7)$$

$$\mathbf{Y}' = (\mathbf{I} + \boldsymbol{S})^{-1}(\mathbf{I} - \mathbf{S}) \quad (8)$$

$$\mathbf{Y} = \langle \frac{1}{\sqrt{Z_o}} \rangle \mathbf{Y}' \langle \frac{1}{\sqrt{Z_o}} \rangle \qquad Z_o = 1\Omega \quad (9)$$

$$Z_s = \frac{1}{Y_{1,1}} \quad (10)$$

$$\mathbf{Z} = \mathbf{Z}^* - \langle \mathbf{Z}_s \rangle \quad (11)$$

Z* is the global impedance matrix

Z is the Global impedance matrix after deembedding

Fig. 6 shows the inductance of a pair of cylindrical wires. The two feeding lumped gap ports introduce parasitic inductances, which can be deembedded by using the above method, which analytically accounts for the DC and AC inductances. The maximum error is reduced by roughly a factor of 3; 4.3% without deembedding and 1.6% with deembedding the parasitic inductance.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

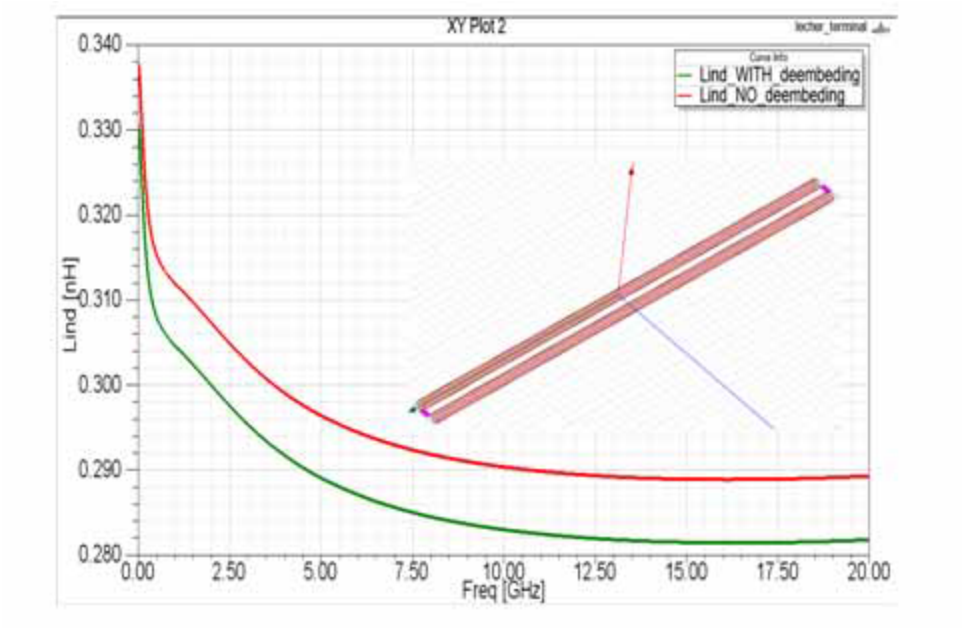


Figure 6: Loop inductance of a pair of cylindrical wires with and without deembedding the parasitic gap port inductance

Conclusions

New numerical constructs are developed for the causal modeling of transient signals in PCBs and interconnects. By introducing the correct treatment of metal traces of finite thickness, accurate DC point solution, causal material models, causal surface roughness models and deembedding parasitic effects of lumped ports, result in accurate and causal broad band frequency sweeps of S parameters. Using these techniques in a finite element code generates accurate and causal simulations provided the mesh is converged to reasonably small tolerance. The techniques described here are implemented in Ansys HFSS. Real-life problems were solved to demonstrate the efficacy of the methods.

References for Accurate and Causal Broad Band Frequency Sweeps

[1] B. Young and A. Bhandal, "Causality Checking and Enhancement of 3D Electromagnetic Simulation Data," Electrical Performance of Electronic Packaging and Systems (EPEPS), 2010 IEEE, 19th Conference, Conference Proceedings, pp. 81-84.

[2] J. Eric Bracken, ANSYS, Inc., "A Causal Huray Model for Surface Roughness", DesignCon 2012, January 2012

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[3] A. R. Djordjevi, R. M. Bilji, V. D. Likar-Smiljani?, and T. K. Sarkar,"Wideband frequency-domain characterization of FR-4 and time-domain causality," IEEE Tran. Electromagnetic Compatibility., vol. 43, pp, 662-667, Nov. 2001.

[4] S. H. Hall and H. L. Heck, High-Speed Digital Designs, John Willey & Sons, Inc.,Hoboken, New Jersey, 2009.

[5] I. Vago and M Gyimesi, Electromagnetic Fields, Academiai Kiado, Budapest, Hungary, 1998.

[6] S. Groiss, "Numerical Analysis of Lossy Cavity Resonators," (In German) PhD Theses, Technical University of Graz, Austria, June 1996.

[7] F. W. Grover, Inductance Calculations, Dover Publications, Inc., Mineola, New York, 1946, 1973.

[8] C. R. Paul, Inductance, Loop and Partial, John Willey & Sons, Inc.,Hoboke, New Jersey, 2010.

[9] P. Pathmanathan, C.M. Jones, S.G. Pytel, D.L. Edgar, P.G. Huray, "Power Loss due to Periodic Structures in High-Speed Packages and Printed Circuit Boards", European Microelectronics Packaging Conference - IMAPS 2011, Brighton, England, September 12 - 15, 2011

Determining PCB Material Properties: Q2D and Nexxim Measurements for HFSS Simulation

This technical note describes how to determine Printed Circuit Board Material Properties (PCB) by using Ansys Electronics Desktop Q2D and Nexxim measurements for accurate simulation with HFSS. The necessary PCB properties are:

- Dielectric Constant (ε_r)
- Loss Tangent ($\tan\delta$)
- Metal Conductivity (σ)
- Surface Roughness Parameters (Hall-Huray Parameters)

In order to obtain the proper values for HFSS Simulation:

- Use the Beatty Standard to characterize loss
- Use proper calibration, or de-embedding to get to actual structure

Use Ansys Q2D and Nexxim to match the measured data.

- Use the properly de-embedded Device Under Test (DUT)
- Use Ansys Q2D to quickly vary the material properties
- Use Nexxim to cascade different line sections together to build the DUT

Validate the Extracted Parameters with HFSS Simulation

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following sections contain detailed information:

- The Beatty Standard and its Use
- De-embedding of the Device Under Test from Test Fixture
- Example using Nexxim and Q2D to obtain PCB properties for HFSS Simulation
- References for Determining PCB Material Properties

The Beatty Standard and its Use

What is the Beatty Standard? as drawn from Reference [1]

- A series resonator between two lines of characteristic impedance Z_0.

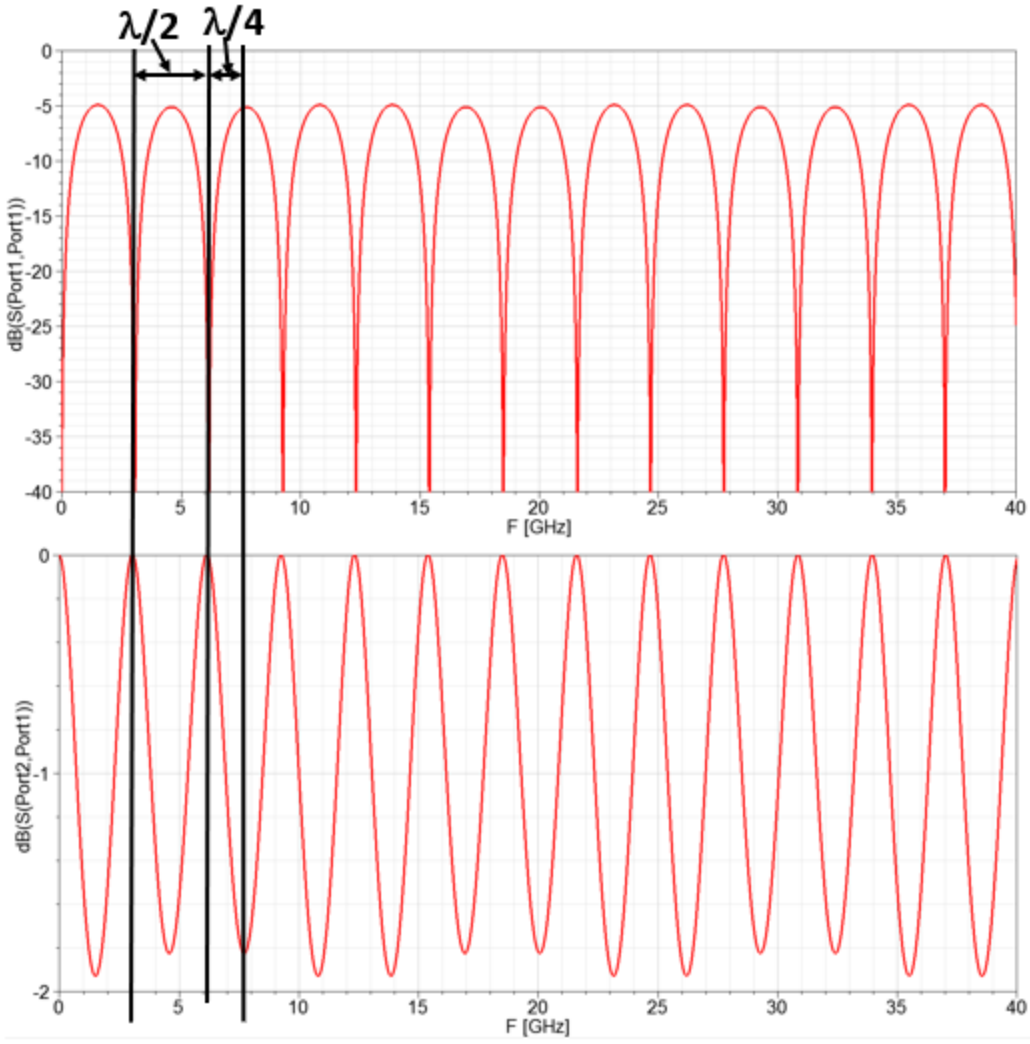
- Z_0 length of line is 0.25 inches. Z_B length of line is 1 inch.

How it performs:

- The resonator will have a periodic response because of transmission line effects
- S_{11} maximums (and S_{21} minimums) will also be λ/2 apart
- S_{11} minimum to maximum (and S_{21} maximum to minimum) will be λ/4 apart

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

How the Beatty Standard Performs: Lossless

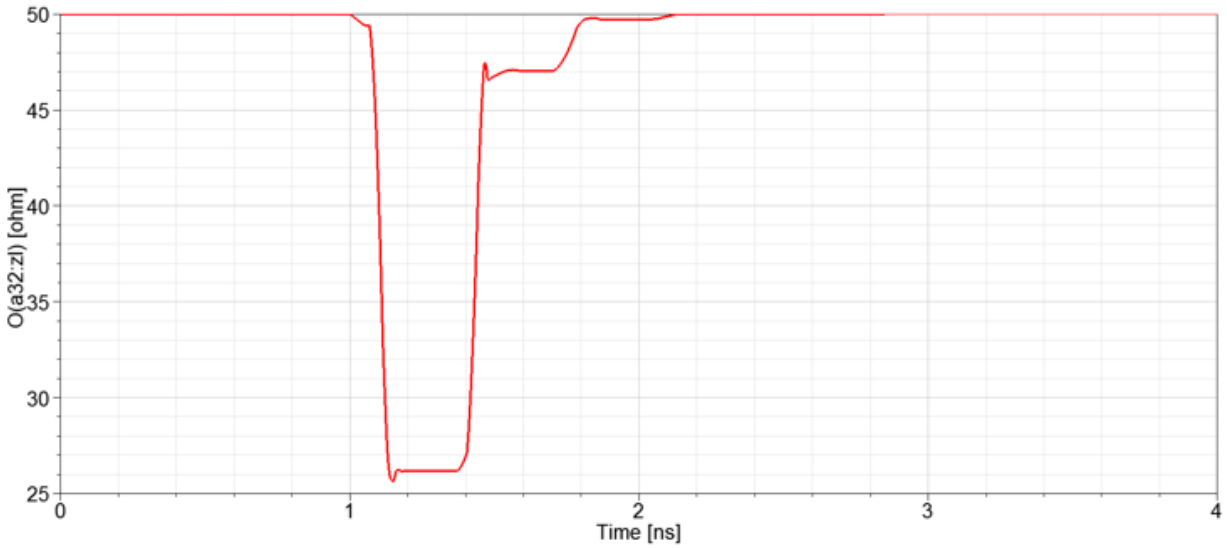


S_{11}

S_{21}

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

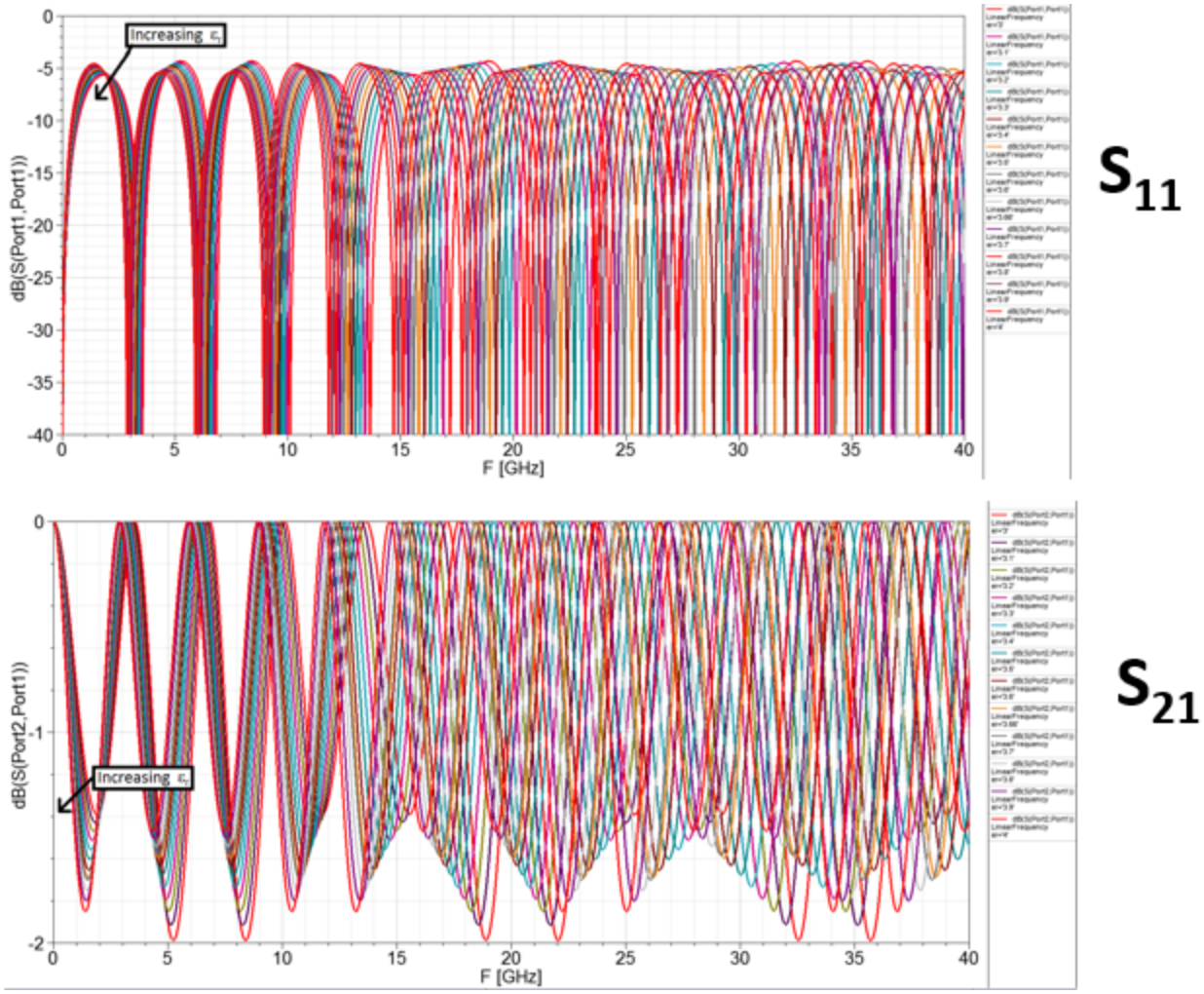
How the Beatty Standard Performs, TDR



What about Changing ε_r?

This plot shows dielectric constant sweep from 3 to 4. Changing ε_r moves the distance between maxima and minima.

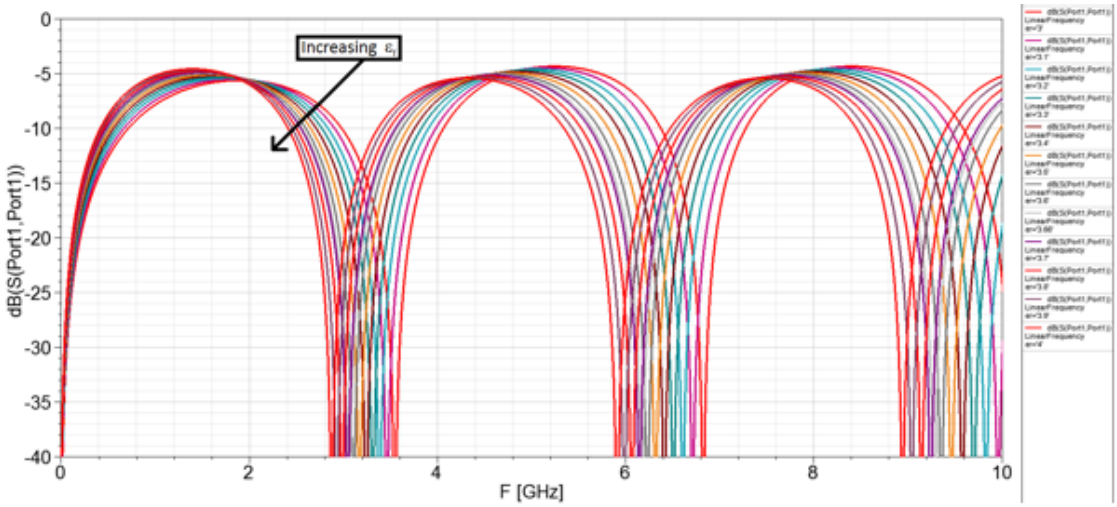
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



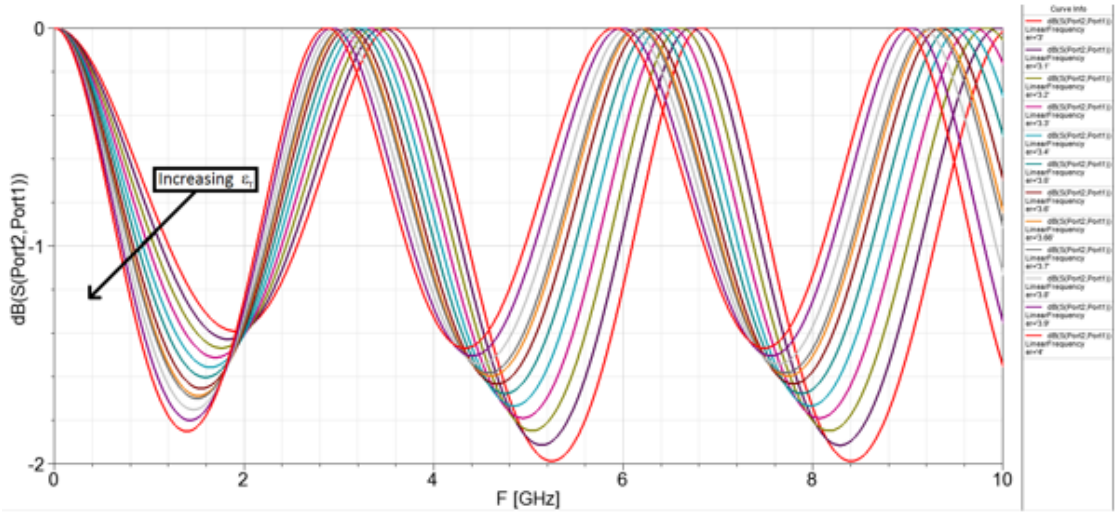
Dielectric Constant, continued: Zoom in, the better to see Changing ε_r

Electrical length is changing with the dielectric constant.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



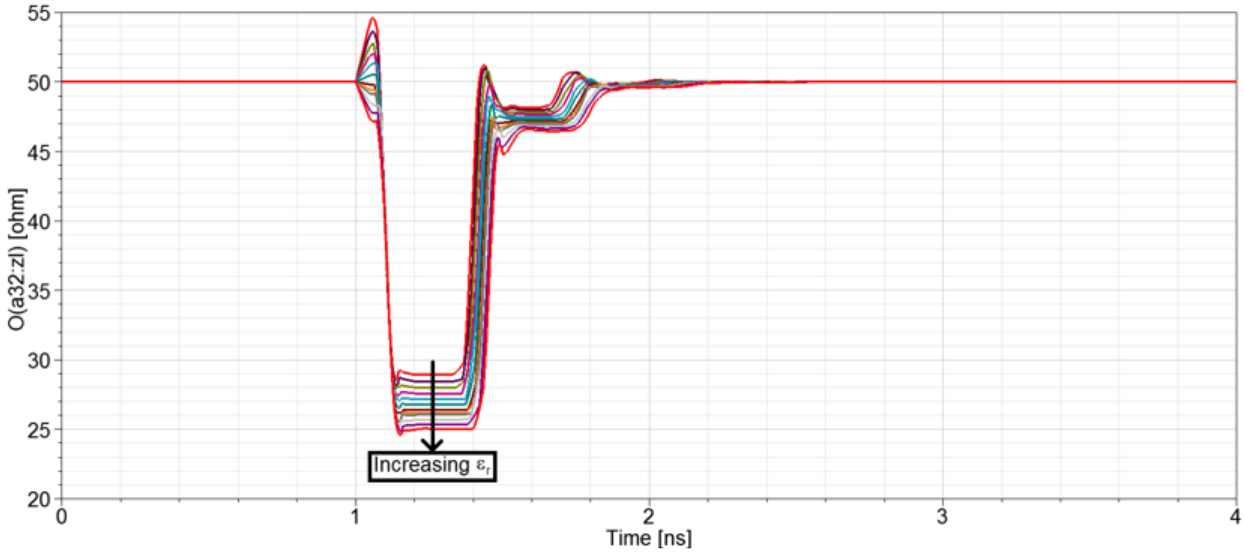
S_{11}



S_{21}

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

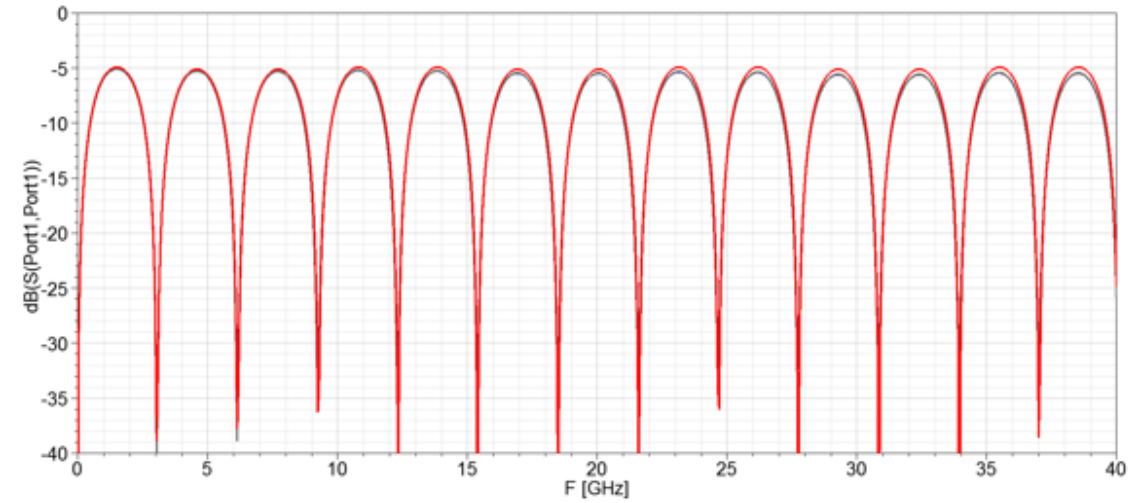
Dielectric Constant, continued. Now Look at TDR to see effects of Changing ε_r



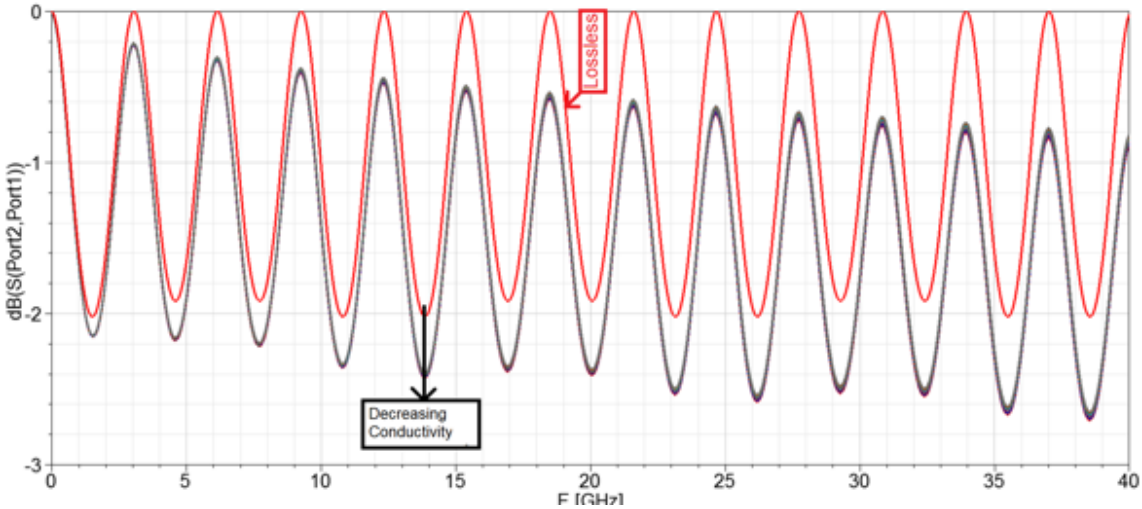
What about Metal Conductivity?

Look at DC, or a low as you can.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



S_{11}



S_{21}

What does this tell us about metal conductivity? There is a change in plot shape due to loss.

- Look at DC to eliminate skin effect and surface roughness effects
- Network Analyzer is not be best tool to measure this low in frequency
- Will likely need to use Impedance Analyzer or precision Ohm-meter.
- Results of DC resistance of structure show below vs. metal conductivity

Conductivity (S/m)	DC Resistance (W)
$5.0x10^7$	0.070643
$5.1x10^7$	0.070241
$5.2x10^7$	0.069855
$5.3x10^7$	0.069483
$5.4x10^7$	0.069125

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

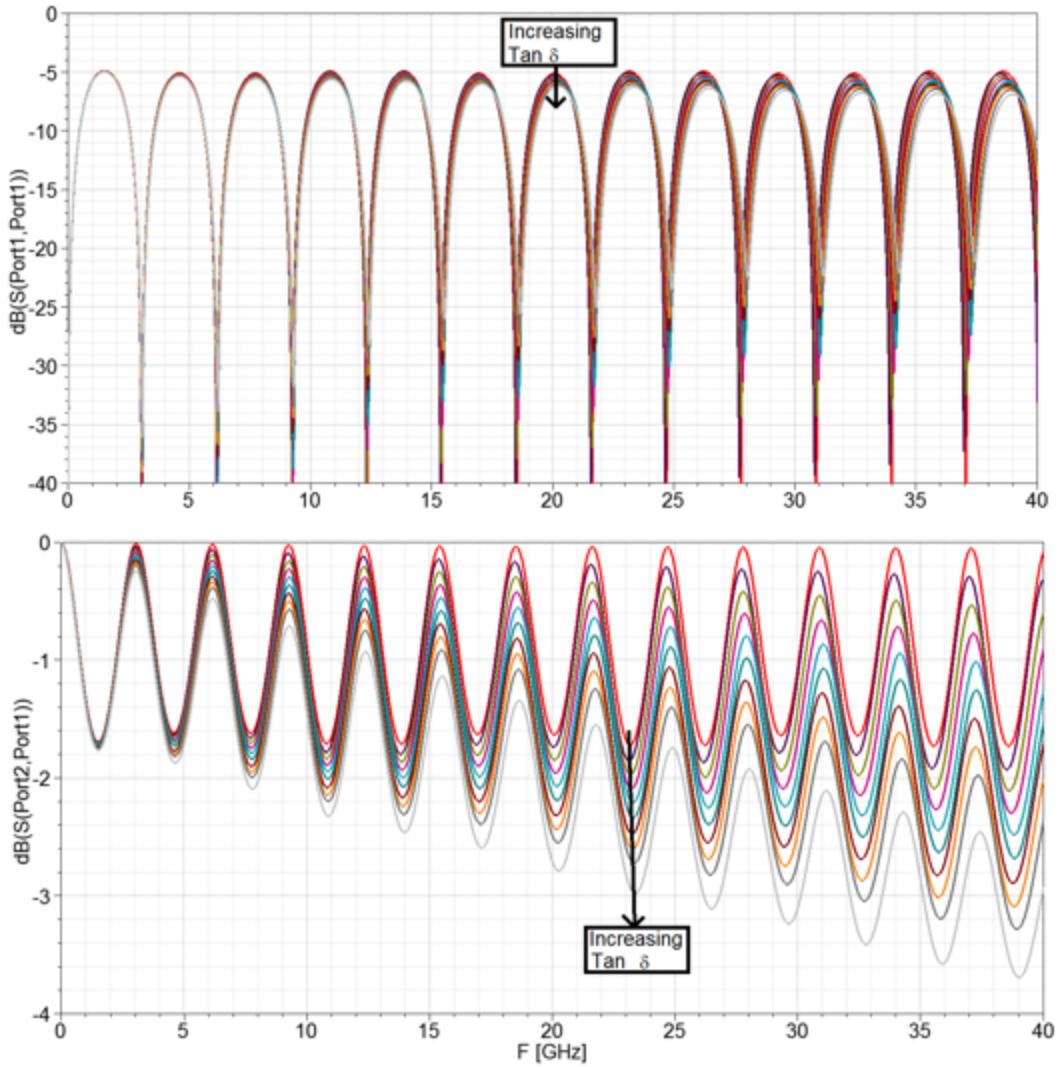
Conductivity (S/m)	DC Resistance (W)
$5.5x10^7$	0.068780
$5.6x10^7$	0.068448
$5.7x10^7$	0.068127
$5.8x10^7$	0.067817
$5.9x10^7$	0.067518
$6.0x10^7$	0.067228

What about Loss Tangent?

The plots shows a change in shape due to substrate loss.

- We know metal conductivity from DC value, so change tan to get loss to match. Sweep from 0.001 to 0.01.
- Less change at “low frequency” and more “high-frequency” loss with tanδ

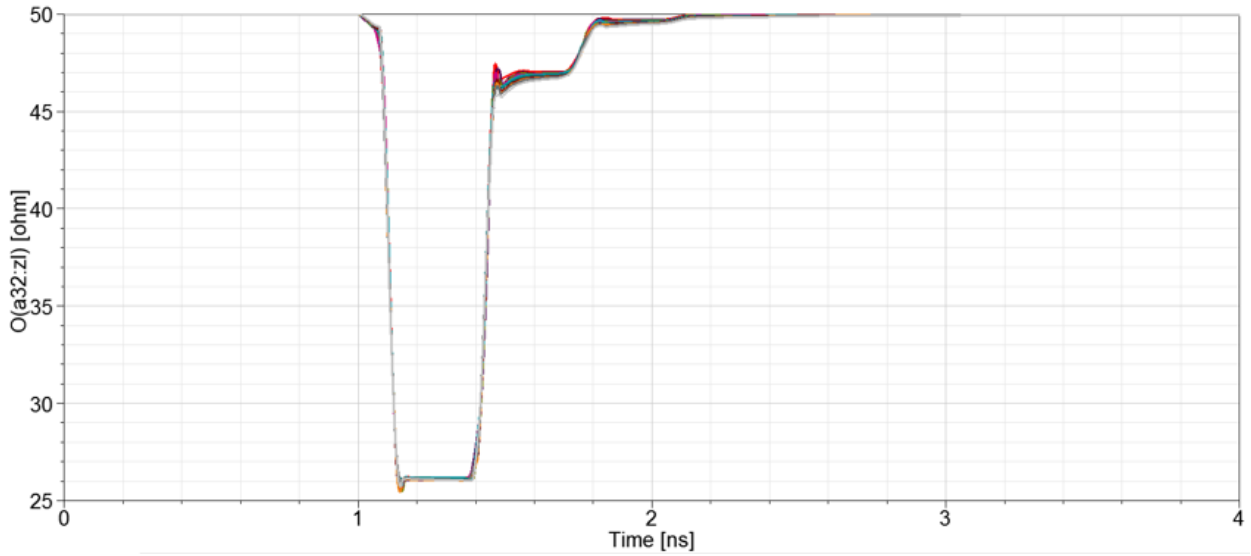
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Now Look at TDR for Loss Tangent

This shows a change in shape due to substrate loss, with little change in TDR.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

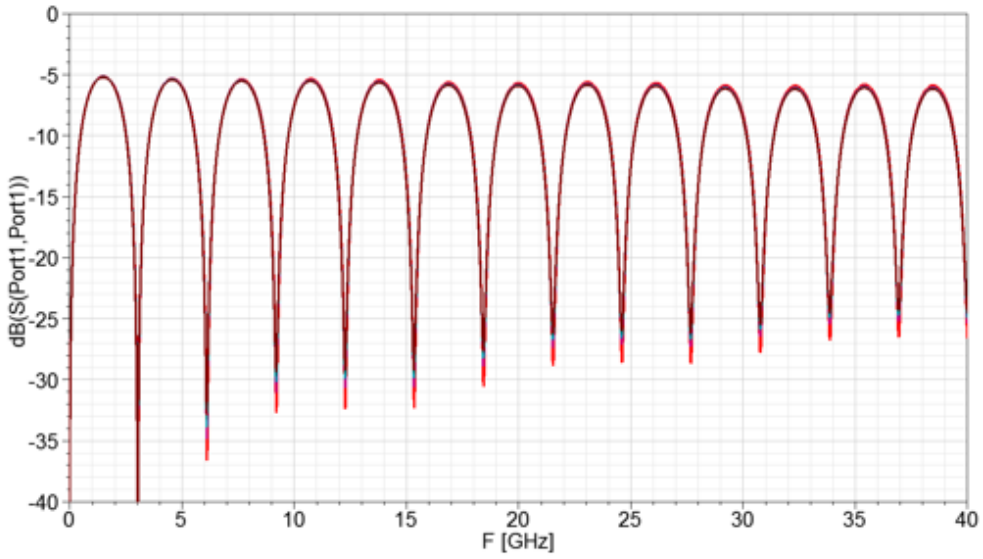


What about Surface Roughness?

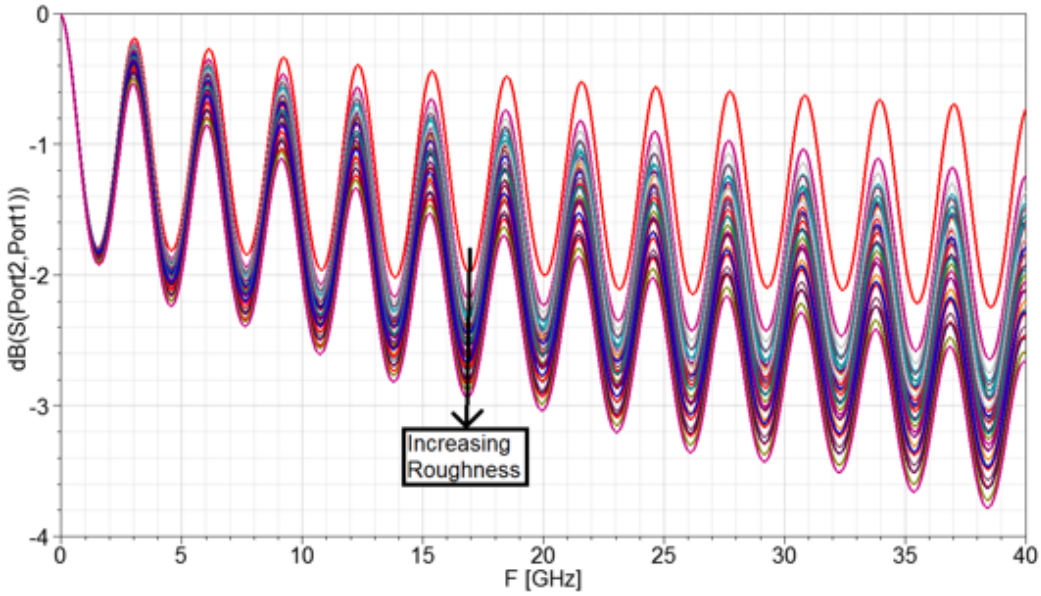
This plot shows a change in shape due to roughness.

- Similar loss as loss tangent at high-frequency.
- More loss at low frequency. Match low frequency first with surface roughness.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



S_{11}



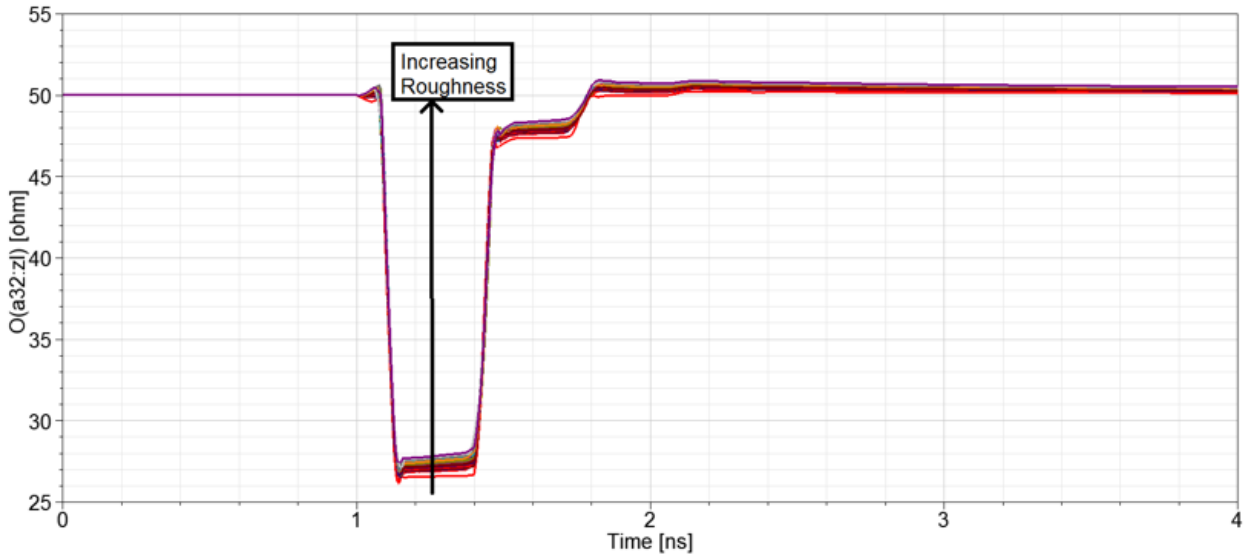
S_{21}

Now Look at TDR for Surface Roughness

The plot shows change in shape due to roughness.

- More change in impedance compared to changing loss tangent.
- May be important to separate roughness from substrate loss.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Observations so Far

- Change ε_r to match spacing between maxima and minima.
- Look at very low frequency (ideally DC) to match conductivity
- Look at “low frequency” to match Surface Roughness
- Look at “high frequency” loss to match Loss Tangent after Surface Roughness is matched.
    - Look at TDR plot to help separate loss if not obvious from Frequency Domain data

Not Looking at:

- Changing line and substrate dimensions
    - Restricting variables to tune for loss
    - Need to verify all the thickness, widths, and any etch with cross-section
- Phase Information and Group Delay
    - To verify propagation time if needed for fine tuning

What Have We Haven’t Looked At

- Changing line and substrate dimensions
    - Restricting variables to tune for loss
    - Need to verify all the thickness, widths, and any etch with cross-section
- Phase Information and Group Delay
    - To verify propagation time

Next

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

De-embedding of the Design Under Test from Test Fixture

De-embedding of Design Under Test from Test Fixture

This section describes de-embedding, that is the impact of the test fixture on the measurement result is removed from the measurement data thereby allowing characterization of the device under test (DUT). This is accomplished by means of:

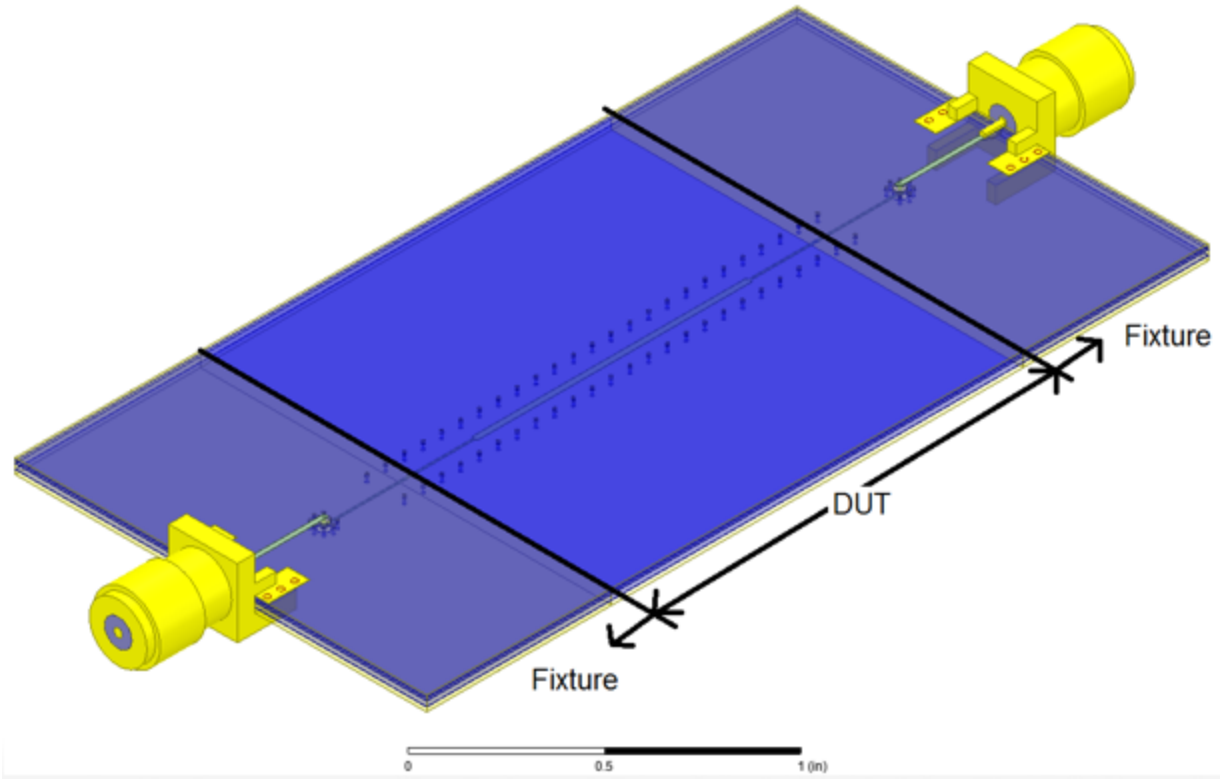
- Custom Calibration Kit
- YZ De-embedding
- Modeling and Simulation
- Automatic Fixture Removal (AFR)

Illustration of Fixture and Device Under Test (DUT)

The total structure includes the following:

- CAD drawing of the Fixtures and Device Under Test (DUT)
- We want the response of just the DUT, but we can only measure at coaxial connectors.
- Notice that the Fixtures are identical. While not necessary, this makes the de-embedding a little easier.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



De-embedding: Custom Calibration

This section describes how you can make a calibration kit on your PCB. In this case, we use Short-Open-Load-Thru (SOLT). You can also use a variant called Short-Open-Load-Reciprocal (SOLR), sometimes called Short, Open, Load, Unknown Thru. The process needs well-defined standards which can be difficult to get at high frequency.

We also use a technique called Use Multiline Thru-Reflect-Line (TRL) described in Reference [3].

- Use multiple Lines to get required bandwidth as each line can only cover a line whose electrical length is between 200 and 1600
- Used by National Institute of Standards and Technology (NIST)
- Can take a lot of space to get lines long enough for low frequency measurements

While there are many other calibration methods, with well defined standards, this is an excellent choice. It can be difficult to make, and characterize, standards at a necessary reference plane.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

De-Embedding: YZ De-embedding Described in Reference [4]

This approach is used mainly for on-die (i.e. on-wafer) measurements.

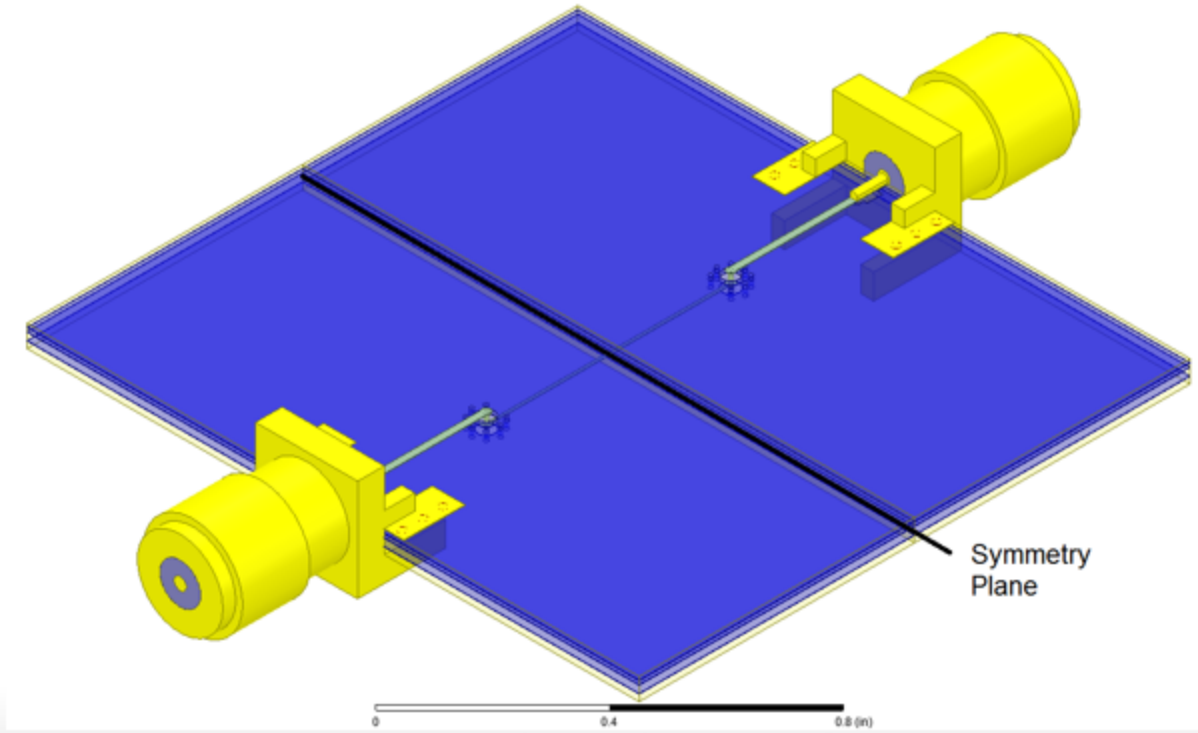
- Measure an Open Standard (i.e. the Y-standard)
- Measure a Short Standard (i.e. the Z-standard)
- Convert the measured device, with fixture, to Y-parameters and subtract Y-parameters of Open. Call this $Y_{Yremove}$. Also, remove the open standard from the Z-standard.
- Convert $Y_{Yremove}$ to Z-parameters and subtract the Z-parameters of the Y-corrected Short Standard. The result is the de-embedded DUT.
- Assumes that the Fixture is electrically small.
- Easy to have electrically small structure on-die, but harder to do on PCB.

De-embedding: Modeling and Simulation

To perform the de-embeding for Automatic Fixture Removal (AFR as described in Reference [5]):

1. Concatenate the Fixture.
2. Make symmetric (as this is much easier).
3. Measure 2X Thru. Use this structure

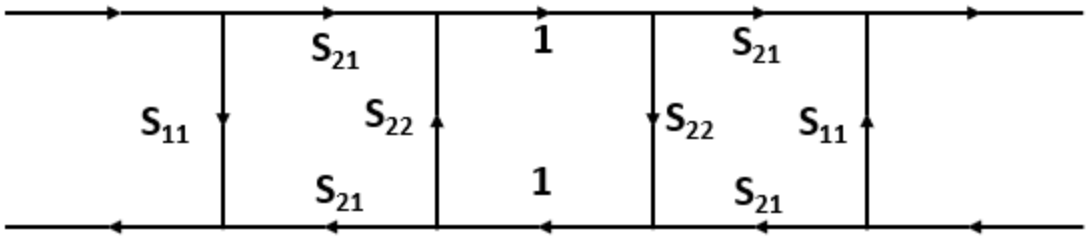
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Create circuit model of each fixture half and concatenate as well
   - Use Lumped Element
   - Or Distributed
   - Match of model to measurement must be excellent, or you will introduce error
5. Simulate with full-wave solver like HFSS.
   - You need to know surface roughness, dielectric properties, etc.
   - Match in S-parameters needs to be excellent, or you will introduce error.

Using Signal Flow Graphs (see References [6, 7]), we can model the 2X Thru.

- Notice the S-parameters in each half.
- 2X Thru has symmetry. Also, $S_{21}=S_{12}$ since they are reciprocal.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These are the S-parameters we are looking for, but we are measuring the 2X Thru, so we only know:

- $S_{11,2X_THRU}=S_{22,2X_THRU}=S_{11}$ since symmetric
- $S_{12,2X_THRU}=S_{21,2X_THRU}=S21$ since reciprocal

So, solve these equations

$$S_{11,2X_THRU} = S_{11} + \frac{S_{22}S_{21}^2}{1 - S_{22}^2} \qquad S_{21,2X_THRU} = \frac{S_{21}^2}{1 - S_{22}^2}$$

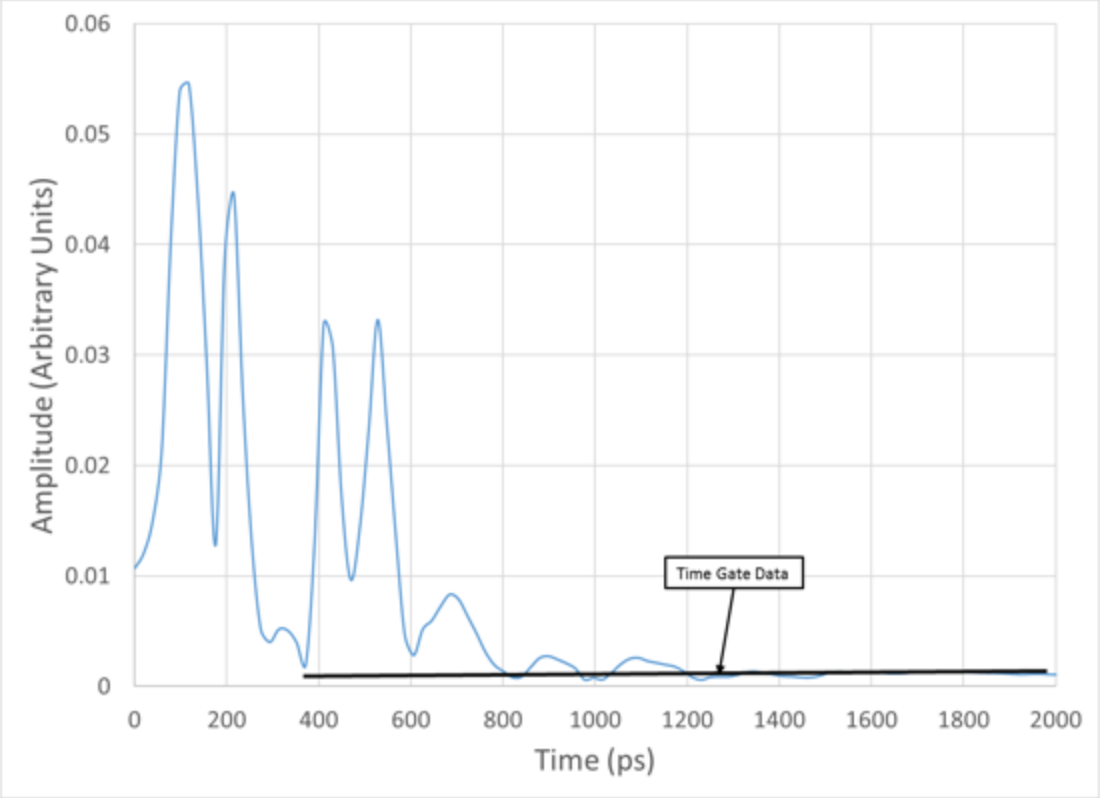
This gives us two equations with three unknowns.

Use time gating to learn S_{11}.

- Take inverse Fourier Transform of $S_{11,2X_THRU}$
- At half-way point in time, set level to that of a matched termination.
- Take Fourier Transform of gated data
  - This is S_{11} of the Signal Flow Graph
  - Now you have two equations and two unknowns to find S_{21} and S_{22}
  - Recall square root of complex number is (in mag/ang format):
  - $\sqrt{Magnitude}\angle\ \theta/2$

Solving is outside the scope of this chapter, but once solved you have the fixtures.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With S-parameters of the fixture

- Use ABCD parameters described in *Reference [6]* to remove fixture effects from measured data to get DUT
- Can use T-parameters described in *Reference [7]* as well.
- Or, use Nexxim with Reciprocal option to de-embed the data for you.

Next

Example of Determining PCB Material Properties: Q2D and Nexxim to HFSS

Example Determining PCB Material Properties: Q2D and Nexxim to HFSS

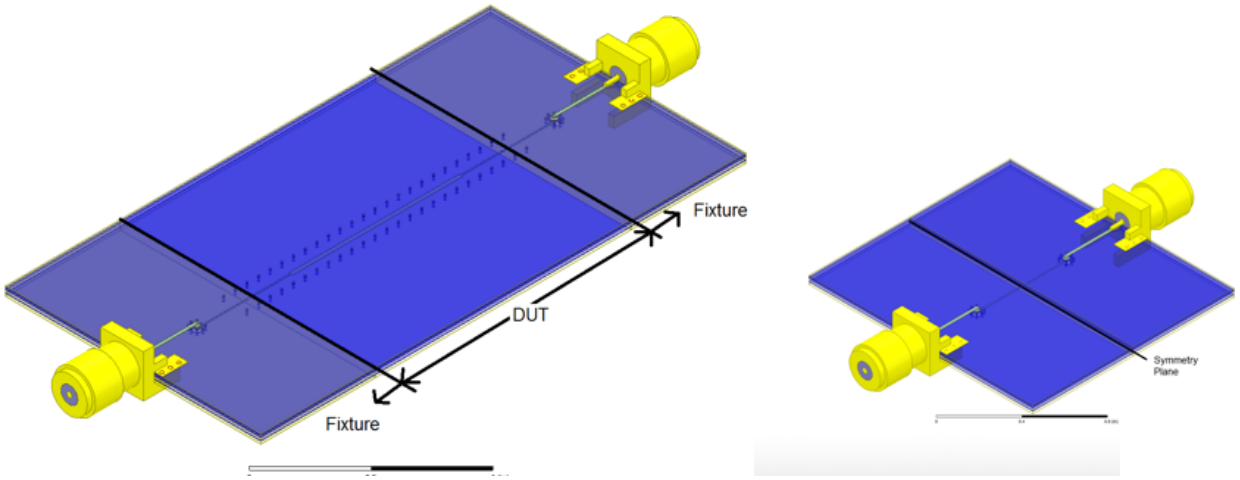
This section describes the procedure for determining the PCB material properties that will be used for an HFSS simulation of a PCB.

Beatty Structure

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- 7 mil wide, 0.25in long 50 ohm line, with 1.4 mil thick trace and 8 mil dielectric above and below. A stripline
- 21 mil wide, 1 inch long line, with 1.4 mil thick trace and 8 mil dielectric above and below. A stripline.

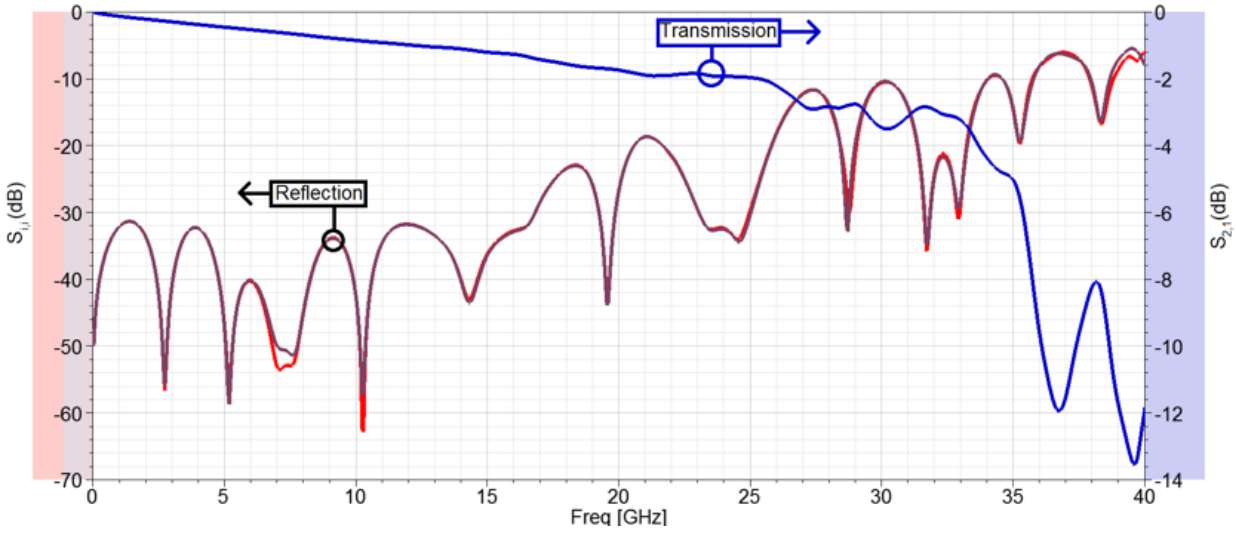
Fixture is a coaxial connector with a via transition down to the stripline.



Note: The Full structure is simulated in HFSS. It is called “Measured” to illustrate that this will typically be measured data

Look at 2X Thru S-parameters. There is a Good response until ~35 GHz.

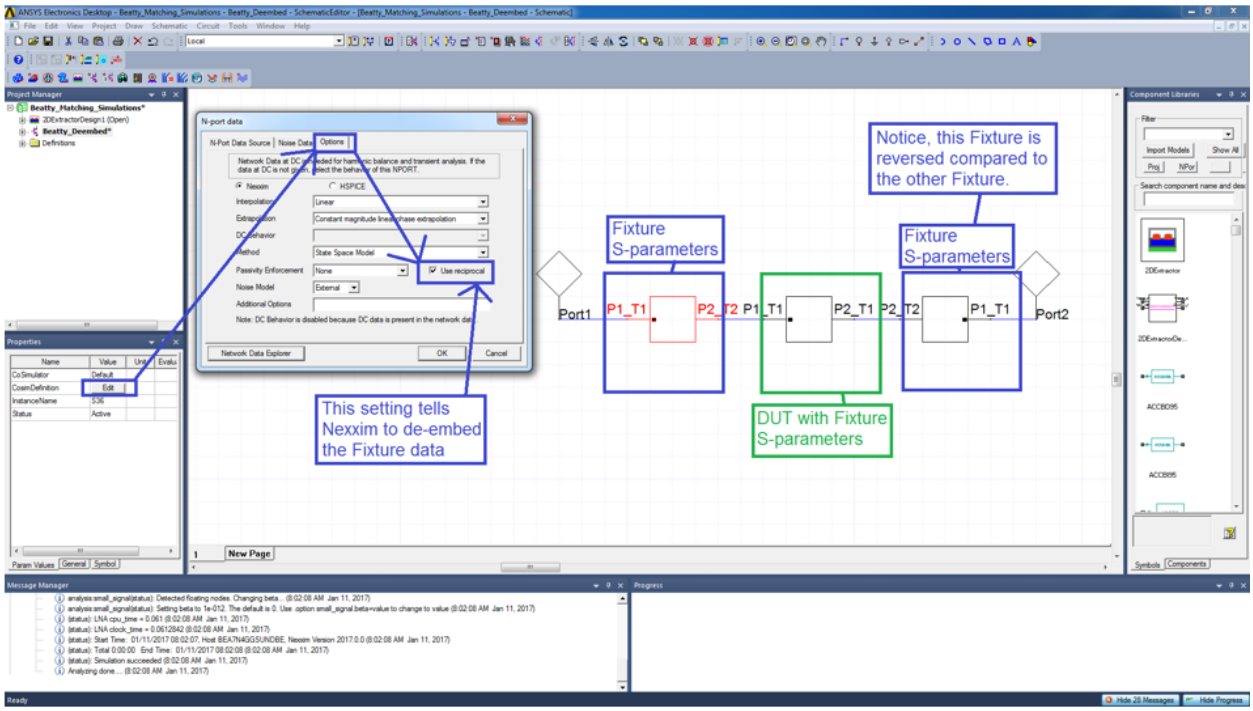
- De-embedding will be questionable above this frequency
- Still good for the “lower” frequency data.



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

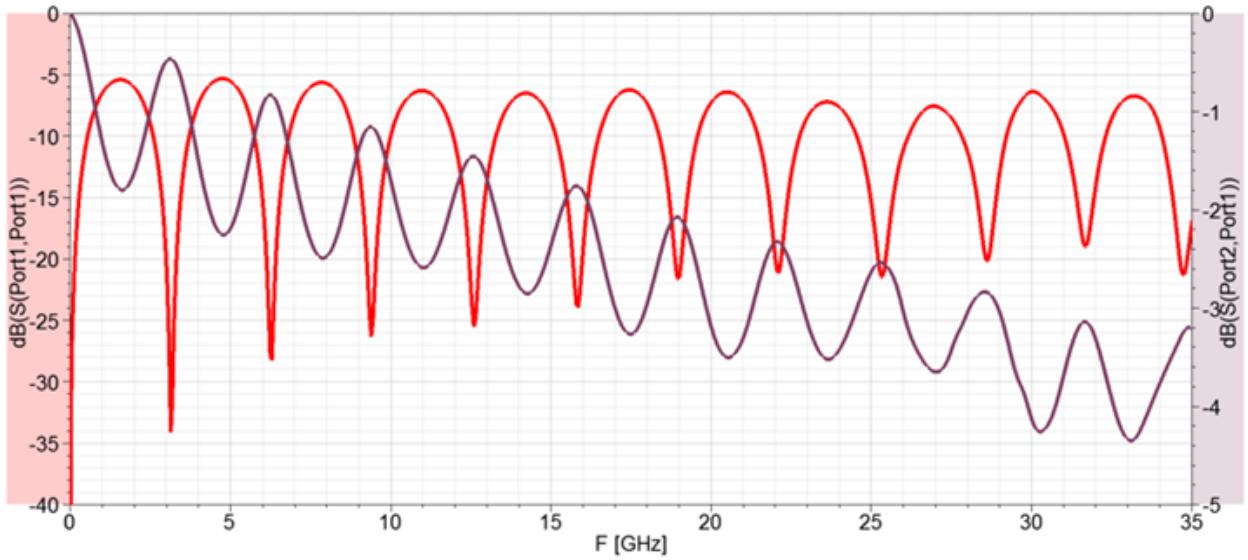
De-embed in Nexxim

- Get Fixture S-parameters using Automatic Fixture Removal (AFR as described in Reference [5]).
- Cascade Fixture S-parameter block (with reciprocal turned on) with measured data to get just the Design Under Test (DUT).



De-embedded Results in Nexxim. We clearly see the resonances

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Q2D Model

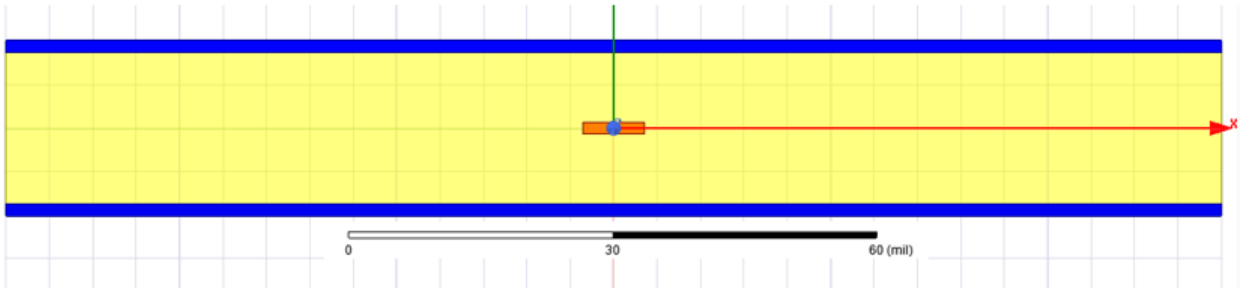
Now make Q2D model of the transmission lines and perform circuit simulations with Nexxim

Fully Parameterized so we can optimize response to extract material properties.

- Width, thickness
- Metal conductivity and surface roughness: Roughness assumed the same on top/bottom of trace and on two GND planes
- Dielectric constant and loss tangent.

Fix the following because they were measured:

- Line width is 7 mils (for 50 Ω sections) and 21 mils for Beatty Section
- Metals are all 1.4 mils thick
- Total substrate thickness (from Top GND to Bottom GND) is 17.4 mils.

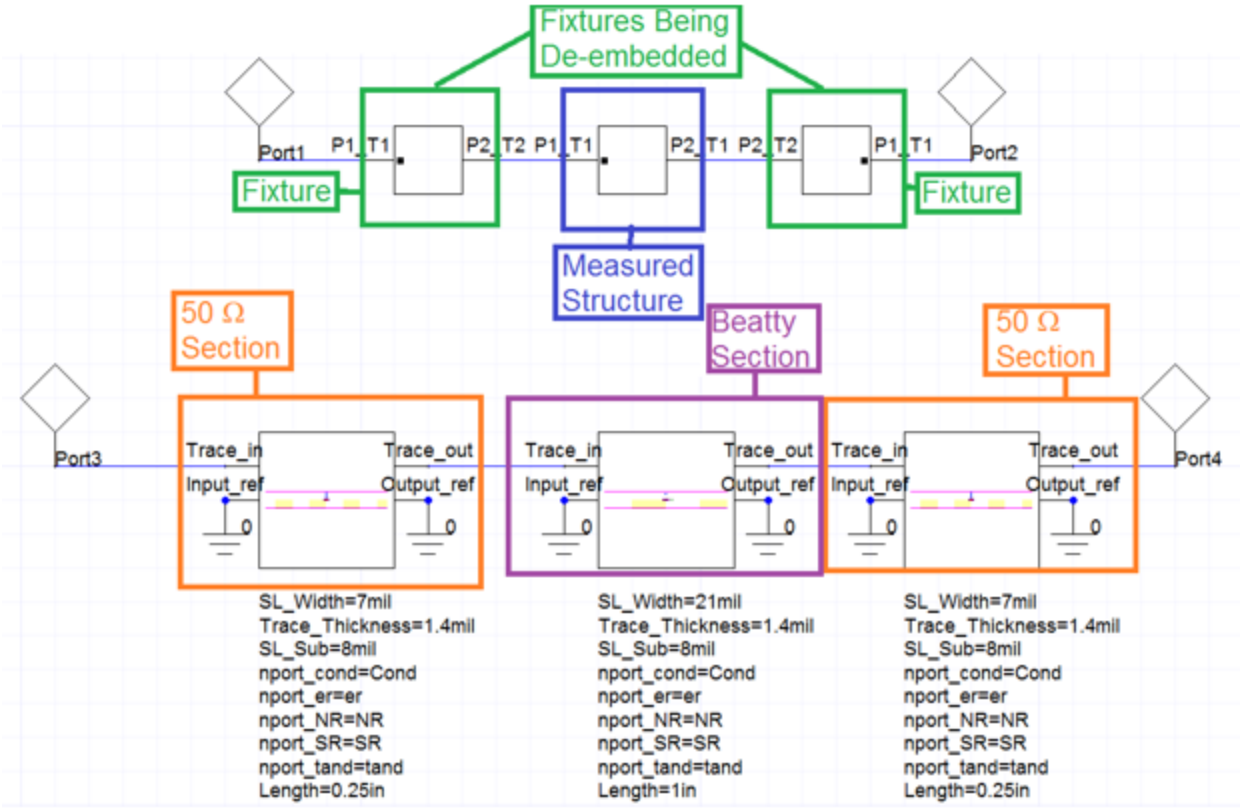


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Nexxim with Q2D Model

Run the Q2D model in Nexxim with the following to compare to our measured results.

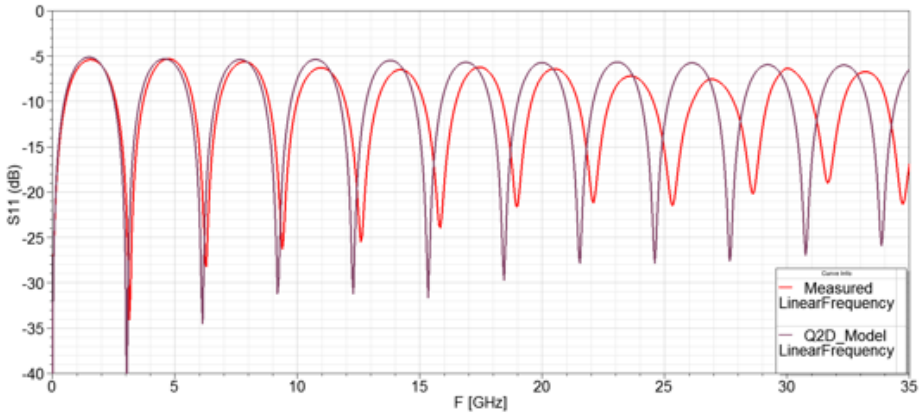
- Copper Conductivity = 5.8×10^7 S/m
- Dielectric Constant = 3.66
- Loss Tangent = 0.004
- Hall-Huray Nodule Radius = 0.4 um
- Hall-Huray Surface Ratio = 1.9



Nexxim Model Results

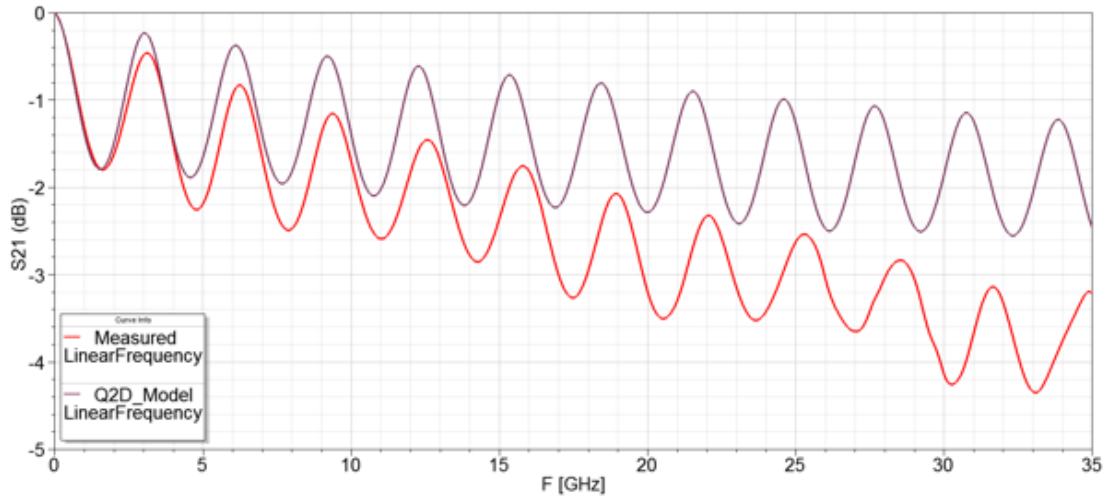
Notice the Q2D model does not match

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



S_{11}

Measured is "Red"



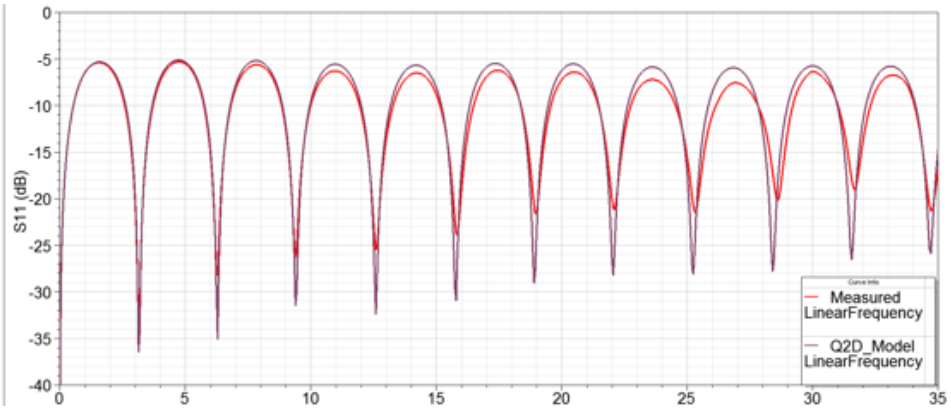
S_{21}

In Nexxim, you need to make ε_r lower so that the maxima/minima are further apart

Nexxim Model with Changed ε_r

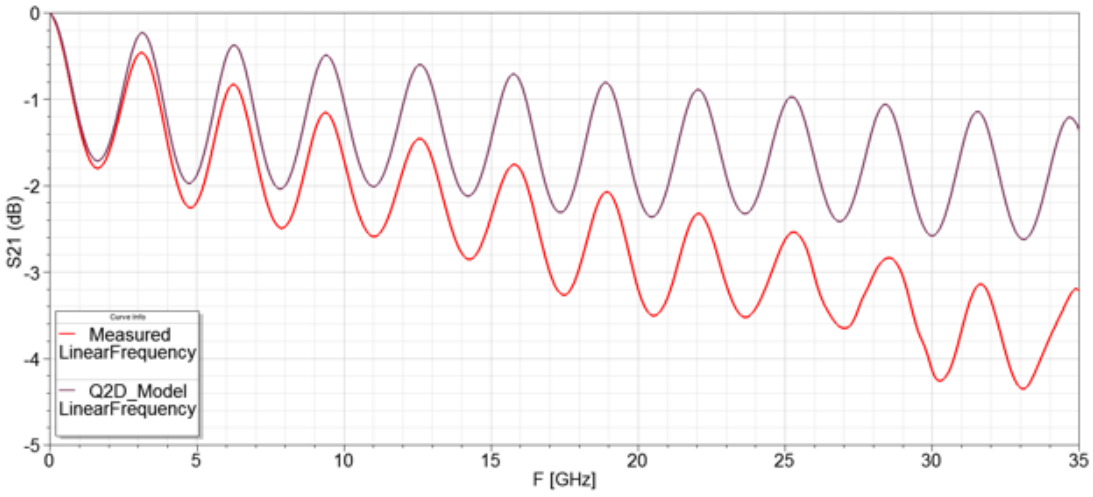
Use Optimetrics to automatically sweep ε_r. Notice ε_r =3.48 is best.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



S_{11}

Measured is "Red"



S_{21}

Now look at DC Resistance

Nexxim Model, Changing Conductivity

Use Optimetrics to automatically sweep Conductivity.

Notice is closest $5.0x10^7$ S/m. This has very little impact on results.

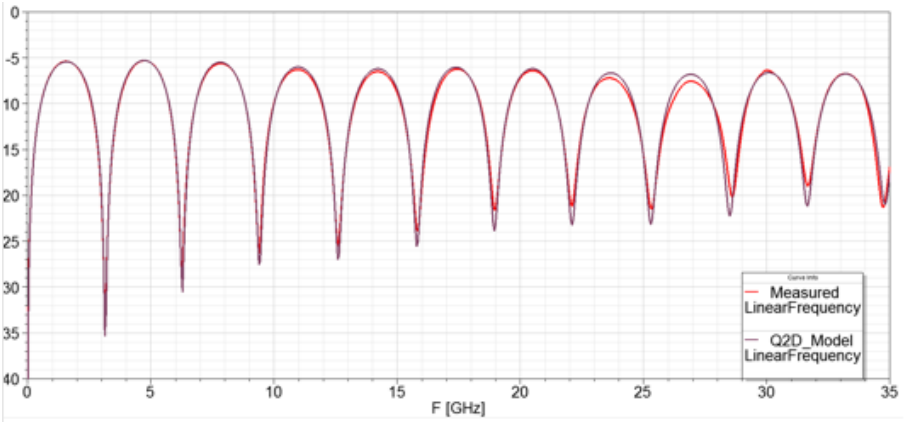
Conductivity (S/m)	DC Resistance (Ω)	Measured Resistance (Ω)
$5.0x10^7$	0.070643	0.070691
$5.1x10^7$	0.070241	
$5.2x10^7$	0.069855	
$5.3x10^7$	0.069483	
$5.4x10^7$	0.069125	
$5.5x10^7$	0.068780	
$5.6x10^7$	0.068448	

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Conductivity (S/m)	DC Resistance (Ω)	Measured Resistance (Ω)
$5.7x10^7$	0.068127	
$5.8x10^7$	0.067817	
$5.9x10^7$	0.097518	
$5.10x10^7$	0.067228	

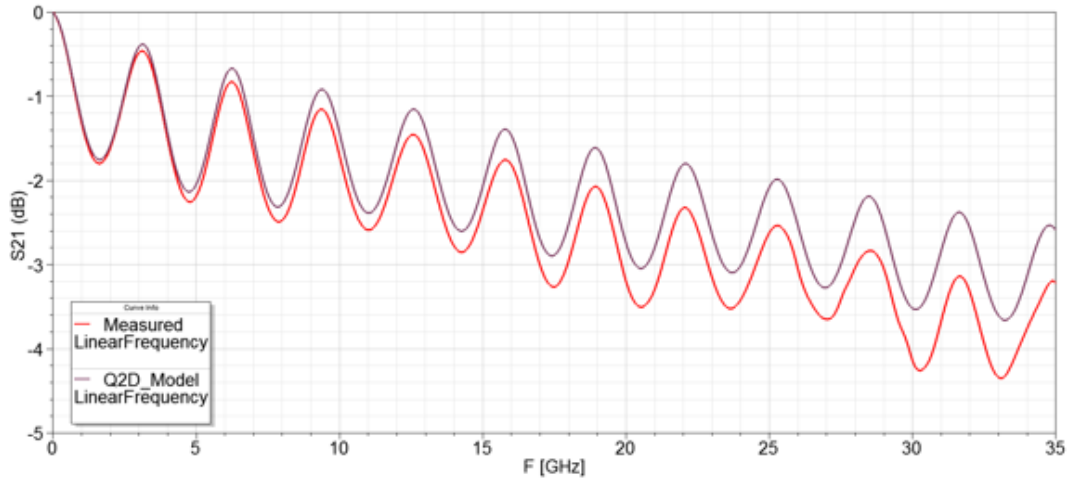
Observations So Far

- We have determined ε_r =3.48 and conductivity of the metal is $5.0x10^7$ S/m
- Add more roughness to make loss match in lower part of band.
- Use Optimetrics to automatically sweep Nodule Radius and Surface Ratio
- Nodule Radius = 0.5um and Surface Ratio = 2.9 appears to be best fit.



S_{11}

Measured is "Red"



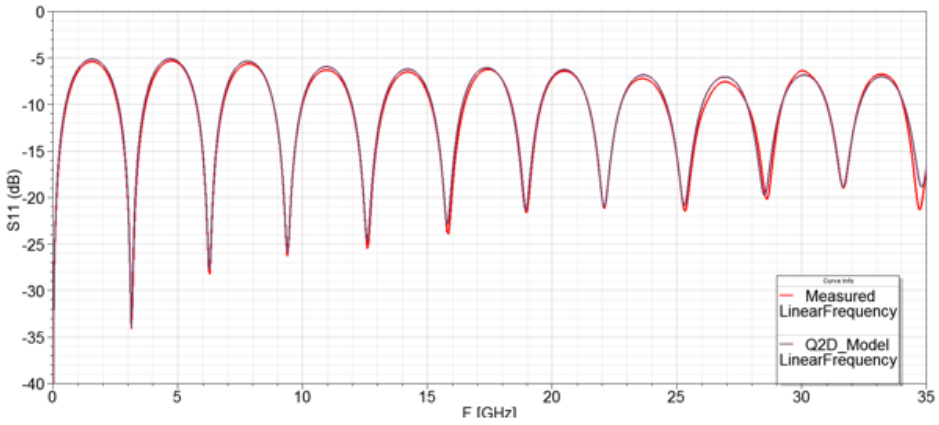
S_{21}

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Change Loss Tangent

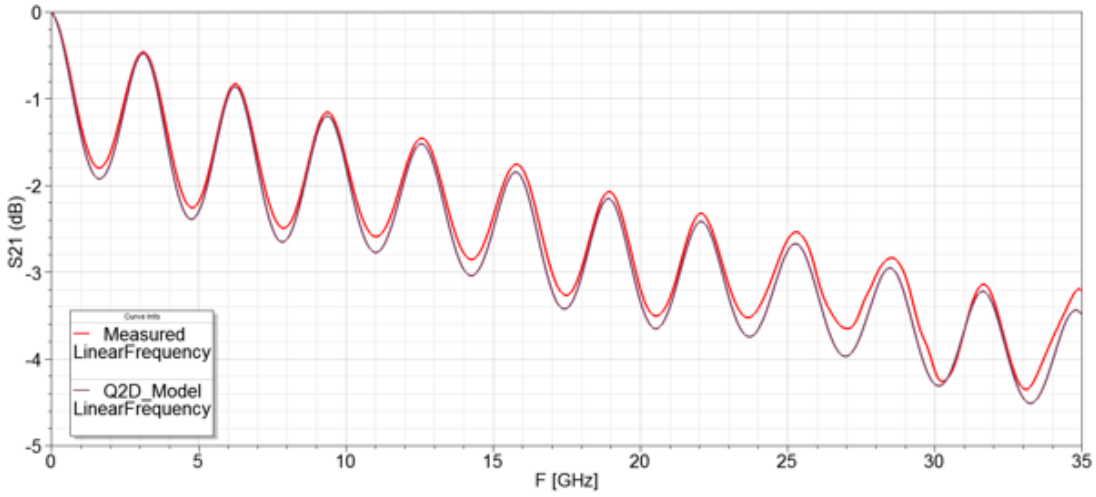
Use Optimetrics to automatically sweep Loss Tangent.

Loss Tangent = 0.008 appears to be best fit at high frequency.



S_{11}

Measured is "Red"



S_{21}

Final Values from Nexxim and Q2D

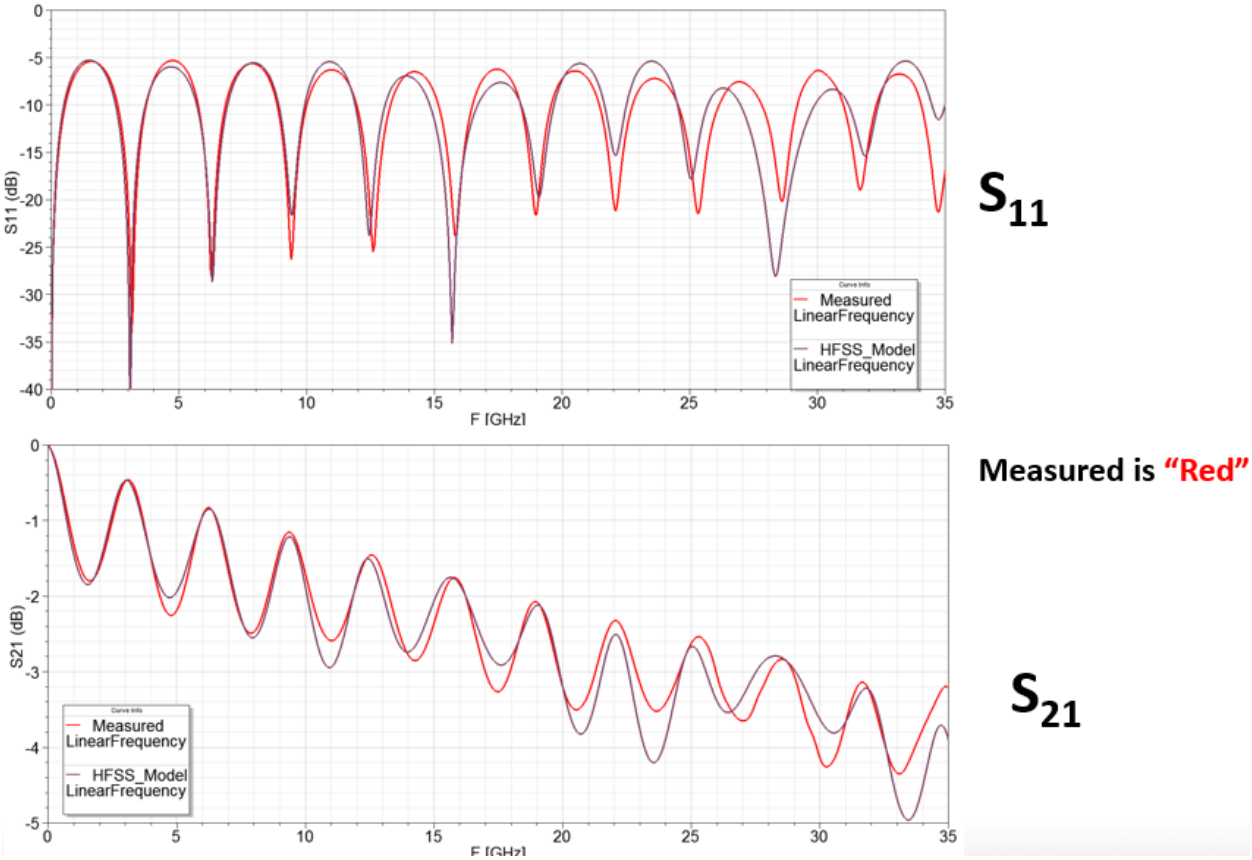
The final tuning gives:

- Copper Conductivity = 5.0×10^7 S/m
- Dielectric Constant = 3.48
- Loss Tangent = 0.008
- Hall-Huray Nodule Radius = 0.5 um
- Hall-Huray Surface Ratio = 2.9

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use Optimizer to match S-parameters as well. The goal of this example is show how each parameter changes the response in order to build your intuition in using them.

Compare HFSS Simulation with Extracted Parameters



Final Tuning Values

- Copper Conductivity = 5.0×10^7 S/m
- Dielectric Constant = 3.48
- Loss Tangent = 0.008
- Hall-Huray Nodule Radius = 0.5 um
- Hall-Huray Surface Ratio = 2.9

Summary of the Method for using Nexxim and Q2D to Determine PCB Parameters for HFSS

- Use the Beatty Standard to characterize loss
- Use proper calibration, or de-embedding, to get to actual structure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use Q2D with Nexxim
    - Deembed using Nexxim
    - Quickly (in seconds) simulate variations of line to match measured data
- Validate extraction with HFSS simulations
    - Ideally have multiple structures to measure and simulate to verify parameters.

References for Method for Determining PCB Material Properties

[1] R.W. Beatty, “2-Port l/4 Waveguide Standard of Voltage Standing-Wave Ratio,” *Electronics Letters*, vol. 9, no. 2, pp. 24-26, January 25, 1972.

[2] H. Barnes, R. Schaefer, and J. Moreira, “Analysis of test coupon structures for the extraction of high frequency PCB properties,” Signal and Power Integrity (SPI), *2013 17th IEEE Workshop* on May 2013.

[3] R.B. Marks, “A Multiline Method of Network Analyzer Calibration,” *IEEE Transactions on Microwave Theory and Techniques 39*, pp. 1205-1215, July 1991.

[4] A. Raghavan, N. Srirattana, J. Laskar, *Modeling and Design Techniques for RF Power Amplifiers*, Wiley, Hoboken, N.J., 2008.

[5] H. Barnes, “Advances in ATE Fixture Performace and Socket Characterization for Multi-Gigabit Applications” in *DesignCon 2012*.

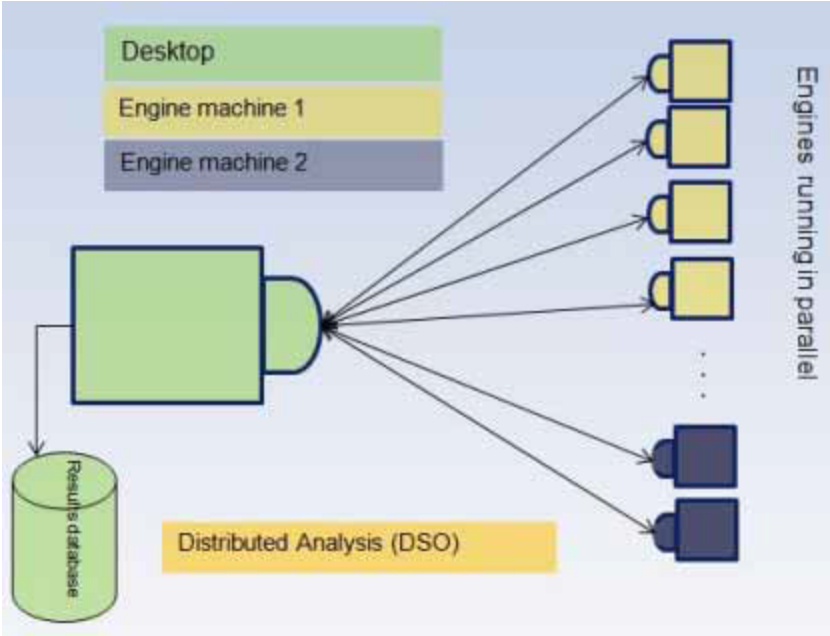
[6] D. M. Pozar, *Microwave Engineering*, Fourth Ed., Wiley, Hoboken, N.J., 2012.

[7] G. Gonzalles, *Microwave Transistor Amplifiers: Analysis and Design*, 2nd ed., Prentice-Hall, Upper Saddle River, N.J., 1997.

Large Scale DSO Theory

The parametric analysis command in Desktop computes simulation results as a function of model parameters, such as geometry dimensions, material properties and excitations. The parametric analysis is either performed on a local machine, where each variation is analyzed serially by a single engine, or distributed across machines through a DSO license. Desktop's DSO analysis runs multiple engines in parallel, thus generating results in a shorter time. In the Regular DSO algorithm, the parametric analysis job (Desktop) runs on master node, which in turn launches one or more distributed-parallel engines on each machine allocated to the job. Desktop distributes parametric variations among these engines running in parallel. As variations are solved, the progress/messages and variation results are sent back to Desktop, where they are persisted into the common results database on master node, as illustrated below:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A single engine can now span multiple machines with auto multi-level DSO.

Regular DSO Bottleneck

As per above illustration, Regular DSO's speedup is limited by the resources of the centralized 'Desktop' bottleneck. It's been observed that DSO becomes unreliable at a certain point, as the number of engines and number of variations is increased. The term 'large scale parallel' can be used to define this tipping point. For a given model, a 'large-scale parallel' job denotes scenarios, in terms of the number of distributed-parallel engines and the number of parametric variations, where the Regular DSO runs into centralized bottlenecks that result in one or both of: progressively smaller speedups, unreliability.

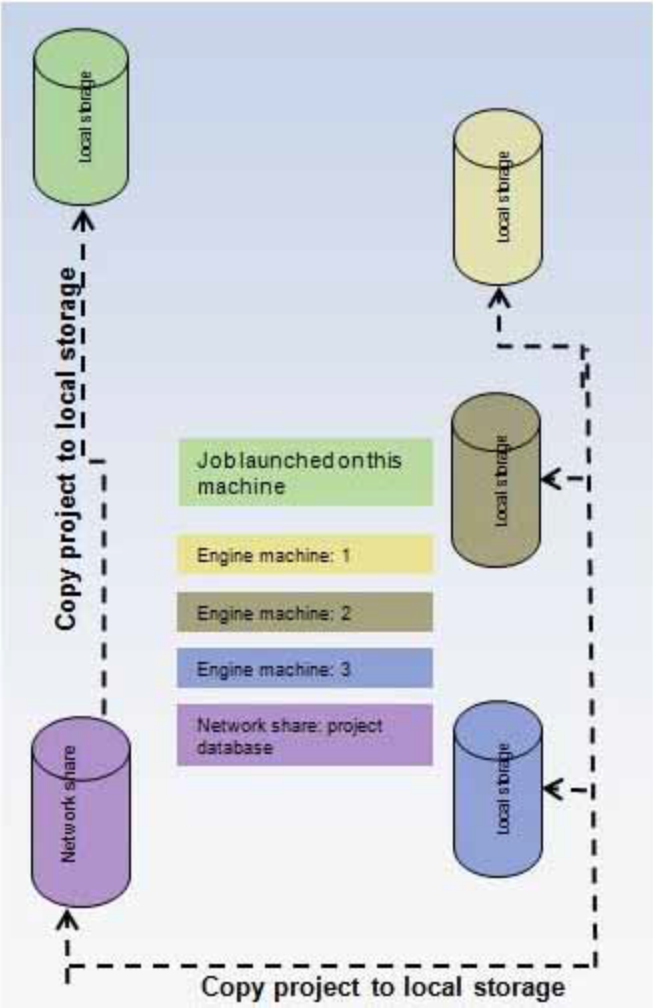
With the advent of economical availability and timely provisioning of compute resources, product designers have access to large compute clusters to run their simulations. And they are throwing larger and larger number of compute resources at simulation jobs, in order to obtain results faster. The parametric DSO needs to meet this challenge and target linear speedup for 'large-scale parallel' jobs. The Large Scale DSO feature is targeted toward 100% reliability and linear speedup of large-scale parallel DSO jobs.

Key Algorithms/Concepts for Large Scale DSO

- **Embarrassingly Parallel algorithm**: Large Scale DSO exploits the embarrassingly parallel nature of parametric runs. The solve of each variation is made fully independent of another variation, by replacing Regular DSO's centralized database with per-engine distributed databases
- **Distributed databases**: For each distributed-parallel engine, the database of the input model is cloned to local storage of engine's compute node (as illustrated in the following

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

picture). Parallel analysis is performed on these cloned models and the analysis results also go the corresponding location on local storage. As each engine has it's own database, there is no 'shared database' contention between any two engines. There is negligible network traffic as analysis/computations are contained in a single machine and use just the 'local resources'.



- **Hierarchical activation of engines**: In the Large Scale DSO algorithm, engine activation is done hierarchically, where the overhead of launching of engines is also shared across the nodes. Such hierarchical activation improves reliability of the activation phase of large-scale parallel jobs. A new 'desktopjob' program encapsulates hierarchical activation, as illustrated in the following pictures. In this approach, the 'root' desktopjob (Root DJ) activates a 'level-one' child desktopjob (DJ(L1)) for each unique node allocated to the DSO job. Each level-one desktopjob activates one or more 'leaf' desktopjobs (DJ(L2)) equal to the number of distributed engines per node. A leaf desktopjob in turn runs Desktop in batch mode, to perform local-machine parametric analysis to solve the variations assigned to this engine.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Decentralized Load balancing:** A parametric table is divided into regions of equal number of variations, with the number of regions equal to the number of engines. In the following illustration, the analysis of 30 parametric variations is distributed among 5 engines. Each engine solves it's assigned region as a 'local machine' parametric analysis. Large Scale DSO job is considered done once all engines are done with their assigned variations.
- **Distributed results postprocessing**: As engine is done with the solve of a variation, it extracts results for the solved variation before progressing to the analysis of next variation. The extracted results are saved to the local storage. When the engine is done with analysis of all variations, the extracted results are transferred from local storage to the results folder of the input project.

Redistribution Feature

Large Scale Distributed Solve Operation could submit a parametric setup to be solved in multiple machines, each machine may launch multiple EM-Desktop processes to solve the assigned variations(Design Points). Variations are distributed to each task(EM-Desktop process) equally, regardless of the machine hardware and each variation's complexity. That may resultin some tasks finishing earlier than others, in some extreme case some tasks may hours behind fastest task.

Now, redistribution occurs when a task finished its own assignment, it calls back to the L2 to ask for new assignment, L2 forward the request to L1, L1 forward it to L0, L0 may pick one of the slow task to remove some variations or return with no more assignment. When a task is picked to remove unsolved variation, L0 calls L1(may be different L1), L1 forward it to the selected L2, L2 makes a request to the EM-Desktop to remove some variations from its queue. EM-Desktop returns the removed variation indexes, or error code if it fails. In L0 if error is returned, it marks the selected task failed to respond the remove request, to void picking it again. L0 returns the result through L1-L2 back the EM-Desktop.

The Ansys Electronics Desktop will enqueue the new assignment and then start solving those new variations. If the return data indicates error, it makes another call to L2 to get new assignment. If no more variations return, the Ansys Electronics Desktop finishs the simulation and exit.

Geometric Objects

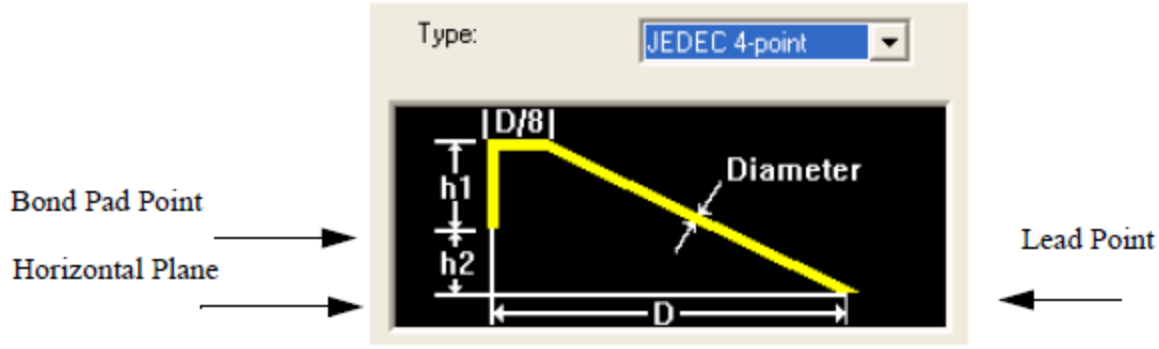
Following are supplemental technical details about working with geometric objects and complicated models:

- Bondwires
- Braided Shield Models for Cable Modeling
- Detecting and Addressing Model Problems to Improve Meshing
- Handling Complicated Models

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bondwires

A bondwire is a thin metal wire that connects a metal signal trace with a chip. You can choose to draw a standard **JEDEC 4-point** bondwire, as shown below:



where

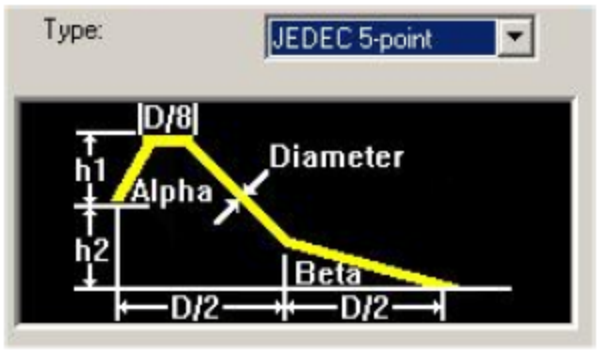
h1 = the height between the bond pad point and the top of the loop.

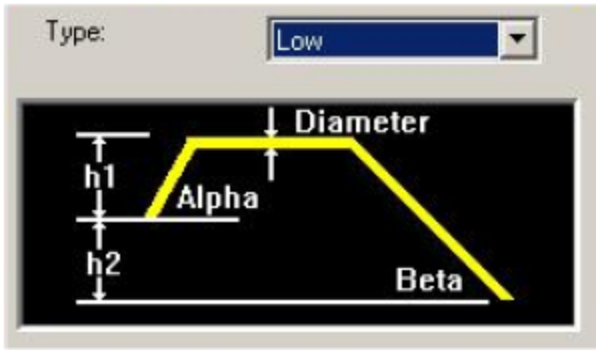
h2 = the height between the lead point and the bond pad point.

diameter = thickness of the wire.

D = the distance between the start and end points, and D/8 is the distance divided by 8.

Or you can choose to draw a **JEDEC 5-point** or **Low** bondwire, as shown below:





where

α = the angle between the horizontal plane and the wire at the bond pad point.

β = the angle between the horizontal plane and the wire at the lead point.

D = the distance between the start and end points, D/2 is the distance divided by 2, and D/8 is the distance divided by 8.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When drawing the bondwire, you first select the bond pad point, a point in 3D space that defines the bond pad position in a horizontal plane. Then you select the lead point, which indicates the distance the wire covers in the horizontal plane. HFSS will use the distance between the bond pad and lead points to calculate the height between the bond pad and the lead point, or *h2*, a value that you can modify in the **Bondwires** dialog box.

For the JEDEC types, notice that the horizontal distance on the wire is calculated as the total length divided by 8. (This horizontal distance is shown in the JEDEC 4 and 5-point figure as "D/8.") For the Low type, the horizontal distance is affected by the Alpha and Beta values as well as the length.

Braided Shield Models in Cable Modeling

The Cable Modeling feature includes an option for braided shields. This section provides technical details on the Kley model used in Cable Modeling.

Definitions

The following definitions are used throughout:

N	number of wires per carrier
C	number of carriers (belts) in the braid
α	weave angle of the braid
d	diameter of individual wires
D_0	diameter under the braid
P	number of “picks” (carrier crossings) per unit length
σ	conductivity of the braid
f	frequency in Hz
$\omega=2\pi f$	frequency in rad/sec

Kley’s Model

Kley further defines

$D_m=D_0+2.5d$	average braid diameter
$G_0=NCd/(2\pi D_m$	“minimal” fill factor
$F=G_0/\cos\alpha$	fill factor (Kley uses G for this)
$B=F(2-F)$	optical coverage
$d_R=0.67d/\sqrt{\cos\alpha}$	“equivalent” wire thickness for DC resistance
$\delta=1/\sqrt{\pi f \mu_0 \sigma}$	skin depth

Kley then derives the transfer impedance of the braid as

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$Z_{T,Kley} = Z_D + j\omega L_T + (1+j)\omega L_s$$

Here Z_D is the diffusion component of the transfer impedance, given by

$$\gamma = \frac{1+j}{\delta}$$

$$Z_D = \frac{4}{\pi d^2 \sigma N C \cos\alpha} \cdot \frac{\gamma d_R}{\sinh \gamma d_R}$$

L_T is the combined hole and braid component of the transfer impedance, given by

$$L_T = \frac{\mu_0}{C}\left[0.875\frac{\pi}{6}(2-\cos\alpha)\cdot(1-F)^3 e^{-\tau_H} - \frac{0.11}{N}\cos(2k_1\alpha)\right]$$

Where

$$\tau_H = 9.6F\sqrt[3]{B^2 d/D_m}$$

$$k_1 = \frac{\pi}{4}\left(\frac{2}{3}G_0 + \frac{\pi}{10}\right)^{-1}$$

$(1+j)\omega L_s$ is the skin impedance of the holes, given by

$$(1+j)\omega L_s = \frac{1+j}{\pi\sigma\delta D_m}\left[10\pi G_0^2 \cos\alpha\cdot(1-F)e^{-\tau_E} - \frac{3.3}{2\pi G_0}\cos(2k_2\alpha)\right]$$

Where

$$\tau_E = 12F\sqrt[3]{B^2 d/D_m}$$

$$k_2 = \frac{\pi}{4}\left(\frac{2}{3}G_0 + \frac{3}{8}\right)^{-1}$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometric Relations

Figure 2 below shows a braided shield, along with some of the parameters that define it. The shield has been flattened, as if it were cut down one side and smoothed out. Defining parameters are *P*, the number of "picks" (holes) per unit length, and the shield's average diameter D_m. The number of carriers (belts of wire) is *C*, and the number of wires per carrier is *N*. Another key parameter is the weave angle, α.

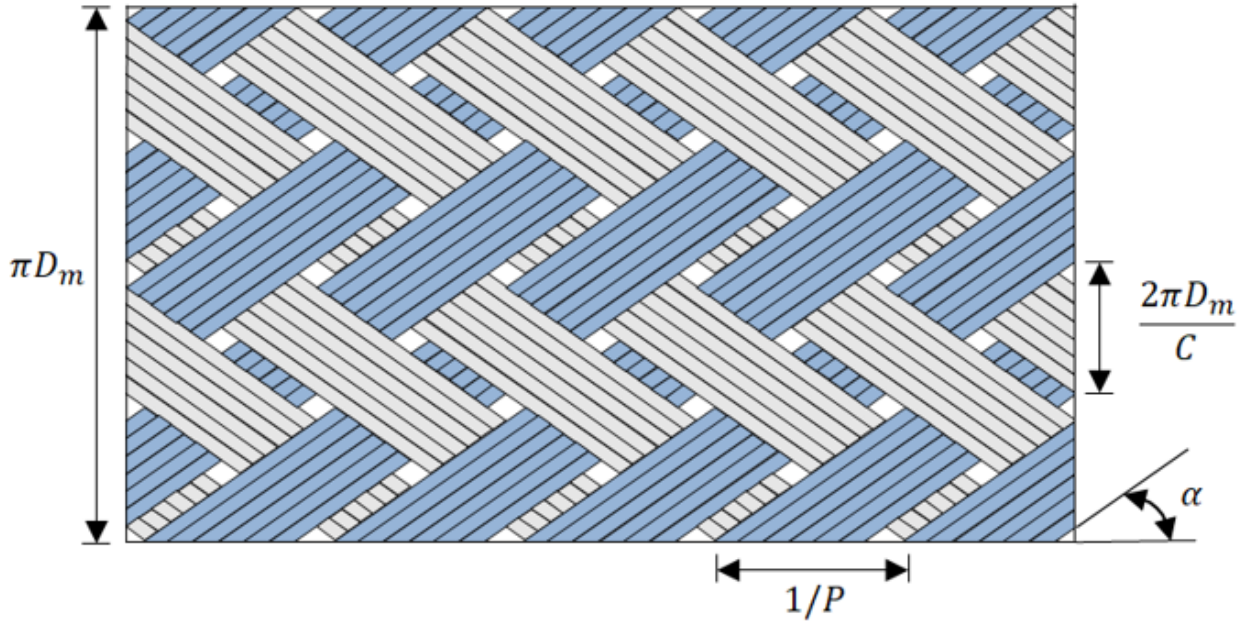


Figure 2. Portion of a braided shield (adapted from Vance).

The value of *P* can be determined by measuring the horizontal distance along the braid from one corner of the diamond-shaped holes to the next corresponding corner, and then taking the reciprocal. The number of carriers *C* can be determined by counting the carriers crossing the left or right edge of the figure. (It is 8 in this example.)

From the figure, the weave angle is related to the other parameters by

$$\alpha = \tan^{-1}\left(\frac{2\pi D_m P}{C}\right)$$

Therefore, if we know three of the parameters $\{\alpha, P, C, D_m\}$ we can always compute the fourth.

Referring now to Fig 3 below, we can derive several more parameters of the braid model. First, the width *W* of the carrier plus the hole is

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$W = \frac{1}{P} \sin \alpha$$

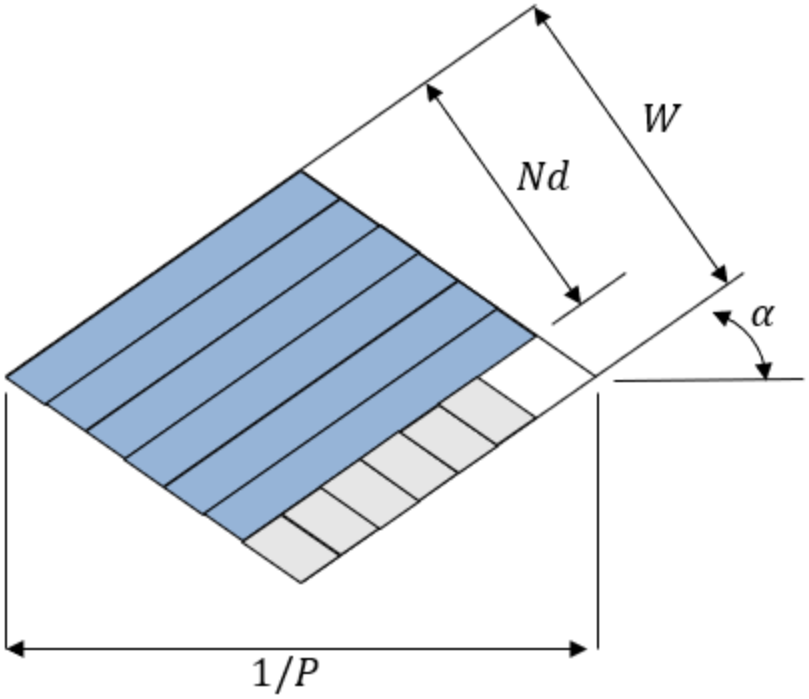


Figure 3. Close-up of one diamond of the braid (adapted from Vance).

Therefore the fractional coverage of the carrier is

$$F = \frac{Nd}{W} = \frac{PNd}{\sin \alpha} = \frac{NCd}{2\pi D_m \cos \alpha}$$

The carrier coverage *F* is different from the optical coverage *B* (the fraction of the braid's surface area covered by carriers) due to the crossing carriers. It can be seen from Figure 3 that the hole and the diamond are similar polygons, and thus the area of the hole is $(1\text{-}F)^2$ times smaller than the area of the diamond. Therefore, the optical coverage is

$$B = 1 - (1 - F)^2 = (2 - F)F$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cable manufacturers often quote the optical coverage of their braided shields. Given **B**, we can solve for *F*, as follows:

$$F = 1 - \sqrt{1 - B}$$

If the weave angle α is not known, but *B*, *N*, *C*, *d* and D_m are given, then we can find *F* as above, and then compute α as

$$\alpha = \cos^{-1}\left(\frac{NCd}{2\pi D_m F}\right)$$

Detecting and Addressing Model Problems to Improve Meshing

The following sections describe a systematic procedure to detect and address model problems that can interfere with the meshing process.

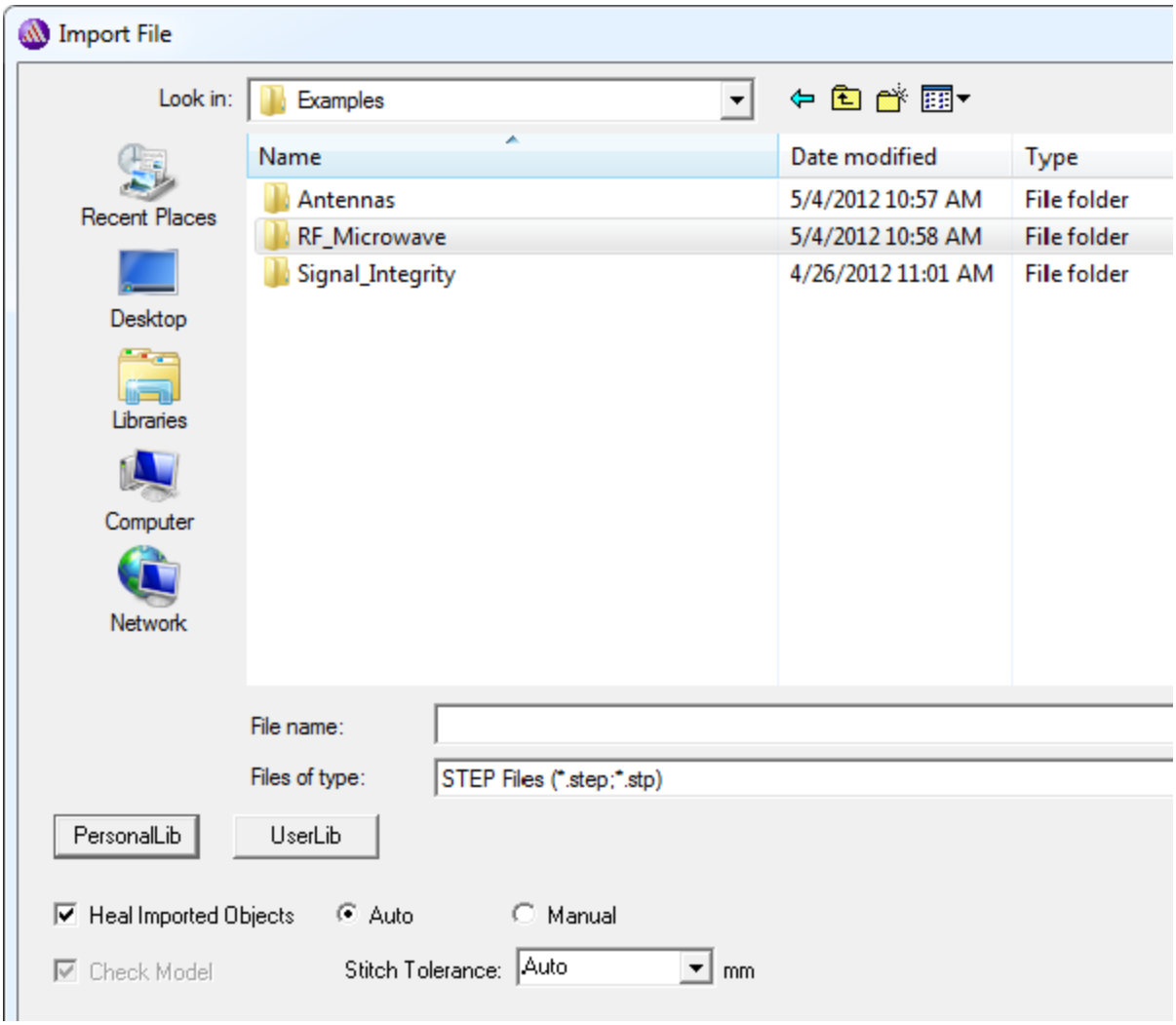
- Technical Notes: Healing During Geometry Import
- Technical Notes: Healing After Geometry Import
- Technical Notes: Removing Object Intersections
- Technical Notes: Removing Small Features
- Technical Notes: Aligning Objects
- Technical Notes: Troubleshooting if Meshing Still Fails

One: Healing during geometry import

In case you don't draw your entire geometry in the Ansys EM environment but wish to import (part of) it, in the **Import File** window you select which geometry file to import. Some formats permit healing during import (See: Healing). Selecting these formats enables a check box at the bottom of this window, "**Heal Imported Objects**." For these supported formats (except for 3D Modeler files), two modes exist, "auto" and "manual". Auto Healing will try to address kernel errors and non-manifold errors, the first two classes of potential problems listed earlier. It will also fix surface normals in the body and updating orientation of body, to avoid having a body with negative volume.

Manual healing adds tolerant stitching, geometry simplification (such as small-feature removal) and tight en gaps settings to this. You can remove small features at this stage if you wish. However, the usual approach is to apply auto-healing at this stage and leave small-feature removal until later.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Two: Healing after geometry import

Healing can only be performed on objects that have no drawing history other than "import". If necessary, object history will be automatically deleted through **Modeler>Purge History**. If that causes a warning that another object will be deleted, you may need to purge the history of that other object first, or purge the histories of several objects simultaneously. If you need to preserve the object history, you should save a separate copy before purging the history.

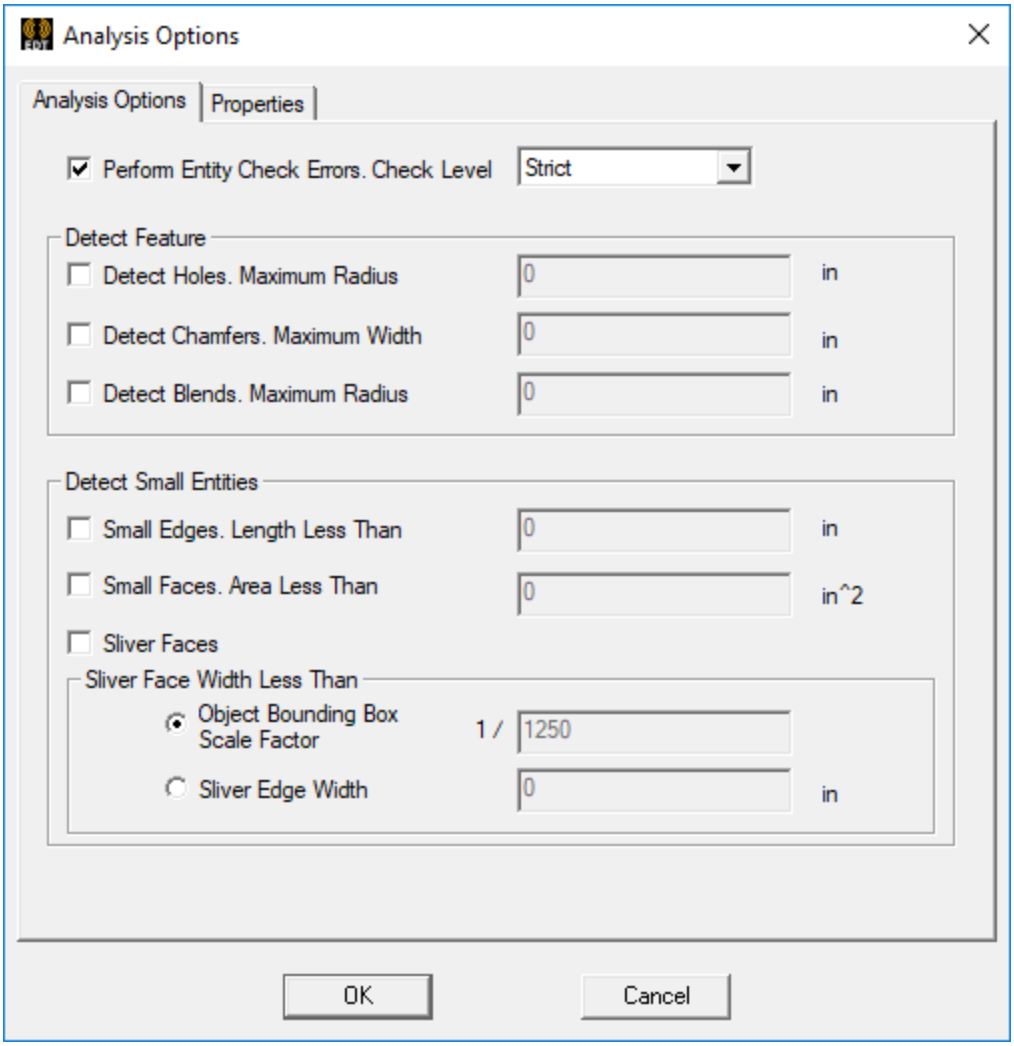
At any time after import, you can perform a Validation Check: **HFSS** > **Validation Check**. This will enable you to focus on bodies and body pairs that need attention before a mesh can be created. Note that you can set the Modeler Validation settings for Warning Only, Basic, and Strong.

1. Select the objects that have kernel errors, such as failing api_check_entity(), and the objects that have non-manifold features (See: Mixed Dimensionality).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Invoke **Modeler->Model Analysis->Analyze Objects**.

This will bring up an Analysis Options dialog where you set the strictness of the entity check, detection and thresholds for holes, chamfers and blends, thresholds for small feature detection.



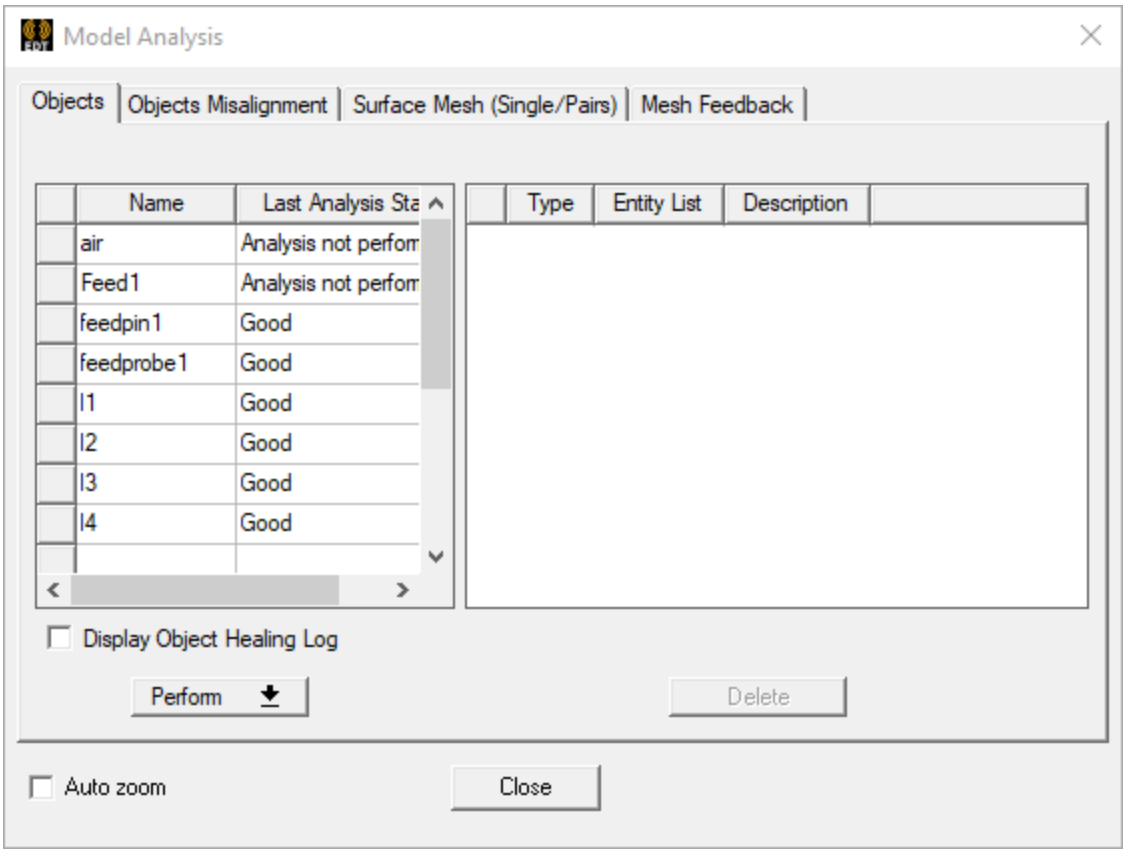
3. When you have made selections, click **OK** to start the analysis.

On completion, the **Model Analysis** dialog is displayed. All bodies in the model are shown in the objects grid along with their status. Bodies can have the following status:

1. Good
2. Null Body
3. Analysis not performed
4. Invalid entities found

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Small-entity errors.



Invalid-entity errors are kernel errors and non-manifold errors. Small-entity errors are small faces, sliver faces and small edges that are optionally detected based on user-defined parameters.

> **Note:**
>
> Invalid-entity errors must be fixed before a mesh can be generated.

To fix invalid entity errors:

1. Choose the bodies that have "Invalid Entities Found."
2. In the same **Model Analysis** window, choose **Perform>Heal Objects**, with or without an optional setting for small-feature removal.

   In most cases, the bodies will be healed and the errors fixed.

3. If errors still persist, choose "offending" faces and edges and click on **Delete**.

   This will replace the selected face/edge entity by a tolerant edge/vertex respectively.

In order to avoid unintended changes, it is good practice to do the following:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

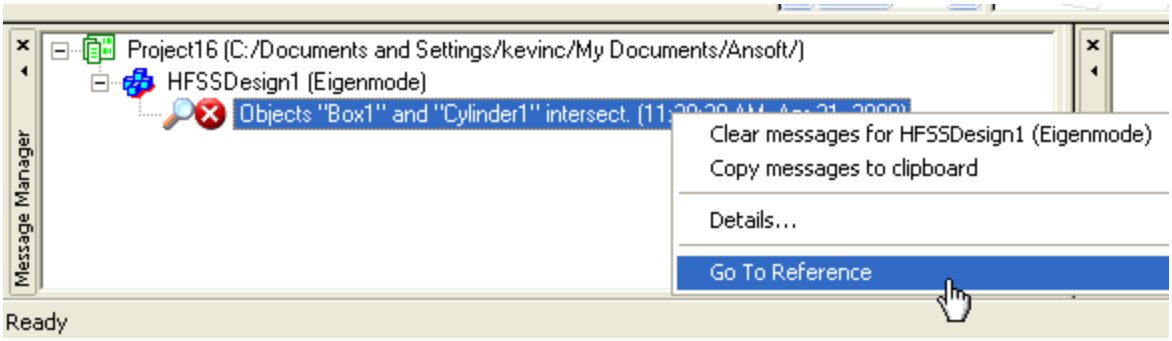
1. At the bottom of the **Model Analysis** window, check the box "Auto Zoom to Selection."
2. Select one face or edge at a time
3. Decide for each face and edge whether you want to delete it.

> **Note:**
>
> Healing causes changes to the geometry and topology of the body being healed. Validation check has to be re-run after healing is done to identify body pairs that intersect. It is possible that after healing, bodies that were disjoint before now overlap.

In some cases the replacement of the face/edge by tolerant edge/vertex will fail. If the object remains invalid, you know at this point what parts of the object are invalid. You will need to change that part of the object manually, either in Ansys EM's drawing environment or in the original CAD tool, to make it pass. Often, the invalid entities are in small details that can be changed without noticeably affecting the results of the electromagnetic analysis. For example, it may be possible to create a small object, well placed in the "offending" region, and to unite it with or subtract it from the problematic object, such that the "offending" details don't exist anymore.

Three: Removing Object Intersections

If there are any intersecting objects, a Validation Check will list them. You must eliminate object intersections before a mesh can be created. After you run a validation check, you can right-click on an intersection error message in the Message window, and select **Go to reference** from the shortcut menu. This selects the intersecting objects.



In complicated models, before making changes, it is good practice to inspect the overlap visually. A way to do this is to:

1. Duplicate both objects and place the copies outside the model.
2. Perform Boolean Intersect on the copies.

   This will show you what causes the intersection and will help you decide how to remove it.
3. Then, delete the copies.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The easiest way to eliminate object overlap is to subtract one object from the other, in the order that leaves the desired material in the region of overlap. If the overlap is very small and you can choose the order of subtraction, choose one that does not create coincident true surfaces, if possible.

Caveat: if as a result of a subtraction the model has pairs of true surfaces that are coincident, that is, smooth curved surfaces that fit exactly one inside the other, you will make it harder for the mesh generator to create a mesh. This is because Parasolid will create segmentations on each of these surfaces, and these segmentations are not guaranteed to fit. Setting a small value for **Surface Deviation** under **Mesh Operations>Assign>Surface Approximation** increases your chance of success in such a case, but it is better to avoid such situations if you can.

A way to eliminate object intersections without subtraction is to split one object in parts, in such a way that some parts are completely enclosed in the other object, and some parts are completely outside the other object. Even for complicated objects, this is possible through a sequence of Boolean operations on the objects and copies of the objects.

At this point, the geometry has no kernel errors, no non-manifold objects and no partial object intersections. A mesh can be created for the electromagnetic analysis.

The procedure for fixing face improper intersections is similar. When it occurs, the error report lists a pair of faces intersecting.

1. You can detach those faces (using **Modeler>Surface>Detach Faces**), and others if necessary.
2. Select the faces and invoke either the **Modeler>Model Healing>Stitch Sheets** command (see Stitch Sheets) or **Modeler>Boolean>Unite** (see Uniting Objects).
3. Once the improper intersections have been corrected, you can stitch the set of faces back to the original model.

Four: Removing Small Features

Even though, in principle, the geometry may be ready for a mesh to be created, it is possible that small features in the geometry lead to a mesh that is unnecessarily large and contains long and thin tetrahedra that make the simulation converge slower. Small features may even cause the mesh generation to fail. By small, we mean details on an object that are thousands of times smaller than the main features of the object, and that, in most cases, are unintended consequences of the drawing history in another CAD tool. Therefore, it is advantageous to remove small features.

You may have noticed that you could have invoked small-feature removal at several earlier stages. There is no objection to doing it earlier. See Object Overlap Settings for Complicated Models. The reason why it is presented here as stage four is that the previous stages were necessary while this one is optional.

To start the small-feature removal:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click HFSS>Modeler>Analysis Options and specify the Entity Check level, make selections for holes, chamfers, and blends, their thresholds, as well as for edges, faces and slivers.
2. Select objects and invoke object analysis through **Modeler>Model Analysis>Analyze Objects**.

Alternatively, without objects selected, use **Modeler>Model Analysis>Show Analysis Dialog>Objects** and select objects from the list. In the **Model Analysis window**, invoke **Perform>Analyze Objects**.

The software will report according to the **Analysis Options** settings, such as holes smaller than a given radius, the smallest edge length and the smallest face area.

3. Upon clicking OK, the analysis is performed.

As a result of the analysis, the software presents a list of all holes, chamfers, blends, faces and edges that don't meet the thresholds set by you in the Analysis Options.

4. Check the box "Auto-Zoom to Selection" at the bottom of the **Model Analysis** window and click on items in the list.

Inspect them visually and decide whether they can be deleted. It is good practice to delete them one by one rather than deleting many at once in order to prevent unintended changes.

Sometimes, an edge or face cannot be deleted, and you get a message notifying you. In that case, either ignore it, or revisit it after deleting some other details first, or revisit it later manually in the 3D drawing environment.

At this point, the geometry has no errors, no non-manifold objects and no partial object intersections. Furthermore, there are fewer small features that were unintended or unimportant for the electromagnetic analysis, so the quality of the model has improved.

Five: Aligning Objects

Objects that touch each other in imported geometries don't always have well-aligned faces. Often, this is a consequence of the limited level of precision in the imported file. Misaligned faces can cause tiny object intersections or tiny gaps between objects, which in turn can lead to an inefficient mesh or even a failure to create the mesh.

To repair such occurrences in an automated way, you can select groups of objects and invoke **Modeler>Model Analysis>Analyze Interobject Misalignment**. This will yield face pairs from different bodies that are slightly misaligned with respect to each other.

In the window that shows this list, check the box "**Auto-Zoom to Selection**" and select face pairs from the list. When you decide that faces should be aligned, click Align Faces. In some cases, face alignment will fail if the topology of the body would change by a large amount after alignment. In that case, you can decide to ignore it, as it may not be a problem, or revisit it later manually in the Modeler environment.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

In complicated models, the **Interobject Misalignment** analysis can take a long time if you select all objects before launching the analysis. If you don't know which pairs of objects to analyze, just let the mesh generator try to make a mesh. If the mesh fails, a list will be presented to you of misalignments that the mesh generator finds suspicious but didn't want to adjust without permission. Not every misalignment in the list is always a problem: this is a list of features that might need your attention.

Note:

As face misalignments between touching objects can cause small object intersections, this alignment capability can already serve a useful role in stage three.

Six: Troubleshooting if meshing still fails

If mesh generation fails, information about the reasons for the failure is presented under **Modeler** > **Model Analysis** > **Show Analysis Dialog** > **Mesh Feedback**.

Again, check the box "Auto Zoom to Selection" and click on the errors in the list. This can give you hints about which parts of the model are causing difficulties.

For instance, there may be self-intersecting bodies or faces. Such errors can have a variety of causes, such as a face that is supposed to be planar, but of which the vertices don't quite lie in the same plane. When you zoom and search you are likely to see what causes the problem.

Also, there may be face misalignments. Once you know they exist, you can inspect them and decide whether to align them under the Objects Misalignment tab.

One of the tabs of the **Model Analysis** window is the **Surface Mesh** tab. Under that tab, you can try to create surface meshes for objects and pairs of objects. Since a surface mesh on selected objects is easier to create than a volume mesh for the whole model, this can help you to identify quickly which objects are causing difficulties and why.

Also, in order to determine which objects are causing difficulties, you can exclude objects temporarily from the model. If the mesh succeeds without them, this helps to identify the reason for failure. To exclude an object temporarily, select it and uncheck "Model" in its properties window. Then try to create the mesh again.

Once you know which objects make the mesh fail, you can try to make small changes to them that don't affect the electrical properties noticeably but help the mesh maker succeed. For example:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Zoom in on details and consider removing details;
- Find coincident true surfaces and move one of the faces over a very short distance so the pair of faces is not coincident anymore;
- Split very complicated objects into multiple less-complicated objects;
- Delete a complicated 3D ground object and create a 2D ground through a boundary condition on the appropriate faces of a dielectric;
- Replace imported objects by objects drawn in Ansys EM's 3D modeling environment. For instance, some CAD tools produce cylinders that consist of two half cylinders that have a seam where they join. The fit is not always perfect.

Finally, for coincident true surfaces, set a very small value for **Surface Deviation** under **Mesh** > **Assign Mesh Operation** > **Surface Approximation**. Parasolid will give them more segments, but you can compensate for that with **Model Resolution**. In a parametric sweep, you can experiment with settings for **Surface Deviation** and **Model Resolution**.

Handling Complicated Models

Complicated models, often imported from a CAD tool or layout tool, may slow down the interface, use a lot of RAM during file I/O and other operations, contain imperfections and object overlaps. After analysis, post processing of such models may be time consuming. The modeler has several options and features that address these problems.

- Interface Options for Complicated Models
- Geometry Imperfections and Complicated Models
- Object Overlap Settings for Complicated Models
- Post Processing Settings for Complicated Models

> **Note:**
>
> Remember that Healing always purges the history. If you heal an object for which you need to recover the history, use the Undo command. If you want to heal an object for which you want to keep the history, you should make a copy of the object for that purpose.

Interface Options for Complicated Models

To improve the speed of the interface when dealing with complicated geometries, do the following:

Under **Tools>Options>Modeler Options**, on the **Display** tab...

- Set "Default View Render" to "Wire Frame". Wire-frame rendering is faster than shaded rendering.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Turn off "Display UV Isolines". For models with curved faces, this will simplify the wire-frame display, so the rendering will be faster.
- Turn off "Visualize History of Objects". This will remove visualization of objects that are part of the model history. For large models, this is faster and uses less memory.

Under **Tools>Options>HFSS Options**, on the **General** tab...

- Turn off "Visualize Boundaries on Geometry". When a boundary or excitation is selected, it will not be shown with a "pattern". This will prevent large delays when selecting.

Under **View>Visualization Settings**

- Use larger deviations to view curved objects in less detail.

Under **Modeler>Import**

- Un-check "Check Model" and "Heal Imported Objects." This helps for complicated models: Validation and healing take considerable time for such models. Use this option to defer checking to a later stage (especially in cases where you know that you want to mesh the model as is).

Under **Tools>Options>General Options**, on the **Project Options** tab

- Turn off "Do Autosave" or set the autosave interval to a larger value (e.g., 50). Auto-save can be time consuming.

Geometry Imperfections and Complicated Models

Modeler>Import

Many formats can be handled. It is recommended to import a version of the geometry that is as close as possible to its source, rather than geometries that have been translated before from one format to another, or that have been imported into another computational tool and later exported from it.

Modeler>Validation Settings

Geometry imperfections are listed as kernel errors when executing **HFSS>Validation Check** and when starting an analysis. It is recommended to attempt to heal objects with such errors. However, HFSS enables you to bypass the errors (not the check itself) by choosing a setting under **Modeler>Validation Settings**. "Warning Only" enables you to ignore all errors. "Basic" enables you to bypass all but the most severe errors. The HFSS mesh generator has been enhanced to handle many geometry errors.

Object Overlap Settings for Complicated Models

HFSS>Set Material Override

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Complicated geometries often have small object overlaps. This option allows some intersections to be resolved automatically in the mesh. If metal intersects dielectric, the metal will override the dielectric in the region of the overlap. If objects with the same material intersect, the smaller object will override the larger. All other intersections will still be treated as errors.

Post Processing for Complicated Models

Under **Tools>Options>General Options**, on the **Miscellaneous Options** tab...

- Turn off "Dynamically update postprocessing data during edits". This will disable expensive updating of existing reports and plots.
- Turn off "Update reports on file open". This will disable expensive updating of reports and plots when opening a project.

Under **HFSS>Fields>Modify Plot Attributes**, on the Plots tab…

- Set Plot Quality to Coarse, and save as default. This will make field plots much faster. The fields will not be as smoothly approximated within each tetrahedron, but this should not be noticeable on very large meshes.

Boundaries Technical Notes

Boundary conditions specify the field behavior on the surfaces of the problem region and object interfaces. This area of the technical notes includes information about the following boundary types:

- Perfect E
- Impedance
- Radiation
- PML
- Finite Conductivity
- Symmetry
- Primary and Secondady (Lattice Pair)
- Lumped RLC
- Anisotropic Impedance
- Layered Impedance
- Infinite Ground Planes

and the following subjects:

- Frequency-Dependent Boundaries
- Default Boundary Assignments

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Perfect E Boundaries

In HFSS, perfect E boundaries represent perfectly conducting surfaces in a structure.

By default, all HFSS model surfaces exposed to the background are assumed to have perfect E boundaries; HFSS assumes that the entire structure is surrounded by perfectly conducting walls. The electric field is assumed to be normal to these surfaces. The final field solution must match the case in which the tangential component of the electric field goes to zero at perfect E boundaries.

The surfaces of all model objects that have been assigned perfectly conducting materials are automatically assigned perfect E boundaries.

> **Note:**
>
> Currents from a Symmetric E boundary will not be included in the ZPI calculation when there is only 1 conductor on the port. Otherwise, the currents will be included in ZPI calculations.

Impedance Boundaries

In HFSS, impedance boundaries represent surfaces of known impedance. The behavior of the field at the surface and the losses generated by the currents flowing on the surface are computed using analytical formulas; HFSS does not actually simulate any fields inside the resistor.

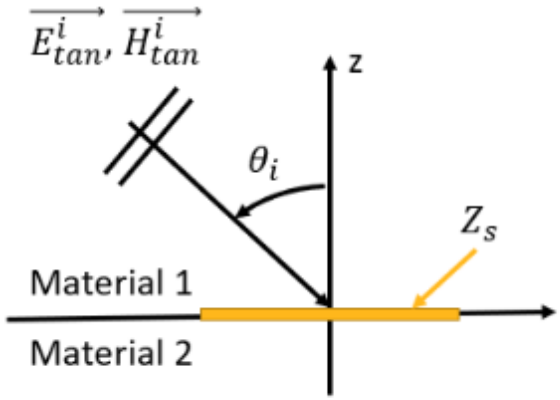
Similar to finite conductivity boundaries, the following condition applies at impedance boundaries:

$$\boldsymbol{E}_{tan} = Z_s(\hat{n} \times \boldsymbol{H}_{tan}) \quad (1)$$

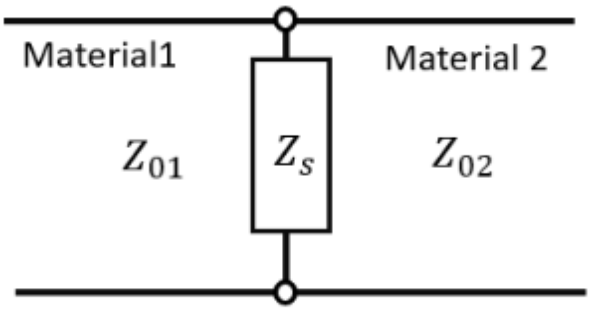
- $\hat{n}$ is the is the unit vector that is normal to the surface.
- E_{tan} is the component of the E-field that is tangential to the surface.
- H_{tan} is the component of the H-field that is tangential to the surface.
- Z_s is the surface impedance of the boundary, $Rs + jXs$, where
- Rs is the resistance in ohms/square.
- Xs is the reactance in ohms/square.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, consider a structure containing two dielectrics separated by a thin-film resistor. This resistor can be represented by an impedance boundary at the surface between the two objects. In the transmission-line equivalent model, the resistor acts as shunt impedance.

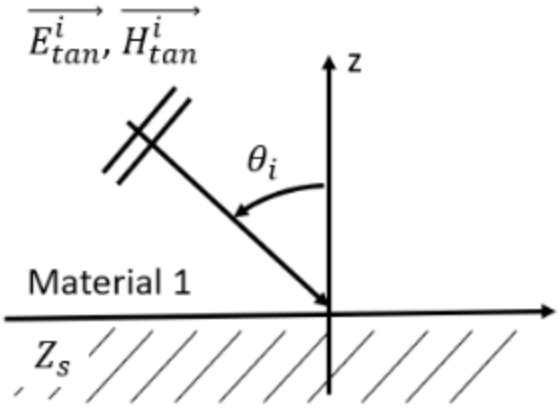


Geometry

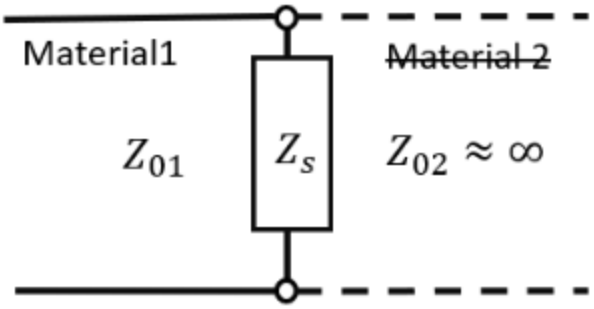


Transmission line equivalent model

Note that the boundary for that geometry is modified when the Solve Inside flag for an object is set to false. When Solve Inside is false, the boundary is replaced by a half-space material with the surface impedance. This means that the object is treated as perfect magnetic conductor.



Geometry



Transmission line equivalent model

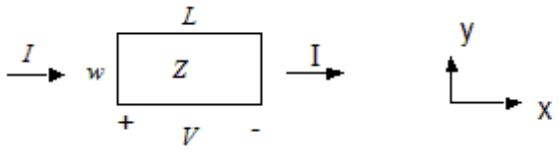
In general, for full-wave solvers, the Solve Inside flag is set automatically based on the material conductivity. The Solve Inside is set to false when the conductivity is greater than Solve Inside Threshold (100000 S/m by default), otherwise it is set to true. Users may override this behavior manually. SBR+ regions do not expose the Solve Inside flag, and such manual override is not available. The only way to modify the behavior of the Solve Inside flag in SBR+ is to modify the global Solve Inside Threshold.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Units of Impedance Boundaries

Impedance on the surface of objects, Z_S, has units of ohms per square. The units *ohms per square* indicate that the impedance, Z_S, is equal to the equivalent circuit impedance, Z, measured between the edges of a square sheet of the material.

For example, a rectangle of length *L* and width *w* has a uniform current, *I*, applied to it. It has a voltage drop, *V*, across it and an equivalent circuit impedance of *Z* ohms.



If the current density, *J*, is uniform over the rectangle then the equation (1)

$$\hat{n} \times \vec{E} = Z_s \hat{n} \times \vec{J} \quad (1)$$

becomes equation (2)

$$E = Z_s J \quad (2)$$

where

- $E = \left|\vec{E}\right|$ on the rectangle, and
- $J = \left|\vec{J}\right|$ on the rectangle.

The circuit quantities and fields are related as follows:

$$V = \int_{x=0}^{L} \vec{E} \cdot \overrightarrow{dL} = EL \quad (3)$$

$$I = \int_{y=0}^{w} \vec{J} \cdot \hat{x} dy = Jw \quad (4)$$

$$Z = \frac{V}{I} = \frac{EL}{Jw} \quad (5)$$

Substituting equation (1) into equation (5) results in the following equation:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$Z = Z_s \frac{L}{w} \quad (6)$$

Thus, when $L = w$, the equivalent circuit impedance is equal to the impedance on one square. Hence the units *ohms per square*.

If in this example $L = 2w$, the impedance would be equal to one-half of the circuit equivalent impedance for the rectangle, or the circuit equivalent impedance of one "square" of the rectangle is equal to the impedance of that square. Therefore, when entering the surface impedance for an object, you must enter the impedance per square.

Radiation Boundaries

When solving radiating and scattering structures in an unbounded, infinite domain, HFSS truncates the problem into a bounded domain and prescribes the appropriate truncation condition. This is generally known as the "radiation boundary condition". Theoretically, the radiation boundary condition should be a "transparent" condition. In other words it should not produce any unphysical reflection as a result of the artificial truncation. HFSS offers three types of radiation boundaries: first-order absorbing boundary condition (ABC), perfectly matched layers (PML), and boundary integral equations (IE).

ABC and PML Boundaries

Both the ABC and PML boundaries attempt to minimize reflections by absorbing all outgoing waves at the truncation boundary. Because of this, they can only be prescribed at convex surfaces. This is because for concave surfaces, outgoing waves will re-enter the problem domain and should therefore not be completely eliminated. While PMLs absorb any kind of waves including guided waves, ABC imitates radiation to homogeneous background space.

ABCs only absorbs normal or near normal incident waves. Thus in order to produce accurate results, it must be placed sufficiently far away from structures. The typical recommendation is at least a quarter wavelength from the radiating source, although in some cases the radiation boundary may be located closer than one-quarter wavelength, such as portions of the radiation boundary where little radiated energy is expected. ABC is a local condition and thus preserving the sparse nature of the FEM formulation.

PMLs absorb all outgoing waves by adding artificial material layers that are designed such that all of the incident waves impinging upon them are completely transmitted with minimal reflections. Thus PMLs can be placed closer than ABCs. Furthermore, the PML absorbs a much wider range of waves in terms of frequency and direction whereas ABC absorbs only normally incident waves accurately. However, PMLs in general makes it more difficult for the iterative solver to reach convergence compared to ABCs. PMLs also preserves the sparse nature of the FEM formulation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Radiating BCs and PMLs are local boundary conditions and inherit the Global Environment Material from the computational domain. They work as if the material of the computational domain were continued to infinity. FE-BI uses the user defined Global Environment material which may or may not be the same as the material of the computational domain. If they are the same, PMLs and FE-BI give the same results. If the materials are different, there is a jump in the materials which can be modeled by FE-BI, but not by PMLs.

Boundary Integral Equations

The boundary integral equations (IE) is an exact transparent condition. The Sommerfeld radiation condition at infinity as required by physics is enforced exactly through the employment of appropriate Green's functions via an integral equation method. This hybridization is commonly known as the finite element-boundary integral (FE-BI) method. Unlike ABC and PML, the IE hybrid region can be of arbitrary shape, both concave and convex thus in some cases allowing the size of the finite element solution domain to be significantly reduced. It can be placed very close to the structures and produce accurate results and from a performance point of view a minimum distance of 0.125 wavelengths is recommended. Since analytic Green's functions are known only for a few specific situations, the IE hybrid region is not allowed to touch any other boundary except an infinite ground plane. In addition the IE hybrid region does not work with symmetry planes. The IE condition is a non-local condition, producing a partial sparse and partial dense system matrix. For this reason when simulating electrically large structures, the IE condition typically requires more RAM and CPU than either ABC or PML assuming all are placed on the same geometry. However by taking advantage of conformal radiation volumes to reduce the overall finite element solution domain utilization of the IE hybrid region will result in more efficient simulation for electrically large open boundary problems.

PML Boundaries

Perfectly matched layers (PMLs) are fictitious materials that fully absorb the electromagnetic fields impinging upon them. These materials are complex anisotropic.

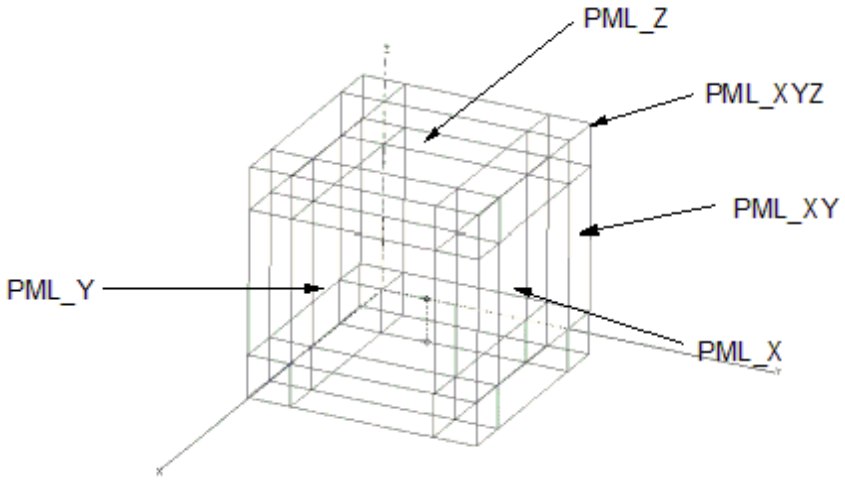
There are two types of PML applications: free space termination and guided wave termination. With free space termination, PMLs are associated with a surface that radiates into free space equally in every direction. PMLs are more appropriate than radiation boundaries in this case because PMLs enable radiation surfaces to be located closer to radiating objects, reducing the problem domain. Any homogeneous isotropic material, including lossy materials like ocean water, can surround the design.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

With reflection-free termination of guided waves, the structure continues uniformly to infinity. Its termination surface radiates in the direction in which the wave is guided. Reflection-free PMLs are appropriate for simulating phased array antennas because the antenna radiates in a certain direction.

Material Tensors Applied at PML Boundaries

PMLs materials are complex anisotropic. An example is shown below.



To ensure that there will not be any reflection at the PML/air interface, the bi-axial diagonal material tensors for x-, y- and z-directed PMLs (PML_X, PML_Y, and PML_Z) are as follows.

For PML_X:

$$\frac{[\varepsilon]}{\varepsilon_0} = \begin{bmatrix} \frac{1}{C} & C & C \end{bmatrix} \qquad \frac{[\mu]}{\mu_0} = \begin{bmatrix} \frac{1}{C} & C & C \end{bmatrix} \tag{1}$$

For PML_Y:

$$\frac{[\varepsilon]}{\varepsilon_0} = \begin{bmatrix} C & \frac{1}{C} & C \end{bmatrix} \qquad \frac{[\mu]}{\mu_0} = \begin{bmatrix} C & \frac{1}{C} & C \end{bmatrix} \tag{2}$$

For PML_Z:

$$\frac{[\varepsilon]}{\varepsilon_0} = \begin{bmatrix} C & C & \frac{1}{C} \end{bmatrix} \qquad \frac{[\mu]}{\mu_0} = \begin{bmatrix} C & C & \frac{1}{C} \end{bmatrix} \tag{3}$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where $C = a - jb$.

The tensors designated as PML_X characterize an x-directed PML corresponding to a PML wall in the yz plane. Similarly, PML_Y and PML_Z are designated tensors for y- and z-directed PMLs.

PMLs of different directions must be joined in order to construct a box with PML walls. To ensure complete coverage where the edges and corners of two PMLs meet, create edge and corner PML objects. The tensors of an edge object joining PML_X and PML_Y are as follows for PML_XY:

$$\frac{[\varepsilon]}{\varepsilon_0} = \begin{bmatrix} 1 & 1 & C^2 \end{bmatrix} \qquad \frac{[\mu]}{\mu_0} = \begin{bmatrix} 1 & 1 & C^2 \end{bmatrix} \quad (4)$$

A similar tensor construction rule is valid for joining x- and z-directed and y- and z-directed PMLs. The tensor for a corner object is a follows for PML_XYZ:

$$\frac{[\varepsilon]}{\varepsilon_0} = \begin{bmatrix} C & C & C \end{bmatrix} \qquad \frac{[\mu]}{\mu_0} = \begin{bmatrix} C & C & C \end{bmatrix} \quad (5)$$

> **Note:**
>
> This example in shows the material in GLOBAL coordinate system, while the Material Manager shows the materials in LOCAL coordinate system assigned to the object, which are then rotated to the global coordinate system by the solver.

Tensor Entries

Entering the matrices of the anisotropic materials doesn't require a special procedure. The usual anisotropic material definitions can be used for any PML structure. However, keep in mind that the efficiency of the PMLs depends on the material values assigned to them.

Setting the complex parameter C ensures that the electromagnetic field decays strongly in the PMLs. Back reflections from the bounding PECs are then kept below a prescribed bound. To accomplish this, the following inequalities have to be satisfied:

$$e \geq \frac{-\ln \rho}{2D_{min}H} = e_{min} \quad (1)$$

$$e \leq \frac{-\ln d}{2D_{max}h} = e_{max} \quad (2)$$

where

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$D_{min} = \alpha_{min} + \beta_{min} = \frac{1}{r_{max}} + \frac{\omega_{min}}{c} \quad (3)$$

$$D_{max} = \alpha_{max} + \beta_{max} = \frac{1}{r_{min}} + \frac{\omega_{max}}{c} \quad (4)$$

- $e = a = b$
- $a$and b are the real and imaginary parts of C.
- $H$is the thickness of the PML object.
- ω_{max} and ω_{min} are the minimum and maximum angular frequencies.
- r_{max} and r_{min} are the minimum and maximum distance of a radiating object to the PML surface.
- $\rho$is the bound for back reflection.
- $d$is the maximum decay characterizing the element. ($d$is approximately $3^* 10^{-3}$.)
- $h$is the thickness of one finite element.
- $c$is the velocity of light in vacuum.

Boundaries at PML Surfaces

After embedding a structure in PMLs, the next step is to specify boundaries on the outer surface of the box. The simplest way is to bound the box either with perfect electric conductors (PECs) or perfect magnetic conductors (PMCs.) In general, use PECs because they reduce the problem size.

Finite Conductivity Boundaries

In HFSS, finite conductivity boundaries represent imperfect conductors.

At such boundaries, the following equation holds:

$$E_{tan} = Z\left(\hat{n} \times H_{tan}\right)$$

where:

- E_{tan} is the component of the E-field that is tangential to the surface.
- H_{tan} is the component of the H-field that is tangential to the surface.
- Z_s is the surface impedance of the boundary. When the thickness is much larger than the skin depth, $(1+j)/(\delta\sigma)$ can be used. When not, Z_s can be calculated from the transmission line model used at a layer impedance boundary condition.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- δ is the skin depth $\sqrt{2/(\omega\sigma\mu)}$ of the conductor being modeled.
- ω is the frequency of the excitation wave in radians/second.
- σ is the conductivity of the conductor.
- μ is the permeability of the conductor.

The fact that the E-field has a tangential component at the surface of imperfect conductors simulates the case in which the surface is lossy.

The surfaces of any objects defined to be non-perfect conductors are automatically set to finite conductivity boundaries.

> **Note:**
>
> HFSS does not attempt to compute the field inside these objects; the finite conductivity boundary approximates the behavior of the field at the surfaces of the objects.

The finite conductivity boundary condition is valid only if the conductor being modeled is a good conductor; that is, if the conductor's thickness is much larger than the skin depth in the given frequency range. If the conductor's thickness is in the range or larger than the skin depth in the given frequency range *and has been defined*, HFSS takes the thickness into account.

Huray Surface Roughness Model

Surface roughness can increase conductor power losses by more than two times the loss associated with smooth conductors. In Ansys Electronics Desktop, a modification of the standard real-valued Huray surface roughness model is implemented to accurately account for these losses in both the frequency and time domains.

This modification fixes a causality problem with the original Huray model by adding some frequency-dependent reactance to the roughness-enhanced surface impedance. The purpose is to provide a complex-valued causal analytic model that exactly matches the loss results of the original model but is also suitable for time domain computations.

The original Huray model was limited to calculations in the frequency domain and did not account for the effects of roughness on the phase of the transmitted signal. These limitations have been eliminated. The enhanced causal model not only provides accurate broadband modeling of the losses in very rough copper foils (typically used in PCB manufacturing) but also accurately predicts increased phase delay due to the surface roughness.

Essentially, the smooth surface impedance (Z_{smooth}) is increased by a complex “Huray” factor (H_c), a function of frequency, given by the following equation (1):

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$H_c\left(j\omega\right) = 1 + \frac{K}{1+\left(\frac{j2\omega}{\omega_0}\right)^{-1/2}}$$

where:

- ω is the frequency in radians/second
- ω_0 is a corner frequency represented by the equation (2):

  $$\omega_0 = \frac{2}{a^2 \mu_r \mu_0 \sigma}$$

- K is a derived constant used to simplify the equation (3):

  $$K = \frac{3}{2} \cdot SR = \frac{6\pi a^2 N}{A_f}$$

  in which:

  - *SR* is the Hall-Huray surface ratio:

    $$\frac{4\pi a^2 N}{A_f}$$

  - *a* is the radius of a spherical nodule
  - μ_r is the relative permeability of the conductor
  - μ_0 is the permeability of free space

    $$\mu_0 = 4\pi \cdot 10^{-7}\ \mathrm{H/m}$$

  - σ is the electrical conductivity in Siemens/m
  - *N* is the number of nodules
  - A_f is a unit cell area

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

*a*and*SR*are the two parameters you must enter in the**Finite Conductivity Boundary**dialog box when you select the Huraysurface roughness model:

For more information about this complex causal Huray model (its necessity, background, derivation, proof of accuracy, and consistency with the original real-valued model), see Reference 6 below.

Once the Huray surface roughness factor is determined, the rough surface impedance (Z_{rough}) is determined by the equation (4):

$$Z_{rough} = H_c \cdot Z_{smooth}$$

References:

1. P.G Huray, S.G Pytel, S.H Hall, F. Oluwafemi, R.I. Mellitz, D. Hua, and P. Ye, "Fundamentals of a 3-D "Snowball" Model for Surface Roughness Power Losses", 11th Annual IEEE SPI Proceedings, May 13 – 16, 2007.
2. S.H Hall, S.G. Pytel, P.G Huray, D. Hua, A. Moonshiram, G. Brist, and E. Sijercic, "Multi-GHz, Causal Transmission Line Modeling Methodology with a Hemispherical Surface Roughness Approach", IEEE Transactions on Microwave Theory and Techniques, December 2007 pp 2614 – 2624.
3. S.G Pytel, P.G Huray, A. Moonshiram, S.H Hall, R.I Mellitz, G. Brist, F. Oluwafemi, H.M Meyer, L. Walker, and M. Garland, "Analysis of Copper Treatments and the Effects on Signal Propagation", 58th Annual IEEE ECTC, May 26 – 30, 2008, pp 1144 – 1149.
4. S.G Pytel, "Multi-gigabit data signaling rates for PWBs including dielectric losses and effects of surface roughness", PhD. Dissertation, University of South Carolina, 2007.
5. P.G Huray, O. Oluwafemi, J. Loyer, E. Bogatin, and X. Ye; "Impact of Copper Surface Texture on Loss: A Model That Works", DesignCon 2010, February 1 - 4, 2010.
6. J. Eric Bracken, Ansys, Inc., "A Causal Huray Model for Surface Roughness", DesignCon 2012, January 2012.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Groiss Surface Roughness Model

The Groiss surface roughness model is a simple function of the surface roughness and skin depth of the conductor.

The smooth surface impedance (Z_{smooth}) is increased by a factor ($fact_R$), given by the equation (1):

$$fact_R = 1 + e^{-\left(\frac{\delta}{2s}\right)^{1.6}}$$

where:

- s is the surface roughness (in length units)
- δ is the skin depth, which is defined as (2):

$$\delta = \frac{1}{\sqrt{\pi f \mu_r \mu_0 \sigma}}$$

in which:

- f is the frequency in Hz
- μ_r is the relative permeability of the conductor
- μ_0 is the permeability of free space

$$\mu_0 = 4\pi \cdot 10^{-7}\ \mathrm{H/m}$$

- σ is the electrical conductivity in Siemens/m

Once the Groiss surface roughness factor is determined, the rough surface impedance (Z_{rough}) is determined as (3):

$$Z_{rough} = fact_R \cdot Z_{smooth}$$

Limitations of the Groiss Model:

The Groiss surface roughness model is non-causal and is therefore limited to the frequency-domain (not suitable for time-domain computations). The Groiss roughness factor is bound by the following limits:

$0 < fact_R \leq 2$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Therefore, it is suitable for relatively highly polished conductors and not suitable for the much rougher copper conducting traces that are typical of manufactured PCBs.

For roughness factors greater than 2 and for time-domain computations, use the Huray Surface Roughness Model.

References:

1. S. Groiss, I. Bardi, O.Biro, K. Preis and K.R. Richter, Graz University of Technology, Kopemikusgasse 24, A-8010 Graz, Austria, "Parameters of Lossy Cavity Resonators Calculated by the Finite Element Method", IEEE TRANSACTIONS ON MAGNETICS, VOL 32. NO 3, MAY 1996

Modified Hammerstad Model

In Ansys Electronics Desktop, the Hammerstad-Jensen surface roughness model is only implemented for metal layers in SIwave designs and for Finite Conductivity boundaries assigned to applicable edges in 2D Extractor designs. For HFSS and HFSS 3D Layout designs, you can map Hammerstad surface roughness to the Huray model when assigning Finite Conductivity boundaries.

This section describes a modified Hammerstad model and how it is expressed as a function of frequency. The Hammerstad model is strictly mathematical. As mentioned above, it is possible to map the Hammerstad surface roughness data to a physics-based Huray model.

The modified Hammerstad model is described by two parameters, the RMS surface roughness Δ and the scale factor *SF*. In the original Hammerstad-Jensen model, the value of *SF* = 2.

The model equation (1) is as follows:

$$\mathrm{H_{HJ}}\left(f\right) = 1 + \left(\frac{2}{\pi}\right)\left(SF - 1\right) arc\tan\left[1.4\left(\frac{\Delta}{\delta(f)}\right)^2\right]$$

$\delta\left(f\right)$ is given by

$$\delta\left(f\right) = \frac{1}{\sqrt{\pi f \mu \sigma}}$$

Therefore, the model equation reduces to (2):

$$\mathrm{H_{HJ}}\left(f\right) = 1 + \left(\frac{2}{\pi}\right)\left(SF - 1\right) arc\tan\left[1.4\pi\Delta^2\mu\delta f\right]$$

Define the model's corner frequency, f_{HJ} using the formula (3):

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$f_{HJ} = \frac{1}{1.4\pi\Delta^2\mu\sigma}$$

Therefore, the modified Hammerstad model can be written as (4):

$$\mathrm{H_{HJ}}\left(f\right) = 1 + \left(\frac{2}{\pi}\right)\left(SF - 1\right)arc\tan\left[\frac{f}{f_{HJ}}\right]$$

> **Note:**
>
> At low frequencies, the value of $H_{HJ}(f)$ approaches 1. At high frequencies, the value approaches *SF*.

References:

1. E. Bogatin, D. DeGroot, P.G. Huray, and Y. Shlepnev. "Which one is better? Comparing options to describe frequency dependent losses." *DesignCon, 2013*.

Obtaining Broadband Matching between Huray and Hammerstad Models

This topic contains the procedure for obtaining broadband matching between Huray and Hammerstad models.

> **Note:**
>
> Before reading this topic, see the sections Huray Surface Roughness Model and Modified Hammerstad Model for the definitions of Scale Factor (SF) and to see how each of these models is expressed as a function of frequency.

The simplest way to create a Hammerstad model that is in some sense close to a Huray model is to ensure the maximum values of the models are the same.

$$SF = 1 + \frac{3}{2}\left(SR\right)$$

$$SR = \frac{2}{3}\left(SF - 1\right)$$

Then, try to line up the models at their 50% crossing points. For the Hammerstad model, that occurs when (1):

$$1 + \left(\frac{2}{\pi}\right)\left(SF - 1\right)arc\tan\left(\frac{f}{f_{HJ}}\right) = \frac{1+SF}{2}$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$f = f_{HJ}$$

For the Huray model, the 50% point is reached when (2):

$$\frac{\frac{f}{f_{HH}}}{\frac{f}{f_{HH}} + \sqrt{\frac{f}{f_{HH}}} + \frac{1}{2}} = \frac{1}{2}$$

By solving for *f*, the equation reduces to the following (3):

$$f = f_{HH}\left(1 + \frac{\sqrt{3}}{2}\right)$$

Therefore, for 50% of points to agree, the following condition must be realized (4):

$$f_{HJ} = f_{HH}\left(1 + \frac{\sqrt{3}}{2}\right) \cong 1.866 f_{HH}$$

Using equation 4 and relating back to the model parameters, the following is obtained (5):

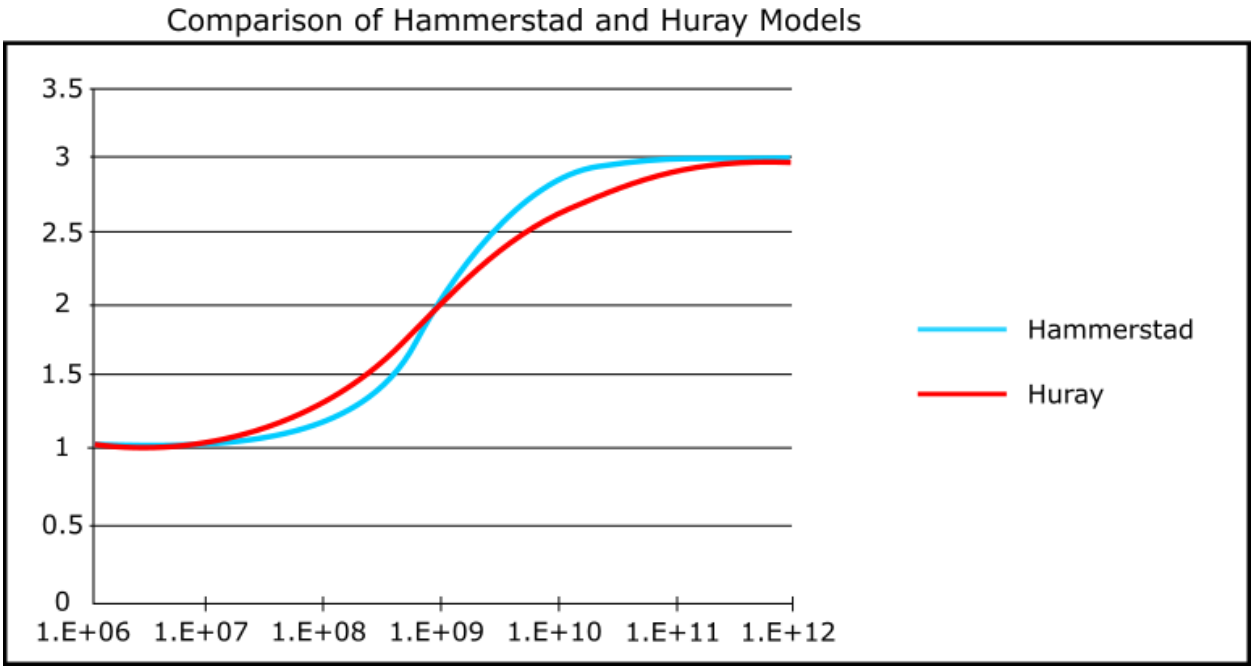
$$\frac{1}{1.4\pi\Delta^2\mu\delta} = \frac{1.866}{\pi a^2 \mu\sigma}$$

Compare the two models. For the Hammerstad model, set *SF*=3 and Δ=1.766m and let the material be copper. This conveniently yields f_{HJ} = 1 GHz.

Using the preceding relations, the corresponding Huray model has *SR*=4/3 and *a*=2.854μm.

Plots of $H_{HJ}(f)$ and $H_{HH}(f)$ are shown in the following figure.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The figure demonstrates the expected match between the two curves at the 50% point (1 GHz), and the expected match at the low and high frequency extremes. It is also clear the Hammerstad model transitions between low and high frequency more rapidly than the Huray model. This is inherent to the mathematical functions used in the model. Since the Huray model is based on the physics of the underlying loss mechanism while the Hammerstad model is a curve fit, it is reasonable to expect the Huray model to be more realistic.

The preceding model-matching procedure potentially may be critiqued as follows:

- The matching is exact at some (three) points and is decent across the entire frequency range.
- The computed snowball radius is actually greater than the RMS surface roughness value. This does not seem realistic.

An alternative procedure is to try matching the two models at low frequencies. The rationale for doing this is the loss measurements used to derive a Hammerstad model are likely to be taken mainly at frequencies well beneath the 50% point. Therefore, the higher frequency values might not be given as much weight in the matching process. This approach is described below.

Obtaining Better Low Frequency Matching

The low frequency match could be improved by shifting the Huray model curve to the right. Such a shift can be accomplished by reducing the value of the snowball radius; this increases the Huray corner frequency f_{HH}.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

No unique method exists to do this since there are many different frequency ranges and error metrics that could be used. Results obtained from some trial and error experiments reveal that a good starting point is using the broadband parameter values:

$$SR = \frac{2}{3}(SF - 1)$$

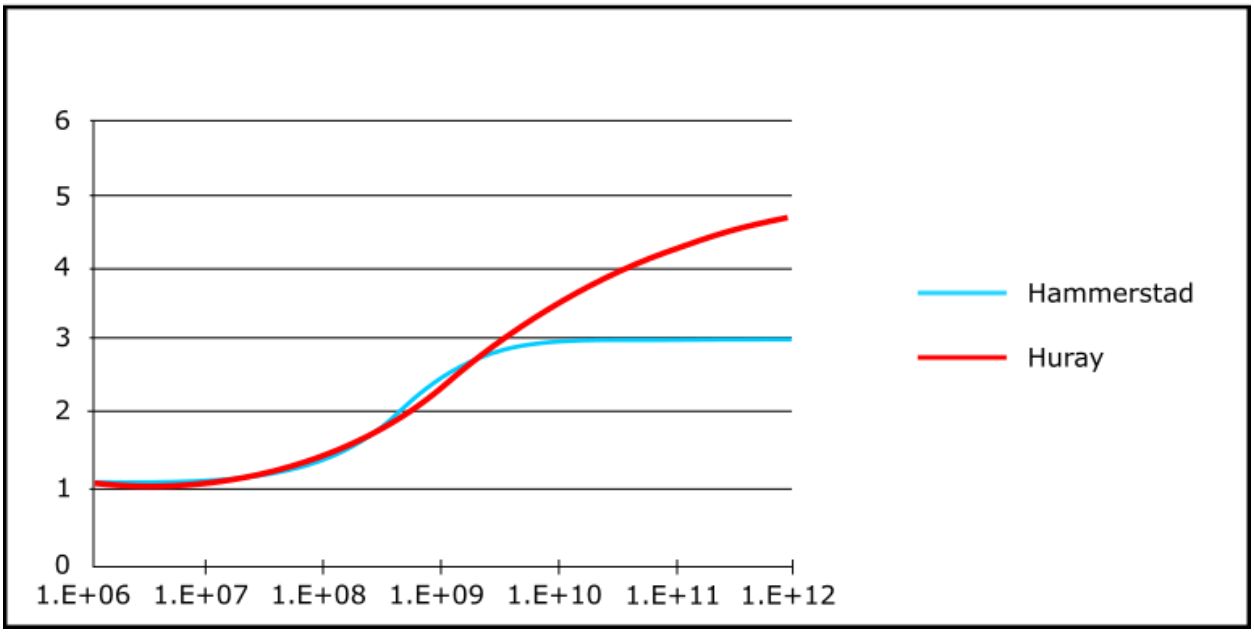
A match qualitatively similar to that shown in the following figure is obtained by choosing:

$$a = 0.65\Delta \quad SR = \frac{4}{3}(SF - 1)$$

For more information, see references below.

Setting *a* = 1.148μm and *SR* = 2.667 results in the following comparison:

Comparison of Hammerstad and Huray Models with Low-Frequency Matching Emphasized at the Expense of High-Frequency



The match between the two is very good up to about 4 GHz, well beyond the 50% crossing point of the Hammerstad model at 1 GHz. Beyond this frequency, the fitted Huray model predicts significantly higher losses (66% greater) than the Hammerstad model. This is the price paid for the improved low frequency match. Of course it was assumed the Hammerstad model may not be accurate in this higher frequency range anyway.

To summarize, the Hammerstad model is fundamentally different from the Huray model. Since Hammerstad is strictly a mathematical model and the Huray model is a physics-based model, *the conversion between the two is not exact*. An approximate conversion is indeed possible, as

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

shown, and the choice of conversion process depends on whether a broadband or a low-pass match is chosen. Two conversion schemes have been described here, but a continuum of different choices is available.

Expect that any usable conversion scheme has Huray model parameters that satisfy the following bounds:

$$0.65\Delta \leq a \leq 2\Delta$$

and

$$\frac{2}{3}\left(SF-1\right) \leq SR \leq \frac{3}{2}\left(SF-1\right)$$

References:

1. E. Bogatin, D. DeGroot, P.G. Huray, and Y. Shlepnev. "Which one is better? Comparing options to describe frequency dependent losses." *DesignCon, 2013*.

Symmetry Boundaries

In HFSS, symmetry boundaries represent perfect E or perfect H planes of symmetry. Symmetry boundaries enable you to model only part of a structure, which reduces the size or complexity of your design, thereby shortening the solution time.

When you are defining a symmetry plane, keep the following requirements in mind:

- A plane of symmetry must be exposed to the background.
- A plane of symmetry must not cut through an object drawn in the **3D Modeler** window.
- A plane of symmetry must be defined on a planar surface.
- Only three orthogonal symmetry planes can be defined in a problem.

Perfect E Vs. Perfect H Symmetry Boundaries

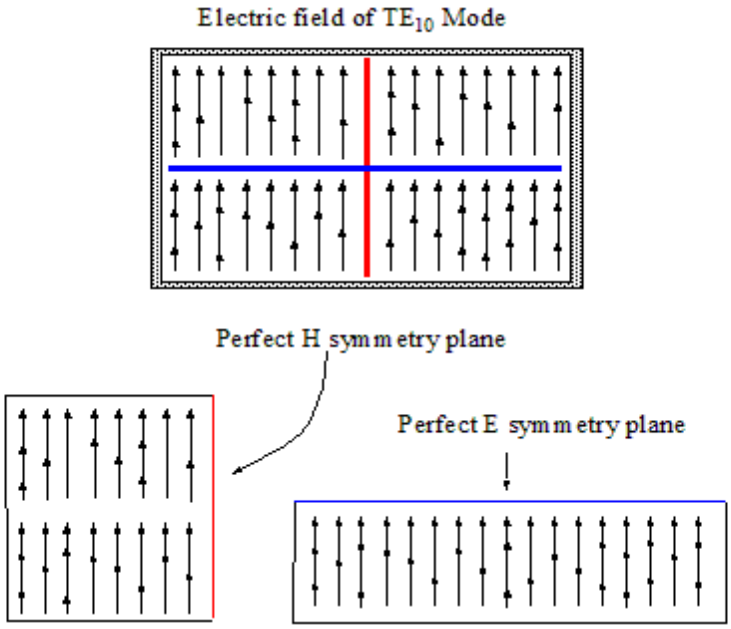
In general, use the following guidelines to decide which type of symmetry boundary to use, a perfect E or a perfect H:

- If the symmetry is such that the E-field is normal to the symmetry plane, use a perfect E symmetry plane.
- If the symmetry is such that the E-field is tangential to the symmetry plane, use a perfect H symmetry plane.

The simple rectangular waveguide shown below illustrates the differences between the two types of boundaries. The E-field of the dominant mode signal (TE_{10}) is shown. The waveguide has two planes of symmetry, one vertically through the center and one horizontally.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The horizontal plane of symmetry is a perfect E surface. The E-field is normal and the H-field is tangential to that surface. The vertical plane of symmetry is a perfect H surface. The E-field is tangential and H-field is normal to that surface.



For common problems, you can usually decide which symmetry boundary to use by reviewing the geometry. For example, if the structure is a microstrip, the flux lines of the E-field run between the ground plane and the conductive strip; therefore, the E-field is tangential to any vertical symmetry plane that slices a microstrip in half.

Symmetry and Port Impedance

If a symmetry plane has been defined, the computed port impedances will not match the port impedance of the full structure unless an impedance multiplier is specified.

> **Note:**
>
> Port impedance is only calculated when a port has been defined. If you are solving a problem without ports, you do not need to specify an impedance multiplier.

Symmetry and Multiple Modes

If you are solving for multiple modes, keep in mind that the orientation of the E- and H-fields may differ from mode to mode. A perfect H symmetry boundary for the dominant mode may be a perfect E symmetry for another mode.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Primary and Secondary Boundaries

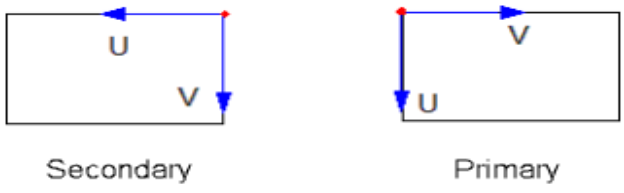
Primary and secondary boundaries (Lattice Pairs) enable you to model planes of periodicity where the E-field on one surface matches the E-field on another to within a phase difference. They force the E-field at each point on the secondary boundary match the E-field to within a phase difference at each corresponding point on the primary boundary. They are useful for simulating devices such as infinite arrays.

Unlike symmetry boundaries, **E** does not have to be tangential or normal to these boundaries. The only condition is that the fields on the two boundaries must have the same magnitude and direction (or the same magnitude and opposite directions).

When creating matching boundaries, keep the following points in mind:

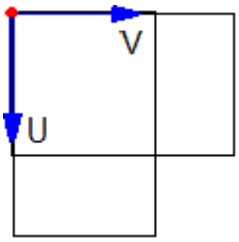
- Primary and secondary boundaries can only be assigned to planar surfaces. These may be the faces of 2D or 3D objects.
- The geometry of the surface on one boundary must match the geometry on the surface of the other boundary. For example, if the primary is a rectangular surface, the secondary must be a rectangular surface of the same size.
- If the mesh on the primary boundary does not match the mesh on the secondary boundary exactly, the solution will fail. Normally HFSS automatically forces the mesh to match on each boundary; however, in some cases, the mesh cannot be forced to match. To prevent the solution from failing, create a virtual object on the secondary boundary that exactly matches any extra object on the primary boundary, or create a virtual object on the primary boundary that exactly matches any extra object on the secondary boundary.
- To make a surface a primary or secondary boundary, you must specify a coordinate system that defines the plane on which the selected surface exists. When HFSS attempts to match the two boundaries, the two coordinate systems must also match each other. If they do not, HFSS will transpose the secondary boundary to match the primary boundary. When doing this, the surface to which the secondary boundary is assigned is also transposed. If, after doing this, the two surfaces do not occupy the same position relative to their combined defined coordinate system, an error message appears.

For example, consider the following figure:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To match the coordinate system of the primary boundary, the coordinate system on the secondary boundary must rotate 90 degrees counterclockwise; however, when this is done, you get the following:



The two surfaces do not correspond and thus the mesh will not match, causing an error message.

- The angle between the axes defined by the u point and v point must be identical for the primary and secondary boundary.

Calculating the E-Field on the Secondary Boundary

The E-field on the secondary boundary is forced to match the E-field on the primary boundary. The magnitude of the E-field on both boundaries is the same; however, the fields may be out of phase with each other.

The function relating the electric field on the secondary boundary, E_S, to the electric field on the primary boundary, E_M, depends on the type of problem you are solving. For example, consider an infinite array simulation for a rectangular array. If the array excited to radiate in the direction (θ, ϕ) in spherical coordinates. The fields above the array experience a phase delay of

$$\Psi = k(\hat{r}_o \bullet \boldsymbol{v}) \quad (1)$$

where

- $\hat{r}_0$ is the unit vector in the direction of scan.
- $\boldsymbol{v}$ is the vector from the secondary boundary to the primary boundary.

To simulate this in the finite element solution, HFSS incorporates phase shifts in the relation between the matching boundaries. That is, the electric field values on the primary boundary will be related to the electric field values on the corresponding points on the secondary boundary. This equation would be the following:

$$\boldsymbol{E}_S = e^{j\Psi}\boldsymbol{E}_M \quad (2)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

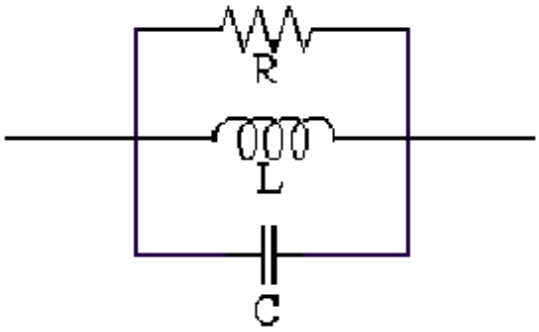
HFSS gives you the option of entering the scan angles, ϕ and θ, when relating E_S to E_M. The phase delay is calculated from the scan angles. However, if you know the phase delay, you may enter that directly.

Anisotropic Impedance Boundaries

Planar screens or grids of large extent with periodic geometry can be replaced by an Anisotropic impedance boundary. The boundary applies a homogeneous characteristic impedance to the surface in an effort to create an equivalent electrical representation of the geometric grid pattern.

Lumped RLC Boundaries

To model any combination of lumped resistor, inductor, and/or capacitor in parallel on a surface, create a lumped RLC boundary. A lumped RLC boundary represents R, L, and C in parallel:



Similar to impedance boundaries, the following condition holds at lumped RLC boundaries:

$$Loss = 0.5 \times Real\left((\vec{E} \times \overline{normal}) \cdot \overline{(\vec{E} \times \overline{normal})} \div \overline{Impedance}\right)$$

Unlike impedance boundaries, you are not required to supply the impedance per square, but you must supply the actual values for R, L, and C. HFSS then determines the impedance per square of the lumped RLC boundary at any frequency.

A Fast frequency sweep is supported for this boundary condition.

Layered Impedance Boundaries

A layered impedance boundary is used to model multiple layers in a structure as one impedance surface. The effect is the same as an impedance boundary condition, except that HFSS calculates the reactance and resistance values for the surface based on data you enter for the layered structure. Surface roughness is also taken into account.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The reactance and resistance values are calculated differently for internal and external layered impedance boundaries. For external layered impedance boundaries, HFSS calculates the impedance for the side of the surface in contact with the computational domain and assigns this value to the boundary. For internal layered impedance boundaries, HFSS calculates the average impedance value for the two sides of the surface that are in contact with the computational domain and assigns this value to the boundary.

The layered impedance boundary is supported for single-frequency solutions and Discrete and Interpolating frequency sweeps.

Impedance Calculation for Layered Impedance Boundary

The impedance of the layered structure is calculated recursively. First the impedance of the lowest layer is calculated, and then the impedance of the layer above is computed from transmission line theory as follows:

$$Z_{input,k} = Z_{wk}\frac{Z_{input,k+1}\cosh(\gamma_k d_k) + Z_{wk}\sinh(\gamma_k d_k)}{Z_{input,k+1}\sinh(\gamma_k d_k) + Z_{wk}\cosh(\gamma_k d_k)}$$

The process is completed when the uppermost layer is reached.

Here, for the k^{th} layer:

- d_k is the layer thickness.
- Z_{wk} is the wave impedance:

$$Z_{wk} = \sqrt{\frac{\mu_0\mu_{rk}}{\epsilon_0\epsilon_{rk}}}$$

- γ_k is the propagation constant:

$$\gamma_k = jk_0\sqrt{\epsilon_{rk}\mu_{rk}}$$

- $k_0=\omega\sqrt{\mu 0\,\epsilon 0}$ is the free space wave number, where $\omega=2\pi f$ is the angular frequency.
- ϵ_{rk} the complex relative permittivity.
- μ_{rk} is the complex relative permeability.

ϵ_{rk} and μ_{rk} are given by the following formulas:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\epsilon_{rk} = \epsilon'_{rk} - j\epsilon''_{rk}$$

$$\mu_{rk} = \mu'_{rk} - j\mu''_{rk}$$

The real parts ϵ'_{rk} and μ'_{rk} are entered directly. The imaginary parts of the complex relative permeability and permittivity may be entered directly, or derived from the conductivity and loss tangents of the materials as follows:

$$\epsilon''_{rk} = \frac{\sigma_k}{\omega\epsilon_0} + \epsilon'_{rk} \tan\delta_{ek}$$

$$\mu''_{rk} = \mu'_{rk} \tan\delta_{mk}$$

Here, for the k^{th} layer:

- σ_k is the bulk electrical conductivity.
- $\tan\delta_{ek}$ is the electric loss tangent.
- $\tan\delta_{mk}$ is the magnetic loss tangent.

Surface Roughness Calculation for Impedance Boundary

The surface roughness is measured as the RMS deviation of the conductor surface from a plane. Surface roughness increases conduction losses. HFSS calculates surface roughness by modifying the conductivity as follows:

$$\sigma_c = \frac{\sigma}{K_w^2} \quad (1)$$

where

σis the material's conductivity.

$$K_w = 1 + \exp\left(-\left(\frac{s}{2h}\right)^{1.6}\right) \quad (2)$$

where, further:

- h is the surface roughness.
- s is the skin depth.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Shell Elements Theory

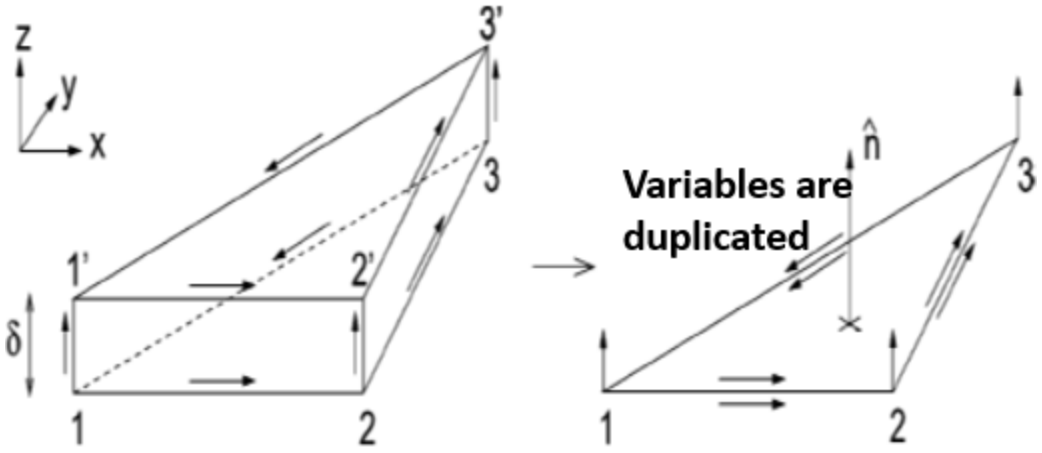
Layered Impedance Boundaries include an option to designate two-sided boundaries as shell elements. For Finite Conductivity boundaries, if DC thickness is set and the user selects two-sided, they can also designate that the boundary be treated as a shell element. If the layered structure is within a 3D model, meshing thin layers and solving inside may be inefficient, so replace the highly conductive thin layer with a shell element. The Shell Element option only applies to HFSS Driven solution types and does not apply to HFSS Eigenmode and Transient solution types. Shell elements are useful for both shielding and radome applications. They can touch or intersect other boundary conditions or materials. They must be valid across a broad frequency range. Example uses include simulation of an antenna inside a car chassis or an antenna inside a dielectric radome, when the chassis and/or radome is modeled using sheets. Shell elements maintain two sets of unknown coefficients for the top and bottom surface.

> **Note:**
>
> The shell element option is applicable for all frequencies and can therefore be used for broadband simulations without the need to modify boundary options depending on the simulation frequency. A sheet can also be a face of a solve inside object.

HFSS uses non-classical shell elements. Classical shell elements appear as prism elements created by doubling tangential components of the electrical field on both sides of a layer. This assumes constant linear or higher order variation in the normal direction. This classical approach is limited in application when the layer is not thin enough to the degree that the field has significant variation in the normal direction or if the field has strong decay in the layer despite it being thin.

Conventional Shell Elements (First Order)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



$$\vec{N}_{12} = \beta \vec{N}_1^t \quad \vec{N}_{23} = \beta \vec{N}_2^t \quad \vec{N}_{31} = \beta \vec{N}_3^t$$
$$\vec{N}_{1'2'} = \beta' \vec{N}_1^t \quad \vec{N}_{2'3'} = \beta' \vec{N}_2^t \quad \vec{N}_{3'1'} = \beta' \vec{N}_3^t$$
$$\vec{N}_{11'} = -L_1^t \nabla\beta \quad \vec{N}_{22'} = -L_2^t \nabla\beta \quad \vec{N}_{33'} = -L_3^t \nabla\beta \quad (1)$$

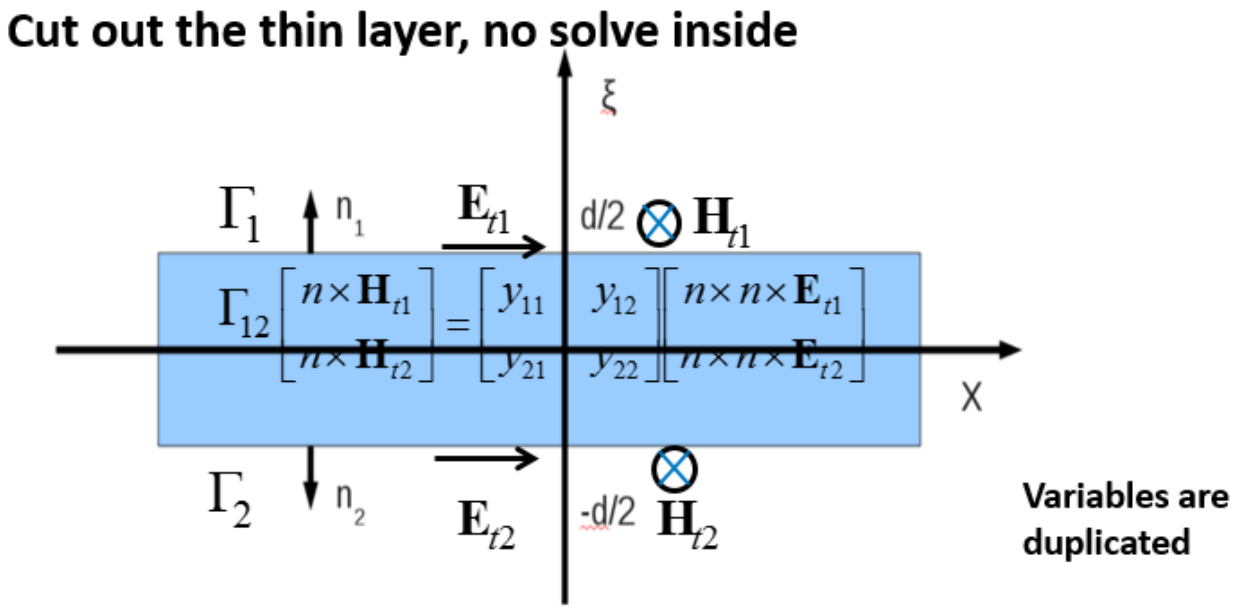
where $\beta' = 1 - \beta$ and $\vec{N}_{12}$ denotes the basis function associated with the edge between node 1 and node 2.

This depicts a degenerated prism element with no 3D mesh assembled on the fly.

Non-Conventional Shell Elements

For non-classical shell elements, we assume that we know the tangential electric and magnetic field on the two sides of the sheet. Rather than evaluate the finite element matrices in the thin prism, we apply a Neuman boundary condition to both sides of the sheet using the circuit ***Y*** matrix of the layers. Due to cases where the structure ***Y*** matrix cannot be characterized by closed analytical formulas, HFSS uses a two stage method which does not require analytical coupling formulas to replace Frequency Select Surfaces (FSS) by a sheet with shell elements. First, we extract a ***Y*** matrix for the FSS by solving a unit cell model of the layer(s). Then we use the ***Y*** matrix to couple the two sides of the layer in the FEM. This approach provides accurate and efficient modeling for an arbitrary thin or thick FSS. This is valid for any broad band frequency and material. (See References).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This uses:

- Use a 1D local transmission line model inside
- No Solve Inside, no 3D mesh
- Neumann/natural boundary condition
- Skin effect - copes with strong decay

Conventional versus Non-Conventional Shell Elements

Conventional shell elements:

- accurate
- good for the mesh maker
- can’t cope with strong decay
- higher order is difficult

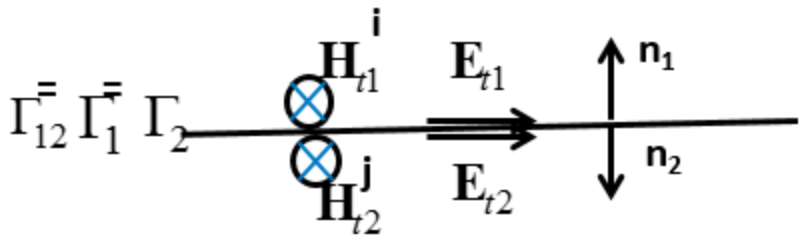
Non-Conventional shell elements

- approximated by a 1D local model
- good for the mesh maker
- copes with strong decay
- no need for higher order

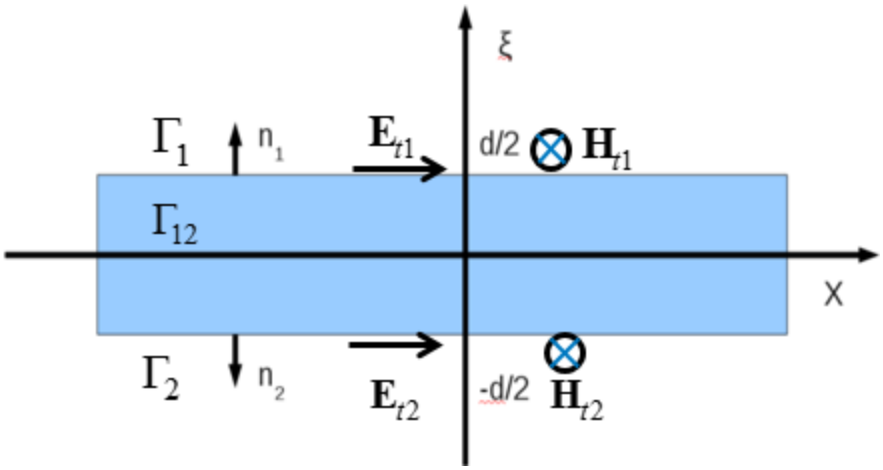
Finite Element Discretization

For sheet shell elements, 3D objects are replaced by a sheet:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For volumetric shell elements, keep the layer of thickness *d*, no solve inside.



Finite Element Discretization

Sheet shell elements:

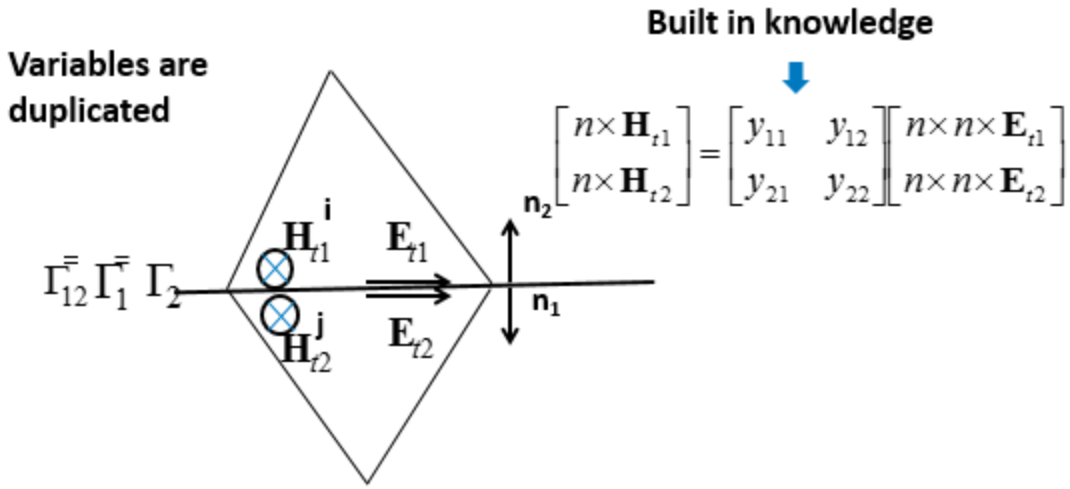
- no mesh inside
- good for the mesh maker
- manual work to get rid of 3D objects

Volumetric shell elements

- good for automation and ease of use
- background mesh inside
- overall mesh quality might be worse

Sheet Shell Elements

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- 3D layer replaced by a sheet
- E DoFs have to be duplicated
- Edges and vertices also can touch the surface

Finite Element Implementation

$$\int_{\Gamma_1} \mathbf{E}_1^* \bullet (\mathbf{n}_1 \times \nu_r curl \mathbf{E}_1) d\Gamma = -j\omega\mu_o \int_{\Gamma_1} (\mathbf{E}_1^* \bullet \mathbf{n}_1 \times \mathbf{H}_1) d\Gamma$$

$$\int_{\Gamma_1} \mathbf{E}_1^* \bullet (\mathbf{n}_1 \times \nu_r curl \mathbf{E}_1) d\Gamma = -j\omega\mu_o \int_{\Gamma_1} (\mathbf{E}_1^* \bullet (\mathbf{n}_1 \times (y_{11}\mathbf{n}_1 \times \mathbf{E}_1) + (y_{12}\mathbf{n}_1 \times \mathbf{E}_2)) d\Gamma$$

$$\int_{\Gamma_2} \mathbf{E}_2^* \bullet (\mathbf{n}_2 \times \nu_r curl \mathbf{E}_2) d\Gamma = -j\omega\mu_o \int_{\Gamma_2} (\mathbf{E}_2^* \bullet (\mathbf{n}_1 \times (y_{22}\mathbf{n}_2 \times \mathbf{E}_2) + (y_{21}\mathbf{n}_2 \times \mathbf{E}_1)) d\Gamma$$

Here $\mathbf{E}^*$ denotes weighting functions in the FEM Galerkin method. In this way, a shell element resembles an outer impedance boundary condition. Notice that the field coupling from one side to the other is captured by the term $y_{ij}\mathbf{n}_i \times \mathbf{E}_j$ in the surface integrals. Furthermore, when assembling a tetrahedron on one side of the sheet, the solver does not include n regular coupling from a tetrahedron on the opposite side except the surface integration contribution just described.

Implementation Issues: Sheet Shell Element Considerations

- Mesh is not modified
- Mesh maker provides the solver with list of shell tets

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

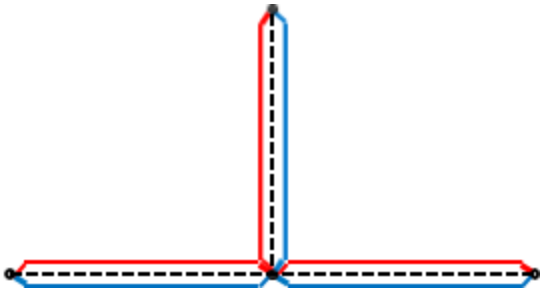
Implementation Issues: FEM and Sheet Shell Elements

Broad frequency range: Singlar matrix at low frequencies when E tangential components are continuous.

Boundary edges always have impedance:

Intersecting edges:

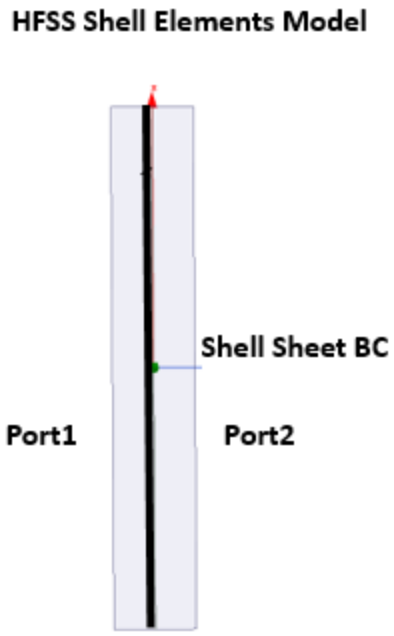
- Impedance on dielectrics
- PEC on metals



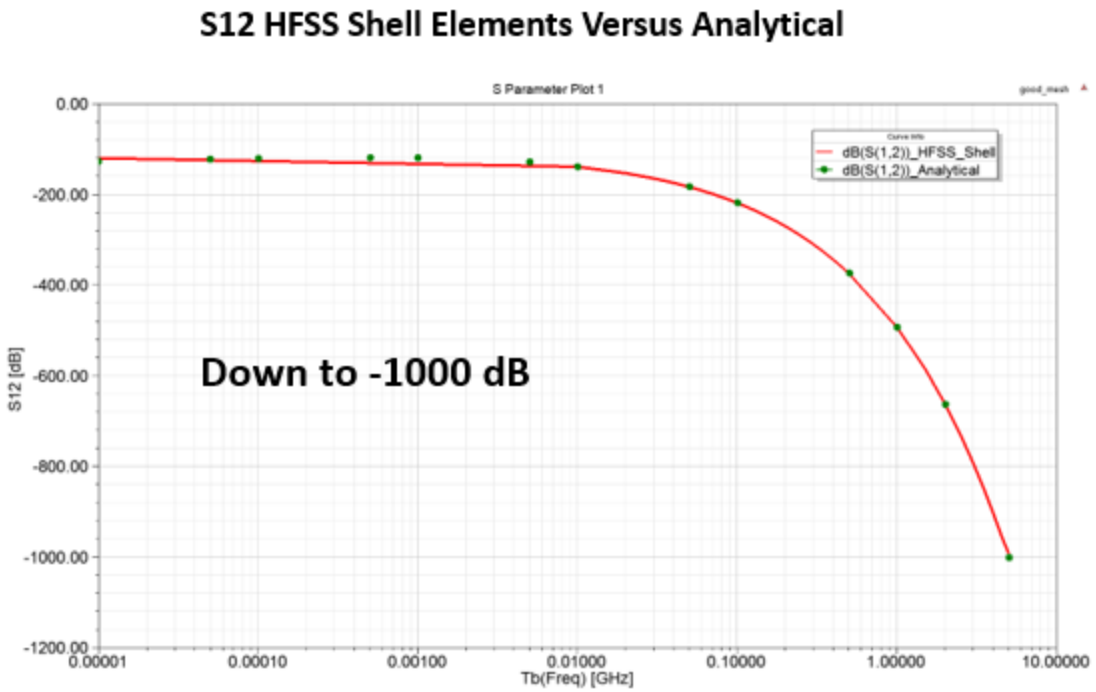
Accuracy

This example models a plane wave incidence on a finite thickness copper slab:

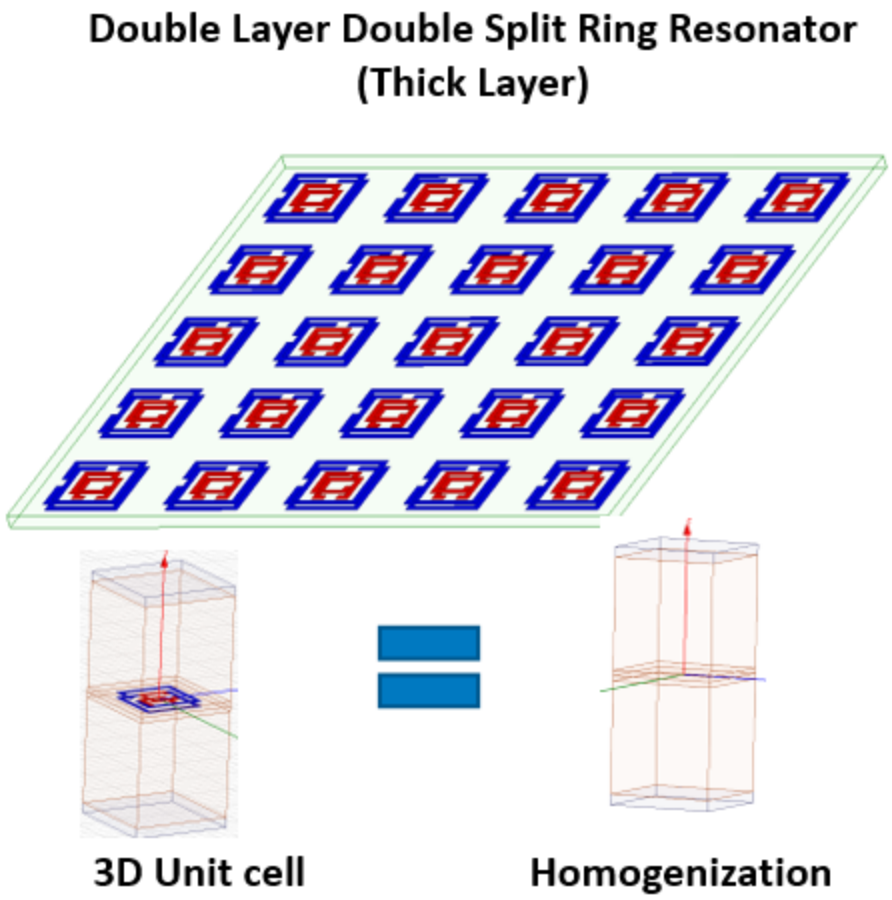
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Simulations comparing HFSS Shell Elements to analytical show accuracy down to -1000 dB.

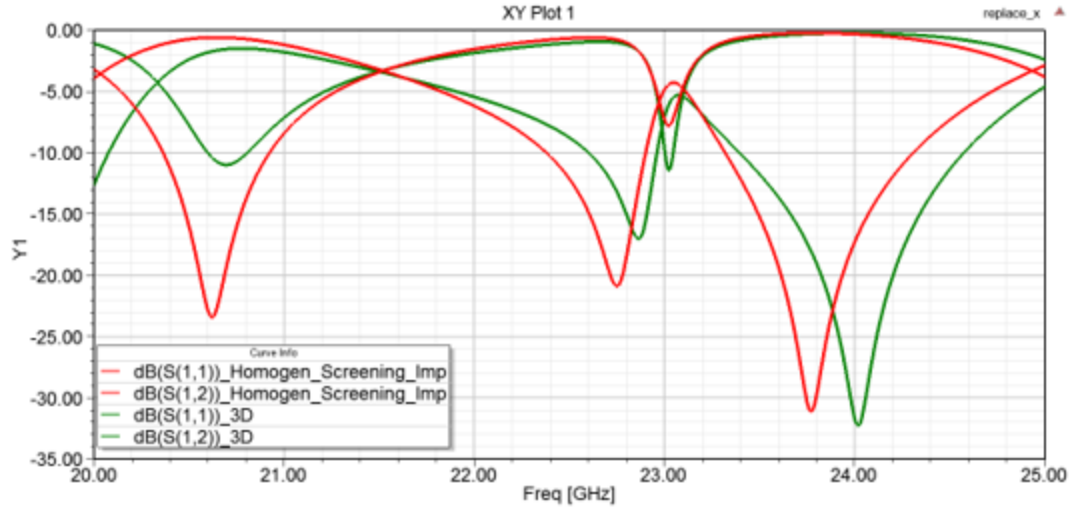


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

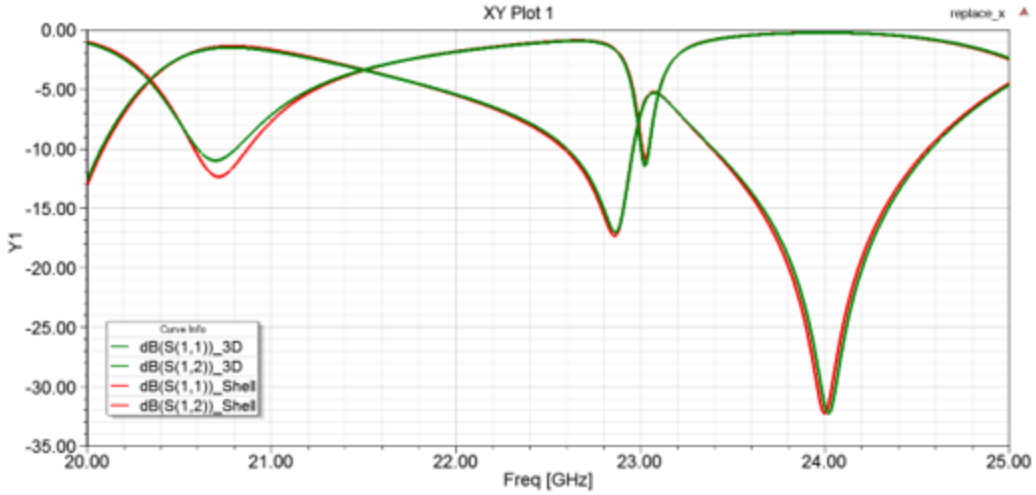


Simulation results compared.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

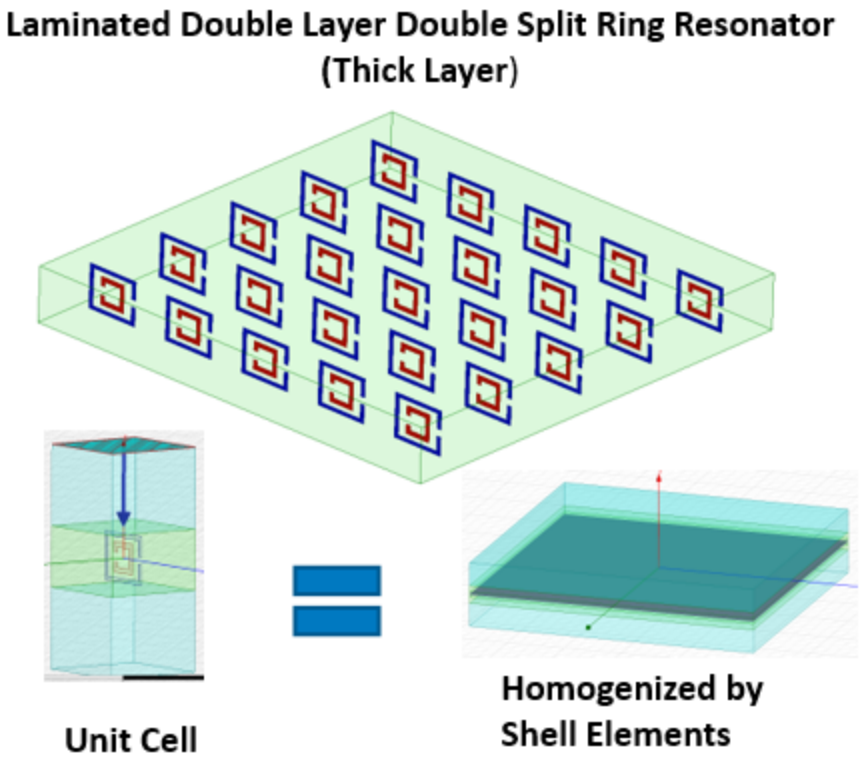


Green curves: 3D Solution
Red curves: Homogenization with Impedance BC



Green curves: 3D Solution
Red curves: Homogenization with Sheet Shell Elements

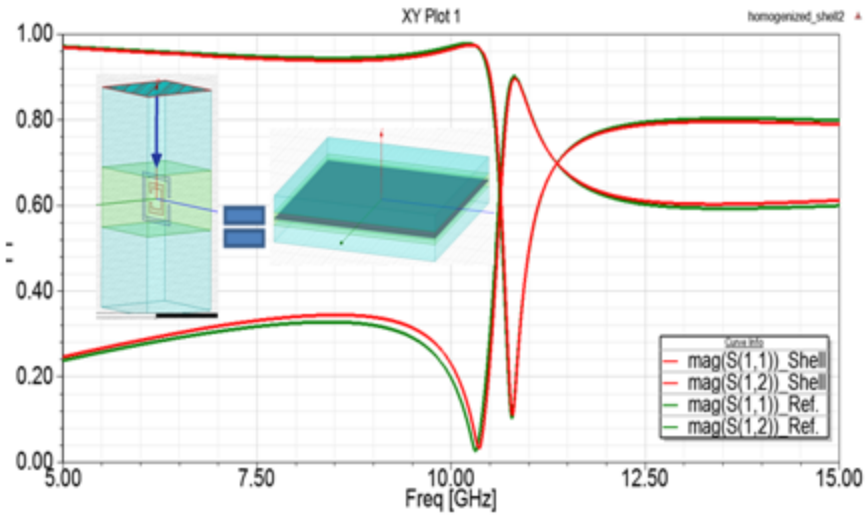
Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Simulation results compared.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Laminated Double Layer Double Split Ring Resonator (Thick Layer)



Green curves: 3D Solution
Red curves: Homogenization with Sheet Shell Elements

Conclusions

- Shell elements are applicable both for conductive and dielectric layers.
- Broad band frequency sweep
- For dielectric layers, the accuracy deteriorates in relation to the how off normal the angle of incident is from the boundary. If the layered impedance boundary or shell element contains at least one high conductivity layer, then the accuracy maintains for a significant extent off normal.
- This dependence is much less when using layers with high conductivity.
- Numerical shell elements work well by using Unit Cell Data Link.
- Volumetric shell elements are accurate even with non-conformal mesh.

References

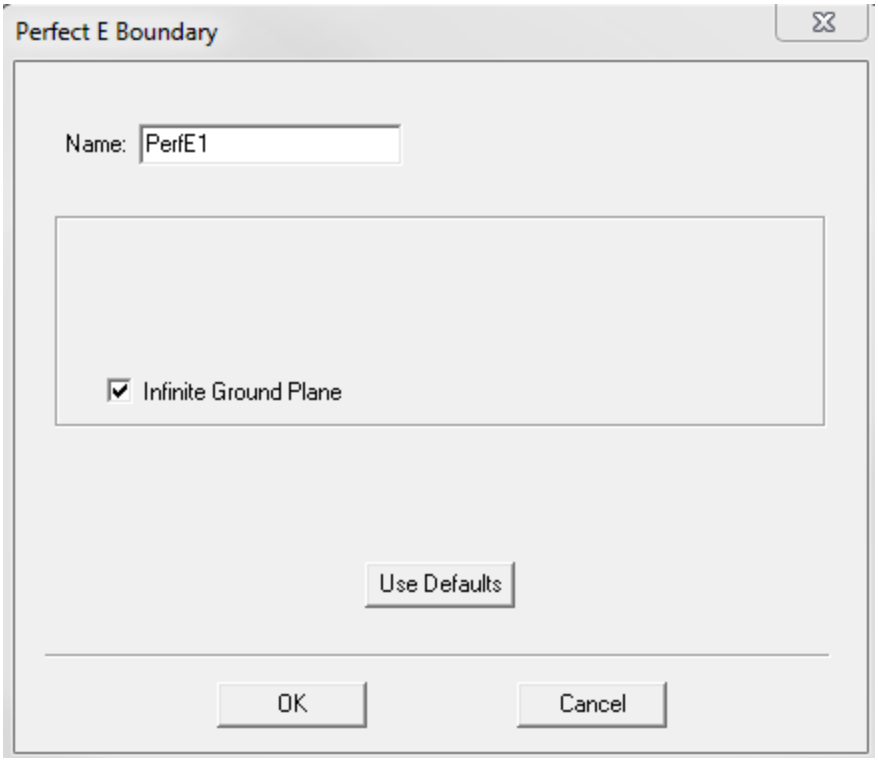
IEEE TRANSACTIONS ON MAGNETICS, VOL. 52, NO. 3, March 2016, "Modeling Periodic Layered Structures by Shell Elements Using the Finite-Element Method" by I. Bardi, G. Peng, and L. E. R. Petersson

Infinite Ground Planes

When defining **Perfect E Boundary**, **Impedance Boundary**, or **Finite Conductivity Boundary**, select the **Infinite Ground Plane** check box to model its effects in the simulation.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following figure shows the **Perfect E Boundary** dialog box with this option selected.



For designs with port excitations but no FE-BI boundary and metallic IE region defined, selecting **Infinite Ground Plane** affects only the calculation of near field and far field radiation during post processing; otherwise, the design has to be re-simulated to model the effects of infinite ground plane.

The boundary condition infinite ground plane divides the problem region into two halves. The entire model resides in the half above the boundary, and the radiated fields are set to zero in the half below the boundary. Antenna parameters involving radiated power are consistent with these properties.

Lossy ground planes can be approximated by selecting the **Infinite Ground Plane** check box when defining **Finite Conductivity Boundary** or **Impedance Boundary**. The effects of these boundary conditions are incorporated in the field solution and in the radiated fields accordingly.

Remember the following requirements when defining an infinite ground plane:

- It must be defined on a planar surface.
- All infinite ground planes and symmetry planes must be mutually orthogonal.
- For impedance, layered impedance, or finite conductivity boundaries, only one infinite ground plane can exist in a design. For perfect E boundary conditions, multiple infinite ground planes are supported and they must be co-planar.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

When PML or a standard radiation boundary is used, the FEM model must contain the ground plane on the bottom surface and the PML or radiation boundary on the other surfaces. To achieve accurate results, this restriction must be followed. This restriction does not apply in the following cases:

- When a FE-BI is used for the radiation boundary and the infinite ground plane is modeled as metallic IE region.
- Finite arrays are analyzed using a Finite Array DDM model.

Frequency-Dependent Boundaries and Excitations

In general, boundary and excitation parameters cannot depend on intrinsic functions. An exception is when a parameter depends on the variable *Freq*, which represents the solution frequency. The following boundary parameters can be assigned an expression that includes *Freq*:

- Impedance boundary - the Resistance and Reactance parameters.
- Finite conductivity boundary - the Conductivity parameter. If a material is specified, the conductivity of the material can be frequency dependent.
- Secondary boundary - the Phase parameter.
- Lumped RLC boundary - Resistance, Inductance, and Capacitance parameters.
- Layered impedance boundary - materials assigned on layers can be frequency dependent.
- Anisotropic Impedance boundary - Resistance and Reactance parameters.

Note:

Dependence on *Freq* is supported for single-frequency solutions and for Discrete and Interpolating frequency sweeps. If a Fast sweep is requested, the solution will be valid for the center frequency, but may not be valid at other frequencies.

Implicit Boundary Assignments

If a boundary has not been assigned to a model surface, one of the following default boundaries will be assigned to the surface:

smetal	A single perfect E boundary is assigned to all objects that do not have **Solve Inside** selected in the **Properties** window and that are perfect conductors.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

i_<*object name*>	A finite conductivity boundary is assigned to each object that does not have **Solve Inside** selected in the **Properties** window and that is not a perfect conductor. <*object name*> is the name of the object on which the boundary is assigned.
outer	A default boundary applied on the outermost surfaces of the model.

These boundary assignments are intrinsic and not displayed in the modeler.

Gradient Model for Surface Roughness

The gradient model was proposed by G. Gold and K. Helmreich in reference [1] below. In their approach, random variations in a rough metal surface are modeled by an assumed *probability density function* (PDF). The PDF describes how likely it is that the surface will be perturbed by some distance above or below its mean height (see Figure 1). Provided that these perturbations are small compared to the wavelength, their effects on a propagating signal can be averaged out. This averaging results in a spatially varying conductivity that is a function of the distance from the mean surface (see Figure 2). This conductivity can be plugged into a one-dimensional version of Maxwell's equations which can then be solved to yield the effective surface impedance.

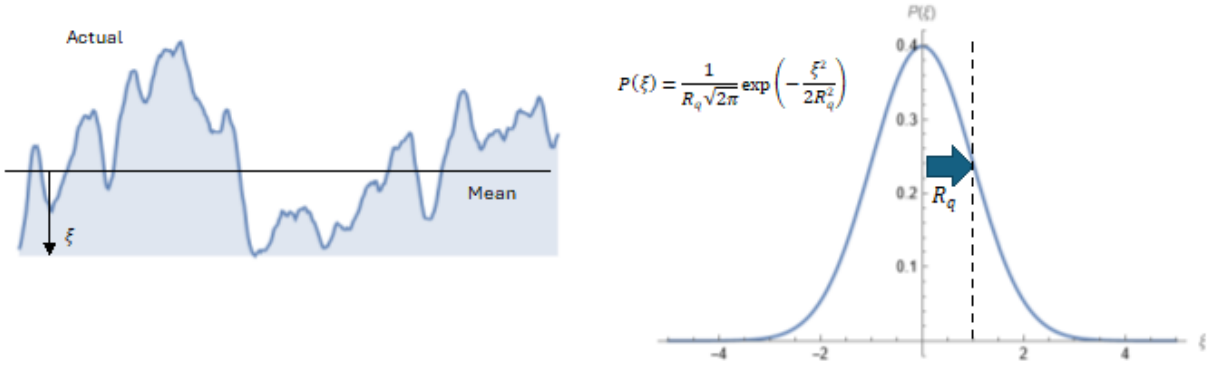


Figure 1: A Rough Suface described with a Gaussian Probability Density Function

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

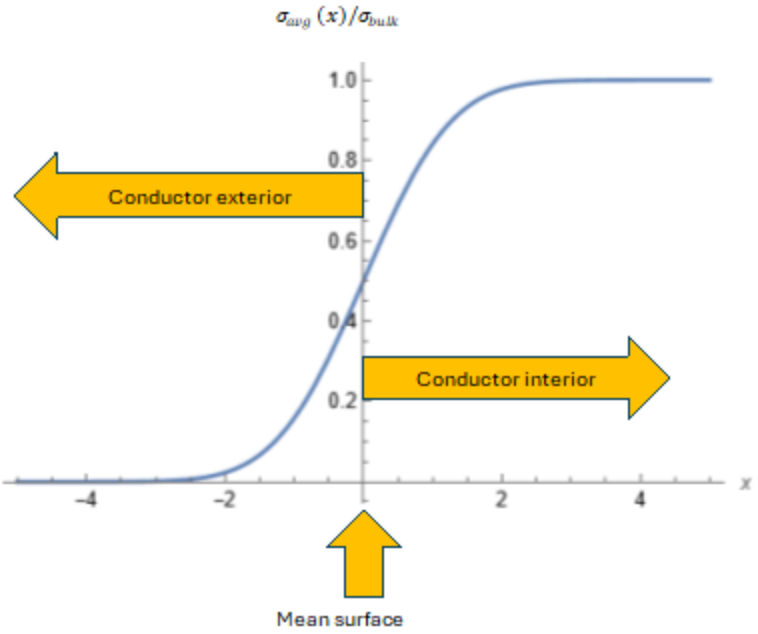


Figure 2: The Spatially Varying Conductivity, $\sigma(x)$ obtained by averaging the PDF.

An advantage of this approach is that the PDF can usually be characterized by a single parameter, the RMS roughness. This is convenient because the RMS roughness is typically measured by manufacturers and reported on their data sheets.

The gradient model in HFSS supports two types of PDF: Gaussian and Rayleigh. The Gaussian PDF is suitable for a low-profile conductor. The Rayleigh model produces somewhat higher losses and may be more appropriate for a high-profile conductor (see reference [2] below).

References

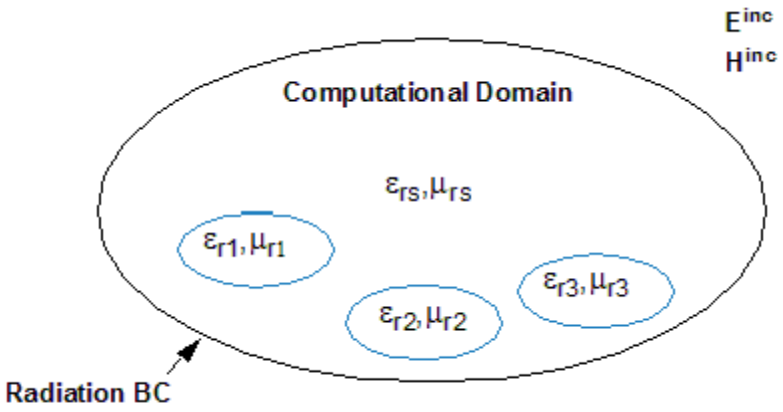
[1] G. Gold and K. Helmreich, “A physical surface roughness model and its applications,” *IEEE Trans. on Microwave Theory and Techniques*, vol. 65, no. 10, Oct. 2017.

[2] D. N. Grujic “Simple and accurate approximation of rough conductor surface impedance,” *IEEE Trans. on Microwave Theory and Techniques*, vol. 70, no. 4, April 2022.

General Approach for Scattered and Total Field Decomposition

Given our compositional domain:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Incident field is known in absence of our computational domain
- Physical (total) field is sought in presence of our computational domain
- Radiation BC applies to the field generated by the objects of our computational domain (scattered field)
- Incident field might be known in the whole domain (scattered field description) or just on the surface ("space ports," total field description)
- HFSS Transient provides the total field. The fair comparison is to use a single frequency in the frequency domain and apply a harmonic excitation in the time domain with the same frequency.

Scattered and total field formulation pros and cons

Scattered field formulation

- Mostly for plane waves and analytical waves
- Cancellation errors when total field is weak
- Problems arise at infinite scatterers (slot in an infinite conducting plate)
- Near/far incident waves are not known in the whole region

Total field formulation

- Applicable for any incident pattern
- Cancellation errors when scattered field is weak
- Infinite scatterers can be treated
- Applicable to near/far field, too, since excitations have to be known on surfaces
- New, space port concept (decomposition differs from the literature).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Where the Solvers Apply Scattered or Total Field Formulations

Scattered Field Formulation

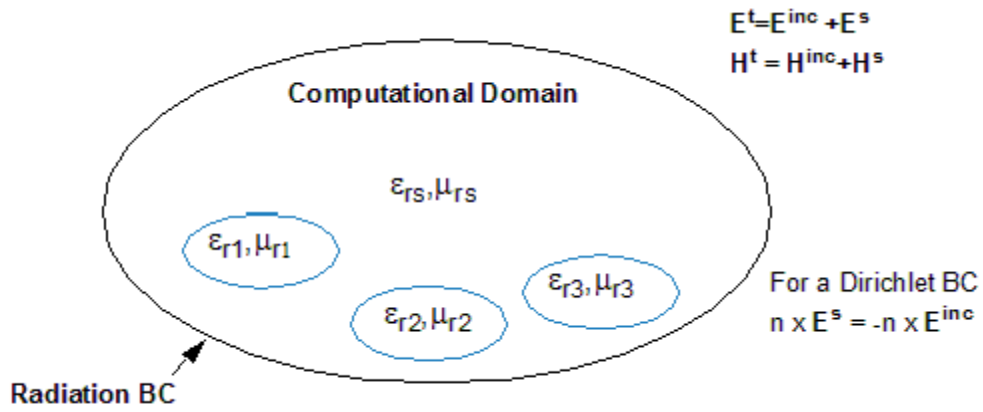
- Monostatic/bistatic RCS of finite scatterers
- Frequency Selective Surfaces (FSSs)

Total Field Formulation

- Monostatic/bistatic RCS of infinite scatterers
- Frequency Selective Surfaces (FSSs)
- When near/far field Link is involved
- EMC/EMI problems

For Scattered Field Decomposition

It is known in the whole domain.



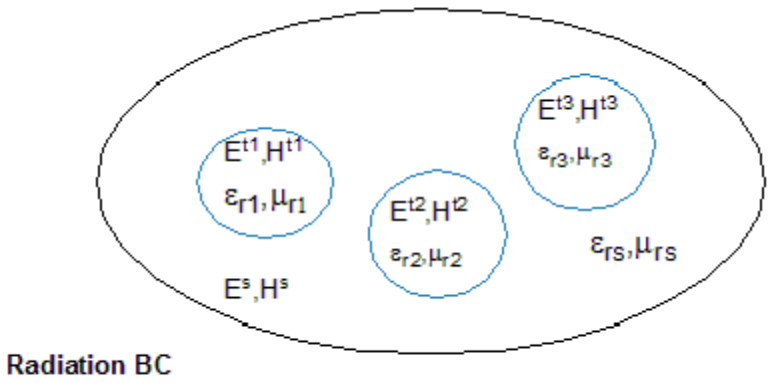
In summary, scattered field descriptions apply:

- Environment material other than vacuum
- Inhomogeneous magnetic material
- Special treatments on
- Ports
- Impedance BCs
- PMC

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

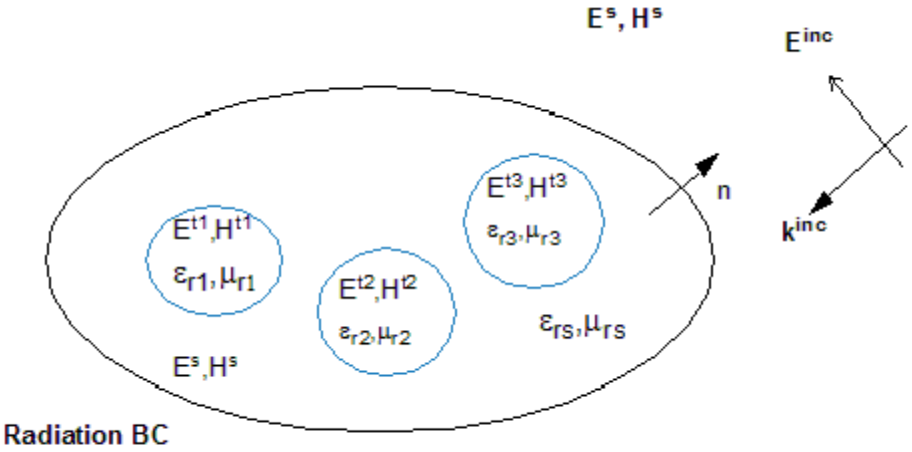
Total Field Description: Usual Approach

When the incident field is known, everywhere, the solvers take the usual approach. Both E and H jump with n x E^{inc} and n x H^{inc} on the interfaces, respectively.



Total Field Description: Unusual Approach

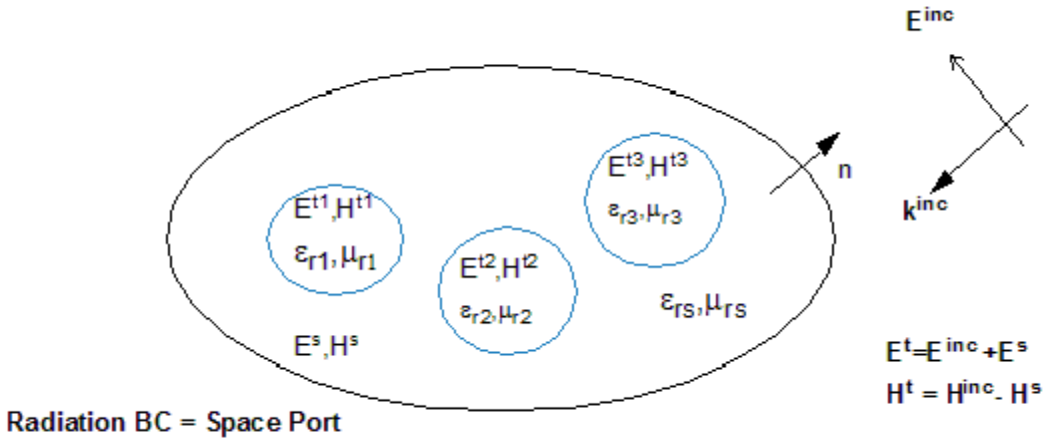
In this case, the incident wave is just on known on the surface. Both E and H jump with n x E^{inc} and n x H^{inc} on the outer surface respectively.



Total Field Description: Space Port Concept

The Radiation Boundary Condition is treated as a space port. The field is decomposed just on the surface of the space port.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Summary for Total and Scattered Fields

- Enhanced scattered field formulation
- Scattered field formulation for classical RCS problems
- Total field formulation with space ports is necessary for numerical incident waves (near/far)
- Total field formulation is necessary to analyze infinite scatterers (infinite plane, iris, etc)
- Total field formulation improves the accuracy in many cases

Excitations

Assigning excitations to an HFSS design enables you to specify the sources of electromagnetic fields and charges, currents, or voltages on objects or surfaces. This area of the Technical Notes includes information on the following topics:

- Wave Ports Theory
- Lumped Ports Theory
- Terminals Theory
- Floquet Ports Theory
- Incident Waves
- Simulating Circuits Containing N-Port Models from a Full-Wave Solver
- Voltage Sources
- Current Sources

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Magnetic Bias Sources
- Dembedding

Wave Ports Theory

By default, the interface between all 3D objects and the background is a perfect E boundary through which no energy may enter or exit. Wave ports are typically placed on this interface to provide a window that couples the model device to the external world.

HFSS assumes that each wave port you define is connected to a semi-infinitely long waveguide that has the same cross-section and material properties as the port. When solving for the S-parameters, HFSS assumes that the structure is excited by the natural field patterns (modes) associated with these cross-sections. The 2D field solutions generated for each wave port serve as boundary conditions at those ports for the 3D problem. In addition to serving as a boundary condition, a wave port also supplies port impedances and propagation constants that are useful in describing waveguides or transmission lines.

A wave port is restricted to reside at an external boundary of a 3D problem. In some instances, it makes sense to get around this restriction by defining a wave port in the interior of a 3D domain by capping the wave port surface with a PEC object. In doing this, in effect the wave port is viewed as residing at the external boundary of a 3D problem. However, in general lumped ports should be used when defining ports in the interior of a 3D domain.

Often there is a need to have a particular field direction of a mode pattern for consistent results. There are two different methods in HFSS to accomplish this task. The most basic method is where, in the presence of an integration line, HFSS resolves 180 degree ambiguity by enforcing the electric field mode pattern to have a positive line integral along the integration line for that mode (Mode Polarity). The second more advanced option is where HFSS aligns electric field mode patterns, when appropriate, in specific directions (Mode Alignment). This second option consists of a manual but general approach and an automated approach for a few port types. For more in-depth discussions refer to the following topics:

- Mode Polarity
- Mode Alignment

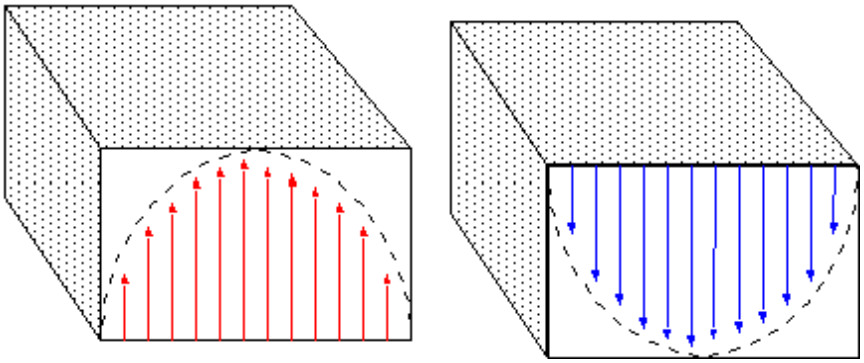
Mode Polarity

When HFSS computes an excitation mode pattern, the direction of the field at $\omega t = 0$ is arbitrary; the field can point in one of at least two directions. This document addresses how to remove the ambiguity of the field direction when a mode is not part of a set of degenerate modes. For the case of degenerate modes, there is no unique mode alignment and this topic is addressed in Mode Alignment.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the following figure, at $\omega t = 0$ the electric field pattern of the mode can point either up (left image) or down (right image). Either direction is correct - unless a preferred direction is specified. This issue becomes of concern when inspecting the phase angle of S-parameters.

For example, consider a 3D model containing two single-mode ports of the type shown in the figure. The phase of the computed S-parameters linking the two ports will flip by 180 degrees if the polarity of the mode at either port is reversed. Therefore when transmission S-parameter phase is of interest in models such as this, the user has the responsibility to properly set the mode polarity at each port.



To specify the polarity of a mode, a mode calibration must be performed by creating an appropriate integration line. For instance, if the preferred direction is up then the integration line should be drawn at the center of the waveguide starting at the bottom and ending at the top. In general, a mode calibration is automatically performed if an integration line has been defined for a mode where HFSS ensures that the real part of the line integral of the electric field of that mode is positive.

Mode Alignment

In some cases the requested set of modes to be used with a wave port contains degenerate modes. For that case, the modes that are degenerate need special care in obtaining a unique orientation of the mode patterns which is important, for example, when a specific orientation for a given mode is intended to excite the structure.

For example, in the case of a square waveguide the first two dominant modes form a degenerate mode set. Therefore, in order to guarantee that the typically desired orientation of the modes (TE_{01} and TE_{10}) are used for an HFSS simulation, the user needs to enable one of the two supported mode alignment options. The first option is general but manual and requires detailed user knowledge of the mode patterns while the other is restricted to a few basic port types where HFSS automatically aligns the modes.

The general mode alignment option is enabled by defining appropriate integration lines for the modes that should be aligned and selecting **Align modes using integration lines** in the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modes tab of the Wave Port setup window. Note that not all modes requested for a wave port need to have an integration line defined - only the ones for which mode alignment is desired. It is important that you draw integration lines that make sense for the modes to be aligned along and that all the modes that form a set of degenerate modes have a line defined. For example, if the first two modes of a square waveguide should be TE_{10} and TE_{01}, you should draw two lines through the center of the waveguide where one line starts at the bottom and ends at the top while the other line starts on the left side and ends on the right side.

You enable the automated mode alignment option by selecting **Align modes analytically using coordinate system** in the **Modes** tab of the Wave Port setup window. By selecting this option, the user will be asked to draw a local coordinate system which is used to automatically align the modes in pre-defined directions. This option is only available for a select few port types discussed in Analytic Port Types.

In order to get intended outcome of either alignment option, it is necessary to include all modes in a set of degenerate modes. For example, assume that modes 3, 4, 5, and 6 are degenerate. You should either include all of them by specifying at least 6 modes or none of them by specifying at most 2 modes.

Analytic Port Types

Analytic mode alignment is available for certain port types by selecting **Align modes analytically using coordinate system** in the **Modes** tab in the Wave Port setup window. If analytic mode alignment is selected for a port that is not supported, the port setup becomes invalid. Here is a summary of the validation for analytic mode alignment:

- The port geometry must correspond to a known analytic type, as follows:
- Rectangular. The port must have four linear edges which meet at right angles. Square ports are a special case of rectangular.
- Circular. The port must have all curved edges. The radius, perimeter, and area of the face must be consistent with those of an ideal circle.
- Faceted Circular. The port must have at least 16 edges, all linear and of equal length. The radius, perimeter, and area of the face must be consistent with those of an ideal polygon.
- Circular or Faceted Coax.

  Explicit. The port face must have one set of inner and one set of outer edges. The inner radius, outer radius, face area and perimeter must be consistent with an ideal coaxial circle or polygon.

  Implicit. The port face itself must be circular or faceted circular, and it must have a circular or faceted circular conducting object touching it and aligned with the face center.

- The port must be assigned to a single face of a 3D object whose material is isotropic. Assignment to sheets is not allowed, as this makes it difficult to determine the object

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

material.

- No other material can contact the port, except for the inner conductor of a coax.
- Only Perfect E and Finite Conductivity boundaries can contact the port.

When reading legacy projects prior to v12, HFSS automatically tests ports with the "Polarize E Field" option enabled. If they qualify for analytic alignment, that option will be selected, and U-V lines will be set from the first two modal integration lines.

Alignment Groups for Wave Ports with Multiple Modes

When assigning wave ports for modal solutions, alignment groups can be used only for a consecutive block of degenerate modes with each block of modes identified by a unique group number. There may be modes at the end of the modes table which have no assigned alignment group, but no unassigned modes in the middle of a block of degenerate modes containing an alignment group definition. Finally, each mode with an alignment group must have an integration line.

Alignment groups allow the solver to collect the modes within a group and ensure that each mode has non-zero voltage on its own integration line and zero voltage on every other integration line in the group. If the modes in a group are not degenerate or its defined integration lines are not appropriately defined, hf3d will terminate with an error message.

Any change in the mode groups will cause invalidation of solutions but not of the mesh.

Lumped Ports Theory

Lumped ports resemble wave ports, but can be located internally and have a complex user-defined reference impedance. Lumped ports are restricted to single mode ports and the S-parameters are always based on the user defined reference impedance. This mainly because no transmission line is being modeled in the interior domain which suggests an interpretation of the lumped port as a measurement probe being connected to the surface of the lumped port with the reference impedance specified by the user.

When a lumped port is used internal to a 3D problem, it makes sense to place the lumped port at a location where the field distribution would approximately be the same as the dominant mode of the port definition in the absence of the lumped port. A typical example of this is when a rectangular port is drawn between a conductor trace and a ground plane with the face oriented perpendicular to both conductors. The port mode resulting from this case is a parallel plate mode which will often closely resemble the field distribution at that location in the absence of the port. Lumped ports must be planar but can have any arbitrary shape within the plane.

A perhaps unsuspected result of using a complex reference impedance is that the S-parameters can be greater than one even for a passive device.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

When a lumped port is used as an internal port, the conducting cap required for a traditional wave port must be removed to prevent short-circuiting the source.

Terminals Theory

HFSS can categorize microwave structures in terms of a black-box that relates voltages and currents flowing in and out of a given structure. The black-box has several terminals, each with an associated voltage/current pair. In HFSS, these terminals reside inside wave ports that enable post processing of a modal representation of the black-box into the terminal representation.

When a terminal project is solved using HFSS, the number of modes for a port is determined by the number of terminals touching the port. If N+1 distinct conductors touch the port, there are N terminals and one reference conductor usually referred to as ground. The modal port representations of the electric and magnetic fields are:

$$\mathrm{E} = \sum_{n=1}^{N} (a_n + b_n)\mathrm{e}_n \quad (1)$$

$$\mathrm{H} = \sum_{n=1}^{N} (a_n - b_n)\mathrm{h}_n \quad (2)$$

where a_n and b_n are unitless complex amplitudes of the ingoing and outgoing modal fields, respectively. The modal black-box representation of a given structure is given by HFSS in terms of the generalized scattering matrix, *S*,

$$\mathrm{b} = S\mathrm{a} \quad (3)$$

where a and b are the unitless complex modal coefficient vectors. In order to obtain a terminal representation of the black-box, N integration paths Ci and N integration loops Li are used to define N voltages and N currents, respectively,

$$= \int_{C_i} (\vec{\mathbf{E}} \cdot \vec{\mathbf{dl}}) = \sum_{n=1}^{N} (a_n + b_n) \int_{C_i} \vec{\mathbf{e}_n} \cdot \vec{\mathbf{dl}} \quad (4)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$= \sum_{n=1}^{N} (a_n + b_n) T_{in} \quad i = 1,2...,N$$

$$i_i = \oint_{C_i} (\mathrm{H} \cdot \mathrm{dl}) = \sum_{n=1}^{N} (a_n - b_n) \oint_{C_i} \mathrm{h}_n \cdot \mathrm{dl} \qquad (5)$$

$$= \sum_{n=1}^{N} (a_n - b_n) U_{in} \quad i = 1,2...,N$$

In matrix notation, the equations become

$$\mathrm{v} = T(\mathrm{a} + \mathrm{b}) \quad (6)$$

$$\mathrm{i} = U(\mathrm{a} - \mathrm{b}) \quad (7)$$

Combining these equations with (3), yields the following circuit or terminal description of the black-box representation of the structure

$$\mathrm{i} = Y\mathrm{v} \quad (8)$$

where the terminal admittance matrix *Y* is given by

$$Y = U(I - S)(I + S)^{-1} T^{-1} \qquad (9)$$

In HFSS, a generalized terminal scattering matrix S_t is supplied to the user..

$$\beta = s_t \alpha \quad (10)$$

where α and β are ingoing and outgoing pseudo waves, respectively. The S_t matrix is obtained by assuming that the voltages and currents are measured by some device with a given renormalizing impedance. This assumption leads to the following relations.

$$\mathrm{v} = Z_{ref}^{1/2} \quad (\alpha + \beta) \qquad (11)$$

$$\mathrm{i} = Z_{ref}^{-1/2} \quad (\alpha - \beta) \qquad (12)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where Zref is a renormalizing impedance matrix representing the measuring device. In HFSS, the user has the option to use the terminal characteristic impedance matrix, *Zo*, as the renormalizing impedance matrix by using the 'Do Not Renormalize' option in the port post processing tab. This means that the measuring device is perfectly matched to the transmission line. By default, all the terminals are individually renormalized which results in Zref being a diagonal renormalizing impedance matrix. An expression for St is obtained by using equations (8) - (12)

$$S_t = (I + Z_{ref}^{1/2} \quad YZ_{ref}^{1/2} \quad)^{-1} (I - Z_{ref}^{1/2} \quad YZ_{ref}^{1/2} \quad) \quad (13)$$

Terminals in HFSS can also be used to create differential pairs as discussed in the Differential Pairs section.

Differential Pairs

A differential pair signal resides on two conducting traces in the presence of a ground plane. To form the differential pair, the transmitter launches a wave that carries equal but opposite voltages on the two traces (e.g., one trace is +1 volt and the other -1 volt). When two traces are routed in close proximity to each other, the waves traveling on those traces exhibits similar amounts of system noise. A differential pair is an effective way to remove noise from a signal because when a receiver processes a differential signal it subtracts the voltages on the two traces. This can dramatically reduce the amount of noise in the differential signal compared to if the traces are driven individually (single-ended signals). In HFSS, a differential pair can be defined for two terminals residing in the same wave port or for terminals that reside in two different wave ports or lumped ports. From the case of wave ports, matched differential pairs can be defined by not renormalizing the wave ports involved in the differential pair definitions.

Computing Differential Pairs

To obtain various differential pair quantities, we first define differential and common voltages v_d and v_c in terms of the singled-ended terminal voltage pairs $v_1$and v_2, (see Setting up Differential Pairs).

The differential and common voltages v_d and v_c are defined by

$$v_d = v_1 - v_2 \quad (1)$$

$$v_c = \frac{v_1 + v_2}{2}$$

Consistent with power conservation, the corresponding differential and common currents, represented as i_d and i_c respectively, are defined by

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$i_d = \frac{i_1 - i_2}{2} \quad (2)$$

$$i_c = i_1 + i_2$$

Equations (1) and (2) can be concisely represented as

$v = Qe$ (3)
$i = Q^{-T}u$

where

$$v = \begin{bmatrix} v_1 \\ v_2 \end{bmatrix} \quad i = \begin{bmatrix} i_1 \\ i_2 \end{bmatrix} \quad e = \begin{bmatrix} v_d \\ v_c \end{bmatrix} \quad u = \begin{bmatrix} i_d \\ i_c \end{bmatrix}$$

- Q is the real, non-singular matrix defined by

$$Q = \begin{bmatrix} \frac{1}{2} & 1 \\ -\frac{1}{2} & 1 \end{bmatrix} \quad (4)$$

- Q^{-T} is the inverse transpose of Q defined by

$$Q^{-T} = \begin{bmatrix} 1 & \frac{1}{2} \\ -1 & \frac{1}{2} \end{bmatrix} \quad (5)$$

Using eq. (3), we may easily transform between single-ended and differential quantities.

Differential Admittance and Impedance Matrices

The terminal admittance (Y) and impedance (Z) matrices discussed in the previous topics in the *Technical Notes* relate single-ended voltages and currents as $i = Yv$ and $v = Zi$, respectively. By first making differential voltage and current definitions, e and u eq. (3) can be used to derive Y_d and Z_d matrices that relate those qualities.

For example, if $i = Yv$, then substituting eq. (3) yields

$Q^{-T}u = YQe$.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solving for u yields

$$u = Q^T Y Q e$$

and the matrix Y_d relating differential quantities e and u is defined by

$$Y_d = Q^T Y Q \quad (6)$$

A similar procedure applies to the terminal Z_d.

Differential S-Matrices

It is clear that an S-matrix can be computed for differential signals because it is possible to compute admittance and impedance matrices for differential signals. The differential S-matrix can be envisioned as relating in-going and out-going waves on imaginary transmission lines attached to the differential ports. Characteristic impedances must be specified for these lines unless they are considered to be matched loaded which is automatically the case when the wave port(s) that contain the single-ended terminals used to define the differential pair is not renormalized.

In the single-ended case, the characteristic impedance for a pair of transmission lines may be written in the form of a matrix relating the voltages and currents on the two (uncoupled) lines,

$$\begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} Z^{(1)}_{ref} & 0 \\ 0 & Z^{(2)}_{ref} \end{bmatrix} \begin{bmatrix} i_1 \\ i_2 \end{bmatrix} \quad (7)$$

where $Z^{(1)}ref$ and $Z^{(2)}ref$ are the user-specified renormalizing impedances. In the differential case, the matrix equation relating differential and common currents and voltages is written as:

$$\begin{bmatrix} v_d \\ v_c \end{bmatrix} = \begin{bmatrix} Z^{(d)}_{ref} & 0 \\ 0 & Z^{(c)}_{ref} \end{bmatrix} \begin{bmatrix} i_d \\ i_c \end{bmatrix} \quad (8)$$

In this case, $Z^{(d)}ref$ and $Z^{(c)}ref$ denote the user-specified differential and common renormalizing impedances, respectively.

Floquet Ports Theory

The Floquet port in HFSS is used exclusively with planar-periodic structures. Chief examples are planar phased arrays and frequency selective surfaces when these may be idealized as infinitely large. The analysis of the infinite structure is then accomplished by analyzing a unit cell. Linked boundaries most often form the side walls of a unit cell, but in addition, at least one "open"

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

boundary condition representing the boundary to infinite space is needed. The Floquet port is a specialized boundary condition to handle this case.

The Floquet port is closely related to a wave port in that a set of modes, here "Floquet modes", is used to represent the fields on the port boundary. Fundamentally, Floquet modes are plane waves with propagation direction set by the frequency and geometry of the periodic structure. Just like Wave modes, Floquet modes have propagation constants and experience cut-off at a sufficiently low frequency.

When a Floquet port is present, HFSS performs a modal decomposition that gives additional information on the performance of the radiating structure. As in the case of a wave port, this information is cast in the form of an S-matrix interrelating the Floquet modes. In fact, if Floquet ports and wave ports are simultaneously present, the S-matrix will interrelate all wave modes and all Floquet modes in the model. Floquet ports can be combined with lumped ports, but not with terminal ports.

Formula Summary for HFSS Floquet Modes

- Planar array lattice is specified by lattice vectors **a** and **b**.
- Lattice angle α defined by $a * b = ab\cos\alpha$
- Unit cell area A
- Global material is lossless, isotropic, and homogeneous, and specified by real ε_r and μ_r.
- Frequency is specified in the form of free-space wave number $k_0 = \frac{2\pi f}{c}$ where f is frequency and c is the speed of light in a vacuum.
- Local coordinate system (LCS) unit vectors $\hat{x}$ and $\hat{y}$ are related to lattice vectors by:

$$\hat{\mathbf{x}} = \hat{\mathbf{a}} \quad (1)$$

$$\mathbf{c} = \mathbf{b} - (\mathbf{b} \cdot \hat{\mathbf{a}})\hat{\mathbf{a}} \quad (2)$$

$$\hat{\mathbf{y}} = \frac{\mathbf{c}}{c} \quad (3)$$

$$\hat{\mathbf{a}} = \frac{\mathbf{a}}{a}$$

Unit vector $\hat{r}_s$ denotes scan direction. In the global coordinate system (GCS) the components of $\hat{r}_s$ are given by ($\sin\theta_s\cos\phi_s$, $\sin\theta_s\sin\theta\phi_s$, $\cos\theta_s$) where scan angles θ_s and ϕ_s are specified as spherical angles in the GCS.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\psi_a = \sqrt{\mu_r \varepsilon_r} k_s \hat{\mathbf{r}} \cdot \mathbf{a} \quad (4)$$

$$\psi_b = \sqrt{\mu_r \varepsilon_r} k_0 \hat{\mathbf{r}}_s \cdot \mathbf{b} \quad (5)$$

$$m,n \varepsilon \{\ldots,-2,-1,0,1,2,\ldots\} \quad (6)$$

If $k_z \neq 0$ (7)

$$k_a = \frac{2\pi m}{a} + \frac{\psi_a}{a} \quad (8)$$

$$k_b = \frac{2\pi n}{b} + \frac{\psi_b}{ab} \quad (9)$$

$$k_x = k_a \quad (10)$$

$$k_y = \frac{k_b}{\sin\alpha} - k_a \cot\alpha \quad (11)$$

$$k_c^2 = k_x^2 + k_y^2 \quad (12)$$

$$k_z = \sqrt{\mu_r \varepsilon_r k0^2 - k_c^2} \quad (13)$$

$$\psi_{mn}(x,y) = \sqrt{\frac{1}{A}} e^{-jk_x x} e^{-jk_y y} \quad (14)$$

$$\Psi_{mn}^{(1)} = \left[\frac{k_y \hat{\mathbf{x}} - k_x \hat{\mathbf{y}}}{k_c}\right] \psi_{mn}(x,y) \qquad (k_c \neq 0) \quad (15)$$

$$\Psi_{mn}^{(2)} = \left[\frac{k_x \hat{\mathbf{x}} + k_y \hat{\mathbf{y}}}{k_c}\right] \psi_{mn}(x,y) \qquad (k_c \neq 0) \quad (16)$$

$$\tilde{\Psi}_{mn}^{(1)} = \sqrt{\frac{1}{A}} (\sin\theta_s \hat{\mathbf{x}} - \cos\theta_s \hat{\mathbf{y}}) \qquad (k_c \neq 0) \quad (17)$$

$$\tilde{\Psi}_{mn}^{(2)} = \sqrt{\frac{1}{A}} (\sin\theta_s \hat{\mathbf{x}} + \cos\theta_s \hat{\mathbf{y}}) \qquad (k_c \neq 0) \quad (18)$$

$$\vec{\mathbf{e}}_{mn}^{TE} = N_{mn}^{TE} \Psi_{mn}^{(1)}$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\vec{\mathbf{h}}_{mn}^{TE} = Y_{mn}^{TE}\hat{\mathbf{z}} \times \vec{\mathbf{e}}_{mn}^{TE} \tag{19}$$

$$Y_{mn}^{TE} = \frac{k_z}{\mu_0 k_0}\frac{1}{\eta_0} \tag{20}$$

$$N_{mn}^{TE} = \sqrt{\mu_r k_0}\frac{1}{\sqrt{|k_z|}}\sqrt{\eta_0} \tag{21}$$

$$\hat{\mathbf{z}} e_{mn}^{TE} = 0 \tag{22}$$

$$\hat{\mathbf{z}} h_{mn}^{TE} = -\hat{\mathbf{z}} Y_{mn}^{TE} \quad N_{mn}^{TE} \quad \left(\frac{k_c}{k_z}\right)\Psi_{mn} \tag{23}$$

$$\vec{\mathbf{e}}_{mn}^{TE} = N_{mn}^{TM}\Psi_{mn}^{(2)} \tag{24}$$

$$\vec{\mathbf{h}}_{mn}^{TM} = Y_{mn}^{TM}\hat{\mathbf{z}} \times \vec{\mathbf{e}}_{mn}^{TM} \tag{25}$$

$$Y_{mn}^{TM} = \frac{\varepsilon_r k_0}{k_z}\frac{1}{\eta_0} \tag{26}$$

$$N_{mn}^{TM} = \frac{1}{\sqrt{\varepsilon_r k_0}}\frac{k_z}{\sqrt{|k_z|}}\sqrt{\eta_0} \tag{27}$$

$$\hat{\mathbf{z}} e_{mn}^{TM} = -\hat{\mathbf{z}} N_{mn}^{TM} \quad \left(\frac{k_c}{k_z}\right)\Psi_{mn} \tag{28}$$

$$\hat{\mathbf{z}} h_{mn}^{TM} = 0 \tag{29}$$

If $k_Z = 0$, use the preceding $k_z \neq 0$ formulas with $k_Z = \alpha$ where $0 < \alpha << 1$ is a small constant.

If $k_C = 0$

$$\vec{\mathbf{e}}_{mn}^{TEM1} = N^{TEM}\tilde{\Psi}^{(1)} \tag{30}$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\vec{\mathbf{h}}_{mn}^{TEM1} = Y^{TEM}\,\hat{\mathbf{z}} \times \vec{\mathbf{e}}_{mn}^{TEM1} \quad (31)$$

$$\vec{\mathbf{e}}_{mn}^{TEM1} = N^{TEM}\tilde{\Psi}^{(2)} \quad (32)$$

$$\vec{\mathbf{h}}_{mn}^{TEM2} = Y^{TEM}\,\hat{\mathbf{z}} \times \vec{\mathbf{e}}_{mn}^{TEM2} \quad (33)$$

$$Y^{TEM} = \sqrt{\frac{\varepsilon_r}{\mu_r}}\frac{1}{\eta_0} \quad (34)$$

$$N^{TEM} = \frac{1}{\sqrt{Y^{TEM}}} \quad (35)$$

Incident Waves Theory

There are different kinds of incident wave sources in HFSS. Analytical wave forms are defined by formulas. The following analytical wave forms exist:

- Plane Wave
- Evanescent Plane Wave
- Elliptically Polarized Plane Wave
- Electric Spherical Wave (Electric Hertzian Dipole)
- Magnetic Spherical Wave (Magnetic Hertzian Dipole)
- Cylindrical Wave
- Gaussian Beam
- Linear Antenna

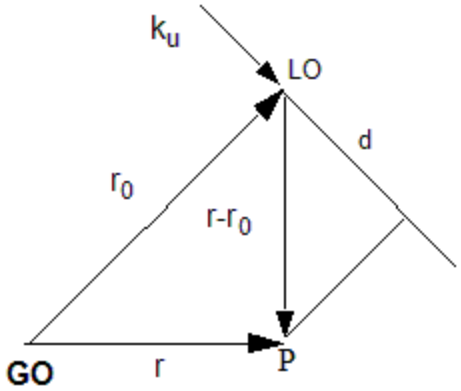
Numerical Wave Forms can come from the solution of another HFSS project or from a non-HFSS project via DataLink. Two numerical wave forms exist:

- Near Field Wave
- Far Field Wave

Plane Wave

An incident plane wave is a wave that propagates in one direction and is uniform in the directions perpendicular to its direction of propagation. The angle at which the incident wave impacts the device is known as the angle of incidence.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The equation that HFSS uses to calculate the incident wave is

$$E_{inc} = E_0 e^{-jk_0 k_u (\hat{r} \cdot r_0)}$$

where

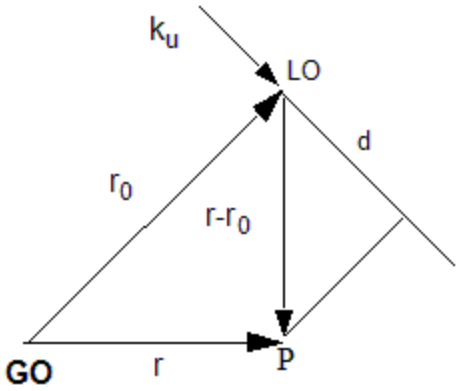
- E_{inc} is the incident wave.
- E_0 is the E-field polarization vector.
- k_0 is the free space wave number. It is equal to $\omega\sqrt{\mu_0 \varepsilon_0}$.
- $\hat{k}$ is the propagation vector. It is a unit vector.
- r is the position vector and is equal to $x\hat{x} + y\hat{y} + z\hat{z}$.

Incident wave excitations are specified in a peak sense. That is, if the incident wave magnitude is 5 V/m, then the real time function of the incident field is **E**(**t**) = 5cos(***k****r***+ωt).

Evanescent Plane Wave

An evanescent plane wave resembles the regular plane wave except that it decays in one direction. Its propagation constant is complex (see the equations).

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

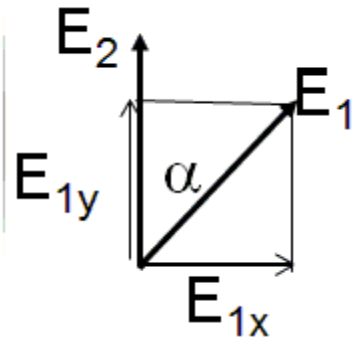


$$d = \left| k_u \bullet (r - r_0) \right| \qquad \alpha = Re(|k|)$$

$$\beta = Im(|k|) \qquad \vec{E} = \vec{E}_0 e^{-\alpha d} e^{-jkd}$$

$$\vec{H}_0 = \frac{k}{\omega \mu_0 \mu_r} (\vec{k_\mu} \times \vec{E}_0) = \frac{\beta - j\alpha}{\omega \mu_0 \mu_r} (\vec{k_\mu} \times \vec{E})$$

Elliptically Polarized Plane Wave



$$E'_1 = E_{10} \cos(\omega t) \qquad (1)$$

$$E'_2 = E_{20} \cos(\omega t + \varphi) \qquad (2)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$E_x^{inc} = E_1 \cos(\omega t) \quad (3)$$

$$E_y^{inc} = E_2 \cos(\omega t + \psi) \quad (4)$$

where

$$E_1 = E_{10} \sin(\alpha) \quad (5)$$

$$E_2 = \sqrt{(E_{10}\cos\alpha + E_{20}\cos\varphi)^2 + (E_{20}\sin\varphi)^2} \quad (6)$$

$$\psi = -arctg\frac{E_{20}\sin\Psi}{E_{10}\cos\alpha + E_{20}\cos\varphi} \quad (7)$$

$$E_a = \frac{1}{2}\left(\sqrt{E_1^2 + E_2^2 - 2E_1E_2\sin\psi} + \sqrt{E_1^2 + E_2^2 + 2E_1E_2\sin\psi}\right) \quad (8)$$

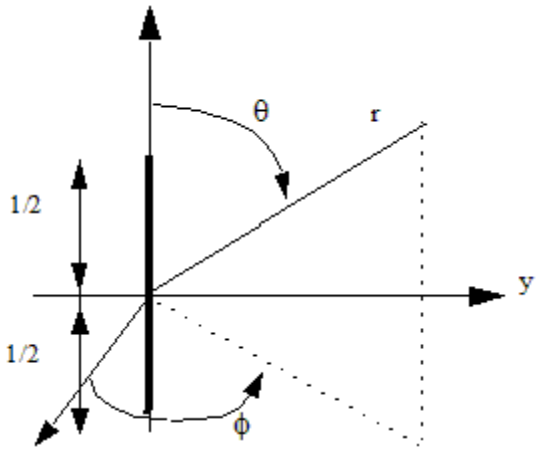
$$E_b = \frac{1}{2}\left(\sqrt{E_1^2 + E_2^2 - 2E_1E_2\sin\psi} - \sqrt{E_1^2 + E_2^2 + 2E_1E_2\sin\psi}\right) \quad (9)$$

Input:

$$ratio = E_b / E_a; \qquad \psi$$

Electric Spherical Wave (Electric Hertzian Dipole)

The analytical formulas of an electric Hertzian dipole are:



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$H_r = 0 \qquad H_\theta = 0$$

$$H_\phi = j\frac{kI_0 l \sin\theta}{4\pi r}\left[1+\frac{1}{jkr}\right]e^{-jkr} \qquad E_r = \eta\frac{I_0 l\cos\theta}{2\pi r^2}\left[1+\frac{1}{jkr}\right]e^{-jkr}$$

$$E_\theta = j\eta\frac{kI_0 l\sin\theta}{4\pi r}\left[1+\frac{1}{jkr}-\frac{1}{(kr)^2}\right]e^{-jkr} \qquad E_\Phi = 0$$

Magnetic Spherical Wave (Magnetic Hertzian Dipole)

The analytical formulas of a magnetic Hertzian dipole are:

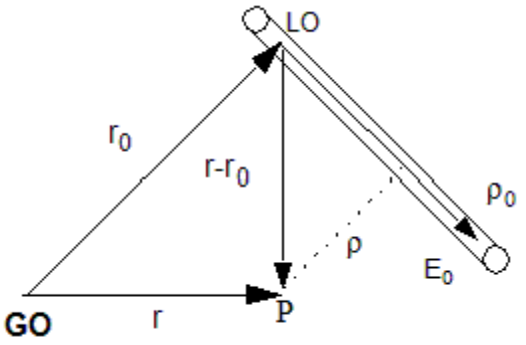
$$E_r = 0 \qquad E_\theta = 0$$

$$E_\phi = -j\frac{kI_0 l\sin\theta}{4\pi r}\left[1+\frac{1}{jkr}\right]e^{-jkr} \qquad H_r = \frac{I_m l\cos\theta}{2\pi\eta r^2}\left[1+\frac{1}{jkr}\right]e^{-jkr}$$

$$H_\theta = j\frac{kI_m l\sin\theta}{4\pi\eta r}\left[1+\frac{1}{jkr}-\frac{1}{(kr)^2}\right]e^{-jkr} \qquad H_\Phi = 0$$

Cylindrical Wave

The analytical formulas of a Cylindrical Wave are:



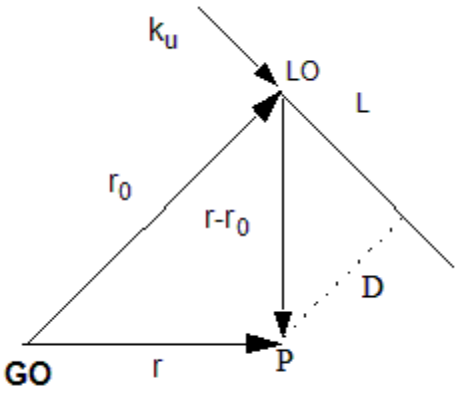
$$E_z(\rho) = \frac{C_1 e^{-jk\rho}}{\sqrt{\rho}} \quad \text{where} \quad C_1 = ZI^e\sqrt{k/8\pi}\,e^{j\pi/4}$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$H_\phi(\rho) = \frac{C_2 e^{-jk\rho}}{\sqrt{\rho}} \quad \text{with} \quad C_2 = C_1 / Z$$

Gaussian Beam

The analytical formulas of a Gaussian Beam are:



Where

$$L = (r - r_0) \bullet k_u D = \left|(r - r_0) \times k_u\right|$$

$$W(L) = W_0 \left[1 + \left(\frac{L\lambda}{\pi W_0^2 n}\right)^2\right]^{0.5}$$

$$R(L) = L\left[1 + \left(\frac{\pi W_0^2 n}{L\lambda}\right)^2\right]$$

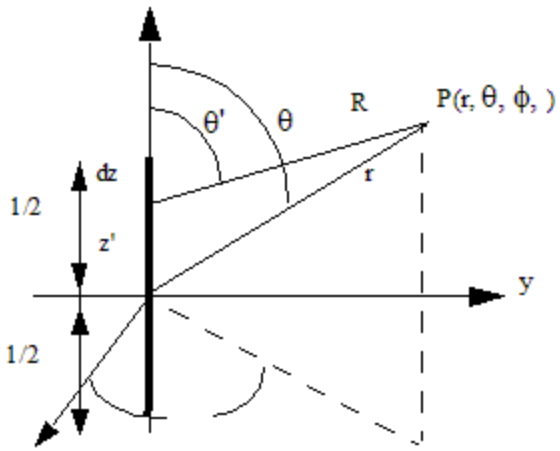
$$\Psi(L) = \text{atan}\left[\frac{L\lambda}{\pi W_0^2 n}\right]^2$$

$$E^{inc} = E_0 \frac{W_0}{W(L)} e^{-D^2/W(L)^2} e^{\left(-jk\bullet(r-r_0) - jk\frac{D^2}{2R(L)} + j\Psi(L)\right)}$$

Linear Antenna

The analytical formulas of a linear antenna are:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



$$E_\theta \cong j\eta \frac{I_0 e^{-jkr}}{2\pi r}\left[\frac{\cos\left(\frac{kl}{2}\cos\theta\right) - \cos\left(\frac{kl}{2}\right)}{\sin\theta}\right]$$

$$H_\phi \cong \frac{E_\theta}{\eta} \cong j\frac{I_0 e^{-jkr}}{2\pi r}\left[\frac{\cos\left(\frac{kl}{2}\cos\theta\right) - \cos\left(\frac{kl}{2}\right)}{\sin\theta}\right]$$

Simulating Circuits Containing N-Port Models from a Full-Wave Field Solver

This note looks at some of the issues involved in taking an N-port black box model from a full-wave field solver, such as HFSS or SIwave, and simulating it in a circuit tool, such as HSPICE, AEDT Circuit, or Nexxim. There are certain subtle differences between these models and the familiar low-frequency RLC equivalent circuit models from parasitic extractor tools like Q3D. In order to explain these differences, we will “open up” the N-port black box model and peek inside to see what is going on with its implementation. Armed with this information, we will then be able to better appreciate the capabilities and limitations of such models.

Some Background on Lumped Models

Consider a Spice subcircuit that consists of two resistors, R1 and R2, as shown below in Figure 1. The resistors in this case might represent the partial resistances for two nets, as computed by

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a tool like Q3D Extractor.

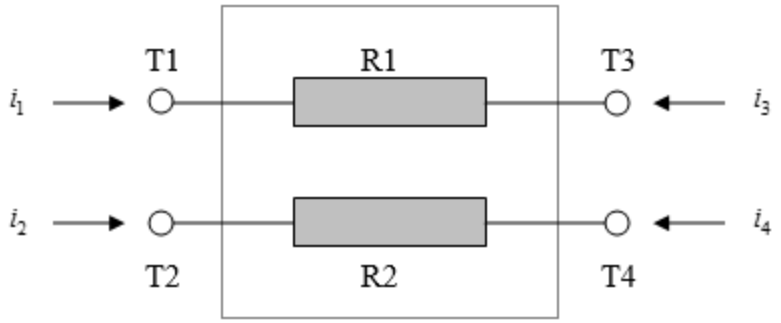


Figure 1: A conventional Spice subcircuit with two resistors.

This subcircuit has four terminals, labeled T1, T2, T3 and T4 in the figure. We can associate a node voltage with each of these terminals. We will call these v_1, v_2, v_3 and v_4, respectively. It should be noted that these node voltages are measured with respect to some global ground reference—node 0 in Spice. We can also define a set of terminal currents, one for each terminal, called i_1, i_2, i_3 and i_4, respectively. For consistency, the direction of these terminal currents is taken to be “into” the subcircuit.

Given the structure of this subcircuit, we have a fair amount of flexibility. We could compute the resistance of the top net by hooking a current source across T1 and T3 and measuring the resulting voltage drop. We could do the same with the bottom net. Or we could measure the loop resistance by shorting T3 and T4 together, and putting a current source across T1 and T2 (and then measuring the resulting voltage drop between T1 and T2.)

You need to be very careful with how you use connections to ground in such a model. Consider the example shown below in Figure 2. On the top, we show an incorrect attempt at measuring the loop resistance. The problem is that the user has grounded both terminals, T2 and T4. This effectively shorts out the resistor R2, and so the apparent resistance will only be R_1. The correct way of setting up the circuit is to remove the ground at terminal T4; this allows us to measure the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

true loop resistance, $R_1 + R_2$.

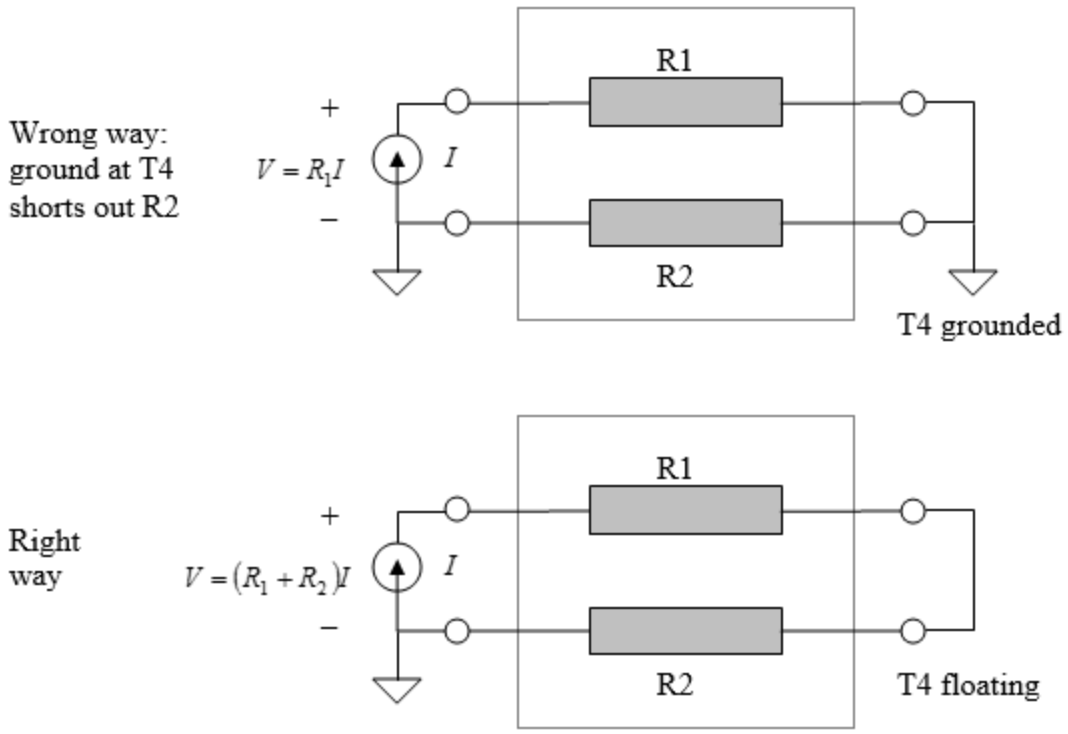


Figure 2: Illustration of the right (bottom) and wrong (top) ways to measure the loop resistance, in the case of a subcircuit consisting of standard RLC Spice elements.

As a general rule, when a conventional RLC circuit model is used, at most one terminal of the subcircuit should be connected to ground. If more than one terminal is grounded, you will be implicitly placing an ideal short circuit across those pairs of grounded terminals. The short circuit will bypass internal current flow paths, often resulting in unexpected behavior. Note that it is possible that none of the subcircuit's terminals will be grounded; the connection to ground may be made through some external circuitry.

A 2-Port Model

Now let's consider creating a 2-port model of this same circuit. We could choose S, Y or Z parameters, or other more exotic descriptions, but these are all basically equivalent. For the purposes of this note, we will use Y-parameters.

The first step is to choose the ports. If we suppose that the two resistors represent a signal conductor and its ground return, then the logical place for the ports is between the two nets. We

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will set up the ports as shown Figure 3.

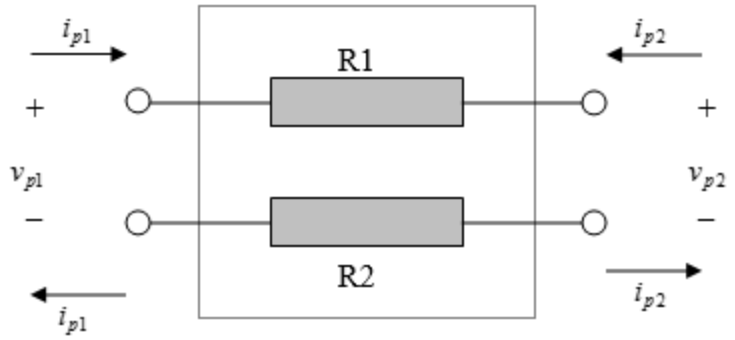


Figure 3: A 2-port model for the subcircuit.

Each port is created using a pair of terminals. Port 1 was created between terminals T1 and T2, and port 2 was created between terminals T3 and T4. We associate a port voltage (v_{p1} and v_{p2}) and a port current (i_{p1} and i_{p2}) with each port. These port voltages and port currents are related to the terminal voltages and currents as follows:

$$v_{p1} = v_1 - v_2 \text{ and } v_{p2} = v_3 - v_4 \qquad (1)$$

$$i_1 = i_{p1} \text{ and } i_2 = -i_{p1} \qquad (2)$$

$$i_3 = i_{p2} \text{ and } i_4 = -i_{p2} \qquad (3)$$

You will immediately notice that the ports have introduced some additional constraints that were not present before. In particular, due to the presence of the ports, the currents in the top and bottom resistors are no longer independent of one another. Whatever current enters the positive reference terminal (T1) for port 1 will flow back out of the negative reference terminal T2. Similarly, the current for port 2 goes into T3 and comes back out of T4. The placement of the ports in this circuit thus forces the bottom resistor to act as the return path for the current in the top resistor.

We now illustrate how the Y-parameters for this 2-port circuit are computed. Two voltage sources are connected at the ports, as shown in Figure 4 below. Illustration of how the Y-parameters for the 2-port circuit are computed. Two voltage sources are applied to control the port voltages, and the resulting port currents are calculated. These voltage sources directly

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

control the port voltages v_{p1} and v_{p2}.

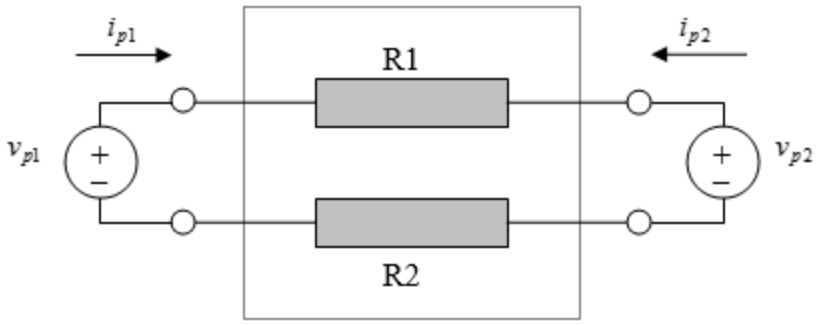


Figure 4: Illustration of how the Y-parameters for the 2-port circuit are computed. Two voltage sources are applied to control the port voltages, and the resulting port currents are calculated.

Using simple circuit analysis, the resulting port currents can be expressed in terms of the port voltages:

$$i_{p1} = \frac{1}{R_1 + R_2} v_{p1} - \frac{1}{R_1 + R_2} v_{p2} \qquad (4)$$

$$i_{p2} = -\frac{1}{R_1 + R_2} v_{p1} + \frac{1}{R_1 + R_2} v_{p2} \qquad (5)$$

These relationships can also be expressed in matrix form as

$$\begin{bmatrix} i_{p1} \\ i_{p2} \end{bmatrix} = \begin{bmatrix} \frac{1}{R_1 + R_2} & \frac{-1}{R_1 + R_2} \\ \frac{-1}{R_1 + R_2} & \frac{1}{R_1 + R_2} \end{bmatrix} \begin{bmatrix} v_{p1} \\ v_{p2} \end{bmatrix} \qquad (6)$$

The 2x2 matrix here is the admittance matrix (or Y-matrix) for the 2-port black box.

Observe that each of the Y-parameters depends on the sum $(R_1 + R_2)$. Even if we measure all 4 matrix entries, we will never be able to determine the value of the individual resistances R_1 and R_2 from the Y-matrix. The information about the individual resistance values has been lost when we condensed the circuit down to a 2-port representation.

It is also interesting to compare the circuit of Figure 4 with the one in Figure 5 below. The Y-parameters of the two circuits will be found to be identical. Therefore, we have no way of

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

distinguishing between the two networks if we have only their 2-port Y-parameter descriptions.

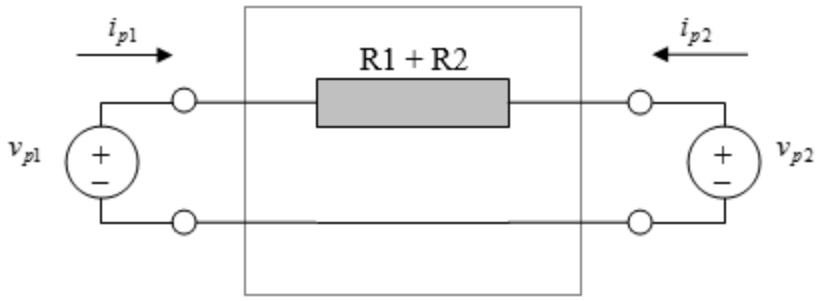


Figure 5. A different circuit that has the same Y-parameters as the one in Figure 4.

Implementing an N-port Network Model in a Circuit Simulator

An N-port network can be implemented within a circuit simulator by introducing N different 2-terminal circuit elements (or branches) as shown Figure 6. For each branch, the port voltage is taken to be the difference between the two node voltages at its terminals, and the port current is taken to be the current flowing in the branch. In addition to creating this element topology, the circuit simulator will be responsible for enforcing the branch relationships expressed in Equation 6 above.

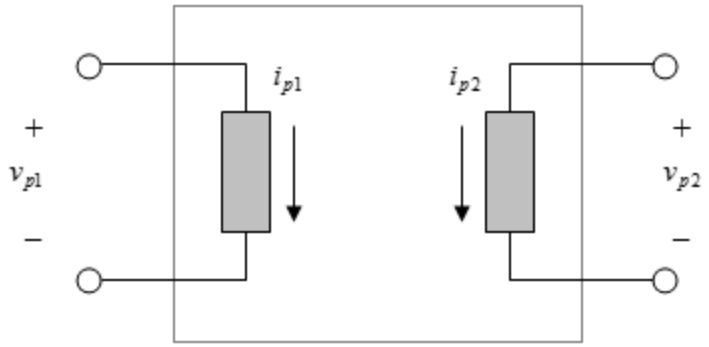


Figure 6. Illustration of how an N-port network is implemented within a circuit simulator, for the case of a 2-port network. In this topology, each port has its own reference node.

Another way of representing an N-port network is to use a common reference terminal, as shown in Figure 7 below. Because we will often choose to ground the negative reference terminal of a port, it may be more convenient to use this representation, because there are roughly half as many terminals to wire up. This is the approach taken by HSPICE (with the S element N-port S-parameter device) and also by Electronics Desktop (for imported S-parameter data.)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

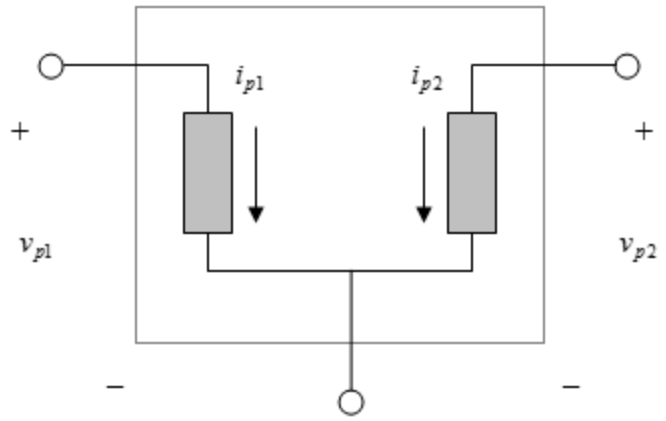


Figure 7. An N-port network with a common reference terminal (illustrated for the case of a 2-port).

Still another possibility is to suppress the reference terminal completely. In this case, the common reference terminal is internally wired to node 0. Only the positive reference terminals are exposed for external connections to the subcircuit. This is the approach taken by Electronics Desktop for dynamically linked field solver projects.

Using an N-Port Model for a Circuit Simulation

Let's consider how we would use this circuit. Suppose that we want to evaluate the loop resistance for the two conductors. We will place a current source in parallel with port 1 and a short circuit across port 2 to create the desired current loop, and then we will measure the resulting voltage across port 1. The resulting circuit is shown in Figure 8. This experiment is essentially the same as the one that we conducted on the "conventional" Spice equivalent circuit illustrated in Figure 2 and so (at first glance) it seems reasonable to use the same circuit topology. However, now that we are working with the 2-port representation we will see some

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

interesting differences in the details, although of course the final result will be the same.

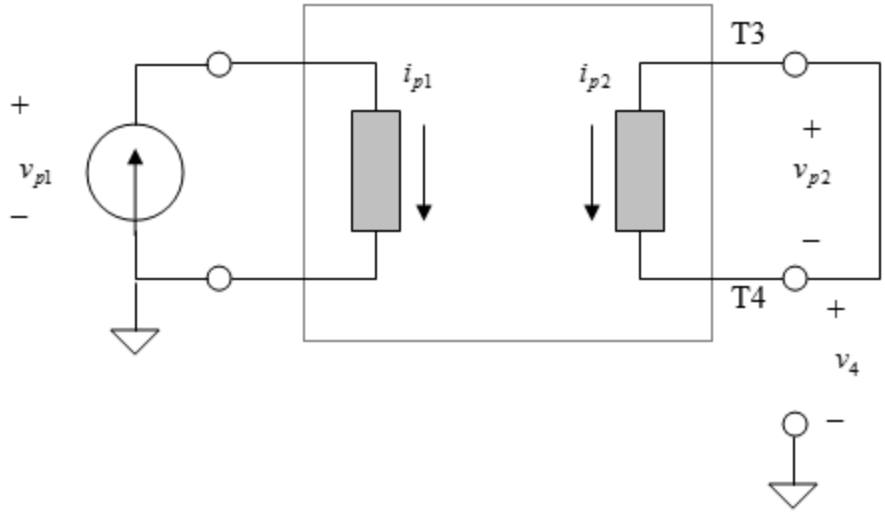


Figure 8. Computing the loop resistance using the 2-port black box model. This circuit looks correct (compare the bottom circuit of Figure 2) but actually causes simulation problems in Spice because the node voltage v4 at terminal T4 is ill-defined.

The first, and most important detail, is that the circuit simulator will probably fail if we try to simulate the 2-port circuit exactly as it is drawn in Figure 8. A typical error message would complain that there is “no DC path to ground” from the node corresponding to terminals T3 or T4. Another possibility would be an error message referring to a singular matrix, time step too small, Gmin stepping failed, or some other numerical problem.

What has gone wrong? The figure actually makes the problem fairly clear. Thanks to the short circuit, there is no question that the port voltage v_{p2} is zero and hence the node voltages at terminals T3 and T4 are clearly the same. The problem is that these two node voltages are “floating” with no connection to ground, and so they could assume any values at all, as long as they are the same. An ill-defined problem of this sort is tantamount to a singular matrix equation—a system of equations with more unknowns than equations. Circuit simulators do not deal gracefully with this ambiguous situation, because they want to return specific values for the node voltages.

To make this a bit more rigorous, consider the problem of writing down the nodal Kirchhoff’s current law (KCL) equation at terminal T4. (This is what the circuit simulator will do internally.) We have a current i_{p2} leaving the node through terminal T3, and the same current i_{p2} entering the node through terminal T4. As a result, the circuit simulator writes the equation

$$i_{p2} - i_{p2} = 0 \qquad (7)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This, of course, is a tautology: 0 = 0. Because this equation effectively contributes no information about the circuit, the simulator is unable to come up with a unique solution.

The problem can be resolved by introducing some sort of DC conduction path between terminal T4 (or T3) and ground. One way would be to add a resistor between T4 and ground. The KCL equation at the node then becomes

$$i_{p2} - i_{p2} + \frac{1}{R} v_4 = 0 \qquad (8)$$

Of course, all that this equation really says is that $v_4 = 0$. But that's enough to give us a well-defined solution to the system of equations, and the circuit simulator will now succeed.

It's interesting to note that the value of the grounded resistor we use is immaterial to the final solution. Because $v_4 = 0$ no matter what value of resistor is used, we may as well simply connect terminal T4 to ground (node 0 in Spice.) The solution to the problem will not change.

Based upon previous experience with Spice, it may be tempting to use a large-valued resistor (perhaps in the giga-Ohm range) instead of a direct connection to ground. But Equation 8 shows that this is a bad idea. If we make the resistance too large, then v_4 / R becomes vanishingly small and the equation again becomes equivalent (in the finite-precision arithmetic that computers use) to 0 = 0. In practice, what will happen is that the linear equation solver in the simulator will return increasingly inaccurate solutions as the value of the grounded resistor is increased. Therefore, using a large grounded resistor is a bad idea.

We should check what value of loop resistance will be returned by the simulator if we ground terminal T4. Looking back at Equation 4, we see that port voltage $v_{p2} = 0$, and therefore

$$I = \frac{1}{R_1 + R_2} v_{p1}$$

and so we will have $v_{p1} / I = R_1 + R_2$. This is the correct value of the loop resistance.

In the case of the standard RLC Spice subcircuit, we argued that you should never connect more than one subcircuit terminal to ground. The reason for observing this practice was to avoid shorting out internal conduction paths in the internal RLC network. We have just seen that this same restriction does not apply to the 2-port model. We grounded two of the terminals, and yet we still obtained the correct loop resistance. How can this be?

Returning to the equivalent circuit model of a 2-port in Figure 6, we see that there are no internal circuit elements connected from the positive or negative terminals of port 1 to the positive or negative terminals of port 2. Therefore, it's clear that we cannot accidentally short out an internal circuit element by grounding, unless we happen to ground both terminals of a single port.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

There is one thing about the solution we obtained using the 2-port model that appears at first glance to be incorrect. If we refer back to the original Spice model (drawn at the bottom of Figure 2), it's clear that the voltage at the "floating" terminal T4 is $R_2 I$, not zero. So is the 2-port model producing the wrong answer—or are we interpreting it incorrectly?

It turns out that the interpretation of the voltage is the problem. If we look back at the Y-matrix expression in Equation 6 , we see that it basically expresses port currents in terms of the port voltages. These port voltages are simply the voltage differences between the terminals, as expressed by Equation 1. We have previously argued (using Figure 5 as an illustration) that the N-port model has no way to tell whether the resistance lies in the upper or lower conductor. All it can do is to predict the correct voltage differences at the ports, given the port currents. So, because we have chosen to anchor terminal T4 to ground, the absolute voltage at this terminal disagrees with the predictions of the original subcircuit model. However, the voltage differences at the ports are in complete agreement.

We give a circuit interpretation of what is going on in Figure 9. Here we are taking the 2-port circuit of Figure 8 (with terminal T4 "grounded") and re-drawing it with the real internal network of 2 resistors.

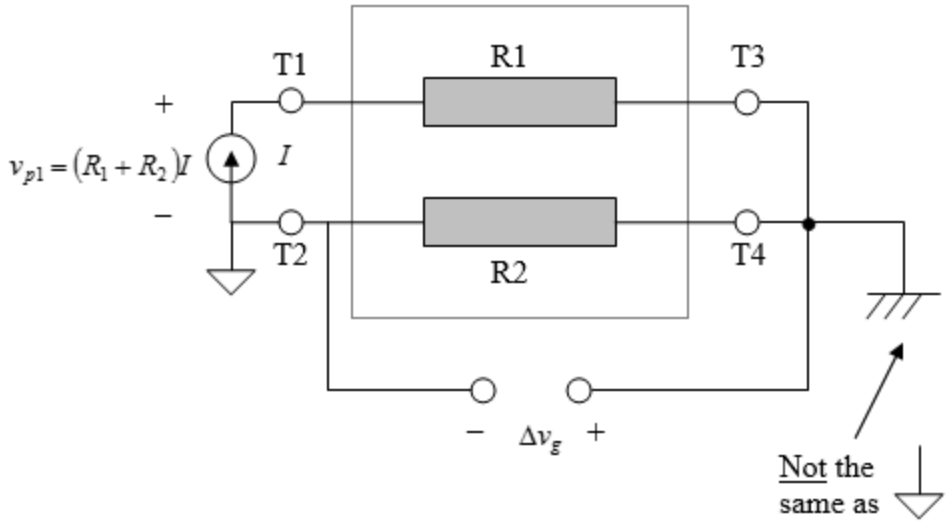


Figure 9. Interpretation of the different grounds being used in the two-port model.

Δv_g between the different grounds can easily be computed as $R_2 I$

Voltage Sources

A voltage source in HFSS can be defined on surface located anywhere in the 3D problem space, but it typically makes sense to place the source on a surface between two conductors such that a user defined total voltage is maintained between the conductors.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A voltage source is implemented in HFSS by forcing the electric field on the source surface to maintain a user-specified voltage drop along the voltage line. You can specify any surface as the source surface, but the prescribed field pattern is best suited for rectangular planar surfaces or non-planar surfaces obtained from the side wall surface of cylinders with uniform cross-sections such as a circular cylinder. The enforced electric field pattern is obtained by projecting a static uniform field onto the source surface with the electric field in the direction of the user defined voltage line.

A voltage source cannot be used to extract S-parameters. For S-parameter computations of an HFSS design involving a mixture of ports and voltage sources, all the voltage sources are shorted by treating the voltage source surface as a perfect electric conductor.

Current Sources

A current source in HFSS can be defined on a surface located anywhere in the 3D problem space, but it typically makes sense to place the source on a surface between two conductors such that it injects the user defined total current onto the conductors. A current source is implemented in HFSS by weakly enforcing the jump across the source surface of the tangential component of the magnetic field, $\Delta\vec{H}$, to be related to a surface current density, $\vec{J_s}$:

$$\hat{n} \times \Delta\vec{H} = \vec{J_s}$$

where $\hat{n}$ is the unit normal of the surface. The surface current density can be expressed as $\vec{J_s} = J_0\hat{u}$ where J_0 is the magnitude and $\hat{u}$ is the unit vector in the direction of current flow.

The unit vector is in the direction of the user defined current flow direction line. The magnitude is obtained by first assuming that the surface is a rectangle such that the width, *w*, is equal to the area divided by the length of the current flow direction line. The magnitude is set equal to the total current specified by the user divided by *w*. Note that any source surface is allowed but considering the definition of the surface current density it is recommended to only use a planar rectangular surface.

A current source cannot be used to extract S-parameters. For S-parameter computations of an HFSS design involving a mixture of ports and current sources, all the current sources are kept opened by treating the current source surface as a natural boundary condition where no special behavior of the electric field is explicitly enforced.

Magnetic Bias Sources

When you create a ferrite material, you must define the net internal field that biases the ferrite by assigning a magnetic bias source. The bias field aligns the magnetic dipoles in the ferrite,

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

producing a non-zero magnetic moment. In HFSS, a ferrite's permeability tensor is a direct result of an applied static magnetic bias field. The static field causes the tensor to assume an hermitian form, with cross coupling terms between field components perpendicular to the bias.

When the applied bias field is assumed to be uniform, you will specify net internal bias field as well as the tensor coordinate system through a rotation from the global coordinate system. When the applied bias field is non-uniform, neither specified coordinate system rotations nor specified bias are allowed. The permeability tensor's local coordinate system and bias field are calculated on a tetrahedron by tetrahedron basis, both of which are determined by the calculated static solution. HFSS references the static solution project as the source of the non-uniform magnetostatic field information during solution generation.

Multiple uniform bias excitations may be included in a design, but only non-uniform bias is allowed. In addition, a design cannot contain a bias excitation of both types. If only one excitation is included in a design, its type can be changed from uniform to non-uniform, and visa versa.

Uniform Applied Bias Fields

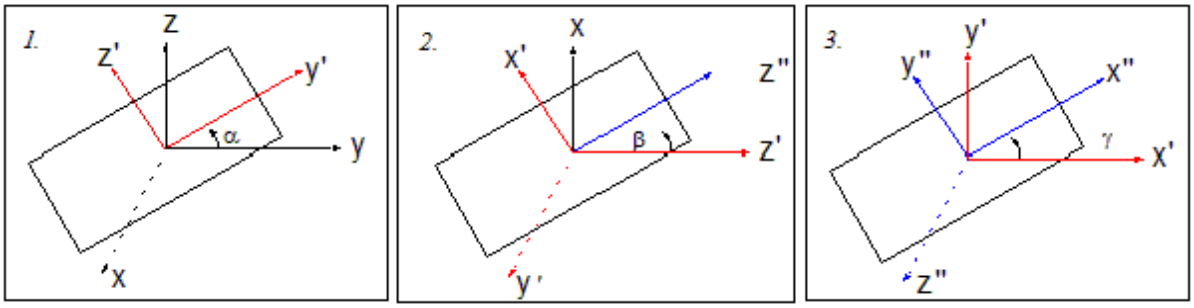
The applied DC bias that causes ferrite saturation is always in the positive z direction of the tensor coordinate system. Initially the tensor coordinate system is assumed to be aligned with the fixed coordinate system; the tensor's z-axis is the same as the model's z-axis. To model other directions of applied bias, the permeability tensor must be rotated so that its z-axis lies in another direction on the fixed coordinate system. This is accomplished by specifying the rotation angles about the axes when you assign a magnetic bias source to a model surface.

The rotation angles should be defined in the **Magnetic Bias Source** dialog box in such a way that the tensor coordinate system is obtained in the following manner:

1. Rotating the tensor coordinate system by α degrees (from the **X Angle**) around the fixed x-axis.
2. Rotating the resulting tensor coordinate system by β degrees (from the **Y Angle**) around the new y-axis.
3. Rotating the new tensor coordinate system by γ degrees (from the **Z Angle**) around the new z-axis.

This concept is illustrated in the following graphic. In the first panel, the permeability tensor is rotated α degrees about the x-axis. In the second panel, the tensor is rotated β degrees about the y'-axis (the new y-axis). In the third panel, the tensor is rotated γ degrees about the z''-axis (the new z-axis). The resulting tensor has the coordinate system (x''y''z'') relative to the fixed coordinate system.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For example, to model the DC bias in the x direction you would rotate the tensor coordinate system so that its z-axis lies along the x-axis of the fixed coordinate system. To do this you would enter **0** for the **X Angle**, **90** for the **Y Angle**, and **0** for the **Z Angle**.

Non-uniform Applied Bias Fields

To accurately model a ferrite in an applied static magnetic bias field, the non-uniform magnetic bias fields must also be calculated. In HFSS, a ferrite's permeability tensor is a direct result of an applied static magnetic bias field. The static field causes the tensor to assume an hermitian form, with cross coupling terms between field components perpendicular to the bias. A uniform bias field is difficult to achieve in practice. Even if the bias field is nearly uniform, a non-ellipsoidal-shaped ferrite material will have non-uniform demagnetization, resulting in non-uniform fields in the ferrite.

Use the magnetostatic solver provided in the Maxwell 3D Field Simulator to generate a solution for non-uniform magnetostatic fields. Once a solution is generated it may be imported into HFSS.

> **Note:**
>
> To specify the non-uniform bias field, you must have purchased the Maxwell 3D Field Simulator. Refer to the Maxwell 3D Field Simulator documentation for instructions on solving for non-uniform magnetostatic fields.

Deembedding

Deembedding means adding or subtracting transmission line. In HFSS only wave ports and Floquet ports are allowed to be deembedded where the S-parameters are modified as described in Deembedding S-Matrices. Fields used for visualization and the calculator can also be modified by selecting Include Post Processing Effects in the Edit Sources panel. The new results appear instantaneously. Be careful if you deembed cutoff-modes because you may get non-physical results.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This section shows an example where subtracting transmission line can be useful and an example where adding transmission line can be useful. A following section describes When HFSS Needs Port Calibration in deembedding.

Consider the aperture-coupled patch antenna shown in Fig. 1. A microstrip trace located below a ground plane feeds a patch antenna located above the ground plane. A slot in the ground plane couples power from the microstrip trace to the antenna.

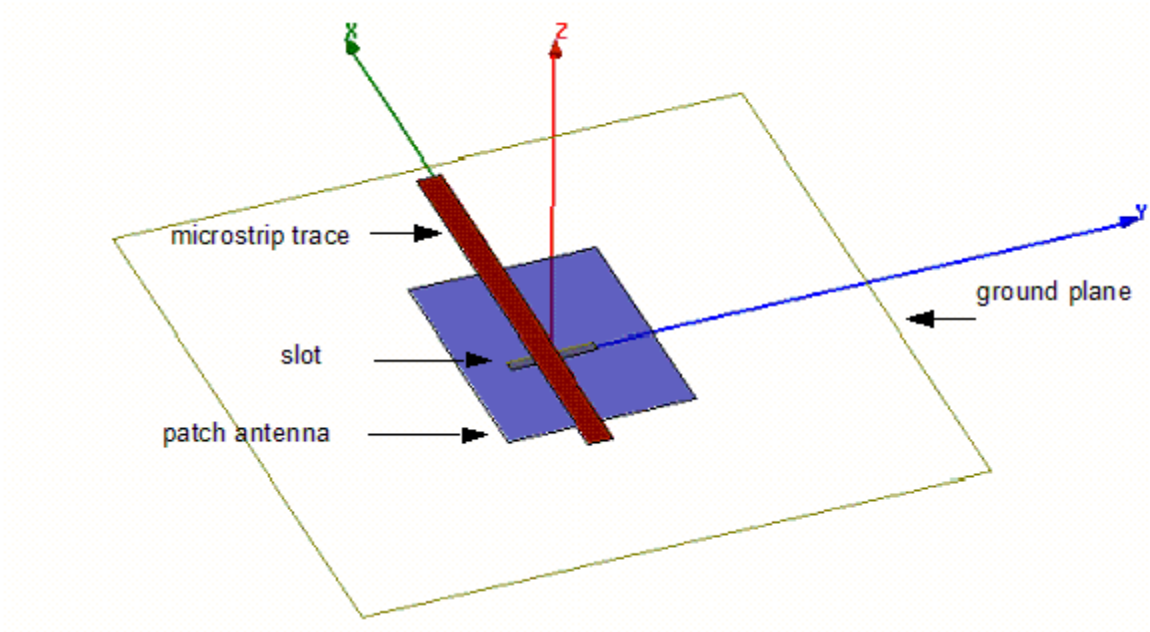
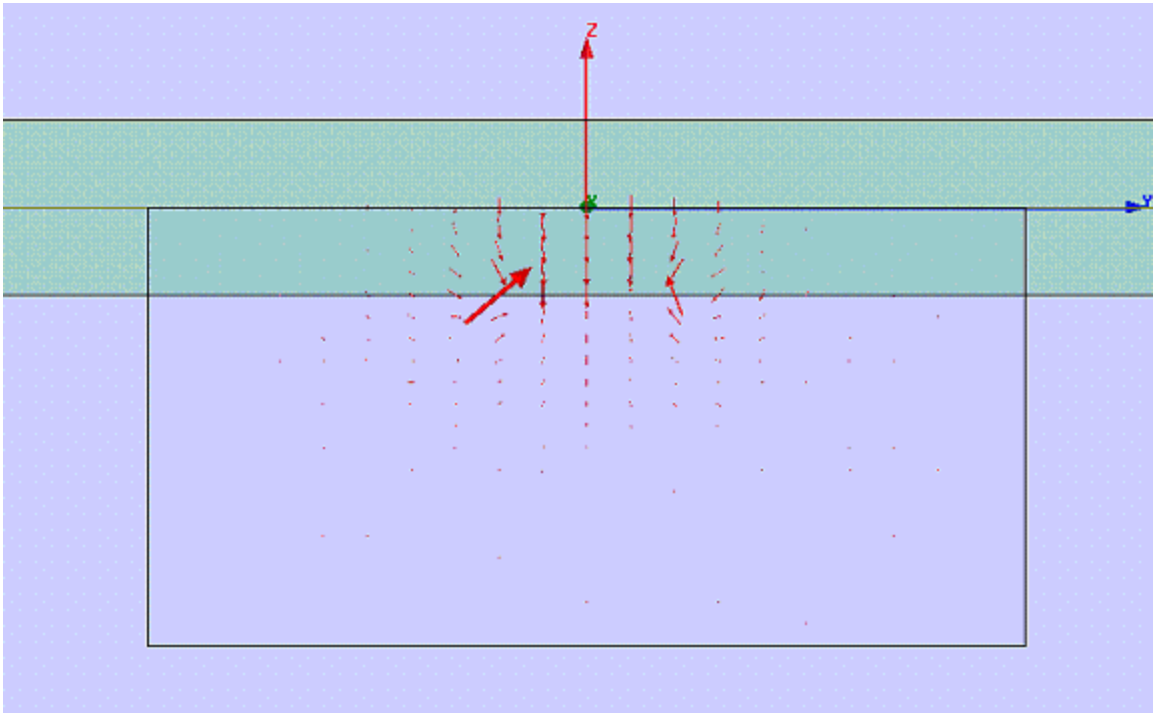


Fig. 1 Aperture-coupled patch antenna

A port is located at the beginning of the trace. It cannot be located very close to the slot, since a port has to be located in a position where a clean transmission-line mode is expected, at some distance from the first discontinuity. The port is shown in Fig. 2. One edge coincides with the ground plane. The other edges are such that the port width is several times the trace width and the port height several times the substrate thickness. The red arrows represent the port field solution. Notice that the three non-ground-plane edges don't influence the port solution noticeably, so this port is large enough.

Fig. 2 Magnitude of S11 in dB

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The S-parameter characteristic is shown in Fig. 3 in a rectangular plot and in Fig. 4 in a Smith chart. Note that the resonance frequency in Fig. 3 corresponds to the point in Fig. 4 where the curve is closest to the center of the chart. Also note that the curve circles the chart several times.

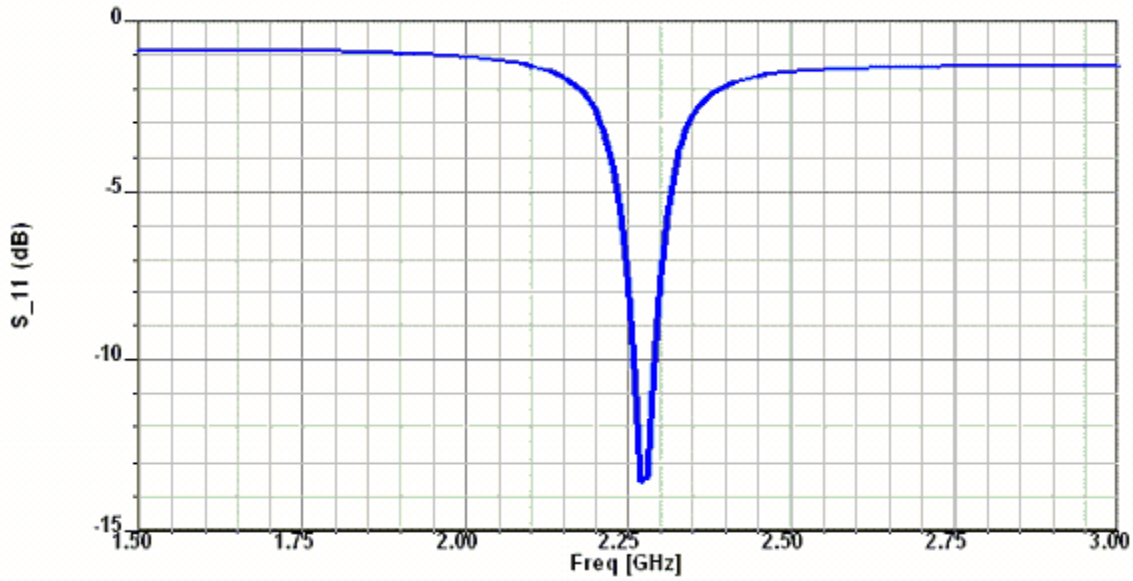


Fig. 3 Magnitude of S11 in dB

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

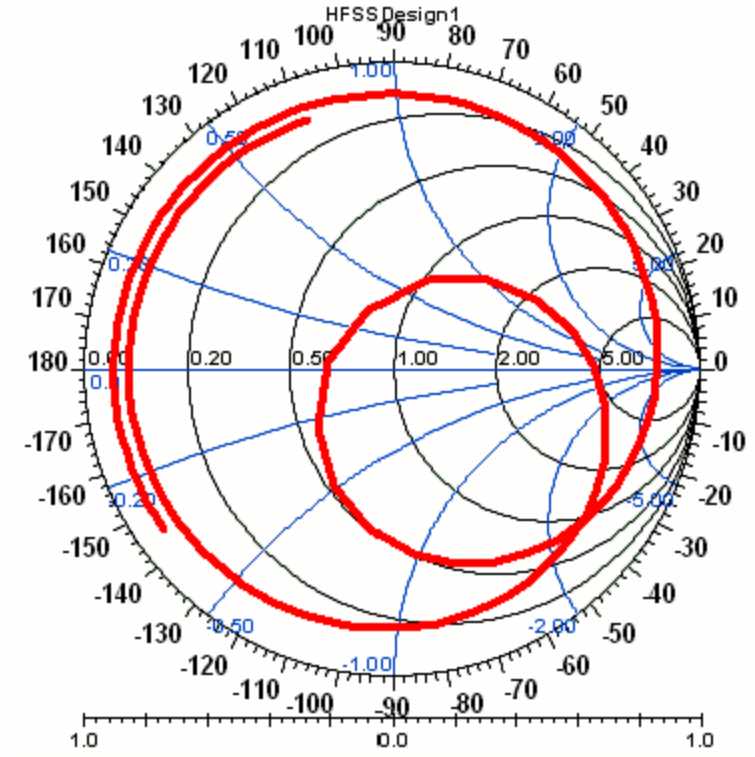


Fig. 4 Smith Chart before deembedding

Antenna designers who need to design a matching circuit close to the feed point may want to see the Smith chart with the reference point located close to the feed point of the antenna, that is. close to the slot, rather than some distance away from the feed point. That can be achieved by deembedding. Fig. 5 shows the Smith chart after deembedding.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

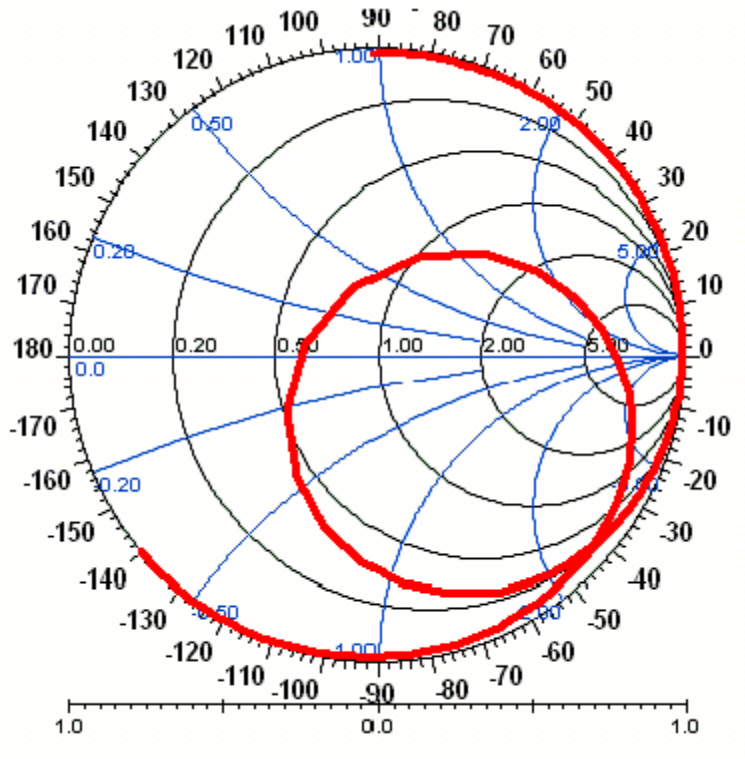


Fig. 5 Smith Chart after deembedding

Note that after this kind of deembedding, where transmission line was removed, the curve circles the Smith chart fewer times.

As an example where adding transmission line may be useful, consider an IC package or a connector mounted on a printed circuit board. Fig. 6 shows a connector on a printed circuit board. The connector designer has included short sections of traces on the board in his model. These are coupled differential lines, and because of the coupling they share one port. Again, there is some distance between the port and the first discontinuity, so the fields that reach the port form clean transmission-line modes. To model the connector accurately, this model is sufficient.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

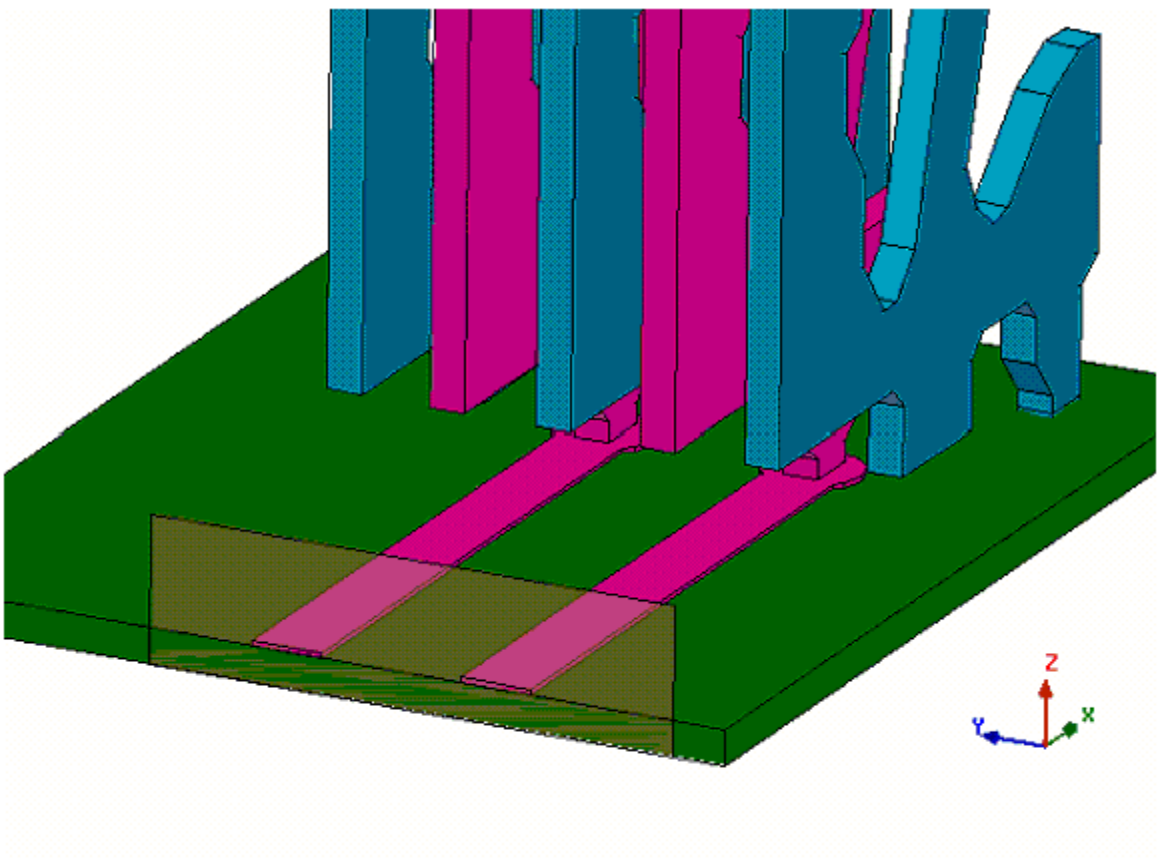


Fig. 6 Connector on printed circuit board

On the board, the traces may continue as uniform transmission lines for some distance. To evaluate system performance, it may be desirable to include that length in the S-parameters that are to be exported to a circuit simulator. With deembedding, uniform transmission line can be added as a matrix post processing step.

Another example where adding transmission-line length through deembedding may be useful is modeling long cables with frequency-dependent materials, for example, in back planes. In HFSS, only a short section would be modeled, and its behavior over the entire frequency range analyzed accurately. Analyzing only a short section has the benefits of both speed and accuracy. Then, with deembedding, the cable can be extended to any desired length. The S-parameters for the full length can be exported to a circuit simulator to be incorporated into a system.

Related Topics

Deembedding Parasitic Port Effects

Deembedding of Design Under Test

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Deembedding Port Inductance Numerically

Assign Wave Ports for Modal Solutions

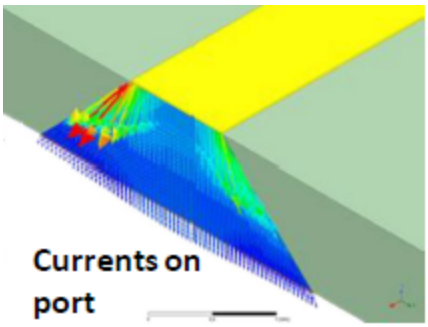
Assign Wave Ports for Terminal Solutions

Assign Floquet Ports

Differential Pair Microstrip

Deembedding Port Inductance Numerically

The port solver impresses current distribution, *J*, onto lump port.

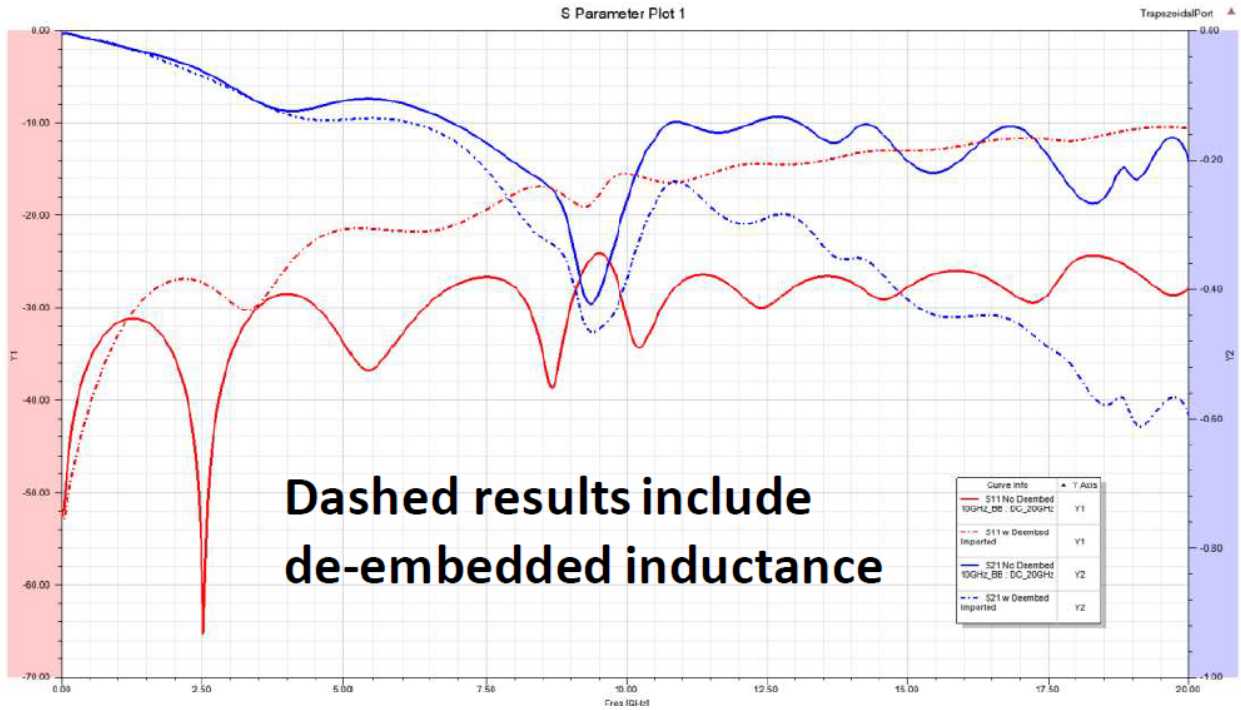


If the sheet comprises N_t triangles, the above surface integrals are written as:

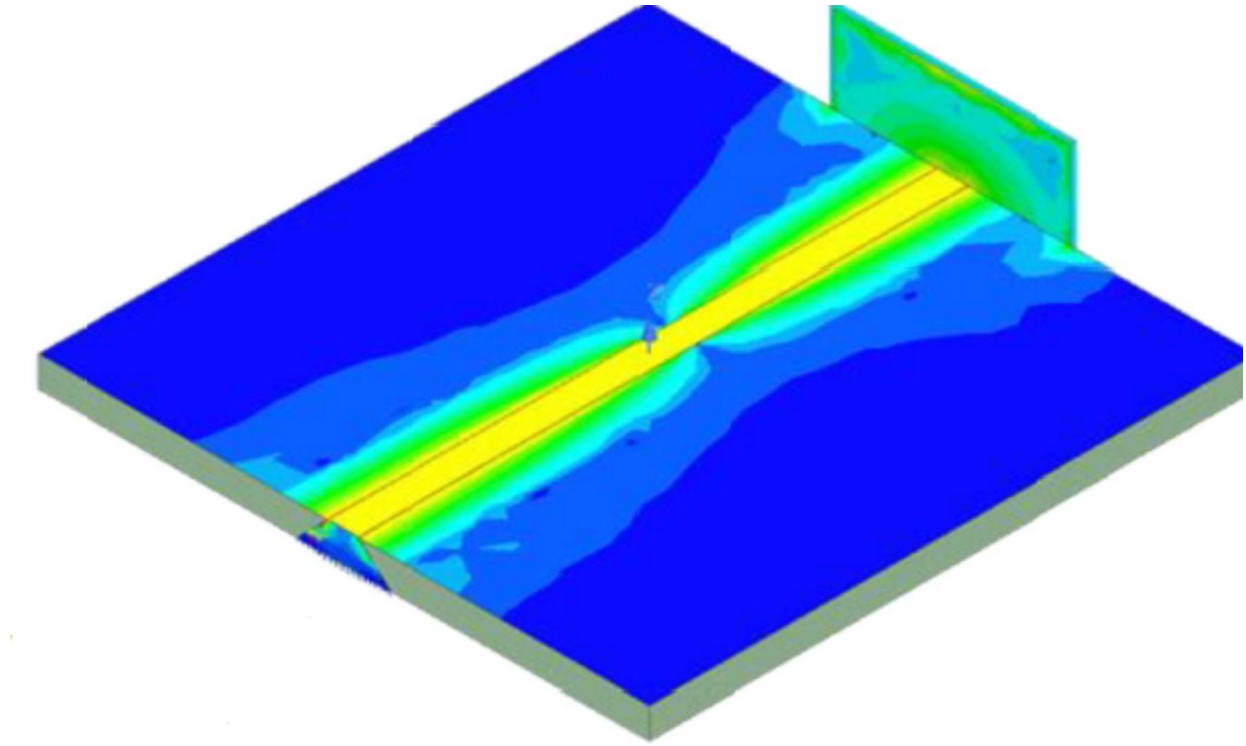
$$L = \frac{\mu_0}{4\pi I^2} \sum_{m=1}^{N_t} \sum_{n=1}^{N_t} \int_{T_m} \int_{T'_n} \frac{\mathbf{J}_s(\mathbf{r}) \cdot \mathbf{J}_s(\mathbf{r}')}{R} dT'_n dT_m$$

Surface integration of *J* upon port triangles returns inductance, *L*.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Microstrip Line with Trapezoidal Lumped and Wave Port Excitations

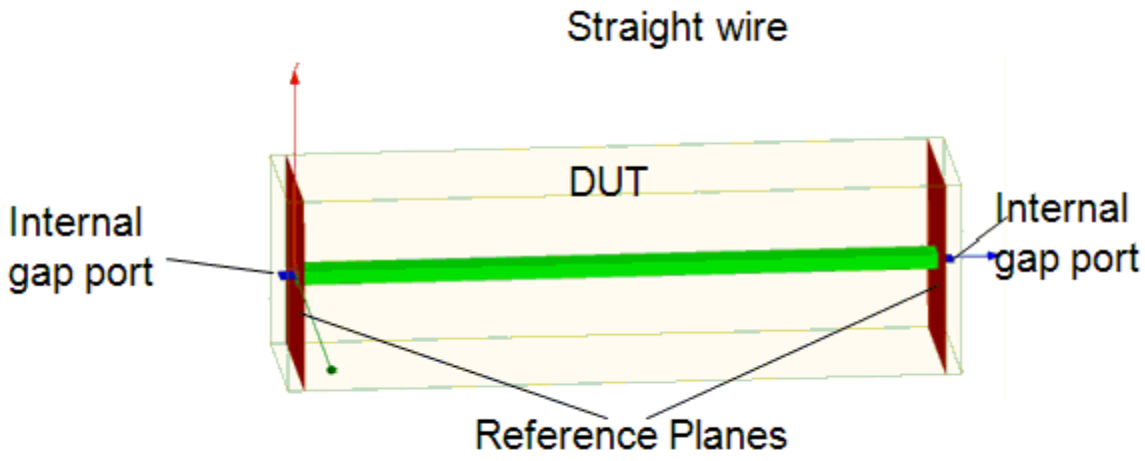


Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

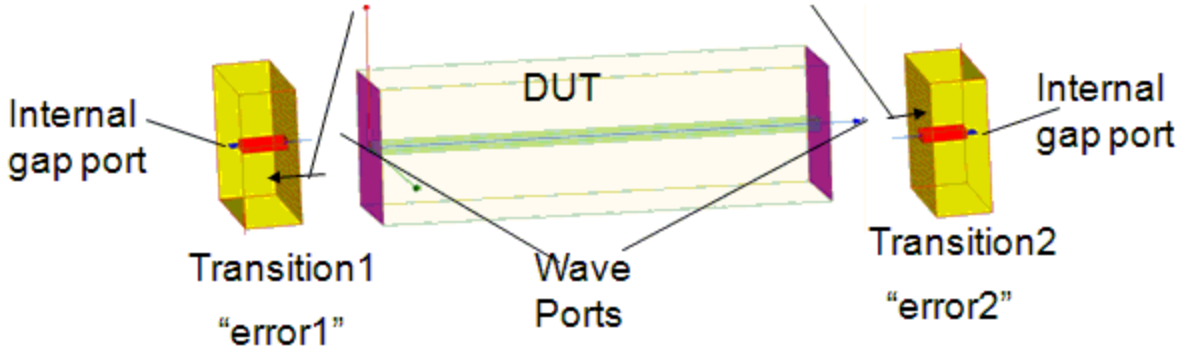
When HFSS Needs Port Calibration (Deembedding)

- If wave ports are used, HFSS does not require any port calibration.
- If internal lumped ports are used, HFSS requires some port calibration (deembedding).
- If lumped or circuit ports are used, parasitic inductance of port geometry can be removed either through calibration or de-embedding.

Deembedding Extension/Transition Effects

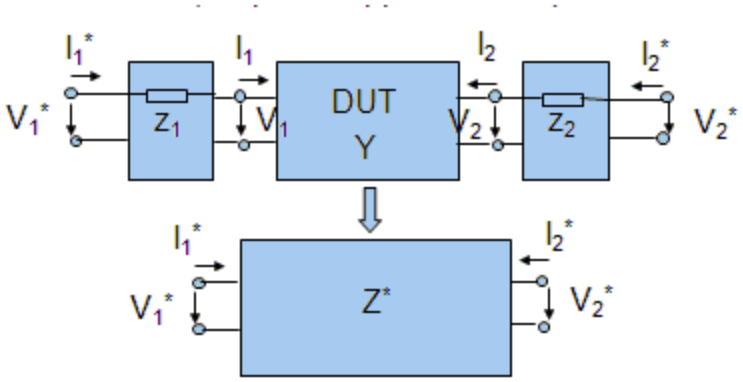


Deembedded to the Discontinuity



Deembedding Extension/Transition Z-matrix Method (Simplified Approximation)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Impedance Matrix Z* is Measured/Simulated

Where $\mathbf{Z_1}$ and $\mathbf{Z_2}$ are known, then calculate/deembed **Z** matrix

Z Matrix Method

Measured/Calculated

$$\mathbf{U}^* = \mathbf{Z}^*\mathbf{I}^* \qquad \mathrm{V}^* = \begin{bmatrix} V_1^* \\ V_2^* \end{bmatrix} \qquad \mathrm{I}^* = \begin{bmatrix} I_1^* \\ I_2^* \end{bmatrix} \qquad \mathrm{Y}^* = \begin{bmatrix} Y_{11}^* & Y_{12}^* \\ Y_{21}^* & Y_{22}^* \end{bmatrix} \tag{1}$$

Dembedded

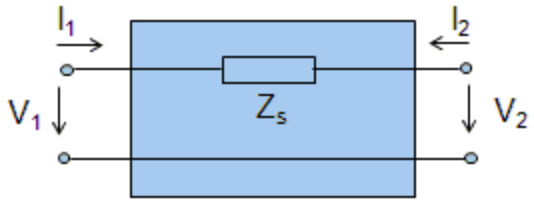
$$\mathbf{U} = \mathbf{Z}\mathbf{I} \qquad \mathrm{V} = \begin{bmatrix} V_1 \\ V_2 \end{bmatrix} \qquad \mathrm{I} = \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} \qquad \mathrm{Y} = \begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix} \tag{2}$$

$$\mathrm{U}^* = \mathrm{U} + (\mathrm{Z_D})\mathrm{I} \qquad Z_D^{\,i} = Z_i \tag{3}$$

$$\mathrm{Z}^*\mathrm{I} = \mathrm{U} + (\mathrm{Z_D})\mathrm{I} \qquad \mathrm{U} = \mathrm{Z}^*\mathrm{I} - (\mathrm{Z_D})\mathrm{I} \tag{4}$$

S matrix calculation from the series impedances to be deembedded

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Formula

(5)*

$$L = \frac{\mu_0 len}{2\pi}\left[\ln\left(u+\sqrt{u^2+1}\right)+u\ln\left(\frac{1}{u}+\sqrt{\frac{1}{u^2}+1}\right)+\frac{1}{3}\left(u^2+\frac{1}{u}-\frac{(u^2+1)^{3/2}}{u}\right)\right]$$

Where:

Input:

w = width of the lumped port [mm]

len = length of the lumped port[mm]

u = len/w

Output:

L[nH]

*From C.R.Paul, *Inductance. Loop and partial*, Wiley, 2010

And

$$Z_s = R_s + jX_s \qquad X_s = \omega L 10^{-9} \quad (6)$$

Currently, $R_s = 0$

$$\mathbf{Y} = -\begin{bmatrix} \frac{1}{Z_s} & -\frac{1}{Z_s} \\ -\frac{1}{Z_s} & \frac{1}{Z_s} \end{bmatrix} \quad (7)$$

$$\mathbf{Y}' = (\mathbf{I}+\mathbf{S})^{-1}(\mathbf{I}-\mathbf{S}) \quad (8)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\mathbf{Y} = \langle \frac{1}{\sqrt{Z_o}} \rangle \mathbf{Y}' \langle \frac{1}{\sqrt{Z_o}} \rangle \qquad Z_o = 1\Omega \tag{9}$$

$$Z_s = \frac{1}{Y_{1,1}} \tag{10}$$

$$\mathbf{Z} = \mathbf{Z}^* - \langle \mathbf{Z}_s \rangle \tag{11}$$

Z* is the global impedance matrix

Z is the Global impedance matrix after deembedding.

Materials Technical Notes

This section of the *Technical Notes* includes information on the following material properties:

Simple Materials

- Relative Permeability
- Relative Permittivity
- Bulk Conductivity
- Dielectric Loss Tangent
- Magnetic Loss Tangent

Frequency Dependent

- Frequency-Dependent Material Properties.
- Debye Materials
- Djordjevic-Sarkar Causal Dielectric Model
- Djordjevic-Sarkar Model Parameter Calculation
- Expressions for Permittivity and Conductivity
- Conductivity and Permittivity Limits

Anisotropic materials

- Anisotropic materials
- Anisotropic relative permeability tensors
- Anisotropic relative permittivity tensors
- Anisotropic conductivity tensors
- Anisotropic dielectric loss tangent tensors
- Anisotropic magnetic loss tangent tensors
- Anisotropic materials and ports

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ferrite materials

- Magnetic Saturation
- Lande G Factor
- Delta H

Relative Permeability

The permeability of dielectrics is assumed to be complex, as follows:

$$\mu = \mu' - j\mu''$$

The relative permeability is a dimensionless quantity, defined as follows:

$$\mu' = \mu_r \mu_0$$

where μ_0 is the permeability of free space.

The permeability can also be expressed as

$$\mu = \mu'\left(1 - j\frac{\mu''}{\mu'}\right)$$

where μ' is the real portion of μ and μ''/μ' is the magnetic loss tangent.

Relative Permittivity

The permittivity of dielectrics is assumed to be complex, as follows:

$$\varepsilon = \varepsilon' - j\varepsilon''$$

which can also be expressed as

$$\varepsilon = \varepsilon'\left(1 - j\frac{\varepsilon''}{\varepsilon'}\right)$$

where ε' is the real portion of ε and from which the relative permittivity is defined as:

$$\varepsilon' = \varepsilon_r \varepsilon_0$$

where $\varepsilon_0$is the permittivity of free space.

$\varepsilon''/\varepsilon'$ is the dielectric loss tangent.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If a material's losses due to bulk conductivity will be significant, such as in semiconductor dielectric materials, an additional bulk conductivity value, σ, must be added. From the time harmonic form of Maxwell's equations, the complex permittivity, ε_c is defined as

$$\nabla \times H = (\sigma + j\omega\varepsilon)E = j\omega\varepsilon_c E$$

where

$$\varepsilon_c = \varepsilon\left(1 - j\tan\delta - j\frac{\sigma}{\omega\varepsilon}\right)$$

Bulk Conductivity

HFSS is capable of including conductivity in the model either as a bulk material loss factor, similar to dielectric loss tangent, or as an impedance boundary condition applied to the outer surfaces of the object. The choice of bulk material loss instead of the boundary condition is made by selecting **Solve Inside** in the **Properties** window.

The choice between bulk material loss and the surface boundary condition is problem dependent. The boundary condition should be applied whenever the conductor is much thicker than the skin depth at the solution frequency. In this case, the unknowns within the conductor are not included in the unknown vector, resulting in a smaller matrix and a faster analysis. However, if the conductor is not thick relative to the skin depth, the bulk material conductivity must be used to arrive at an accurate solution. With this assumption, the wave equation reduces to

$$\nabla\times\left(\frac{-1}{j\omega\mu}\nabla\times\boldsymbol{E}\right) = (j\omega\varepsilon + \sigma)\boldsymbol{E}$$

The threshold where HFSS will automatically choose whether an object will be solved inside can be accessed via the **Tools>Options>HFSS Options**.

Dielectric Loss Tangent

To represent a dielectric that dissipates the power of a high-frequency electric field, enter a dielectric loss tangent, $\varepsilon''/\varepsilon'$, property value for the material. The smaller the loss tangent, the less lossy the material.

The dielectric loss tangent may vary with frequency. To simulate the variances, define a function for the dielectric loss tangent.If a wide-band frequency sweep is being solved, the Frequency Dependent Material property may be appropriate.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Magnetic Loss Tangent

To represent a dielectric that dissipates the power of a high-frequency magnetic field, enter a magnetic loss tangent, μ''/μ', property value for the material. The smaller the loss tangent, the less lossy the material.

> **Note:**
>
> If you plan to do a fast frequency sweep for a design that includes dielectrics, make sure that the dielectric or magnetic loss tangent does not vary significantly over the requested frequency range. If they do, the results may not be what you expect. In cases where the loss tangent does vary significantly over the frequency range in which you are interested, copy and solve the design several times, adjusting the loss tangent and associated frequency range for the copied design so that the loss tangent is relatively stable over the design's requested frequency range.

Ferrite Materials

Ferrite materials are used to model the interaction between a microwave signal and a material whose magnetic dipole moments are aligned with an applied bias field. The gyrotropic quality of the ferrite is evident in the permeability tensor which is Hermitian in the lossless case. The Hermitian tensor form leads to the non-reciprocal nature of the devices containing microwave ferrites. If the microwave signal is circularly polarized in the same direction as the precession of the magnetic dipole moments, the signal interacts strongly with the material. When the signal is polarized in the opposite direction to the precession, the interaction will be weaker. Because the interaction between the signal and material depends on the direction of the rotation, the signal propagates through a ferrite material differently in different directions.

If you assign a ferrite material to an object, you must assign a magnetic bias source to the object.

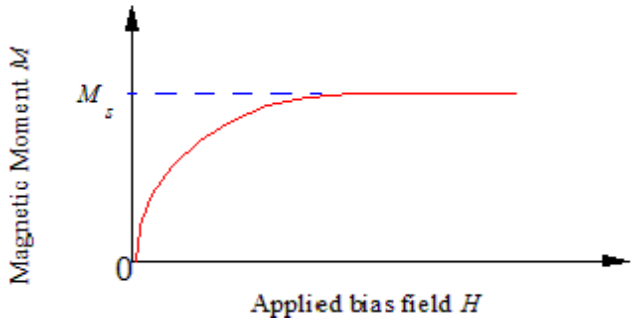
- Magnetic Saturation
- Lande G Factor
- Delta H
- Ferrite Permeability Tensor in HFSS

Magnetic Saturation

A material with a non-zero magnetic saturation is considered to be a ferrite. When a ferrite is placed in a uniform magnetic field, the magnetic dipole moments of the material begin to align with the field. As the strength of the applied bias field increases, more of the dipole moments align. The magnetic saturation, M_s, is a property that describes the point at which all of the

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

magnetic dipole moments of the material become aligned. At this point, further increases in the applied bias field strength do not result in further saturation. The relationship between the magnetic moment, *M* and the applied bias field, *H*, is shown below.



You specify $4\pi M_s$ when entering a value for Magnetic Saturation in the Materials database for HFSS.

Lande G Factor

The Lande *g* factor is a ferrite property that, on a microscopic level, describes the total magnetic moment of the electrons according to the relative contributions of the orbital moment and the spin moment. When the total magnetic moment is due entirely to the orbital moment, *g* is equal to one. When the total magnetic moment is due entirely to the spin moment, *g* is equal to two. For most microwave ferrite materials, *g* has a range from 1.99 to 2.01. The Lande *g*factor is dimensionless.

Delta H

Delta H is the full resonance line width at half-maximum, which is measured during a ferromagnetic resonance measurement. It relates to how rapidly a precessional mode in the biased ferrite will damp out when the excitation is removed. The factor ΔH doesn't appear in the permeability tensor; instead, the factor α appears. The factor α is computed from

$$\alpha = -\frac{\gamma \mu_0 \Delta H}{2\omega}$$

The factor α changes the κ and χ terms in the permeability tensor from real to complex, which makes the tensor complex non-symmetric (where it had been hermitian for lossless ferrites).

Enter the full resonance line width at half maximum in the **Delta H** value box. Delta H is measured at a specific frequency. That frequency needs to be entered by the user at the interface. 9.4 GHz is the frequency where Delta H is typically measured.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ferrite Permeability Tensor in HFSS

Gyrotropic Permeability

The ferrite capability of HFSS is based on the Polder susceptibility tensor small signal approximation of the Landau-Lifshitz equation of motion of a magnetic dipole in a uniform bias field [1] [2].

The formulas are given in CGS units. The formulas assume a saturated ferrite sample, and both M_s and H_o should be set to 0 for the unbiased demagnetized case.

$$\bar{\mu} = \mu_0 \begin{bmatrix} 1+\chi & j\kappa & 0 \\ -j\kappa & 1+\chi & 0 \\ 0 & 0 & 1 \end{bmatrix} \quad (1)$$

Where

$$\chi = \frac{\omega_o \omega_m}{\omega_o^2 - \omega^2} \quad (2)$$

$$\kappa = \frac{\omega \omega_m}{\omega_o^2 - \omega^2} \quad (3)$$

With

$$\omega = 2\pi f \quad (4)$$

$$\omega_o = \gamma \mu_o H_o \quad (5)$$

$$\omega_m = \gamma \mu_o M_s \quad (6)$$

If M_s is not = 0 this tensor overrides any value of permeability
even if magnetic bias is set to 0.

And

$$\gamma = g_l \gamma_e \quad (7)$$

γ_e is half of the electron charge to mass ratio and g_l is the Lande g factor. The Lande g factor is typically between 1 and 2, with 1 corresponding to orbital angular momentum and 2 for spin.

If the ferrite has magnetic losses, we replace ωo by $\omega o + j\omega\alpha$ where α is computed from the ferromagnetic resonance linewidth:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\alpha = \frac{\mu_o \gamma \Delta H}{4\pi f_{FMR}} \quad (8)$$

When HFSS assembles the finite element matrices for ferrite materials it computes the permeability tensor, 1, based on several different inputs:

1. **Frequency** - ω
2. **Material properties** - all of which are specified in the material manager
   a. Saturation Magnetization - *M*s
   b. Lande g factor - *g*l
   c. Loss factor - computed from Δ*H* and *f*FMR
3. **Magnetostatic bias field** - Magnetic Bias source, either:
   a. Uniform bias - *H*o and direction specified in the interface
   b. Non-uniform bias - *H*o and local tensor $\hat{z}$ direction determined by the magnetostatic field solution from Maxwell3D. When the Magnetic Bias source is nonuniform, the permeability tensor will be different in each ferrite tetrahedron.

References

[1] David Pozar, Microwave Engineering, Addison-Wesley, 1990.

[2] Daniel D. Stancil, Theory of Magnetostatic Waves, Springer-Verlag, 1992.

Anisotropic Materials

Anisotropic materials have characteristics that vary with direction. These characteristics are defined by their anisotropy tensors. You must define three diagonals each for anisotropic permittivity, electric loss tangent, conductivity, permeability, and magnetic loss tangent. Each diagonal represents a tensor of your model along an axis. The alignment of the materials axis with respect to the object is discussed in Assigning Material Property Types.

- Anisotropic relative permeability tensors
- Anisotropic relative permittivity tensors
- Anisotropic conductivity tensors
- Anisotropic dielectric loss tangent tensors
- Anisotropic magnetic loss tangent tensors
- Anisotropic materials and ports

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Anisotropic Relative Permeability Tensors

The permeability tensor for an anisotropic material is described by

$$[\mu] = \begin{bmatrix} \mu_1\mu_0 & 0 & 0 \\ 0 & \mu_2\mu_0 & 0 \\ 0 & 0 & \mu_3\mu_0 \end{bmatrix}$$

where

- μ_1 is the relative permeability along one axis of the material's permeability tensor.
- μ_2 is the relative permeability along the second axis.
- μ_3 is the relative permeability along the third axis.
- μ_0 is the permeability of free space.

The relationship between *B* and *H* is:

$$\begin{matrix} x \rightarrow \mu \\ y \rightarrow v \\ z \rightarrow w \end{matrix} \quad \begin{bmatrix} B_x \\ B_y \\ B_z \end{bmatrix} = [\mu] \begin{bmatrix} H_x \\ H_y \\ H_z \end{bmatrix}$$

To specify the relative permeability for an anisotropic material, enter the μ_1, μ_2, and μ_3 values in the **Value** boxes of the **T(1,1)**, **T(2,2)**, and **T(3,3)** rows, respectively. If the relative permeability is the same in all directions, use the same value for μ_1, μ_2, and μ_3. These values can also be entered as variables.

Anisotropic Relative Permittivity Tensors

The permittivity tensor for an anisotropic material is described by

$$\varepsilon = \begin{bmatrix} \varepsilon_1\varepsilon_0 & 0 & 0 \\ 0 & \varepsilon_2\varepsilon_0 & 0 \\ 0 & 0 & \varepsilon_3\varepsilon_0 \end{bmatrix}$$

where

- ε_1 is the relative permittivity of the material along one tensor axis.
- ε_2 is the relative permittivity along the second axis.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- ε_3 is the relative permittivity along the third axis.
- ε_0 is the permittivity of free space.

The relationship between *E* and *D* is then

$$\begin{matrix} x \to \mu \\ y \to v \\ z \to w \end{matrix} \quad \begin{bmatrix} D_x \\ D_y \\ D_z \end{bmatrix} = [\varepsilon] \begin{bmatrix} E_x \\ E_y \\ E_z \end{bmatrix}$$

To specify the relative permittivity for an anisotropic material, enter the ε_1, ε_2, and ε_3 values in the **Value** boxes of the **T(1,1)**, **T(2,2)**, and **T(3,3)** rows, respectively. These values can also be entered as variables.

Anisotropic Conductivity Tensors

The conductivity tensor for an anisotropic material is described by

$$[\sigma] = \begin{bmatrix} \sigma_1 & 0 & 0 \\ 0 & \sigma_2 & 0 \\ 0 & 0 & \sigma_3 \end{bmatrix}$$

where

- σ_1 is the conductivity along one axis of the material's conductivity tensor.
- σ_2 is the conductivity along the second axis.
- σ_3 is the conductivity along the third axis.

The relationship between *J* and *E* is then:

$$\begin{matrix} x \to \mu \\ y \to v \\ z \to w \end{matrix} \quad \begin{bmatrix} J_x \\ J_y \\ J_z \end{bmatrix} = [\sigma] \begin{bmatrix} E_x \\ E_y \\ E_z \end{bmatrix}$$

To specify the conductivity for an anisotropic material, enter the σ_1, σ_2, and σ_3 values in the **Value** boxes of the **T(1,1)**, **T(2,2)**, and **T(3,3)** rows, respectively. The values of σ_1 and σ_2 apply to axes that lie in the xy cross-section being modeled. The values of σ_3 apply to the z-component. These values affect current flowing in dielectrics between the conductors. These values can also be entered as variables.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Anisotropic Dielectric Loss Tangent Tensors

The dielectric loss tangent tensor for an anisotropic material is described by

$$[\bar{\varepsilon}] = \begin{bmatrix} \varepsilon'_1(1 - j\tan\delta_1) & 0 & 0 \\ 0 & \varepsilon'_2(1 - j\tan\delta_2) & 0 \\ 0 & 0 & \varepsilon'_3(1 - j\tan\delta_3) \end{bmatrix}$$

where

- $\tan\delta_1$ is the ratio of the imaginary relative permittivity to the real relative permittivity in one direction.

  $$\tan\delta_1 = \frac{\varepsilon''_1}{\varepsilon'_1}$$

- $\tan\delta_2$ is the ratio of the imaginary relative permittivity to the real relative permittivity in the second direction.

  $$\tan\delta_2 = \frac{\varepsilon''_2}{\varepsilon'_2}$$

- $\tan\delta_3$ is the ratio of the imaginary relative permittivity to the real relative permittivity in the third orthogonal direction.

  $$\tan\delta_3 = \frac{\varepsilon''_3}{\varepsilon'_3}$$

- ε'_1, ε'_2, and ε'_3 are the real relative permittivity's specified earlier.ε

The relationship between *D* and *E* will then be

$$\begin{matrix} x \to \mu \\ y \to v \\ z \to w \end{matrix} \quad \begin{bmatrix} D_x \\ D_y \\ D_z \end{bmatrix} = [\bar{\varepsilon}] \begin{bmatrix} E_x \\ E_y \\ E_z \end{bmatrix}$$

To specify the electric loss tangent for an anisotropic material, enter the $\tan\delta_1$, $\tan\delta_2$, and $\tan\delta_3$ values in the **Value** boxes of the **T(1,1)**, **T(2,2)**, and **T(3,3)** rows, respectively. These values can also be entered as variables.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Anisotropic Magnetic Loss Tangent Tensors

The magnetic loss tangent tensor for an anisotropic material is described by

$$[\bar{\mu}] = \begin{bmatrix} \mu'_1(1 - j\tan\delta_{M1}) & 0 & 0 \\ 0 & \mu'_2(1 - j\tan\delta_{M2}) & 0 \\ 0 & 0 & \mu'_3(1 - j\tan\delta_{M3}) \end{bmatrix}$$

where

- $\tan\delta_{M1}$ is the ratio of the imaginary relative permeability to the real relative permeability in one direction.

  $$\tan\delta_{M1} = \frac{\mu''_1}{\mu'_1}$$

- $\tan\delta_{M2}$ is the ratio of the imaginary relative permeability to the real relative permeability in the second direction.

  $$\tan\delta_{M2} = \frac{\mu''_2}{\mu'_2}$$

- $\tan\delta_{M3}$ is the ratio of the imaginary relative permeability to the real relative permeability in the third orthogonal direction.

  $$\tan\delta_{M3} = \frac{\mu''_3}{\mu'_3}$$

- μ'_1, μ'_2, and μ'_3 are the real relative permeabilities specified earlier.

The relationship between *B* and *H* will then be

$$\begin{matrix} x \to \mu \\ y \to v \\ z \to w \end{matrix} \begin{bmatrix} B_x \\ B_y \\ B_z \end{bmatrix} = [\bar{\mu}] \begin{bmatrix} H_x \\ H_y \\ H_z \end{bmatrix}$$

To specify the magnetic loss tangent for an anisotropic material, enter the $\tan\delta_{M1}$, $\tan\delta_{M2}$, and $\tan\delta_{M3}$ values in the **Value** boxes of the **T(1,1)**, **T(2,2)**, and **T(3,3)** rows, respectively. These values can also be entered as functions.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Anisotropic Materials and Ports

An anisotropic material can be in contact with a port under the following conditions:

- One principal axis of the anisotropic material is aligned normal to the port.

Debye Materials

If the material is a lossy dielectric with a lower frequency near DC, use the **Loss Model Material Input** dialog box to specify the material's conductivity at DC or, if you prefer, its loss tangent value at the lower frequency. HFSS also enables you to specify the lossy dielectric material's high frequency/optical permittivity.

In materials commonly encountered in microwave applications ion and dipole polarization dominate. These polarization types can be described by Debye's relaxation polarization model:

$$\varepsilon_{rcomplex} = \varepsilon_{roptical} + \frac{(\varepsilon_{rstatic} - \varepsilon_{roptical})}{1 + j\omega\tau}$$

where

τ = the relaxation time.

$\varepsilon_{rstatic}$= the static permittivity.

$\varepsilon_{roptical}$ = the high frequency/optical permittivity.

HFSS uses the values you specify in the **Loss Model Material Input** dialog box in Debye's equation above to determine the relative permittivity at any frequency.

Debye's model is valid for most microwave applications. If the frequency exceeds the limit of Debye's model, other models that take atomic and electron polarization into account are available.

Frequency-dependent materials are appropriate for problems solved using a discrete or interpolating sweep. In a fast sweep, frequency dependent material properties are evaluated at the center frequency which may result in less accurate results at the sweep extremes, depending on the sweep range and frequency dependence of material property.

In the simulation of high speed connectors or PCB boards, it is important to take the losses into account. Especially, in the case of transient analysis, where the improper specification of the frequency dependence of the materials would lead to unphysical results. This section discusses the loss mechanism of dielectric materials. Magnetic losses could also be taken into account, but materials commonly encountered in these applications are non-magnetic.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A lossy dielectric material is characterized by two measured values at a certain frequency: dielectric constant ε_r and loss tangent $tg\delta$. There are two problems with the specification of the frequency dependence of dielectric materials:

- The frequency range, in which the solution has to be calculated, is much wider than the range where measured material data are available. It is very common to have just two measured points. A low frequency point, which can be taken as DC value and a higher frequency point, which serves to describe the behavior of the material at high frequency. The question arises: What is an appropriate model across the measured range as well as outside the measured range?
- A measurement always suffers from errors. If the measured data are not consistent, it could cause unphysical effects during the time domain analysis. A question arises again: how to check whether the measured data are consistent, and how to adjust them if they are not.

To answer the questions raised we need to discusses the polarization loss mechanism of dielectric materials.

Figure 1 shows the different polarization mechanisms occur in the frequency domain. It can be seen from the figure that up to the microwave region, ion and dipole polarization dominate. These polarization types can be described by Debye's relaxation polarization model:

$$\varepsilon_{rcomplex} = \varepsilon_{roptical} + \frac{(\varepsilon_{rstatic} - \varepsilon_{roptical})}{1 + j\omega\tau} \quad (1)$$

where

τ is the relaxation time,$\varepsilon_{rstatic}$,and $\varepsilon_{roptical}$ are the static and high frequency permittivity, respectively. Debye's model is valid for the most of microwave applications. If the frequency exceeds the limit of Debye's model, there exists other models which take the atomic and electron polarization into account [1].

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

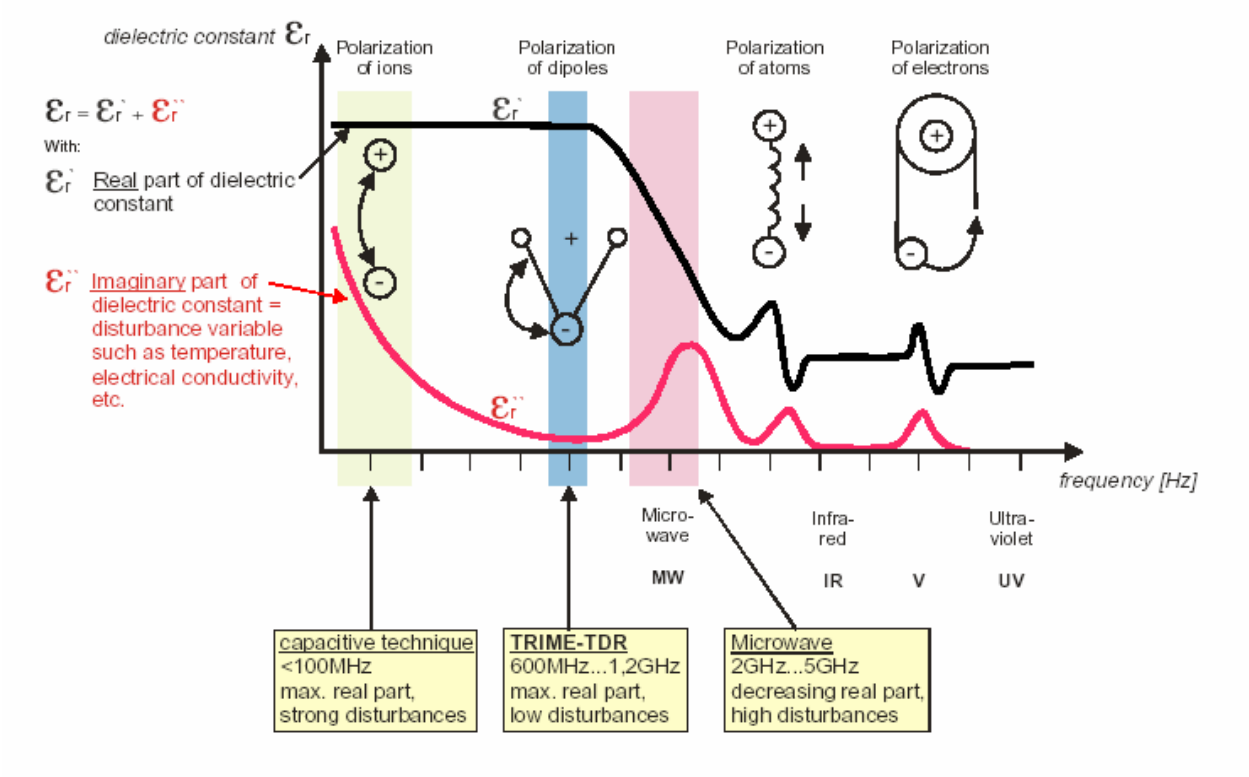


Figure 1: Polarization mechanisms

The question is now how to fit Debye's model to the actual material characteristics. In order to do this, let us express the real part of the dielectric constant and the conductivity from Eq. (1):

$$\varepsilon_r = \varepsilon_{roptical} + \frac{(\varepsilon_{rstatic} - \varepsilon_{roptical})}{1 + (\omega\tau)^2} \quad (2)$$

$$\sigma = \sigma_o + \frac{\omega^2 \varepsilon_o \tau (\varepsilon_{rstatic} - \varepsilon_{roptical})}{1 + (\omega\tau)^2} \quad (3)$$

where

σ_o is the DC conductivity.

Eqs (2) and (3) have 4 parameters: σ_o, $\varepsilon_{rstatic}$, $\varepsilon_{roptical}$ and τ.

Usually, measured data are available at two frequencies for a regular material. The measured data are the dielectric constant and the loss tangent. One set of the measured data is at low frequency

($f_1 \approx 1MHz$, $\varepsilon_{r1}, \sigma_1$) and the other set is at higher frequency ($f_2 \approx 1 \sim 2\,GHz$, $\varepsilon_{r2}, \sigma_2$).

The static or dc values can be considered as the low frequency measured data. So,

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\varepsilon_{rstatic} = \varepsilon_{r1} \quad (4)$$

and

$$\sigma_o = \omega_1 \varepsilon_o \varepsilon_{r1} \tan \delta_1 \quad (5)$$

The critical need is to predict the high frequency behavior of the material.

If we know $\varepsilon_{roptical}$ from measurement, the high frequency behavior of Debye's model is set. We just need to calculate the relaxation time as:

$$\tau = \frac{b - \sqrt{b^2 - 4}}{2\omega_2} \quad (6)$$

where

$$b = \frac{\omega_2 \varepsilon_o (\varepsilon_{rstatic} - \varepsilon_{roptical})}{\sigma_2} \quad (7)$$

and

$$\sigma_2 = \omega_2 \varepsilon_o \varepsilon_{r2} \tan \delta_2 - \sigma_o \quad (8)$$

Knowing τ, Eqs. (2) and (3) prescribe the material characteristic in the whole frequency region.

If we do not know $\varepsilon_{roptical}$, it can be calculated by simultaneously solving the following two equations to get $\varepsilon_{roptical}$ and τ:

$$\varepsilon_{r2} = \varepsilon_{roptical} + \frac{(\varepsilon_{rstatic} - \varepsilon_{roptical})}{1 + (\omega\tau)^2} \quad (9)$$

$$\sigma_2 = \sigma_o + \frac{\omega^2 \varepsilon_o \tau (\varepsilon_{rstatic} - \varepsilon_{roptical})}{1 + (\omega\tau)^2} \quad (10)$$

where ε_{r2} is measured and σ_2 is determined by Eq.(8).

Solving equations (9) and (10), we get:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\tau = \frac{\varepsilon_o \left| \varepsilon_{rstatic} - \varepsilon_{r2} \right|}{\left| \sigma_2 - \sigma_o \right|} \quad (11)$$

This method also yields the parameters of the Debye's model, but the accuracy of the method, especially at high frequencies, depends on the accuracy and the consistency of the measurement at

frequency f_2.

Typical Debye's model material characteristics can be seen in Fig. 2, where $\varepsilon' = \varepsilon_r$.

$$\varepsilon'' = \frac{\omega\tau(\varepsilon_{rstatic} - \varepsilon_{roptical})}{1 + (\omega_2\tau)^2} \quad (12)$$

$$\varepsilon_{roptical} = \varepsilon_{rstatic} - \left| \varepsilon_{rstatic} - \varepsilon_{r2} \right| \left(\frac{1 + (\omega_2\tau)^2}{(\omega_2\tau)^2} \right) \quad (13)$$

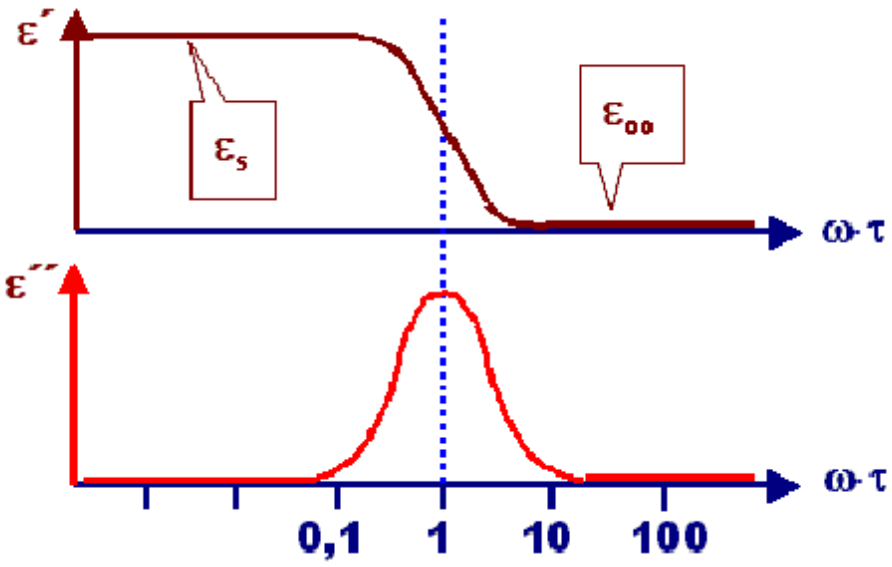


Figure 2: Material characteristics of a Debye model

The model was verified by using a double strip line filled by FR4 material. A 10 mils section of the line was modeled by HFSS and de-embedded into a 34 inch long line. The structure of HFSS model can be seen in Fig. 3.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

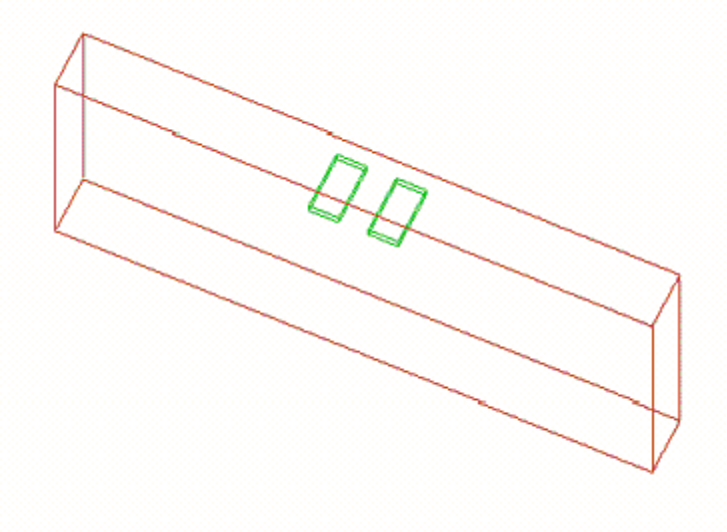


Figure 3: A 10 mils long structure of a double strip line

The input data panel of HFSS for FR4 can be seen in Fig. 4:

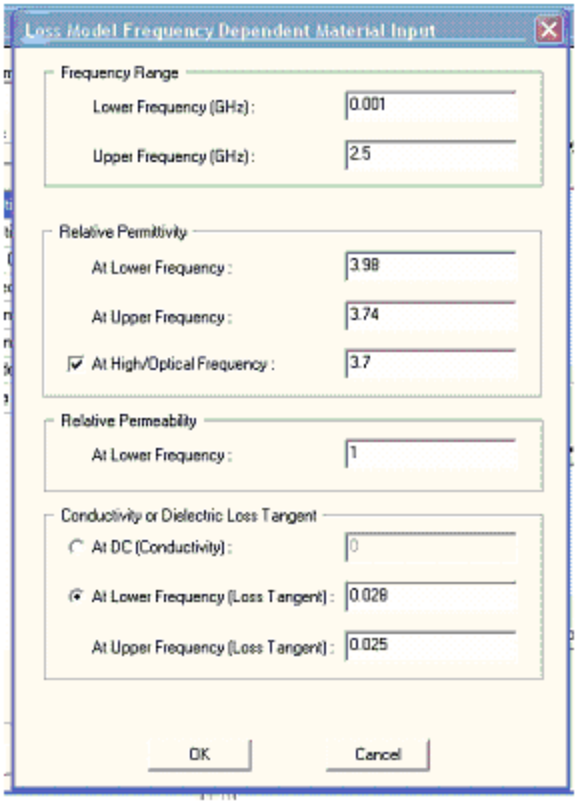


Figure 4: Input data panel for FR4 material in HFSS

It can be seen from the panel that:

$$\varepsilon_{rstatic} = 3.98 \quad \varepsilon_{roptical} = 3.7 \quad \tan\delta_2 = 0.025 \quad \tan\delta_1 = 0.028$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The calculated and measured frequency response of S_{12} can be seen in Fig. 5. The agreement between the calculated and measured values is good.

The transient response of the line to an input pulse also has been calculated. The schematic arrangement of the model in Maxwell Spice is shown in Fig. 6. The output signal is plotted in Fig. 7. The green curve is the transient response of an ideal reference line (lossless) of the same length.

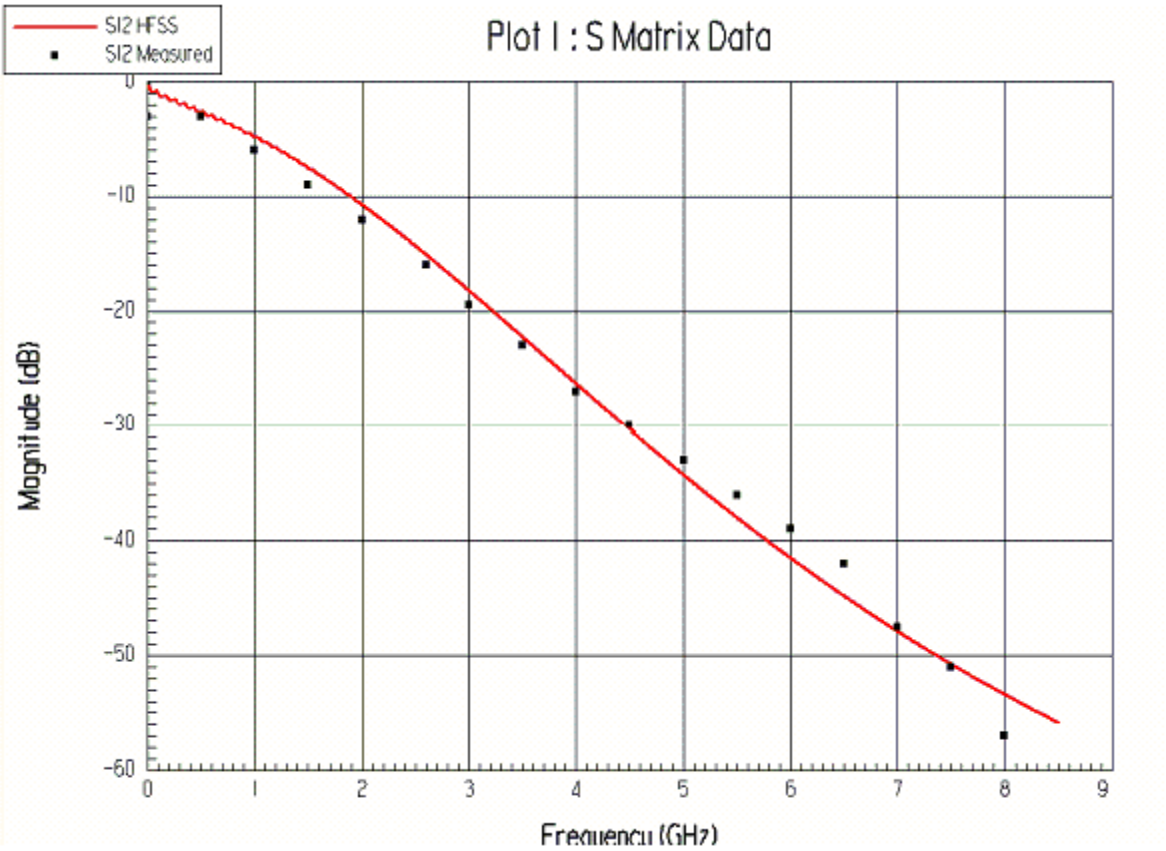


Figure 5: Calculated and measured frequency response of S_{12}

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

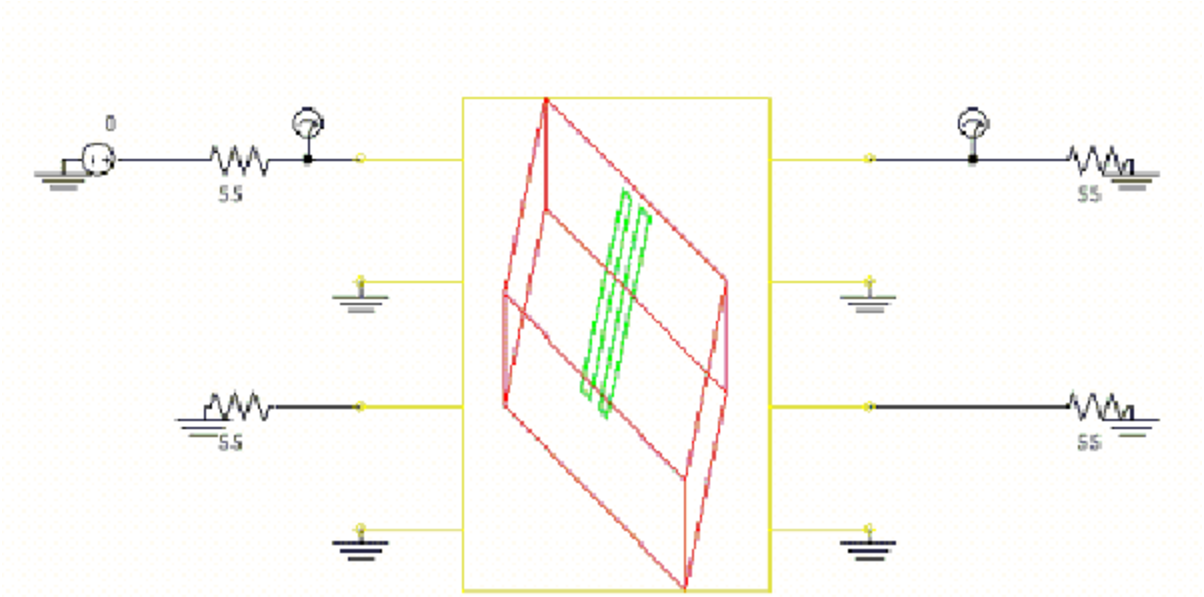
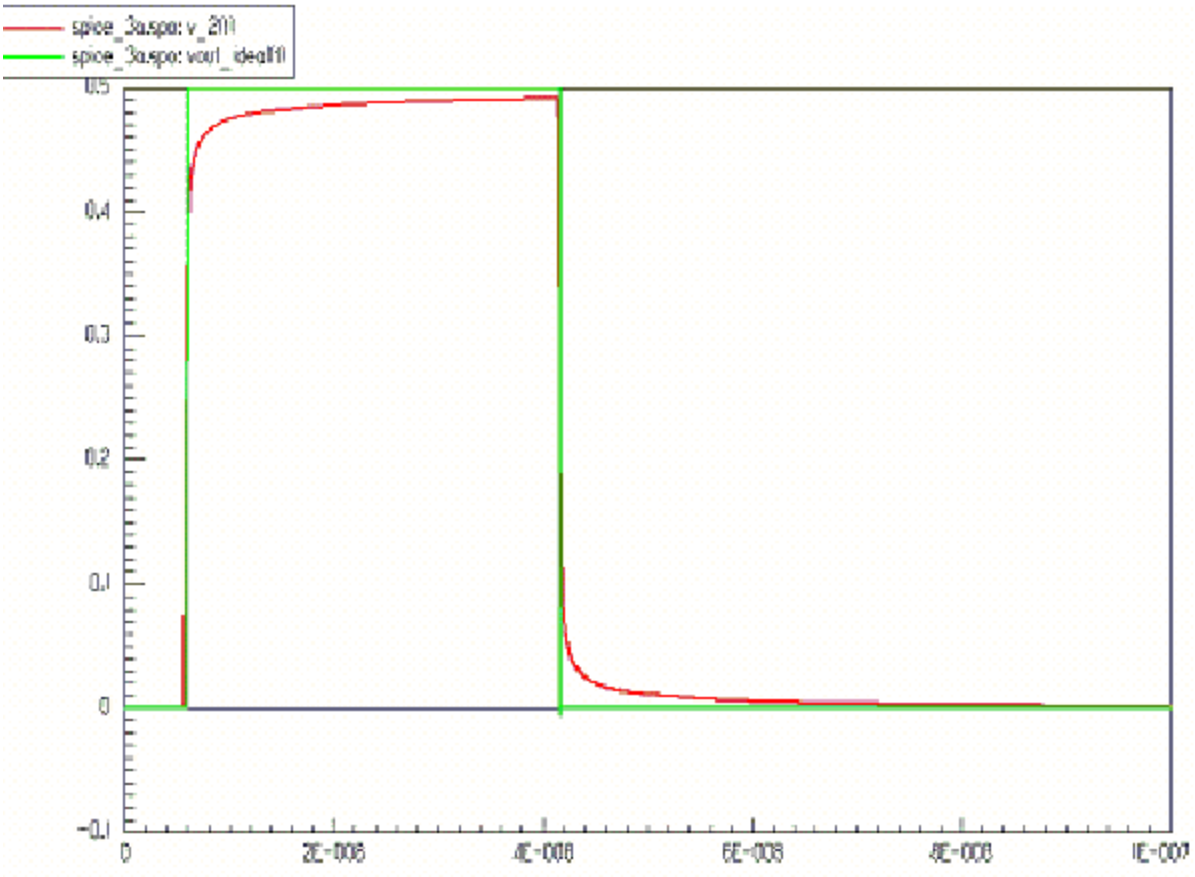


Figure 6: Schematic arrangement in Maxwell Spice



Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7: Transient response by Maxwell Spice

Debye's material loss model predicts the dielectric losses in the whole frequency range within an acceptable accuracy. The model can be set up by using low frequency and higher frequency measured dielectric constants and loss tangents and the optical dielectric constant. If the latter is not available, the model might be less accurate at high frequencies. Using Debye's model, no unphysical phenomena can be observed at the transient response.

References

[1] E.U. Condon and Hugh Odishaw, *Handbook of Physics*, McGraw Hill Book Company, Inc., New York Toronto London, 1958. Pp. 4-113 - 4-119.

[2] G.R. Strobl, *The Physics of Polymers,* Chapter 5, 2nd Ed. Springer, NY, 1977.

Multipole Debye Model

The classical Debye model is based on a single pole or time constant. The Multi-Pole Debye Model attempts to generalize the Debye model by adding multiple poles providing greater flexibility in matching arbitrary measurement data.

A lossy dielectric has a complex relative permittivity denoted as follows:

$$\varepsilon(j\omega)$$

It can be expressed as follows:

$$\varepsilon(j\omega) = \varepsilon_r(j\omega) + j\varepsilon_i(j\omega) = \varepsilon_r(j\omega)[1 - j\delta(j\omega)] = \varepsilon_r(j\omega) - j\varepsilon_r(j\omega)\delta(j\omega)$$

where

$\varepsilon_r(j\omega)$= real valued permittivity and $\delta(j\omega)$ = loss tangent

If the relative permittivity and loss tangent measured at a finite set of frequency points are known, a rational function model can be fit to this data. This approximating rational function can be expressed in pole-residue form as

$$\tilde{\varepsilon}(j\omega) = k_0 + \sum_{i=1}^{N} \frac{k_i}{j\omega - p_i}$$

where

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

p_i = the i-th pole and
k_i = the corresponding residue

Both are potentially complex valued quantities.

N = number of poles which needs to be determined.
k_0 = a high frequency limit, or "optical" permittivity and is also a fitting parameter.

The fitting problem can be expressed as follows: minimize the sum of the least squares differences:

$$E = \sum_{n=1}^{N_f} |\tilde{\epsilon}(j\omega_n) - \epsilon(j\omega_n)|^2$$

where

N_f = number of frequencies
ω_n = measurement frequencies that you supply.

HFSS will pre-process the measured data to ensure that it obeys the Hilbert transform (Kramers-Kronig) causality conditions. It then determines a set of stable poles that provide a good model to the frequency variation of the measured data. Finally the residues are determined through least-squares fitting.

The resulting model ensures a causal response for the dielectric when used in time domain simulation. This is because the impulse response of the multi-pole Debye model can be expressed analytically in the time domain as follows:

$$E(t) = k_0\delta(t) + \sum_{i=1}^{N} k_i \exp(p_i t)\, u(t)$$

where

$\delta(t)$ =Dirac Delta function
$u(t)$ = the Heaviside step-unit function

The above impulse response is zero for time t < 0 and so, it is casual.

Recall the approximating multi-pole Debye model function:

$\tilde{\epsilon}(j\omega)$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After computing this approximating multi-pole Debye model, HFSS splits the model into its real and imaginary parts. The real part corresponds to the approximation for the relative permittivity and the imaginary part divided by the real part gives the approximation for the (negative of the) loss tangent. The resulting (very complicated) expressions are then filled in for the new material model created.

Djordjevic-Sarkar Model Parameter Calculation

The model parameters can be calculated if the following information about the material is known:

- DC conductivity, σ_{DC} (optional; default value is 0)
- DC permittivity, ε_{DC} (optional; you have to choose the default. See below)
- A triplet of real numbers, consisting of
- the measurement frequency ω_1
- the real permittivity ε_1 at this frequency
- the loss tangent tan δ_1 at this frequency

From the model equation, observe that

$$\begin{aligned}\varepsilon(\omega_1) &= \varepsilon_\infty + \frac{\Delta\varepsilon}{\ln(\omega_B/\omega_A)}\ln\left(\frac{\omega_B + j\omega_1}{\omega_A + j\omega_1}\right) + \frac{\sigma_{DC}}{j\omega_1\varepsilon_0} \\ &\cong \varepsilon_\infty + \frac{\Delta\varepsilon}{\ln(\omega_B/\omega_A)}\ln\left(\frac{\omega_B + j\omega_1}{j\omega_1}\right) + \frac{\sigma_{DC}}{j\omega_1\varepsilon_0} \\ &= \varepsilon_\infty + \frac{\Delta\varepsilon}{\ln(\omega_B/\omega_A)}\ln\left(\frac{\sqrt{\omega_B^2 + \omega_1^2}}{\omega_1}\right) - j\left[\frac{\Delta\varepsilon}{\ln(\omega_B/\omega_A)}\cdot\tan^{-1}\left(\frac{\omega_B}{\omega_1}\right) + \frac{\sigma_{DC}}{\omega_1\varepsilon_0}\right]\end{aligned} \tag{3}$$

We know the imaginary part of $\varepsilon(\omega_1) = -\varepsilon_1 . \tan\delta_1$. Therefore we get,

$$\varepsilon_1 \tan\delta_1 = \frac{\Delta\varepsilon}{\ln(\omega_B/\omega_A)}\cdot\tan^{-1}\left(\frac{\omega_B}{\omega_1}\right) + \frac{\sigma_{DC}}{\omega_1\varepsilon_0} \tag{4}$$

and so

$$K = \frac{\Delta\varepsilon}{\ln(\omega_B/\omega_A)} = \frac{\varepsilon_1 \tan\delta_1 - \dfrac{\sigma_{DC}}{\omega_1\varepsilon_0}}{\tan^{-1}\left(\dfrac{\omega_B}{\omega_1}\right)} \tag{5}$$

The following equation for the real part also holds:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\varepsilon_1 = \varepsilon_\infty + K \ln\left(\frac{\sqrt{\omega_B^2 + \omega_1^2}}{\omega_1}\right) \quad (6)$$

Inserting the above expression allows us to solve for the high-frequency permittivity:

$$\varepsilon_\infty = \varepsilon_1 - K \ln\left(\frac{\sqrt{\omega_B^2 + \omega_1^2}}{\omega_1}\right) \quad (7)$$

If you know the DC value of real permittivity, you can find $\Delta\varepsilon = \varepsilon_{DC} - \varepsilon_\infty$ and use the above expression to calculate the lower transition frequency.

$$\omega_A = \frac{\omega_B}{\exp\left(\frac{\Delta\varepsilon}{K}\right)} \quad (8)$$

Check the computed value of lower transition frequency to make sure that it is much less than the measurement frequency. If not, raise the DC permittivity; this will reduce the value of the lower transition frequency. If you do not know the DC permittivity value, you may choose one yourself to satisfy the condition:

$$\omega_A << \omega_1$$

A simple heuristic is to choose $\Delta\varepsilon = 10 \cdot \tan\delta_1 \cdot \varepsilon_\infty$.

For more information, see Djordjevic-Sarkar Causal Dielectric Model.

Expressions for Permittivity and Conductivity

The final expressions are as follows:

Relative Permittivity:

$$\varepsilon(f) = \varepsilon_\infty + \frac{K}{2} \ln\left(\frac{f_B^2 + f^2}{f_A^2 + f^2}\right) \quad (9)$$

Conductivity:

$$\sigma(f) = \sigma_{DC} + 2\pi f \varepsilon_0 K \cdot \left[\tan^{-1}\left(\frac{f}{f_A}\right) - \tan^{-1}\left(\frac{f}{f_B}\right)\right] \quad (10)$$

Here $f_A = \omega_A/2\pi$ and $f_B = \omega_B/2\pi$ is the frequency in Hz.

- In the above expressions:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$f_B = 10^{12} / 2\pi \tag{11}$$

$$K = \frac{\varepsilon_1 \tan\delta_1 - \frac{\sigma_{DC}}{\omega_1 \varepsilon_0}}{\tan^{-1}\left(\frac{\omega_B}{\omega_1}\right)} \tag{12}$$

$$\varepsilon_\infty = \varepsilon_1 - \frac{1}{2} K \ln\left[\left(\frac{\omega_B}{\omega_1}\right)^2 + 1\right] \tag{13}$$

$\Delta\varepsilon = \varepsilon_\infty - \varepsilon_{DC}$ if the DC permittivity is known; else $\Delta\varepsilon = 10 \cdot \tan\delta_1 \cdot \varepsilon_\infty$

$$f_A = \frac{f_B}{\exp\left(\frac{\Delta\varepsilon}{K}\right)} \tag{14}$$

Conductivity and Permittivity Limits

The slope parameter K must be > 0. Therefore we must have

$$\varepsilon_1 \tan\delta_1 - \frac{\sigma_{DC}}{\omega_1 \varepsilon_0} > 0 \tag{15}$$

This can occur if you enter a DC conductivity higher than the effective conductivity at the measurement frequency. This is impossible in the Djordjevic model, because its conductivity increases monotonically with frequency. Another way for it to happen is for the measurement frequency to be too low (Hz instead of GHz, for example) or for the permittivity value to be unreasonably low.

For the model to be passive, we must have $\varepsilon_\infty \geq 1$

Therefore from (5) and (7) the following is true.

$$\varepsilon_\infty = \varepsilon_1 - \frac{\varepsilon_1 \tan\delta_1 - \frac{\sigma_{DC}}{\omega_1 \varepsilon_0}}{2\tan^{-1}\left(\frac{\omega_B}{\omega_1}\right)} \ln\left[\left(\frac{\omega_B}{\omega_1}\right)^2 + 1\right] \geq 1 \tag{16}$$

Define the quantity

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\tan \delta_{\max} = \frac{2 \tan^{-1}\left(\frac{\omega_B}{\omega_1}\right)}{\ln\left[\left(\frac{\omega_B}{\omega_1}\right)^2 + 1\right]} \tag{17}$$

Using this in 16 simplifies the expression as follows.

$$\varepsilon_1 \cdot \tan \delta_{\max} - \left(\varepsilon_1 \tan \delta_1 - \frac{\sigma_{DC}}{\omega_1 \varepsilon_0}\right) \geq \tan \delta_{\max} \tag{18}$$

Then, the condition for passivity can be written as follows:

$$\varepsilon_1 \cdot \left(\tan \delta_{\max} - \tan \delta_1\right) \geq \tan \delta_{\max} - \frac{\sigma_{DC}}{\omega_1 \varepsilon_0} \tag{19}$$

For this to hold, with $\varepsilon > 0$, the following two conditions are necessary

$$\tan \delta_{\max} - \tan \delta_1 > 0 \quad \text{or} \quad \tan \delta_1 < \tan \delta_{\max} \tag{20}$$

$$\varepsilon_1 \geq \varepsilon_{1,\min} = \frac{\tan \delta_{\max} - \frac{\sigma_{DC}}{\omega_1 \varepsilon_0}}{\tan \delta_{\max} - \tan \delta_1} \tag{21}$$

We see from the expression for tan δ_{max} given in (17) that the maximum loss tangent is a function of only the upper corner frequency and the measurement frequency.

Reducing the upper corner frequency will generally increase the value of tan δ_{max} at least as long as the following condition is realized.

ω_B is much greater than ω_1

But reducing the upper corner frequency is not very attractive, because it has not been observed in experimental data. Therefore it is an upper bound on the loss tangents that the Djordjevic-Sarkar model can handle. Equation (21) indicates that if the permittivity is too low, then the optical permittivity will go below 1. If this condition is violated, two possible fixes are to raise the value of ε or to reduce the measurement frequency. But by the same argument as before, reducing the upper corner frequency is not attractive and it is the minimum value of the permittivity. If you reduce the value of the measured loss tangent; this makes the denominator in (21) larger and lowers the value of $\varepsilon_{1,min}$.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Force per Unit Area in HFSS

This is mainly for the purpose of mapping surface force density in HFSS to Work Bench Mechanical. We will also provide it for post processing within HFSS. This feature should work regardless of HFSS solution types as long as surface forces exist. Surface forces may exist when one side is conductor but the other is not, or finite conductivity and layered impedance boundary.

The task is to calculate the force per unit area ($\mathbf{f}_s$) distribution on the surface or on a section of the surface of a solid body in HFSS. We presume the solution is known, so both the electric and the magnetic field is available in any point.

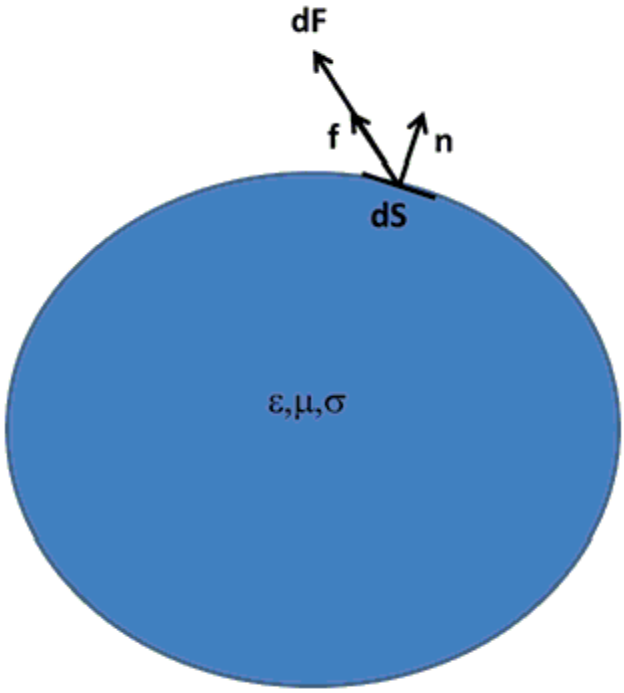


Fig. 1: Force and force per unit area on a solid body

Fig. 1 shows a solid body of homogeneous or inhomogeneous material characteristics (ε,μ, σ). **dS** is an elementary section of its surface and **n** is the normal vector of the surface. If **dF** [N] is the force acting on the elementary surface **dS**, then:

$$\mathbf{f}_s = \frac{\mathbf{dF}}{dS} \quad (1)$$

where $\mathbf{f}_s$ is the force per unit area. Knowing the electromagnetic field, **dF** can be calculated as:

$$\mathbf{dF} = \mathbf{T}^{em}\mathbf{n}\,dS \quad (2)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

then

$$\mathbf{f}_s = \mathbf{T}^{em}\mathbf{n} \quad (3)$$

where $\mathbf{T}^{em}$ is the electromagnetic Maxwell's stress tensor, which consists of the electric and the magnetic stress tensor as:

$$\mathbf{T}^{em} = \mathbf{T}^{e} + \mathbf{T}^{m} \quad (4)$$

(5)

$$\mathbf{T}^{e} = \begin{bmatrix} D_1E_1 - 0.5DE & D_1E_2 & D_1E_3 \\ D_1E_2 & D_2E_2 - 0.5DE & D_2E_3 \\ D_1E_3 & D_2E_3 & D_3E_3 - 0.5DE \end{bmatrix}$$

(6)

$$\mathbf{T}^{e} = \begin{bmatrix} B_1H_1 - 0.5BH & B_1H_2 & B_1H_3 \\ B_1H_2 & B_2H_2 - 0.5BH & B_2H_3 \\ B_1H_3 & B_2H_3 & B_3H_3 - 0.5BH \end{bmatrix}$$

$$DE = D_1E_1 + D_2E_2 + D_3E_3 \quad (7)$$

$$BH = B_1H_1 + B_2H_2 + B_3H_3 \quad (8)$$

Indices 1,2 and 3 mean the first, second and third component. For example, in Cartesian coordinates, the meaning is x, y and z.

Using the notations:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(9)

$$\mathbf{T}^{em} = \begin{bmatrix} T_{11}^{em} & T_{12}^{em} & T_{13}^{em} \\ T_{21}^{em} & T_{22}^{em} & T_{23}^{em} \\ T_{31}^{em} & T_{32}^{em} & T_{33}^{em} \end{bmatrix}; \qquad n = \begin{bmatrix} n_1 \\ n_2 \\ n_3 \end{bmatrix}$$

The force per unit area from Eq. (3) is:

(10)

$$\mathbf{f}_S = \mathbf{T}^{em}\mathbf{n} = \begin{bmatrix} T_{11}^{em} n_1 & + T_{12}^{em} n_2 & + T_{13}^{em} n_3 \\ T_{21}^{em} n_1 & + T_{22}^{em} n_2 & + T_{23}^{em} n_3 \\ T_{31}^{em} n_1 & + T_{32}^{em} n_2 & + T_{33}^{em} n_3 \end{bmatrix}$$

Note two things:

Stress tensor is defined on the surface of **S**

Force density might have both normal and tangential components on the surface

The filed quantities in formulas (1) - (10) are time dependent or static. We want to have the forces in steady state time harmonic case, too, by using the complex notations as the field quantities are defined in HFSS. It is derived for the magnetic stress tensor. The derivation for the electric stress tensor can be got by replacing B_k and H_k by D_k and E_k, respectively, where index k refers to the k-th component. The real time function of the k-th component of a complex field quantity can be written as:

$$B_k = Re(B_{ck} e^{j\omega t}) = \frac{1}{2}(B_{ck} e^{j\omega t} + B_{ck}^* e^{-j\omega t}) \quad (11)$$

where subscript c denotes complex quantity and superscript '*' means complex conjugate.

The stress tensor just contains products of $B_k H_k$ where k =1,2,3 and l =1,2,3. Applying Eq. (10), we get:

(12)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$B_k H_k = \frac{1}{4}(B_{ck} e^{j\omega t} + B_{ck}^* e^{-j\omega t})(H_{cl} e^{j2\omega t} + H_{cl}^* e^{-j2\omega t}) =$$

$$\frac{1}{4}(B_{ck} H_{cl}^* + B_{ck}^* H_{cl} + B_{ck}^* H_{cl}^* e^{-j2\omega t})$$

Eq. (12) has a constant component and a sinusoidal component of 2ω. If we are interested in the time average force, Eq. (12) should be integrated for one period and divided by the period time:

$$\langle B_k H_k \rangle = \frac{1}{T}\int_0^T \frac{1}{4}(B_{ck} H_{cl}^* + B_{ck}^* H_{cl} + B_{ck} H_{cl} e^{j2\omega t}) dt = \frac{1}{2}\mathrm{Re}(B_{ck} H_{cl}^*) \quad (13)$$

Performing the calculations for *k*=1,2,3 and *l*=1,2,3 the elements of the magnetic and electric stress tensor can be calculated as:

$$\langle B_k H_k \rangle = \frac{1}{2}\mathrm{Re}(B_{ck} H_{cl}^*); \qquad \langle D_k E_k \rangle = \frac{1}{2}\mathrm{Re}(D_{ck} E_{cl}^*) \quad (14)$$

The time average surface force density is:

$$\mathbf{f}_S = \mathbf{T}^{em} = \begin{bmatrix} \langle T_{11}{}^{em} \rangle n_1 + \langle T_{12}{}^{em} \rangle n_2 + \langle T_{13}{}^{em} \rangle n_3 \\ \langle T_{21}{}^{em} \rangle n_1 + \langle T_{22}{}^{em} \rangle n_2 + \langle T_{23}{}^{em} \rangle n_3 \\ \langle T_{31}{}^{em} \rangle n_1 + \langle T_{32}{}^{em} \rangle n_2 + \langle T_{33}{}^{em} \rangle n_3 \end{bmatrix} \quad (15)$$

The volume force acting on a closed surface is:

$$\langle \mathbf{F} \rangle = \oint_S \langle \mathbf{f}_s \rangle dS \quad (16)$$

If a surface is selected to calculate the force, smoothing should not go through this surface.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The calculations just need post processing operations.

The relativistic electromagnetic momentum (radiation pressure), which is associated with the volume sources, has been neglected in the formulas. If it is necessary, it can be taken into account

Energy Normalization of Eigen Solutions in HFSS

The energy of a system is the sum of the electric and magnetic energy:

$$W = W_e + W_m \tag{1}$$

$$W_e = \frac{1}{2}\int_V \mathbf{E} \bullet \mathbf{D} dV \quad \text{and} \quad W_{em} = \frac{1}{2}\int_V \mathbf{H} \bullet \mathbf{B} dV \tag{2}$$

The filed quantities in formulas (1)-(2) are time dependent. We want to have the energy in steady state time harmonic case, by using the complex notations as the field quantities are defined in HFSS. The real time function of a complex field quantity can be written as:

$$\mathbf{D} = \mathrm{Re}(\mathbf{D}_c e^{j\omega t}) = \frac{1}{2}(\mathbf{D}_c e^{j\omega t} + \mathbf{D}_c^* e^{-j\omega t}) \tag{3}$$

where subscript c denotes complex quantity and superscript '*' means complex conjugate.

The energy contains products of **E** and **D**. Applying Eq. (3), we get:

$$\mathbf{D} \bullet \mathbf{E} = \frac{1}{4}(\mathbf{D}_c e^{j\omega t} + \mathbf{D}_c^* e^{-j\omega t}) \bullet (\mathbf{E}_c e^{j\omega t} + \mathbf{E}_c^* e^{-j\omega t}) = \frac{1}{4}(\mathbf{D}_c \bullet \mathbf{E}_c^* + \mathbf{D}_c^* \bullet \mathbf{E}_c + \mathbf{D}_c \bullet \mathbf{E}_c e^{j2\omega t} + \mathbf{D}_c^* \bullet \mathbf{E}_c^* e^{-j2\omega t}) \tag{4}$$

Eq. (4) has a constant component and a sinusoidal component of 2ω. If we are interested in the time average energy, Eq. (4) should be integrated for one period and divided by the period time:

$$\langle \mathbf{D} \bullet \mathbf{E} \rangle = \frac{1}{T}\int_0^T \frac{1}{4}(\mathbf{D}_c \bullet \mathbf{E}_c^* + \mathbf{D}_c^* \bullet \mathbf{E}_c + \mathbf{D}_c \bullet \mathbf{E}_c e^{j2\omega t} + \mathbf{D}_c^* \bullet \mathbf{E}_c^* e^{-j2\omega t}) dt = \tag{5}$$

$$\frac{1}{2}\mathrm{Re}(D_c \bullet E_c^*)$$

Performing algebra, the elements of the electric and magnetic energy can be written as:

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$W_m = \frac{1}{2}\mathrm{Re}(B_c H_c^*) \qquad W_e = \frac{1}{2}\mathrm{Re}(D_c E^*_c) \quad (6)$$

Generally, the normalized field can be written as the solution field times a constant:

$$\mathbf{E}_{norm} = c\mathbf{E}_{solve} \quad (7)$$

For normalization to $\mathbf{E}_{max}$=1

$$c = \frac{1}{\mathbf{E}_{solve_max}} \qquad \mathbf{E}_{norm} = \frac{\mathbf{E}_{solve}}{\mathbf{E}_{solve_max}} \quad (8)$$

For normalization to $\mathbf{W}_{norm}$ =1

$$c = \sqrt{\frac{1}{W_{solve}}} \qquad \mathbf{E}_{norm} = \sqrt{\frac{1}{W_{solve}}}\mathbf{E}_{solve} \quad (9)$$

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29 - Glossary

This section defines the terminology used in the Ansys desktop help topics. Terms are listed in alphabetical order.

A **B** **C**
D **E** **F**
G **H** **I**
J **K** **L**
M **N** **O**
P **Q** **R**
S **T** **U**
V **W** **X**
Y **Z**

Glossary: A

A/D

See Analog-to-Digital.

ACPR

Adjacent channel power ratio.

Active Substrate

A hybrid or multichip module substrate formed from a semiconductor. Termed active because components such as transistors can be fabricated directly into the substrate.

Active Trimming

The process of trimming components such as resistors while the circuit is under power. Such components are fabricated directly onto the substrate of a hybrid or multichip module, and the trimming is usually performed using a laser beam.

Active-High

A signal whose active state is considered to be a logic 1.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Active-Low

A signal whose active state is considered to be a logic 0.

Actuator

A transducer that converts an electronic signal into a physical equivalent. For example, a loudspeaker is an actuator which converts electronic signals into corresponding sounds.

Adaptive Hardware

Refers to devices which allow new design variations to be "compiles" in real-time, which may be thought of as dynamically creating subroutines in hardware (see also Virtual Hardware and Cache Logic).

Additive Process

A process in which conducting material is added to specific areas of a substrate. Groups of tracks, individual tracks, or portions of tracks can be built up to precise thicknesses by iterating the process multiple times with selective masking.

Address Bus

A unidirectional set of signals used by a computer to point to memory locations in which it is interested.

Analog

A continuous value that most closely resembles the real world and can be as precise as the measuring technique allows.

Analog Circuit

A collection of components used to generate or process analog signals.

Analog-to-Digital (A/D)

The process of converting an analog value into its digital equivalent.

Anisotropic Adhesive

Special adhesives which contain minute particles of conductive material. These adhesives find particular application with the flipped-chip techniques used to mount bare die on the substrates of hybrids, multichip modules, or circuit boards. The conducting particles are only brought in contact with each other at the sites where the raised pads on the die are pressed down over their corresponding pads on the substrate, thereby forming good electrical connections between the pads.

Anti-Fuse Technology

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A programmable logic device technology in which conducting paths (anti-fuses) are grown by applying signals of relatively high voltage and current to the device's inputs.

Anti-Pad

The area of copper etched away around a via or a plated through-hole on a power or negative signal plane, thereby preventing an electrical connection being made to that plane.

Application-Specific Integrated Circuit (ASIC)

A device whose function is determined for a particular application or group of applications.

Application-Specific Standard Part (ASSP)

Refers to an integrated circuit created by a device manufacturer using ASIC technologies, and for these components to be sold as standard parts to anybody who wants to buy them.

ASIC

See Application-Specific Integrated Circuit.

ASIC Cell

A logic function in the cell library defined by the manufacturer of an application-specific integrated circuit.

Assertion-Level Logic

Special symbols which are used to more precisely indicate the function of gates with active-low inputs.

ASSP

See Application-Specific Standard Part.

Asynchronous

A signal whose data is acknowledged or acted upon immediately, irrespective of any clock signal.

Atto

Unit qualifier (symbol = a) representing one millionth of one millionth of one millionth, or 10-18. For example, 3aS stands for 3 x 10-18 seconds.

Attenuator

A passive device used to reduce signal strength while maintaining proper input and output impedance.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Glossary: B

Backplane

The medium used to interconnect a number of circuit boards. Typically refers to a special, heavy-duty printed or discrete wired circuit board.

Ball Grid Array (BGA)

A packaging technology similar to a pad grid array, in which a device's external connections are arranged as an array of conducting pads on the base of the package. However, in the case of a ball grid array, small balls of solder are attached to the conducting pads.

Bandpass

The frequency limits between half-power points of a signal or filter.

Bare Die

An unpackaged integrated circuit.

Base

Refers to the number of digits in a numbering system. For example, the decimal numbering system is said to be base-10. May also be referred to as the "radix".

Basic Cell

A pre-defined group of unconnected components that is replicated across the surface of a gate array.

BER

Bit error rate.

BGA

See Ball Grid Array.

BiCMOS

A technology in which the function of each logic gate is implemented using low-power CMOS, while the output stage is implemented using high-drive bipolar transistors

Binary Encoding

A form of state assignment for state machines that requires the minimum number of state variables.

BiNMOS

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A relatively new low-voltage integrated circuit technology in which complex combinations of bipolar and NMOS transistors are used to form sophisticated output stages providing both high speed and low static power dissipation.

Bi-quinary

A system which utilizes two bases, base-2 and base-5, to represent decimal numbers. Each decimal digit is represented by the sum of two parts, one of which has the value of decimal zero or five, and the other the values of zero through four. The abacus is one practical example of the use of a bi-quinary system.

Bit

Abbreviation of binary digit.

Bipolar Junction Transistor (BJT)

A family of transistors.

Blackbox

Same as N-Port.

Blind Via

A via that is only visible from one side of the substrate.

Bobble

A small circle used on the inputs to a logic gate symbol to indicate an active-low input or control, or on the outputs to indicate a negation (inversion) or a complementary signal. Some engineers prefer to use the term bubble.

Bounce Pad

A special pattern etched onto the power or negative signal plane of a microwire circuit board to be used in conjunction with a laser beam which is employed to create blind vias. The laser beam evaporates the epoxy forming the outer layers of the board and continues down to the bounce pad which reflects, or bounces, it back up, thereby terminating the via.

Braze

To unite or fuse two pieces of metal by heating, or with a hard solder with a high melting point.

Bulk Storage

Refers to some form of media, typically magnetic, such as tape or a disk drive which can be used to store large quantities of information relatively inexpensively.

Buried Via

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A via used to link conducting layers internal to a substrate. Such a via is not visible from either side of the substrate.

Bundle

A set of signals related in some way that makes it appropriate to group them together for ease of representation or manipulation. May contain both scalar and vector elements; for example, {a,b,c,d[5:0]}.

Bus

A set of signals performing a common function and carrying similar data. Typically represented using vector notation; for example, address[7:0].

Byte

A group of eight binary digits, or bits.

Glossary: C

Capacitance

A measure of the ability of two adjacent conductors separated by an insulator to hold a charge when a voltage differential is applied between them. Capacitance is measured in units of Farads.

CDMA

Code division multiple access.

Cell Library

The collective name for the set of logic functions defined by the manufacturer of an application-specific integrated circuit. The designer decides which types of cells should be realized and connected together to make the device perform its desired function.

Ceramic

An inorganic, nonmetallic material, such as alumina, beryllia, steatite, or forsterite, which is fired at a high temperature and is often used in electronics as a substrate or to create component packages.

CGA

See Column Grid Array.

Channel

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(1)The area between two arrays of basic cells in a channeled gate array.

(2)The gap between the source and drain regions in a MOS transistor.

Channeled Gate Array

Application-specific integrated circuit organized as arrays of basic cells. The areas between the arrays are known as channels.

Channel-Less Gate Array

Application-specific integrated circuit organized as a single large array of basic cells. May also be referred to as a "sea of cells" or a "sea of gates" device.

Checksum

The final cyclic-redundancy-check value stored in a linear feedback shift register (or software equivalent). Also known as a "signature" in the guided-probe variant of functional test.

Chip

Popular name for an integrated circuit.

Chip-On-Board (COB)

A process in which unpackaged integrated circuits are physically and electrically attached to a circuit board, and are then encapsulated with a "glob" of protective material such as epoxy.

Chip-On-Chip (COC)

A process in which unpackaged integrated circuits are mounted on top of each other. Each die is very thin and it is possible to have over a hundred dies forming a 3D cube.

Chip-On-Flex (COF)

Similar to chip-on-board (COC), except that the unpackaged integrated circuits are attached to a flexible printed circuit.

Circuit Board

The generic name for a wide variety of interconnection techniques, which include rigid, flexible, and rigid-flex boards in single-sided, double-sided, multilayer, and discrete wired configurations.

CMOS

Logic gates constructed using both NMOS and PMOS transistors connected in a complementary manner.

Coaxial Cable

A conductor in the form of a central wire surrounded first by a dielectric (insulating) layer, and then by a conducting tube which serves to shield the central wire from external interference.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coefficient of Thermal Expansion

Defines the amount a material expands and contracts due to changes in temperature. If materials with different coefficients of thermal expansion are bonded together, changes in temperature will cause shear forces at the interface between them.

Cofired Ceramic

A substrate formed from multiple layers of "green" ceramic that are bonded together and fired at the same time.

Column Grid Array (CGA)

A packaging technology similar to a pad grid array, in which a device's external connections are arranged as an array of conducting pads on the base of the package. However, in the case of a column grid array, small columns of solder are attached to the conducting pads.

Combinatorial

A digital function whose output value is directly related to the current combination of values on its inputs. Also known as combinational.

Compiled Cell Technology

A technique used to create portions of a standard cell application-specific integrated circuit. The masks used to create components and interconnections are directly generated from Boolean representations using a silicon compiler. May also be used to create data-path functions and memory functions.

Complementary Output

Refers to a function with two outputs carrying complementary logical values. One output is referred to as the true output and the other as the complementary output.

Complex Programmable Logic Device (CPLD)

A device that contains a number of PLA or PAL functions sharing a common programmable interconnection matrix.

Component

Components are items placed on schematics and layouts to represent electrical elements and sub circuits. The component information defines a signal-processing function or source, or secondarily a data source, data channel, or similar entity. Components have pins for connections, bitmaps in the project tree, and properties for simulation. The information that constitutes a component includes its component chooser bitmap, schematic symbol, layout footprint, pin properties, parameters and parameter values, and netlist string definition. A component can be associated with more than one simulation if it can be analyzed in more than one simulator.

Computer-Generated Hologram (CGH)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refers to a slice of quartz or similar material into which three-dimensional patterns are cut using a laser. The angles of the patterns cut into the quartz are precisely calculated for use in the optical communication strategy known as holographic interconnect. All of these calculations are performed by a computer, and the laser used to cut the three-dimensional patterns into the quartz is also controlled by a computer. Thus, the slice of quartz is referred to a computer-generated hologram.

Conductive Ink Technology

A technique in which tracks are screen printed directly onto the surface of a circuit board using a conductive ink.

Configurable Hardware

A product whose function may be customized once or a very few times (see also Reconfigurable Hardware, Remotely Reconfigurable Hardware, Dynamically Reconfigurable Hardware, and Virtual Hardware).

Conjunction

Propositions combined with an AND operator; for example, "You have a parrot on your head AND you have a fish in your ear." The result of a conjunction is true if all the propositions comprising that conjunction are true.

Conversion Loss

The ratio in dB of the IF output of a mixer to the RF input power. All conversion loss measurements and specification are normally based on the mixer being terminated on all ports and a stated LO signal power level being applied.

Cost Function

In an optimization setup, a cost function is based on goal values specified for at least one solution quantity. Optimetrics changes the design parameter values to fulfill the cost function. In HFSS, the cost function can be based on any solution quantity that HFSS can compute, such as field values, S-parameters, and eigenmode data.

CPLD

See Complex Programmable Logic Device.

CRC

See Cyclic Redundancy Check.

CSIC

See Customer-Specific Integrated Circuit.

Customer-Specific Integrated Circuit (CSIC)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

An alternative and possibly more accurate name for an ASIC, but this term is rarely used in the industry and shows little indication of finding favor with the masses.

Chemical Vapor Deposition (CVD)

A process for growing thin films on a substrate, in which a gas containing the required molecules is converted into a plasma by heating it to extremely high temperatures using microwaves. The plasma carries atoms to the surface of the substrate where they are attracted to the crystalline structure of the substrate. This underlying structure acts as a template. The new atoms continue to develop the structure to build up a layer on the substrate's surface.

Chemical Vapor Infiltration (CVI)

A process similar to chemical vapor deposition (CVD) but, in this case, the process commences by placing a crystalline powder of the required substance in a mold. Additionally, thin posts, or columns, can be pre-formed in the mold, and the powder can be deposited around them. When exposed to the same plasma as used in the CVD technique, the powder coalesces into a polycrystalline mass. After the CVI process has been performed, the posts can be dissolved leaving holes through the crystal for use in creating vias. CVI processes can produce layers twice the thickness of those obtained using CVD techniques at a fraction of the cost.

CW

Continuous wave; refers to an unmodulated sine-wave signal.

Cyclic Redundancy Check (CRC)

A calculation used to detect errors in data communications, typically performed using a linear feedback shift register. Similar calculations may be used for a variety of other purposes such as data compression.

Glossary: D

D/A

See Digital-to-Analog.

Data Bus

A bi-directional set of signals used by a computer to convey information from a memory location to the central processing unit and vice versa. More generally, a set of signals used to convey data between digital functions.

Data-Path Function

A well-defined function such as an adder, counter, or multiplier used to process digital data.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dBm

Power level relative to 1mV rms.

DC Balance

Stream of data encoded to ensure an equal balance of 1 or 0 . 8b10b encoding has been developed to ensure DC balancing.

DC Coupling

A method of coupling two different circuits together, allowing them to share both the static DC and varying AC characteristics of a signal.

Decoder (digital)

A logic function that uses a binary value, or address, to select between a number of outputs and to assert the selected output by placing it in its active state.

Deep Sub-Micron

Typically taken to refer to integrated circuits containing structures which are smaller than 0.5 microns.

DeMorgan Transformation

The transformation of a Boolean expression into an alternate, and often more convenient, form.

Design

Designs are the building blocks of projects, and can be Circuit designs or 3D planar EM models. Designs consist of schematics or geometrical models, model data, solution setup information, output graphs and tables, and other pieces of information that go into describing simulation of electrical circuits. A design is the largest single simulatable entity in a project.

Design Variation

A single combination of variable values that is solved during a parametric or optimization setup.

Device

A discrete, separate electrical entity such as a diode, a capacitor or a packaged transistor.

Die

(1)An unpackaged integrated circuit. In this case, the plural of die is also die (in much the same way that "a shoal of herring" is the plural of "herring"). (2)A piece of metal with a design engraved or embossed on it for stamping onto another material, upon which the design appears in relief.

Die Separation

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The process of separating individual die from the wafer by marking the wafer with a diamond scribe and fracturing it along the scribed lines.

Die Stacking

A technique used in specialist applications in which several bare die are stacked on top of each other to form a sandwich. The die are connected together and then packaged as a single entity.

Dielectric Layer

(1) An insulating layer used to separate two signal layers.

(2) An insulating layer used to modify the electrical characteristics of an MCM-D substrate.

Diffusion Layer

The surface layer of a piece of semiconductor into which impurities are diffused to form P-type and N-type material. In addition to forming components, the diffusion layer may also be used to create embedded traces.

Digital

A value represented as being in one of a finite number of discrete states called quanta. The accuracy of a digital value is dependent on the number of quanta used to represent it.

Digital Circuit

A collection of logic gates used to process or generate digital signals.

Digital Signal Processor (DSP)

A primarily digital component used to process either digital or analog signals. In the case of the latter, the signal may first be conditioned, then converted into a digital equivalent using an analog-to-digital (A/D) converter function. The signal conditioning and A/D functions may either be external to the DSP or resident in the device. A typical DSP application might be the compression/decompression of video data.

Digital-to-Analog (D/A)

The process of converting a digital value into its analog equivalent.

Diode

A two-terminal device that only conducts electricity in one direction; in the other direction it behaves like an open switch. The term diode is typically taken to refer to a semiconductor device, although alternative implementations such as vacuum tubes are available.

Diode-Transistor Logic (DTL)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Logic gates implemented using particular configurations of diodes and bipolar junction transistors. For the majority of today's designers, diode-transistor logic is of historical interest only.

Discrete Device

Typically taken to refer to an electronic component such as a resistor, capacitor, diode, or transistor that is presented in an individual package. More rarely, the term may be used in connection with a simple integrated circuit containing a small number of primitive gates.

Discrete Wire Board (DWB)

A form of circuit board in which a special computer-controlled wiring machine ultrasonically bonds extremely fine insulated wires into the surface layer of the board. This discipline has enjoyed only limited recognition, but may be poised to emerge as the technology-of-choice for high-speed designers.

Discrete Wire Technology

The technology used to fabricate discrete wire boards.

Doping

The process of inserting selected impurities into a semiconductor to create P-type or N-type material.

Double-Sided

A printed circuit board with tracks on both sides

DPSK

Differential phase-shift keying.

DQPSK

Differential quadrature phase-shift keying.

DRAM

See Dynamic RAM.

DSP

See Digital Signal Processor.

DTL

See Diode-Transistor Logic.

DUT

Device under test.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dynamic Flex

A type of flexible printed circuit which is used in applications that are required to undergo constant flexing such as ribbon cables in printers.

Dynamic RAM (DRAM)

A memory device in which each cell is formed from a transistor-capacitor pair. Called dynamic because the capacitor loses its charge over time, and each cell must be periodically recharged if it is to retain its data.

Glossary: E

E2PROM

See electrically Erasable Programmable Read-Only Memory.

EBE

See Electron Beam Epitaxy.

ECL

See Emitter-Coupled Logic.

Edge port

A place in a layout or footprint geometry through which excitation signals enter and leave the structure.

Edge-Sensitive

An input that only affects a function when it transitions from one logic value to another.

EEPROM

See electrically Erasable Programmable Read-Only Memory.

electrically Erasable Programmable Read-Only Memory (EEPROM or E2PROM)

A memory device whose contents can be electrically programmed by the designer. Additionally, the contents can be electrically erased allowing the device to be reprogrammed. Also known as EEPROM and E2PROM.

Electromigration

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(1)A process in which structures on an integrated circuit's substrate are eroded by the flow of electrons in much the same way as land is eroded by a river (also known as subatomic erosion). (2)The process of forming transistor-like regions in a semiconductor using an intense magnetic field.

Electron Beam Epitaxy (EBE)

A technique for creating thin films on substrates in precise patterns, in which the substrate is first coated with a layer of dopant material before being placed in a high vacuum. A guided beam of electrons is fired at the substrate causing the dopant to be driven into it, effectively allowing molecular-thin layers to be "painted" onto the substrate where required.

Electron Beam Lithography

An integrated circuit fabrication process in which fine beams of electrons are used to draw extremely high-resolution patterns directly into the resist without the use of a mask.

Electro-Static Discharge (ESD)

The process of moving around can generate static electricity. The term electro-static discharge refers to a charged person, or object, discharging static electricity. Although the current associated with such a static charge is low, the electric potential can be in the millions of volts and can severely damage electronic components. CMOS devices are particularly prone to damage from static electricity.

EM

Electromagnetic.

Emitter-Coupled Logic (ECL)

Logic gates implemented using particular configurations of bipolar junction transistors.

Enzyme

One of numerous complex proteins which are produced by living cells and catalyze biochemical reactions at body temperatures.

EPROM

See Erasable Programmable Read-Only Memory.

Equivalent Gate

A concept in which each type of logic function is assigned an equivalent gate value for the purposes of comparing functions and devices. However, the definition of an equivalent gate varies depending on who you're talking to.

Equivalent Integrated Circuit

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A concept used to compare the component density supported by diverse interconnection technologies such as circuit boards, hybrids, and multichip modules

Erasable Programmable Read-Only Memory (EPROM)

A memory device whose contents can be electrically programmed by the designer. Additionally, the contents can be erased by exposing the die to ultraviolet light through a quartz window mounted in the top of the component's package.

ESD

See Electro-Static Discharge.

Etching

The process of selectively removing any material not protected by a resist using an appropriate solvent or acid. In some cases the unwanted material is removed using an electrolytic process.

Eutectic Bond

A bond formed when two pieces of metal, or metal-coated materials, are pressed together and vibrated at ultrasonic frequencies.

Euler Angles

Euler angles are used in Ansoft software to carry out a coordinate transformation from one coordinate system to another. The Swiss mathematician and physicist Leonhard Euler first developed the classical rotation theorem to describe rotations in 3D space. The angles used are Euler angles and can be used to describe any 3D rotation. These angles, given by (ö, è, ø) represent a series of sequential rotations about two axis of the coordinate system. The first rotation (ö) represents a rotation about the Z-axis of the source coordinate system (X, Y, Z) which results in an intermediate coordinate system denoted by (X'', Y'', Z''). The second rotation (è) represents a rotation of the intermediate coordinate system about the X''-axis, again resulting in an intermediate coordinate system denoted by (X', Y', Z'). The third and final rotation (ø) represents a rotation about the Z'-axis of the intermediate coordinate system. The final rotation completes the rotation and results in the "target" coordinate system denoted (X, Y, Z).

For further information see, Eric W. Weisstein, "Euler Angles." From *MathWorld* - A Wolfram Web Resource.

http://mathworld.wolfram.com/EulerAngles.html .

Eye Diagram

Eye diagrams are commonly used to analyze signal integrity issues with communications channels. The bits are superimposed at unit intervals representing the duration of each bit.

Eye Mask

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The size of the eye opening in the center of an eye diagram indicates the amount of voltage and timing margin available to sample this signal. Thus, for a particular electrical interface, a fixed reticule or window could be placed over the eye diagram showing how the actual signal compares to minimum criteria window, know as the eye mask. If a margin rectangle with width equal to the required timing margin and height equal to the required voltage margin fits into the opening, then the signal has adequate margins. Voltage margin can often be traded off for timing margin.

Glossary: F

Fan-Out Via

In the case of surface mount devices attached to double-sided or multilayer boards, each component pad is usually connected by a short length of track to a via which forms a link to other conducting layers, and this via is known as a fan-out via. The term fan-out via is generally also taken to include any vias that fall inside the device's footprint (under the body of the device). Some designers attempt to differentiate these vias from those that fall outside the device's footprint by referring to them as fan-in vias, but this is not an industry-standard term.

Falling-Edge

A transition from a logic 1 to a logic 0. Also known as a negative edge.

Falltime

The time it takes for a waveform to transition from the high logic state to the low logic state. Falltime is usually measured from 90% of the total signal swing to 10% of the signal swing.

Fan-In and Fan-Out Vias

In the case of surface mount devices attached to double-sided or multilayer boards, each component pad is usually connected by a short length of track to a via which forms a link to other conducting layers, and this via is known as a fan-out via. The term fan-out via is generally also taken to include any vias that fall inside the device’s footprint (under the body of the device). Some designers attempt to differentiate these vias from those that fall outside the device’s footprint by referring to them as fan-in vias, but this is not an industry-standard term.

Femto

Unit qualifier (symbol = f) representing one thousandth of one millionth of one millionth, or 10-15. For example, 3fS stands for 3 x 10-15 seconds.

FET

See Field-Effect Transistor.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Field-Effect Transistor (FET)

A transistor whose control, or gate, signal creates an electro-magnetic field which turns the transistor ON or OFF.

Field-Programmable Gate Array (FPGA)

A programmable logic device which is more versatile than traditional programmable devices such as PALs and PLAs, but less versatile than an application-specific integrated circuit. Some field-programmable gate arrays use fuses such as those found in programmable logic devices, but others are based on SRAM equivalents.

Field-Programmable Interconnect Chip (FPIC)

An alternative, proprietary name for a field-programmable interconnect device (FPID).

Field-Programmable Interconnect Device (FPID)

A device which is used to connect logic devices together, and which can be dynamically reconfigured in the same way as standard SRAM-based FPGAs. Because each FPID may have around 1,000 pins, only a few such devices are typically required on a circuit board.

Filter

Filters are used to block out undesired frequencies. There are two types of filters: band pass and rejection. A band pass filter permits only the desired range to pass through, while the rejection filter attenuates an undesired range of frequencies.

Firmware

Refers to programs, or sequences of instructions, that are hard-coded into non-volatile memory devices.

First-In First Out (FIFO)

A memory device in which data is read out in the same order that it was written in.

Flash (e.g., Gold Flash)

An extremely thin layer of gold with a thickness measured on the molecular level which is either electroplated or chemically plated onto a surface.

FLASH Memory

An evolutionary technology that combines the best features of the EPROM and E2PROM technologies. The name FLASH is derived from the technology's fast reprogramming time compared to EPROM.

Flex

See Flexible Printed Circuit.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flexible Printed Circuit (FPC or Flex)

A specialist circuit board technology, often abbreviated to "flex", in which tracks are printed onto flexible materials. There are a number of flavors of flex, including static flex, dynamic flex, and rigid flex.

Flipped Chip

A generic name for processes in which unpackaged integrated circuits are mounted directly onto a substrate with their component-sides facing the substrate.

Flipped TAB

A combination of flipped chip and tape automated bonding.

Footprint

The area occupied by a device mounted on a substrate.

Fourier Analysis

A mathematical procedure used to determine the collection of sine waves (differing in frequency and amplitude) that is necessary to make up the square-wave pattern under consideration.

FPC

See Flexible Printed Circuit.

FPGA

See Field-Programmable Gate Array.

FPIC

See Field-Programmable Interconnect Chip.

FPID

See Field-Programmable Interconnect Device.

FR4

The most commonly used insulating base material for circuit boards. FR4 is made from woven glass fibers which are bonded together with an epoxy. The board is cured using a combination of temperature and pressure which causes the glass fibers to melt and bond together, thereby giving the board strength and rigidity. The first two characters stand for "Flame Retardant". FR4 is technically a form of fiberglass, and some people do refer to these composites as fiberglass boards or fiberglass substrates, but not often.

Free-Space Optical Interconnect

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A form of optical interconnect in which laser-diode transmitters communicate directly with photo-transistor receivers without employing optical fibers or optical waveguides.

Full Custom

An application-specific integrated circuit in which the designer has complete control over every mask layer used to fabricate the device. The manufacturer does not provide a cell library or pre-fabricate any components on the substrate.

Functional Latency

Refers to the fact that, at any given time, only a portion of the logic functions in a device or system are typically active (doing anything useful).

Functional Test

A test strategy in which signals are applied to a circuit's inputs, and the resulting signals which are observed on the circuit's outputs are compared to known good values.

Fuse

See Fusible-Link Technology.

Fusible-Link Technology (Fuse)

A programmable logic device technology which employs links called fuses. Individual fuses can be removed by applying pulses of relatively high voltage and current to the device's inputs.

Fuzz-Button

A small ball of fibrous gold used in one technique for attaching components such as multichip modules to circuit boards. Fuzz-buttons are inserted between the pads on the base of the package and their corresponding pads on the board. When the package is forced against the board, the fuzz-buttons compress to form good electrical connections. Even when the pressure is removed, the fuzz-buttons act in a similar manner to Velcro and continue to hold the component in place. One of the main advantages of the fuzz-button approach is that it allows broken devices to be quickly removed and replaced. Even though fuzz-button technology would appear to be inherently unreliable, it is used in such devices as missiles, so one can only assume that it is fairly robust.

Glossary: G

Gain

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The ratio of the power output to the power input of the amplifier in dB. The gain is specified in the linear operating range of the amplifier where a 1 dB increase in input power gives rise to a 1 dB increase in output power. Gain = 20*log(S21)

Gate Array

An application-specific integrated circuit in which the manufacturer pre-fabricates devices containing arrays of unconnected components organized in groups called basic cells. The designer specifies the function of the device in terms of cells from the cell library and the connections between them, and the manufacturer then generates the masks used to create the metallization layers.

Glue Logic

Simple logic gates used to interface more complex functions together.

Gold Flash

An extremely thin layer of gold with a thickness measured on the molecular level which is either electroplated or chemically plated onto a surface.

Goal

In an optimization setup, a goal is the value of a solution quantity that you want to be achieved during the optimization. A goal is represented as one row in the cost function table. Each cost function defined in an optimization setup must include at least one goal.

Gray Code

A sequence of binary values in which each pair of adjacent values differs by only a single bit; for example, 00, 01, 11, 10.

Green Ceramic

Unfired, malleable ceramic.

Ground Bounce

Momentary noise on the device negative signal plane causing a 0 signal to erroneously be seen as a 1. Ground bounce is caused by simultaneously switching outputs (SSO).

Guard Condition

A Boolean expression associated with a state transition in a state diagram or state table. The expression must be satisfied for that state transition to be executed.

Guided Probe

A form of functional test in which the operator is guided in the probing of a circuit to isolate a faulty component or track.

Guided-Wave

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A form of optical interconnect, in which optical waveguides are fabricated directly on the substrate of a multichip module. These waveguides can be created using variations on standard opto-lithographic thin-film processes.

Glossary: H

Hard Macro (Macro Cell)

A logic function defined by the manufacturer of an application-specific integrated circuit. The function is described in terms of the simple functions provided in the cell library and the connections between them. The manufacturer also defines how the cells forming the macro will be assigned to basic cells and the routing of tracks between the basic cells.

Hardware

Generally understood to refer to any of the physical portions constituting an electronic system, including components, circuit boards, power supplies, cabinets, and monitors.

Harmonic

Integer multiples of the fundamental frequency of interest commonly produced by a non-linear amplifier.

Harmonic Balance

A frequency domain analysis technique for simulating nonlinear circuits and systems. This method assumes the input stimulus consists of a relatively few steady state sinusoids. Therefore the solution can be expressed as a sum of steady state sinusoids that includes the input frequencies in addition to any significant harmonics or mixing terms. A circuit with a single input source will require a single tone HB simulation. The harmonic balance simulation is ideal for situations where transient simulation methods are problematic, such as:

- Components modeled in frequency domain, for instance (dispersive) transmission lines
- Circuit time constants large compared to period of simulation frequency
- Circuits with lots of reactive components

Harmonic balance methods, therefore, are the best choice for most microwave circuits excited with sinusoidal signals (e.g., mixers, power amplifiers).

Harmonic Tuning

Impedance-matching at the harmonic frequencies for enhanced performance and efficiency.

Hertz (Hz)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Unit of frequency. One Hertz equals one cycle, or one oscillation, per second.

Heterojunction

The interface between two regions of dissimilar semiconductor materials. The interface of a hetrojunction has naturally occurring electric fields which can be used to accelerate electrons, and transistors created using hetrojunctions can switch much faster than their counterparts of the same size.

Hexadecimal

Base-16 numbering system. Each hexadecimal digit can be directly mapped onto four binary digits, or bits.

High Impedance State

The state on a signal that is not being driven by any value. A high-impedance state is indicated by the character Z.

Holographic Interconnect

A form of optical interconnect based on a thin slice of quartz, into which three-dimensional images are cut using a laser beam. Thus, the quartz is referred to as a computer-generated hologram, and this interconnection strategy is referred to as holographic.

Homojunction

An interface between two regions of semiconductor having the same basic composition but opposing types of doping. Homojunctions dominate current processes because they are easier to fabricate than heterojunctions.

Hybrid

An electronic sub-system in which a number of integrated circuits (packaged and/or unpackaged) and discrete components are attached directly to a common substrate. Connections between the components are formed on the surface of the substrate, and some components such as resistors and inductors may be fabricated directly onto the substrate.

Hydrogen Bond

The electrons in a water molecule are not distributed equally, because the oxygen atom is a bigger, more robust fellow which grabs more than its fair share. The end result is that the oxygen atom has an overall negative charge, while the two hydrogen atoms are left feeling somewhat on the positive side. This unequal distribution of charge means that the hydrogen atoms are attracted to anything with a negative bias; for example, the oxygen atom of another water molecule. The resulting bond is known as a hydrogen bond.

Hz

See Hertz.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Glossary: I

IBIS

IBIS (I/O Buffer Information Specification) is a standard for electronic behavioral specifications of integrated circuit input/output analog characteristics. The core of an IBIS model is a table of current versus voltage and I/O switching timing information. Xilinx IBIS models contain tables for typical, slow/MIN (weak transistors, high temperature, low voltage) and fast/MAX (strong transistors, low temperature, high voltage) process corners. IBIS models are derived from SPICE simulation results and/or lab measurements. The benefit for IBIS model user is fast and accurate simulation while preserving IC vendors intellectual property (information about circuit and process details).

IC

See Integrated Circuit.

ICR

See In-Circuit Reconfigurable.

Impedance

The resistance to the flow of current caused by resistive, capacitive, or inductive devices (or undesired elements) in a circuit.

Impedance Matching

Function of ensuring that the impedance of the transmitter, the receiver, and the transmission line are identical. Mismatched impedances could result in signal reflections, ringing, overshoot, undershoot, and stairstep waveforms.

Incident Voltage

The user specified voltage at an input.

In-Circuit Reconfigurable (ICR)

An SRAM-based, or similar component which can be dynamically reprogrammed on-the-fly while remaining resident in the system.

Inductance

A property of a conductor that allows is to store energy in a magnetic field which is induced by a current flowing through it. Inductance is measured in units of Henries (the base unit is a Henry).

Insertion Loss

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Insertion Loss (dB) is defined as the drop in power as a signal enters an RF component. This value not only includes the reflected incoming signal, but also the attenuation of the component.

In-System Programmable (ISP)

An E2-based, FLASH-based, or similar component which can be reprogrammed while remaining resident on the circuit board.

Integrated Circuit (IC)

A device in which components such as resistors, capacitors, diodes, and transistors are formed on the surface of a single piece of semiconductor.

Ion

A particle formed when an electron is added to, or subtracted from, a neutral atom or group of atoms.

Ion Implantation

A process in which beams of ions are directed at a semiconductor to alter its type and conductivity in certain regions.

Isolation

The ratio (expressed in dB) of the power level at one port compared to the resulting power level of the output port.

ISP

See In-System Programmable.

Glossary: J

JEDEC

See Joint Electronic Device Engineering Council.

Jitter

The jitter of a periodic signal is the delay between the expected transition of the signal and the actual transition. Jitter is a zero mean random variable. When worst case analysis is undertaken the maximum value of this random variable is used.

Jitter Tolerance

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Jitter tolerance is defined as the peak-to-peak amplitude of sinusoidal jitter applied on the input that causes a predefined, acceptable loss at the output. For example jitter applied to the input of an OC-N equipment interface that causes an equivalent 1dB optical power penalty.

Jitter Transfer

Jitter transfer is defined as the ratio of jitter on the output of a device to the jitter applied on the input of the device, versus frequency. Jitter transfer is important in applications where the system is utilized in a loop-timed mode, where the recovered clock is used as the source of the transmit clock.

Joint Electronic Device Engineering Council (JEDEC)

A council which creates, approves, arbitrates, and oversees industry standards for electronic devices. In programmable logic, the term JEDEC refers to a textual file containing information used to program a device. The file format is a JEDEC approved standard and is commonly referred to as a JEDEC file.

Jumper

A small piece of wire used to link two tracks on a circuit board.

Glossary: K

Karnaugh Map

A graphical technique for representing a logical function. Karnaugh maps are often useful for the purposes of minimization.

Kelvin Scale of Temperature

A scale of temperature which was invented by the British mathematician and physicist William Thomas, first Baron of Kelvin. Under the Kelvin, or absolute, scale of temperature, 0 K (corresponding to -273oC) is the coldest possible temperature and is known as absolute zero.

Kilo

Unit qualifier (symbol = K) representing one thousand, or 103. For example, 3KHz stands for 3 x 103 Hertz.

Kirchhoff's current law

The sum of all currents entering a node is equal to the sum of all currents leaving the node.

Kirchhoff's voltage law

The directed sum of the electrical potential differences around a circuit must be zero.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Glossary: L

Laminate

A material constructed from thin layers or sheets. Often used in the substrate of circuit boards.

Large-Scale Integration (LSI)

Refers to the number of logic gates in a device. By one convention, large-scale integration represents a device containing 100 to 999 gates.

Laser Diode

A special semiconductor diode which emits a beam of coherent light.

Last-In First-Out (LIFO)

A memory device in which data is read out in the reverse order to which it was written in.

Latch-Up Condition

A condition in which a circuit draws uncontrolled amounts of current, and certain voltages are forced, or "latched-up", to some level. Particularly relevant in the case of CMOS devices which can latch-up if their operating conditions are violated.

Lateral Thermal Conductivity

Good lateral thermal conductivity means that the heat generated by components mounted on a substrate can be conducted horizontally across the substrate and out through its leads.

Layers (and Stackup)

Layers are used in the layout editor to organize and isolate sets of geometry or other visual indicators. Signal, Negative Signal, and Dielectric are common physical layers, while Symbol (to show component symbols in layout), Error, and Ratsnest (to show connectivity) are non-physical layers. The stackup contains additional properties of the physical layers, such as material, thickness, and elevation. Geometrical information on these layers is used to generate masks for manufacturing.

LDMOS

Laterally diffused metal oxide semiconductor.

Lead

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(1) A metallic element (chemical symbol Pb).

(2) A metal conductor used to provide a connection from the inside of a device package to the outside world for soldering or other mounting techniques. Leads are also commonly called pins.

Lead Frame

A metallic frame containing leads and a base to which an unpackaged integrated circuit is attached. After encapsulation, the outer part of the frame is cut away and the leads are bent into the required shapes.

Lead Through-Hole (LTH)

A technique for populating circuit boards in which component leads are inserted into plated through-holes. Often abbreviated to "through-hole" or "thru-hole". When all of the components have been inserted, they are soldered to the board, usually using a wave soldering technique.

Level-Sensitive

An input whose effect on a function depends only on its current logic value or level, and is not directly related to it transitioning from one logic value to another.

LFSR

See Linear Feedback Shift Register.

Library

A library is a collection of one or more components or component dependencies (materials, symbols, footprints, or padstacks) stored in a container file. A library must be configured to a circuit before use, either by the user (manually) or by loading technology files (automatically). User libraries and Personal libraries are used to add foundry support, user defined models, and any custom set of components or simulation models. See the Library Overview topic for more information.

LIFO

See Last-In First-Out.

Limiting Level

The input power level when the output power is goes into compression and no longer becomes linear.

Line

Used to refer to the width of a track; for example, "This track has a line-width of 0.12mm."

Linear Feedback Shift Register (LFSR)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A shift register whose data input is generated as an XOR or XNOR of two or more elements in the register chain.

Linearity

Describes how closely an output signal is to a perfectly scaled multiple of a corresponding input signal.

Load Pull

Automated measurement of RF performance of a device under test (at a constant frequency) by varying the source and load impedance presented to the device

Logic Function

A mathematical function that performs a digital operation on digital data and returns a digital value.

Logic Gate

The physical implementation of a logic function.

Logic Synthesis

A process in which a program is used to optimize the logic used to implement a design.

Low-Fired Cofired

Similar in principle to standard cofired ceramic substrate techniques. However, low-fired cofired uses modern ceramic materials with compositions that allow them to be fired at temperatures as low as 650oC to 750oC. Firing at these temperatures in an inert atmosphere such as nitrogen allows non-refractory metals such as copper to be used to create tracks.

LSI

See Large-Scale Integration.

LTH

See Lead Through-Hole.

Glossary: M

Mask Programmable

A device such as a read-only memory which is programmed during its construction using a unique set of masks.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximal Displacement

A linear feedback shift register whose taps are selected such that changing a single bit in the input data stream will cause the maximum possible disruption to the register's contents.

Maximal Length

A linear feedback shift register that sequences through (2n - 1) states before returning to its original value.

Maxterm

The logical OR of the inverted variables associated with an input combination to a logical function.

MBE

See Molecular Beam Epitaxy.

MCM

See Multichip Module.

Medium-Scale Integration (MSI)

Refers to the number of logic gates in a device. By one convention, medium-scale integration represents a device containing 13 to 99 gates.

Meg

Unit qualifier (symbol = M) representing one million, or 106. For example, 3MHz stands for 3 x 106 Hertz.

Metallization Layer

A layer of conducting material on an integrated circuit that is selectively deposited or etched to form connections between logic gates. There may be several metallization layers separated by dielectric (insulating) layers.

Metal-Oxide Semiconductor (MOS)

A family of transistors where the controlling terminal is connected to a plate that is separated from the semiconduxtor by an insulating layer. This plate was originally made out metal (we now use polysilicon, or poly) and the insulator is an oxide -- hence the "metal-oxide" appellation.

Meta-Stable

A condition where the outputs of a logic function are oscillating uncontrollably between undefined values.

Micro

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Unit qualifier (symbol = u) representing one millionth, or 10-6. For example, 3uS stands for 3 x 10-6 Seconds.

Microwave

The range in the electromagnetic spectrum from 300 MHz to 30 GHz (with corresponding wavelengths from 100 cm to 1 cm).

Microwire

A trade name for one incarnation of discrete wire technology. Microwire augments the main attributes of multiwire with laser-drilled blind vias, allowing these boards to support the maximum number of tracks and components.

Millman's method

The voltage on the ends of branches in parallel is equal to the sum of the currents flowing in every branch divided by the total equivalent conductance.

Minterm

The logical AND of the variables associated with an input combination to a logical function.

MMIC

Monolithic Microwave Integrated Circuit

Mod or Modulus

Refers to the number of states that a function such as a counter will pass through before returning to its original value. For example, a function that counts from 00002 to 11112 has a modulus of 16 and would be called a modulo-16 or mod-16 counter.

Molecular Beam Epitaxy (MBE)

A technique for creating thin films on substrates in precise patterns, in which the substrate is placed in a high vacuum, and a guided beam of ionized molecules is fired at it, effectively allowing molecular-thin layers to be "painted" onto the substrate where required.

MOS

See Metal-Oxide Semiconductor.

MOSFET

Metal-oxide semiconductor field-effect transistor.

MSI

See Medium-Scale Integration.

Multichip Module (MCM)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A generic name for a group of advanced interconnection and packaging technologies featuring unpackaged integrated circuits mounted directly onto a common substrate.

Multilayer

A printed circuit board constructed from a number of very thin single-sided and/or double-sided boards which are bonded together using a combination of temperature and pressure.

Multiplexer (digital)

A logic function that uses a binary value, or address, to select between a number of inputs and conveys the data from the selected input to the output.

Multiwire

A trade name for one incarnation of discrete wire technology.

Multizone

A stackup that contains zones or areas, each of which contains a subset of the layers in the stackup.

Mutual Capacitance

The capacitance between two conductors (one considered aggressor, the other victim) when all other conductors are connected together and then regarded as an ignored ground. It describes the amount of coupling due to the electric field. The mutual capacitance will inject an often undesired current into the victim line proportional to the rate of change of voltage on the aggressor line. Mutual Capacitance it a cause of crosstalk.

Mutual Inductance

The inductance between two conductors (one considered aggressor, the other victim) placed close enough that the magnetic field induced by a current flowing into the aggressor line encompasses the victim. The mutual inductance will inject an often undesired voltage noise onto the victim proportional to the rate of change of the current on the aggressor line.

Glossary: N

Nano

Unit qualifier (symbol = n) representing one thousandth of one millionth, or 10^{-9}. For example, 3nS stands for 3 x 10^{-9} Seconds.

Nanobot

A molecular-sized robot (see Nanotechnology below)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Nanophase Materials

A form of matter which was only recently discovered, in which small clusters of atoms form the building blocks of a larger structure. These structures differ from those of naturally occurring crystals, in which individual atoms arrange themselves into a lattice.

Nanotechnology

Nanotechnology is an elusive term that is used by different research-and-development teams to refer to whatever it is that they're working on at the time. However, irrespective of their particular area of interest, nanotechnology always refers to something extremely small. One of the more exciting branches of nanotechnology that has been suggested as having potential in the future is that of micro-miniature electronic products that assemble themselves.

N-channel MOS (NMOS)

Refers to the order in which the semiconductor is doped in a MOS device. That is, which structures are constructed as N-type versus P-type material.

Negative-Edge

A transition from a logic 1 to a logic 0. Also known as a falling edge.

Negative Ion

An atom or group of atoms with an extra electron.

Negative Logic

A convention which dictates the relationship between logical values and the physical voltages used to represent them. The more negative potential is considered to represent TRUE and the more positive potential is considered to represent FALSE. Also known as negative true logic.

Negative Resist

A process where ultraviolet radiation passing through the transparent areas of a mask causes the resist to be cured. The uncured areas are then removed using an appropriate solvent.

Negative Signal Plane

A conducting layer in, or on, a substrate providing a grounding, or reference, point for components. There may be several negative signal planes separated by insulating layers.

Negative-True

A convention which dictates the relationship between logical values and the physical voltages used to represent them. The more negative potential is considered to represent TRUE and the more positive potential is considered to represent FALSE. Also known as negative logic.

Nibble

See Nybble.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NMOS

See N-channel MOS.

Noise Figure / Noise Factor

The Noise Factor of a transducer at a specified input frequency is the ratio of (a/b) where “a and b” are:

(a) the available Signal to Noise Ratio (SNR) at the signal generator terminals per unit bandwidth when the temperature of the input termination (generator or source) is 290 K and the bandwidth is limited by the transducer, to

(b) the available SNR per unit bandwidth at the output terminals of the transducer.

Traditionally:

Noise Figure NF = 10 log(noise factor F)

Noise Temperature (T_e) = T_o(F - 1)

Where:

- T_e is the noise temperature
- T_o is standard temperature 290 K
- F is noise factor

Noise Floor

This is defined as the lowest possible input to a chain or a component, that will produce a detectable output.

Noise Temperature

This is the amount of thermal noise in a chain or a component. Noise Factor and Noise Temperature (T_e) are related as follows:

Noise Temperature (T_e) = (F - 1)T_o

Where:

- T_e is the noise temperature
- T_o is standard temperature 290 K
- F is noise factor

For example, a noise figure of 2.0 dB is equivalent to a Noise Temperature of 170 K.

Nominal Design

The original model on which Optimetrics analyses are based.

Non-Volatile

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A memory device which does not lose its data when power is removed from the system.

Non-Volatile RAM

A device which is generally formed from an SRAM die mounted in a package with a very small battery, or as a mixture of SRAM and EEPROM cells fabricated on the same die.

Norton's theorem

Any two-terminal collection of voltage sources and resistors is electrically equivalent to an ideal current source in parallel with a single resistor.

NPN (N-type - P-type - N-type)

Refers to the order in which the semiconductor is doped in a bipolar junction transistor.

N-Port

An N-port component is typically characterized by network parameter data contained in the N-port itself in spreadsheet form or (more usually) by network parameter data contained in an external file.

N-type

A piece of semiconductor doped with impurities that make it amenable to donating electrons.

Nybble

A group of four binary digits, or bits (also called a nibble).

Glossary: O

Octal

Base-8 numbering system. Each octal digit can be directly mapped onto three binary digits, or bits.

Ohm

Unit of resistance. The Greek letter omega, *, is often used to represent ohms; for example, 1M* indicates one million ohms.

Ohm's law

The voltage across a resistor is the product of its resistance and the current flowing through it.

One-Hot Encoding

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A form of state assignment for state machines in which each state is represented by an individual state variable.

One-Time Programmable

A device such as a PAL, PLA, or PROM that can only be programmed a single time and whose contents cannot be subsequently erased.

Optical Interconnect

The generic name for interconnection strategies based on opto-electronic systems, including fiber-optics, free-space, guided-wave, and holographic techniques.

Optical Lithography

A process in which radiation at optical wavelengths (usually in the ultraviolet range) is passed through a mask, and the resulting patterns are projected onto a layer of resist coating the substrate material.

Optical Mask

A sheet of material carrying patterns that are either transparent or opaque to the wavelengths used in an optical-lithographic process. Such a mask can carry hundreds of thousands of fine lines and geometric shapes.

Opto-Electronic

Refers to a system which combines optical and electronic components.

Organic Resist

A material which is used to coat a substrate and is then selectively cured to form an impervious layer. These materials are called organic because they are based on carbon compounds as are living creatures.

Organic Solvent

A solvent for organic materials such as those used to form organic resists.

Organic Substrate

Substrate materials such as FR4, in which woven glass fibers are bonded together with an epoxy. These materials are called organic because epoxies are based on carbon compounds as are living creatures.

Overglassing

One of the final stages in the integrated circuit fabrication process in which the entire surface of the wafer is coated with a layer of silicon dioxide or silicon nitride. This layer may also be referred to as the barrier layer or the passivation layer. An additional lithographic step is required to pattern holes in this layer to allow connections to be made to the pads.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Glossary: P

Package

Leaded assembly (inside of which one or more dies are mounted and connected) for use in larger circuits.

Pad

An area of metallization on a substrate used for probing or to connect to a via, plated through-hole, or an external interconnect.

Pad Grid Array (PGA)

A packaging technology in which a device's external connections are arranged as an array of conducting pads on the base of the package.

Padstack

Refers to any pads, anti-pads, and thermal relief pads associated with a via or a plated through-hole as it passes through the layers forming the substrate.

Padcap

A special flavor of circuit board used for high-reliability military applications. Distinguished by the fact that the outer surfaces of the board have pads but no tracks. Signal layers are only created on the inner planes, and tracks are connected to the surface pads by vias.

Parallel-In Serial-Out (PISO)

Refers to a shift register in which the data is loaded in parallel and read out serially.

Parasitic Effects

The effects caused by undesired resistance, capacitance, or inductance inherent in the material or topology of a track or component.

Passive Trimming

A process in which a laser beam is used to trim components such as thick-film and thin-film resistors on an otherwise unpopulated and unpowered hybrid or multichip module substrate. Probes are placed at each end of a component to monitor its value while the laser evaporates some of the material forming the component.

Pass-Transistor Logic

A technique for connecting MOS transistors such that data signals pass between their source and drain terminals. Pass-transistor logic minimizes the number of transistors required to

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

implement a function, and is typically employed by designers of cell libraries or full-custom integrated circuits.

PGA

See either Pad Grid Array or Pin Grid Array.

Photo-Transistor

A special transistor which converts an optical input in the form of light into an equivalent electronic signal in the form of a voltage or current.

Pico

Unit qualifier (symbol = p) representing one millionth of one millionth, or 10^{-12}. For example, 3pS stands for 3 x 10^{-12} Seconds.

PIN Diode

A diode where a thin layer exists between the N and P regions. Rectification with pin diodes is limited. They actually behave more like a variable resistor that changes based upon the DC bias.

Pin Grid Array (PGA)

A packaging technology in which a device's external connections are arranged as an array of conducting leads, or pins, on the base of the package.

PISO

See Parallel-In Serial-Out.

Place-Value

Refers to a numbering system in which the value of a particular digit depends both on the digit itself and its position in the number.

PlasmA gaseous state in which the atoms or molecules are dissociated to form ions.

Plated Through-Hole (PTH)

(1) A hole in a double-sided or multilayer board that is used to accommodate a through-hole component lead and is plated with copper.

(2) An alternative name for the lead through-hole technique for populating circuit boards in which component leads are inserted into plated through-holes.

PMOS (P-channel MOS)

Refers to the order in which the semiconductor is doped in a MOS device. That is, which structures are constructed as P-type versus N-type material.

PNP (P-type - N-type - P-type)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refers to the order in which the semiconductor is doped in a bipolar junction transistor.

Polysilicon Layer

An internal layer in an integrated circuit used to create the gate electrodes of MOS transistors. In addition to forming gate electrodes, the polysilicon layer can also be used to interconnect components. There may be several polysilicon layers separated by dielectric (insulating) layers.

Populating

The act of attaching components to a substrate.

Positive Logic

A convention which dictates the relationship between logical values and the physical voltages used to represent them. The more positive potential is considered to represent TRUE and the more negative potential is considered to represent FALSE. Also known as positive true logic.

Positive Resist

A process where radiation passing through the transparent areas of a mask causes previously cured resist to be degraded. The degraded areas are then removed using an appropriate solvent.

Positive-Edge

A transition from a logic 0 to a logic 1. Also known as a rising edge.

Positive-True

A convention which dictates the relationship between logical values and the physical voltages used to represent them. The more positive potential is considered to represent TRUE and the more negative potential is considered to represent FALSE. Also known as positive logic.

Power Amplifier

A class of amplifier with the primary purpose of delivering high output power (usually accompanied by significant dissipated power).

Power Plane

A conducting layer in or on the substrate providing power to the components. There may be several power planes separated by insulating layers.

Prepreg

Non-conducting semi-cured layers of FR4 used to separate conducting layers in a multilayer circuit board.

Primitives

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Simple logic functions such as BUF, NOT, AND, NAND, OR, NOR, XOR, and XNOR may be referred to as primitive logic gates or primitives.

Product Term

A set of literals linked by an AND operator.

Programmable Array Logic (PAL)

A programmable logic device in which the AND array is programmable but the OR array is pre-defined.

Programmable Logic Array (PLA)

The most user-configurable of the traditional programmable logic devices, because both the AND and OR arrays are programmable.

Programmable Logic Device (PLD)

The generic name for a device constructed in such a way that the designer can configure, or "program" it to perform a specific function.

Programmable Read-Only Memory (PROM)

A programmable logic device in which the OR array is programmable but the AND array is pre-defined. Usually considered to be a memory device whose contents can be electrically programmed (once) by the designer.

Project

A container that groups designs and their associated settings, including report definitions, in a file with a .adsn extension. To the fullest extent possible, projects are portable in that they include, rather than merely refer to, the library elements (graphical symbols, materials, footprints, and so on) of the components and models they contain. Multiple projects can be open simultaneously.

PROM

See Programmable Read-Only Memory.

Pseudo-Random

An artificial sequence of values that give the appearance of being random.

PTH

See Plated Through-Hole.

P-type

A piece of semiconductor doped with impurities that make it amenable to accepting electrons.

Pulling

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The difference between the maximum frequency of a VCO when the phase angle of the load impedance reflection coefficient varies through 360 degrees.

Pulsed Radar

A transmit and receive system used for ranging and detection that transmits a train of short bursts of high microwave signals and receives return signals reflected from a target.

Pushing

The change in frequency when the supply voltage changes, expressed in MHz/V.

Glossary: Q

Q

Quality factor; a measure of stored energy/dissipated energy. Also a measure of bandwidth.

QAM

Quadrature amplitude modulation.

QFP

See Quad Flat Pack.

QPSK

Quadrature phase-shift keying.

Quad Flat Pack (QFP)

The most commonly used package in surface mount technology to achieve a high lead count in a small area. Leads are presented on all four sides of a thin square package.

Quinary

Base-5 numbering system.

Glossary: R

Radio Frequency (RF)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The range in the electromagnetic spectrum loosely defined from 30 MHz to 3 GHz (with corresponding wavelengths from 1000 cm to 10 cm).

Radix

Refers to the number of digits in a numbering system. For example, the decimal numbering system is said to be radix-10. May also be referred to as the "base".

Rats Layer

A non-physical, default layer in the stackup that displays a drawing of logical connections between different components, circuit elements, and net connections. A single connection is called a rat and all of the connections on a rat layers is a rat's nest.

Reed-Müller Logic

Logic functions implemented using only XOR and XNOR gates.

Refractory Metal

Metals such as tungsten, titanium, and molybdenum which are capable of withstanding extremely high, or refractory, temperatures.

Remotely Reconfigurable Hardware

A product whose function may be customized remotely, by telephone or radio, while remaining resident in the system (see also Configurable Hardware, Reconfigurable Hardware, Dynamically Reconfigurable Hardware, and Virtual Hardware).

Reflection

The appearance of a previously transmitted signal on the transmission line causing interference with the current signal. Reflections are caused by a poorly terminated or discontinuous transmission line, where the signal energy is not fully absorbed within the receiver and is therefore transmitted back toward the transmitter.

Resist

A material which is used to coat the substrate and is then selectively cured to form an impervious layer.

Resistor-Transistor Logic (RTL)

Logic gates implemented using particular configurations of resistors and bipolar junction transistors. For the majority of today's designers, resistor-transistor logic is of historical interest only.

Return Loss

Return Loss (dB) is defined as a ratio of the incoming signal to the same reflected signal as it enters a component.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Return Loss (dB) = 10 * LOG10(Reflected Power/Incident Power)

RF

See Radio Frequency.

RF Power

A class of engineering. Circuits and signals primarily concerned with power levels ranging from a few watts to tens of thousands of watts in the RF spectrum.

RF Power Transmitter

A discrete packaged transistor used in the amplification of RF power.

Rigid Flex

Hybrid constructions which combine standard rigid circuit boards with flexible printed circuits, thereby reducing the component count, weight, and susceptibility to vibration of the circuit, and greatly increasing its reliability.

Ringing

Common name for the waveform that is seen when a transmission line ends at a high impedance discontinuity. The signal first overshoots, then dips down below the target value, and continues this with decreasing amplitude until it converges on the target voltage.

Risetime

The time it takes for a signal to rise from 10% of its total logic swing to 90% of its total logic swing.

Rising-Edge

A transition from a logic 0 to a logic 1. Also known as a positive edge.

RTL

See Resistor-Transistor Logic.

Glossary: S

Sample Rate

Time increment of analysis. Sometimes referred to as sampling rate.

Sampling

The process of converting an analog signal into a series of digital values.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scalar Notation

A notation in which each signal is assigned a unique name; for example, a3, a2, a1, and a0.

Scaling

A technique for making transistors switch faster by reducing their size. This strategy is known as scaling, because all of the transistors features are typically reduced by the same proportion.

Schematic

Common name for a circuit diagram.

Scrubbing

The process of vibrating two pieces of metal, or metal coated materials, at ultrasonic frequencies to create a friction weld.

Seed Value

An initial value loaded into a linear feedback shift register or random number generator.

Sensor

A transducer that detects a physical quantity and converts it into a form suitable for processing. For example, a microphone is a sensor which detects sound and converts it into a corresponding voltage or current.

Sequential

A function whose output value depends not only on its current input values, but also on previous input values. That is, the output value depends on a sequence of input values.

Side-Emitting Laser Diode

A laser diode constricted at the edge of an integrated circuit's substrate such that, when power is applied, the resulting laser beam is emitted horizontally; that is, parallel to the surface of the substrate.

Sign Bit

The most significant binary digit, or bit, of a signed binary number. If set to a logic 1, this bit represents a negative quantity.

Signal Conditioning

Amplifying, filtering, or otherwise processing a signal.

Signal Layer

A layer carrying tracks in a circuit board, hybrid, or multichip module. See also wiring layer.

Signature

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refers to the checksum value from a cyclic-redundancy-check when used in the guided-probe form of functional test.

Signature Analysis

A guided-probe functional-test technique based on signatures.

Signed Binary Number

A binary number in which the most-significant bit is used to represent a negative quantity. Thus, a signed binary number can be used to represent both positive and negative values.

Sign-Magnitude

Negative numbers in standard arithmetic are typically represented in sign-magnitude form by prefixing the value with a minus sign; for example, -27. For reasons of efficiency, computers rarely employ the sign-magnitude form. Instead, they use signed binary numbers to represent negative values.

Silicon Bumping

The process of depositing additional metallization on a die's pads to raise them fractionally above the level of the Barrier Layer.

Silicon Chip

Although a variety of semiconductor materials are available, the most commonly used is silicon and integrated circuits are popularly known as silicon chips, or simply chips.

Silicon Compiler

The program used in compiled cell technology to generate the masks used to create components and interconnections. May also be used to create data-path functions and memory functions.

Single-Sided

A printed circuit board with tracks on only one side.

Sintering

A process in which ultra-fine metal powders weld together at temperatures much lower than those required for larger pieces of the same materials.

SIPO (Serial-In Parallel-Out)

Refers to a shift register in which the data is loaded in serially and read out in parallel.

SISO (Serial-In Serial-Out)

Refers to a shift register in which the data is both loaded in and read out serially.

Skew

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Time delay between different bits transmitted at the same time, measured at the receiver.

Skin Effect

In the case of high frequency signals, electrons are only conducted on the outer surface, or skin, of a conductor. This phenomenon is known as the skin effect.

Small-Scale Integration (SSI)

Refers to the number of logic gates in a device. By one convention, small-scale integration represents a device containing 1 to 12 gates.

SMD

See Surface Mount Device.

SMOBC

See Solder Mask Over Bare Copper.

SMT

See Surface Mount Technology.

SNR

Signal-to-noise ratio.

Soft Macro (Macro Function)

A logic function defined by the manufacturer of an application-specific integrated circuit. The function is described in terms of the simple functions provided in the cell library and connections between them. The assignment of cells to basic cells and the routing of the tracks is determined at the same time, and using the same tools, as for the other cells specified by the designer.

Solder Bumping

A flipped chip technique in which spheres of solder are formed on the die's pads. The die is flipped and the solder bumps are brought into contact with corresponding pads on the substrate. When all the chips have been mounted on the substrate, the solder bumps are melted using reflow soldering or vapor-phase soldering.

Solder Mask

A layer applied to the surface of the substrate that prevents solder from sticking to any metallization except where holes are patterned into the mask.

Solder Mask Over Bare Copper (SMOBC)

A technique in which the solder mask is applied in advance of the tin-lead plating. This results in lighter circuit boards because the tin-lead alloy is only used to plate the pads.

Solution

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A solution is the successful result of an analysis, or imported results available for plotting.

Space

Used to refer to the width of the gap between adjacent tracks.

SPICE

Simulated Program for Integrated Circuit Emulation

SRAM

See Static RAM.

SS-CDMA

Spread-spectrum code-division multiple access.

SSI

See Small-Scale Integration.

Stackup

An arrangement of physical signal and dielectric layers that is used in the design of circuit boards. In Circuit, the stackup editor lists conceptual non-stackup layers with the physical stackup layers.

Standard Cell

An application-specific integrated circuit which, unlike a gate array, does not use the concept of a basic cell and does not have any pre-fabricated components. The manufacturer creates custom masks for every stage of the device's fabrication allowing each logic function to be created using the minimum number of transistors.

State Assignment

The process by which the states in a state machine are assigned to the binary patterns that are to be stored in the state variables.

State Diagram

A graphical representation of the operation of a state machine.

State Machine

The actual implementation (in hardware or software) of a function that can be considered to consist of a set of states through which it sequences.

State Table

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A tabular representation of the operation of a state machine. Similar to a truth table, but also includes the current state as an input and the next state as an output.

State Transition

An arc connecting two states in a state diagram.

State Variable

One of a set of registers whose values represent the current state occupied by a state Static Flex.

Statement

A sentence that asserts or denies an attribute about an object or group of objects.

Static Flex

A type of flexible printed circuit which can be manipulated into permanent three-dimensional shapes for applications such as calculators and high-tech cameras which require efficient use of volume and not just area.

Static RAM (SRAM)

A memory device in which each cell is formed from four or six transistors configured as a latch or a flip-flop. The term static is used because, once a value has been loaded into an SRAM cell, it will remain unchanged until it is explicitly altered or until power is removed from the device.

Steady State

A condition in which nothing is changing or happening.

Subatomic Erosion

A process in which structures on an integrated circuit's substrate are eroded by the flow of electrons in much the same way as land is eroded by a river (also known as electromigration)

Substrate

Generic name for the base layer of an integrated circuit, hybrid, multichip module, or circuit board. Substrates may be formed from a wide variety of materials, including semiconductors, ceramics, FR4 (fiberglass), glass, sapphire, or diamond depending on the application. Note that the term substrate has traditionally not been widely used in the circuit board world, at least not by the people who manufacture the boards. However, there is an increasing tendency to refer to a circuit board as a substrate by the people who populate the boards. The main reason for this is that circuit boards are often used as substrates in hybrids and multichip modules, and there is a trend toward a standard terminology across all forms of interconnection technology.

Subtractive Process

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A process in which a substrate is first covered with conducting material, then any unwanted material is subsequently removed, or subtracted.

Superconductor

A material with zero resistance to the flow of electric current.

Surface Mount Device (SMD)

A component whose packaging is designed for use with surface mount technology.

Surface Mount Technology (SMT)

A technique for populating hybrids, multichip modules, and circuit boards, in which packaged components are mounted directly onto the surface of the substrate. A layer of solder paste is screen printed onto the pads and the components are attached by pushing their leads into the paste. When all of the components have been attached, the solder paste is melted using either reflow soldering or vapor-phase soldering.

Surface-Emitting Laser Diode

A laser diode constricted on an integrated circuit's substrate such that, when power is applied, the resulting laser beam is emitted directly away from the surface of the substrate.

Sweep Definition

Also called *variable sweep definition*. A set of variable values within a range that Optimetrics drives the Electronics Desktop to solve when a parametric setup is analyzed. A parametric setup can include one or more sweep definitions.

Synchronous

(1)A signal whose data is not acknowledged or acted upon until the next active edge of a clock signal. (2)A system whose operation is synchronized by a clock signal.

System Gain

The net loss of a system as a measure of reliability with respect to system parameters. It measures the difference between the output power and minimum input power required for satisfactory performance. It represents a system net loss, and is represented as a negative dB value that is larger than or equal to the summed gains and losses of a signal propagating in a system from transmitter to receiver.

Glossary: T

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tap

A register output which is used to generate the next data input to a linear feedback shift register.

Tape Automated Bonding (TAB)

A process in which transparent flexible tape has tracks created on its surface. The pads on unpackaged integrated circuits are attached to corresponding pads on the tape which is then stored in a reel. Silver-loaded epoxy is screen printed on the substrate at the site where the device is to be located and onto the pads to which the device's leads are to be connected. The reel of TAB tape is fed through an automatic machine which pushes the device and the TAB leads into the epoxy. When the silver-loaded epoxy is cured using reflow soldering or vapor-phase soldering, it forms electrical connections between the TAB leads and the pads on the substrate.

TDM

Time-division multiplexing.

TDMA

Time-division multiple access.

Technology File

A technology file is a collection of information, specifiable by name at design-creation time using the Choose Layout Technology dialog box, that specifies material stackup properties for layout, substrates for simulation, and component libraries for design creation. A technology (.asty) file may be saved from the current project via the File menu. A technology file initializes a design with a set of data to avoid repeated entry of commonly used data. This data can consist of layers and stackup information for layout, configured libraries of components, and substrate definition (s) for circuit analysis. Users and foundries can customize Technology files for their own manufacturing process and simulation models.

Tertiary Logic

An experimental technology in which logic gates are based on three distinct voltage levels. The three voltages are used to represent the tertiary digits 0, 1, and 2, and their logical equivalents FALSE, TRUE, and MAYBE.

Thermal Impedance

Relates the temperature rise for a given dissipated power (which employs an analogy to the voltage-current relationship of impedance).

Thermal Relief Pad

A special pattern etched around a via or a plated through-hole to connect it into a power or ground plane. A thermal relief pad is necessary to prevent too much heat being absorbed into the power or ground plane when the board is being soldered.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thermal Tracking

Typically used to refer to the problems associated with optical interconnection systems whose alignment may be disturbed by changes in temperature.

Thevenin's theorem

Any two-terminal combination of voltage sources and resistors is electrically equivalent to a single voltage source in series with a single resistor.

Thick-Film Process

A process used in the manufacture of hybrids and, to a lesser extent, multichip modules in which signal and dielectric (insulating) layers are screen-printed onto the substrate.

Thin-Film Process

A process used in the manufacture of hybrids and multichip modules in which signal layers and dielectric (insulating) layers are created using opto-lithographic techniques.

Time-Of-Flight

The time taken for a signal to propagate from one logic gate or opto-electronic component to another.

Tin-Lead Plating

An electroless plating process in which exposed areas of copper on a circuit board are coated with a layer of tin-lead alloy. The alloy is used to prevent the copper from oxidizing and provides protection against contamination.

Tinning

An abbreviation of tin-lead plating, which is an electroless plating process in which exposed areas of copper on a circuit board are coated with a layer of tin-lead alloy. The alloy is used to prevent the copper from oxidizing and provides protection against contamination.

Toggle

Refers to the contents or outputs of a logic function switching to the inverse of their previous logic values.

TOI

Third-order intercept point.

Total Voltage

Total voltage = Incident voltage + reflected voltage, that is $V^T = V^a + V^b$. See discussion in Excitations in the Time Domain.

Trace

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A conducting connection between electronic components. May also be called a track or a signal. In the case of integrated circuits, such interconnections are often referred to collectively as metallization.

Tracks

A conducting connection between electronic components. May also be called a trace or a signal. In the case of integrated circuits, such interconnections are often referred to collectively as metallization.

Transducer

A device that converts input energy of one form into output energy of another.

Transistor

A three-terminal semiconductor device that, in the digital world, can be considered to operate like a switch.

Tri-State Function

A function whose output can adopt three states: 0, 1, and Z (high-impedance) The function does not drive any value in the Z state and, in many respects, may be considered to be disconnected from the rest of the circuit.

Truth Table

A convenient way to represent the operation of a digital circuit as columns of input values and their corresponding output responses.

TTL (Transistor-Transistor Logic)

Logic gates implemented using particular configurations of bipolar junction transistors.

Glossary: U

ULA

See Uncommitted Logic Array.

Ultra-Large-Scale Integration (ULSI)

Refers to the number of logic gates in a device. By one convention, ultra-large-scale integration represents a device containing a million or more gates.

ULSI

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Ultra-Large-Scale Integration.

Uncommitted Logic Array (ULA)

One of the original names used to refer to gate array devices. This term has largely fallen into disuse.

Unsigned Binary Number

A binary number in which all the bits are used to represent positive quantities. Thus, an unsigned binary number can only be used to represent positive values.

Undershoot

The percentage a waveform falls below its lowest determined value before setting at the correct value.

Glossary: V

Vapor-Phase Soldering

A surface mount process in which a substrate carrying components attached by solder paste is lowered into the vapor-cloud of a tank containing boiling hydrocarbons. This melts the solder paste thereby forming good electrical connections. However, vapor-phase soldering is becoming increasingly less popular due to environmental concerns.

Vaporware

Refers to either hardware or software that exist only in the minds of the people who are trying to sell them to you.

Variable Sweep Definition

Also called *sweep definition*. A set of variable values within a range that Optimetrics drives HFSS or Q3D to solve when a parametric setup is analyzed. A parametric setup can include one or more sweep definitions.

Vector Notation

A notation used in logic simulation and synthesis in which a single name is used to reference a group of signals, and individual signals within the group are referenced by means of an index; for example, a[3:0] = a[3], a[2], a[1], and a[0].

Very-Large-Scale Integration (VLSI)

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refers to the number of logic gates in a device. By one convention, very-large-scale integration represents a device containing 1,000 to 999,999 gates.

Via

A hole filled or lined with a conducting material which is used to link two or more conducting layers in a substrate.

Virtual Hardware or Virtual Logic

An extension of dynamically configurable hardware based on a new generation of FPGAs which were introduced around the beginning of 1994. In addition to supporting the dynamic reconfiguration of selected portions of the internal logic, these devices also feature: no disruption to the device's inputs and outputs; no disruption to the system-level clocking; the continued operation of any portions of the device that are not undergoing reconfiguration; and no disruption to the contents of internal registers during reconfiguration, even in the area being reconfigured (see also Configurable Hardware, Reconfigurable Hardware, Remotely Reconfigurable Hardware, and Dynamically Reconfigurable Hardware).

Virtual Memory

A trick used by a computer's operating system to pretend that it has access to more memory than is actually available. For example, a program running on the computer may require ten mega-bytes to store its data, but the computer may have only five mega-bytes of memory available. To get around this problem, whenever the program attempts to access a memory location that does not physically exist, the operating system performs a slight-of-hand and exchanges some of the contents in the memory with data on the hard disk.

VLSI

See Very-Large-Scale Integration.

Volatile

Refers to a memory device which loses any data it contains when power is removed from the system; for example, random-access memory in the form of SRAM or DRAM

Glossary: W

Wafer Probing

The process of testing individual integrated circuits while they still form part of a wafer. An automated tester places probes on the device's pads, applies power to the power pads, injects a

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

series of signals into the input pads, and monitors the corresponding signals returned from the output pads.

Wave Soldering

A process used to solder circuit boards populated with through-hole components. A wave generating mechanism maintains a wave of hot, liquid solder traveling back and forth across the surface of a tank. The populated circuit boards are passed over the wave soldering machine on a conveyor belt. The velocity of the convener belt is carefully controlled and synchronized such that the solder wave brushes across the bottom of the board only once.

Waveguide

A transparent path bounded by non-transparent, reflective areas, which is fabricated directly onto the surface of a substrate. Used in the optical interconnection strategy known as guided-wave.

W-CDMA

Wideband code-division multiple access. Typically defined with 5 MHz channels and 3.84 MHz carrier signals.

Wire Bonding

The process of connecting the pads on an unpackaged integrated circuit to corresponding pads on a substrate using wires that are finer than a human hair. Wire bonding may also be used to connect the pads on an unpackaged integrated circuit, hybrid, or multichip module to the leads of the component package.

Wiring Layer

A layer carrying wires in a discrete wired board. See also signal layer.

Word

A group of signals or logic functions performing a common task and carrying or storing similar data; for example, a value on the data bus could be referred to as a data word.

Glossary: X

X-Ray Lithography

Similar in principle to optical lithography, but capable of constructing much finer features due to the shorter wavelengths involved. However, X-ray lithography requires an intense source of X-rays, is more difficult to use, and is considerably more expensive than optical lithography.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Glossary: Y

Yield

The number of devices that work as planned, specified as a percentage of the total number actually fabricated.

Glossary: Z

Zepto (z)

The symbol used to represent the high-impedance state in tri-state logic.

Zone

A spatial area on a printed circuit board that may contain a subset or all of the layers in the board’s stackup.

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Index

3

3D component library 6-317, 13-117, 13-160

3D modeler window 2-97

history tree 2-101

commands 2-112

controlling the view 2-104

organizing 2-103

selecting objects 2-109

upgrading modeling kernel 2-113

3D models

changing background 7-2

clip planes 7-52

curved surface 7-51

fitting 7-37

hiding objects 7-39

lighting 7-56

modifying view 7-1

orientation 7-18

panning 7-8

rendering solids 7-47

rendering wireframes 7-47

rotating 7-25

showing objects 7-41

spinning 7-25

user interface options 7-3

zooming 7-9

A

aborting an analysis 9-248, 9-282

absorbing boundary, for ports 16-83

ACA solver, selection guidelines 16-60

accepted power 28-138

ACT extensions 2-52, 2-150

adaptive analysis

and frequency sweeps 16-87

bypassing 16-2, 16-21, 16-37, 16-62

maximum refinement per pass 16-74

setting convergence on real frequency only 16-65

setting lambda refinement 16-72

setting matrix convergence 16-75

setting maximum delta Energy 16-65

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

setting maximum delta S 16-63
setting maximum number of passes 16-62
setting minimum number of converged passes 16-75
setting minimum number of passes 16-74
setting up 16-43, 16-47
Adaptive Lanczos-Padé Sweep (ALPS) 28-28
adaptive passes, completed and remaining 23-5
adaptive solution
single frequency 28-26
admittance matrix
for differential pairs 28-300
method of calculation 28-119
alignment groupings, modes for wave ports 28-297
analysis
aborting 9-248, 9-282
distributed 9-9
parametric (large scale DSO) 9-35
remote 9-1
analysis configurations
in the registry 3-80
Analysis Setup, enable/disable 16-15
analytic port types 28-296
analyze all 22-3
animations 23-370
anisotropic materials
about 28-341
and ports 28-346
conductivity tensor 28-343
electric loss tangent tensor 28-344
magnetic loss tangent tensor 28-345
permeability tensor 28-342
permittivity tensor 28-342
anisotropic tensors 5-20
anisotropy
conductivity 28-343
electric loss tangent 28-344
magnetic loss tangent 28-345
permeability 28-342
permittivity 28-342
Ansys Cloud 9-252
Ansys Cloud Burst 9-264
Ansys Discovery
integration with 6-150
link options 3-33
Ansys Electronics Desktop
batchoptions 2-129
files 3-1
getting started with 2-1

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

launching 2-5
menu bar 2-56
overview 2-7
projects 3-1
ribbon interface 2-8, 2-10, 2-151
running from a command line 2-129
running from Windows remote terminal 2-150
status bar 2-114
system requirements 2-5
user interface 2-7
window displays 2-13
Ansys HPC Platform Services Support for HPC Job Management 9-267
Ansys SpaceClaim
integration with 6-156
Ansys technical support 1-263
Ansys Workbench
feedback iterator 26-44
integration with 26-1
material data transfer 26-34
antenna arrays
defining custom 23-527
defining parametric 23-523
Antenna Parameter Overlays 23-553
antenna parameters
exporting 23-536, 23-544
Save for Overlay 23-551
antenna parameters, decay factor 28-141
antenna parameters, front to back ratio 28-135
antenna properties
accepted power 28-138
axial ratio 28-133
incident power 28-139
max far-field data 28-125
maximum radiation intensity 28-135
overview 28-131
peak directivity 28-135
peak gain 28-136
peak realized gain 28-136
polarization 28-132, 28-132
polarization ratio 28-134
radiated power 28-137
radiation efficiency 28-140
system efficiency 28-141
total efficiency 28-141
antenna properties, computing 23-530
archive
restoring 3-161

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

saving project as 3-157

Array Conformance in SBR+ 28-100

array factor

and power normalizations 28-130

calculation 28-126

scan angle 28-126

Array, Create 3D Component Array command 21-22, 21-31, 21-59

SBR+ Regions 21-54

Array, Create Array command 21-7

Array, Create Array Through CSV command 21-15

Array, Export Array to CSV command 21-19

arrays

defining custom 23-527

defining parametric 23-523

defining regular 23-523

arrays, virtual based on unit cells 21-1

assigning arrays, HFSS 21-1

assigning boundaries, overview 17-1

assigning circuit element

overview 20-1

assigning excitations

overview 19-1

attenuation constant 28-18

auto-complete

variables and properties 2-124

auto-complete, variables and properties 2-124

auto-save 3-101

average SAR, calculation of 28-51

axial ratio 28-133

B

background color 7-2

basis functions

low-order solution basis 16-77

mixed order 16-72

overview 28-3

basis functions, mixed order 16-77

batchoptions 2-129

command line examples 3-76

desktop settings 2-143

directories and library paths 2-142

examples 2-140

file format 3-76

in the registry 3-80

override registry entry 2-149

temp directory 2-143

beta options 3-6

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

bondwires
drawing 6-76
overview 28-234
boundaries
and frequency dependence 28-287
anisotropic impedance 17-52, 17-58
assigning 17-1
default assignments 28-287
deleting 17-71
duplicating with geometry 17-73
editing properties of 17-70
hiding 17-74
moving to different surfaces 17-71
reassigning 17-72
reprioritizing 17-72
setting default values 17-77
types of 17-1
Boundary, Zoom to selected 17-5
box
setting up near-field 23-491
braided shield models
technical details 28-236

C

calculated expressions, plotting 23-314
capacitors, modeling lumped 17-50
causality, enforce for interpolating sweeps 16-95
CFIE, with IE solvers 28-113
characteristic impedance
and renormalized S-parameters 28-116
specifying impedance lines 19-22
Zpi impedance 28-15
Zpv impedance 28-16
Zvi impedance 28-17
chip model analyzer (CMA) 23-645
circuit
inserting Ensemble 3-218
circuit types, modeling 17-50
circular polarization 28-133
clean stop 9-248, 9-282, 22-13
cleaning up solutions 23-35
clip plane 7-52
clip planes 7-55
clipboard
importing model from 6-358
options 3-44
closing
Property Window 2-34
color scheme
changing 3-10

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

combine sweeps 23-24
command-line syntax 2-129
command properties
modifying 2-29
commands
align faces 6-138
chamfer 6-413
copy 6-357
cut 6-361
fillet 6-411
heal 6-131
hide objects 7-2
pan 7-8
paste 6-357
perform explicit subtractions 6-140
project sheet 6-393
purge history 6-422
redo 3-167
remove edges 6-140
remove faces 6-139
rotate view 7-25
scale 6-373
separate bodies 6-410
show objects 7-2
simplify 6-125
spin view 7-25
stitch sheets 6-137
thicken sheet 6-384
undo 3-167
wrap sheet 6-385
zoom 7-9
complex
propagation constant 28-11
weight 28-126
component libraries 6-156, 6-317, 13-116, 13-160
components
cable modeling 6-290
editing definitions 6-209
editing properties 6-206
inserting 6-192
legacy projects using 6-215
updating 3-166
updating definitions 6-213
viewing properties of 6-199
window 2-37
composite excitation, solution type 3-180
conductivity 28-337
configuration files
example uses 3-66
user options 3-69
using to set options 3-57
using UpdateRegistry command 3-60

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

constants
list of 3-237
convergence
in solution process 28-3
output variable 16-67
real frequency only 16-65
convergence criteria
matrix 16-75
maximum number of passes 16-62
minimum converged passes 16-75
minimum number of passes 16-74
setting delta Energy 16-65
setting maximum delta S 16-63
convergence data
mag margin 28-20
maximum delta Energy results 23-7
maximum delta S results 23-6
number of passes completed 23-5
phase margin 28-21
plotting 23-9
viewing the mag margin 23-7
viewing the max delta (Mag S) 23-8
viewing the max delta (phase S) 23-8
viewing the maximum delta frequency 23-9
viewing the phase margin 23-7
coordinate systems
default planes 6-475
deleting 6-483
face 6-473
modifying 6-481
moving to end 6-480
object 6-476
overview 6-464
relative 6-466
working 6-465
coordinates
assigning to objects 6-449
expressing 6-482
setting coordinate systems 6-464
copying and pasting objects 6-357
cost function
setting a goal 8-119
CPU time
viewing for solution tasks 23-9
vs. real time 23-9
Create 3D Component Array Command 21-22, 21-31, 21-59
SBR+ Regions 21-54

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Array Command 21-7

Create Array Through CSV Command 21-15

Create Surface Mesh Summary 23-367

creating

open region 17-7

creeping waves

SBR+ Technical Overview 28-92

cross-sections

creating 6-392

cross-sections, plotting spherical 28-121

current flow direction 19-201

current flow lines 17-50

current sources

scaling the magnitude 23-260

setting the phase 23-260

technical note 28-322

curves

equation-based 6-24

settings 3-39

tooltip 3-44

custom arrays, geometry file setup 28-129

custom arrays, setting up 23-527

D

data types for convergence, sweep setup 16-95

dataset expressions 3-298

datasets

adding 3-262

adding 3D 3-266

cloning 3-275

editing 3-273

exporting 3-278

importing 3-271

overview 3-261

plot properties 3-280

removing 3-279

using SheetScan 3-283

DC thickness, assigning 5-8

de-embedding, S-matrices 19-32

debug logging 2-152

Debye materials, technical notes 28-346

Debye’s model 28-346

decay factor, antenna parameter 28-141

deembedding, port calibration 28-332

deembedding, technical notes 28-324

defaults

assigned boundaries 28-287

basic field quantities 28-47

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SAR settings 23-409

setting for boundary values 17-77

setting for excitation values 17-77

degenerate modes 28-14

Delete, Solution setup 16-15

deleting

field overlay plots 23-360

delta E, See maximum delta Energy 16-65

delta frequency, maximum 23-9

delta H 28-338

delta S, See maximum delta S 16-63

delta temperature units 3-300

derivatives, calculating for variables. 16-85

derived field quantities, plotting 23-314

design area 2-97

design of experiments 8-161

designs

copying and pasting 3-157

deleting 3-165

design notes 3-168, 3-169

inserting in project 3-173

nominal 8-1

read-only 3-165

setting up 3-172, 3-216

updating components 3-166

viewing details in the project tree 2-21

DesignXplorer 8-219

desktop-engine connection 3-18

desktop configuration 3-6

targeted 3-6

dielectric loss tangent 28-337

differential pairs 28-300

setting up 19-59

Differential Pairs

admittance and impedance matrices 28-301

computing 28-300

computing the S-matrix 28-300

reports for Transient Network 12-24

directive gain, antenna properties 28-131

DirectiveGain 23-521, 23-530, 28-135

directivity, peak 28-135

directories

options 3-14

discrete frequency sweep 28-29

Discrete sweeps

saving fields 16-103

saving fields at single

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

frequencies 16-107
specifying single frequencies 16-107
distributed
memory, HFSS 16-115
distributed analysis
command line options 9-249
configuration 9-10
optimal settings 9-32
overview 9-9
two-level distribution 9-32
Djordevic Sarkar , Parameters 28-335, 28-358
documentation file
inserting 3-167
domain decomposition
theory 28-66
drawing objects
polylines 6-20
Driven Modal solutions, overview 28-31
Driven Modal solutions, setting 3-174
Driven Terminal solutions, overview 28-31
Driven Terminal solutions, setting 3-174
DSO
large scale 9-35
command line syntax 9-46
configuration 9-54
monitoring 9-53
outputs 9-49
post-processing 9-51
prerequisites 9-37
process limits on Windows 9-65
troubleshooting 9-65
user interface 9-40
duplicating
excitations 17-74
integration lines 19-225

E

E-field, specifying direction of 19-22
E-fields
calculating on secondary boundary 28-271
mapping to other surfaces 28-269
edges
creating objects from 6-405
detaching 6-391
flattening 6-413
moving 6-397
removing 6-140

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

rounding 6-411

selecting 6-439

Edit Layers window

HFSS 3D Layout 3-218

effective wavelength 28-18

EFIE

Electric Field Integral Equation 28-113

Eigenmode solutions, overview 28-31

Eigenmode solutions, setting 3-174

Eigenmode solutions, viewing solution data 23-35

Eigenmode, setting Maximum delta Frequency 16-65

eigenmodes

minimum frequency to search for 16-20

specifying number of 16-21

elliptically polarized plane wave 28-308

emission test

conclusions and discussion 28-180

mathematical background 28-154

radiated field computations 28-177

setup and results 28-149

technical notes 28-147

encryption

3D components 6-171

energy error 16-65

Enforce Causality, setting in interpolating sweeps 16-95

Enforce Passivity, setting in interpolating sweeps 16-95

epsilon 28-19

equivalent circuit export options 23-27

error tolerance, for interpolating sweeps 16-99

errors

disk space warning 3-15

errors,logging 2-152

excitations

and frequency dependence 28-287

circuit elements 20-1

deleting 19-205

duplicating with geometry 17-73

editing properties of 19-204

fields 28-11

hiding 17-74

invalidated 19-205

moving to another surface 19-205

scaling the magnitude 23-260

setting default values 17-77

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

setting the phase 23-260
types of 19-1
Excitations for SBR+
SBR+ Technical Overview 28-72
Export Array to CSV Command 21-19
exporting
2D model files 3-136
3D model files 3-136
array metadata 3-140
datasets 3-278
equivalent circuit data files 23-27
field overlay plots 23-359
graphics files 3-139
materials 5-80
maximum field data 23-536, 23-544
options files 3-153
W-element data 23-30
expressions
dataset 3-298
delta temperature units 3-300
intrinsic functions 3-295
operators 3-294
overview 3-294
piecewise linear functions 3-296

F

faces
aligning 6-138
covering 6-390
creating objects from 6-404
detaching 6-391
moving 6-397
removing 6-139
selecting 6-434
uncovering 6-391
far-field pattern 28-126
far field region, definition for post processing 23-484
far field, definition 28-119
far fields
and radiation boundaries 23-493, 23-495
calculation of 28-119
computing antenna parameters 23-530
computing max data 23-530
defining antenna arrays 23-521
infinite spheres 23-508, 23-516
plotting on spherical cross-sections 28-121, 28-121
fast frequency sweeps 28-27
Fast sweeps, modifying matrix data 23-14
Fast sweeps, saving fields 16-103

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ferrites
and delta H 28-339
and lande G 28-339
and magnetic saturation 28-338
assigning bias field 19-201
assigning magnetic bias sources 19-201
field overlay plots 23-314
on lines 23-356
SAR settings 23-409
scalar plots 23-317
vector plots 23-322
field overlay plots, SAR 28-51
field overlays
plotting 23-281
field pattern direction, specifying 28-294
field plots
exporting 23-361
field quantities
defaults 28-47
phase angle 23-438
plotting derived 23-314
within tetrahedra 28-2
field solutions
linking from simulations 19-149
saving at all frequency points 16-103
fields calculator
complex vector electric field 23-438
general commands 23-435
input commands 23-423
output commands 23-450
overview 23-411
registers 23-420
scalar commands 23-439
stack commands 23-418
vector commands 23-444
Fields Calculator Cookbook 23-442
fields, saving at specific frequency points 16-107
file formats
.dsp 23-360
.m files 23-20
.sNp 23-22
.spc 23-27
data table 23-22
Ensemble ver. 6+ 23-20
HFSS ver. 6+ 23-20
Libra 23-22
Neutral Model Format 23-20
nmf 23-23
Touchstone 23-22

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

files
about 3-1
exporting 3-135
importing 3-103
finite array patterns 28-126
finite conductivity boundaries
and frequency dependence 28-287
finite element mesh
overview 28-1
finite element method, overview 28-1
fitting objects 7-37
floquet ports, technical note 28-302
free space lambda refinement 16-72
free space termination, and PMLs 28-255
Freq variable 28-287
frequency 28-12
fast sweep overview 28-27
selective surface 17-15
sweeps, error tolerance 16-99
sweeps, maximum solutions 16-100
frequency-dependent materials
data points 5-49
debye model 5-38, 5-43
defining 5-32
djordjevic-sarkar model 5-42
multipole debye model 5-39
piecewise linear 5-36
frequency-dependent materials, and lossy dielectrics 28-346
frequency points
choosing for full-wave SPICE 16-110
deleting from solution 16-109
inserting in solution 16-109
Frequency Sweep
disable/enable 16-112
range for full-wave SPICE 16-110
frequency sweeps
adding to designs 16-87
and adaptive analysis 16-87
and bypassing adaptive analysis 16-2, 16-21, 16-37, 16-62
and lumped RLC boundaries 28-272
Fast sweep overview 28-28
Interpolating sweep overview 28-30
linear step frequencies 16-103
overview 28-27
selecting type 16-90
setting points to solve 16-102
setting single frequencies 16-107
settings for 16-87
Fresnel (SBR+) boundaries 13-73, 13-78,

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13-108, 13-113, 13-113, 13-153, 17-65, 17-66

front to back ratio, antenna parameter 28-135

FSS surfaces, assigning reference surface for radiation boundaries 17-15, 17-19, 17-24

full-wave SPICE

choosing frequencies 16-110

guidelines for calculating maximum frequency 16-111

guidelines for calculating step size 16-111

functions

defining 3-301

intrinsic 3-295

list of 3-301

piecewise linear 3-296

G

gain

peak 28-136

peak realized 28-136

GainAccepted 23-521, 23-530, 28-135

GainInput 23-521, 23-530, 28-135

gamma, complex propagation constant 28-18

GDSII

importing 3-105

layer mapping files 3-108, 3-109

tech file colors 3-110

Geometry and Boundaries in SBR+

SBR+ Technical Overview 28-78

GO blockage

SBR+ Technical Overview 28-95

goal

setting weight of 8-119

goal weight, setting 8-119

gotolink create apo 23-530

GPU 9-28

installation and setup 9-283

Gradient Model for Surface Roughness 28-288

Gradient Surface Roughness Boundary 17-62

grid engine (GE) 9-113

grids

settings 3-40

Groiss surface roughness 28-261

ground planes, lossy 28-285

grouping

options 3-32

groups

selecting 6-445

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

H

help 2-31
HFIE, with IE solvers 28-113
HFSS
  assigning arrays 21-1
  distributed memory solutions 16-115
  getting started 3-171, 3-171
  introduction 3-171
  SBR+ Technical Overview 28-69
HFSS-IE
  integral equation method 28-113
HFSS 3D Layout
  inserting designs 3-218
HFSS, mesh operations 10-1
hide selection 7-39
hiding
  excitations 17-74
  windows 2-14
high performance computing
  Ansys Cloud 9-252
  Ansys Cloud Burst 9-264
  configuring Ansys Electronics for 9-77
  diagnostics 9-308
  grid engine (GE) 9-113
  HPC Platform Services Support 9-267
  integration 9-74
  load sharing facility (LSF) 9-129
  Microsoft Windows HPC 9-78
  overview 9-1
  portable batch system (PBS) 9-139
  remote simulation management (RSM) 9-147, 9-290
  schedulers 9-76
  SLURM Scheduler (Linux Utility) 9-165
  terminology 9-75
  third party schedulers 9-171
highlighting selections 3-28
history tree 2-101
  commands 2-112
  controlling the view 2-104
  options 3-31
  organizing 2-103
  selecting objects 2-109
  Upgrade Version 2-113
  upgrading modeling kernel 2-113
HPC
  setting options 3-46
HPC Analysis options, setting 16-113
HSPICE, exporting to 23-27
Huray surface roughness 28-259
hybrid regions 18-1

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I

i_<object name> boundary type 28-287

impedance

and symmetry 28-269

characteristic, calculating 28-15

multiplier 19-207

impedance boundaries

and frequency dependence 28-287

conditions applied at 28-251

overview 28-251

impedance boundary, units 28-252

impedance lines, See integration lines 19-22

impedance matrix

for differential pairs 28-300

method of calculation 28-119

impedance multiplier

and symmetry planes 28-269

impedance multipliers

perfect boundaries 28-17

imperfect conductors, modeling 17-31

Implicit solver 28-64

importing

data tables 3-133

datasets 3-271

files 3-103

incident power 28-139

incident waves

near fields calculation 28-119

scaling the magnitude 23-260

scaling the phase 23-260

inductors, modeling lumped 17-50

infinite arrays 28-126

defining custom 23-527

infinite sphere, defining 23-508, 23-516

initial mesh, source from current or other design 16-69

insert

circuits, HFSS 3D Layout 3-218

inserting circuit

HFSS 3D Layout 3-218

inserting designs 3-173

integral equation method, technical note 28-113

integration lines

defining 19-22

deleting 19-226

duplicating 19-225

modifying 19-227

reversing direction of 19-226

swapping endpoints 19-226

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

interactive scheduler jobs
design types 9-74
DSO configuration 9-70
options 9-68
overview 9-67
interpolating
sweep 28-30
sweeps, error tolerance 16-99
Interpolating sweeps, modifying matrix data 23-14
intersecting objects 6-404
introduction to HFSS 3-171
invalid excitations 19-205
Iterative matrix solver 28-23

J

jobs
aborting 9-248, 9-282
advanced submission options 9-215
exporting 9-221
importing 9-221
monitoring 9-236
multi-step submission 9-221
scripting 9-234
submitting 9-210, 22-5
submitting using command line 9-169, 9-219
Windows to Linux 9-227

K

keyboard shortcuts
3D modeler 2-116
custom 2-122
desktop 2-115
general 2-114
layout editor 2-118
netlist editor 2-119
report window 2-119
schematic editor 2-117

L

lambda 28-18
lambda refinement, setting 16-72
large scale DSO 9-35
command line syntax 9-46
configuration 9-54
monitoring 9-53
outputs 9-49
post-processing 9-51
prerequisites 9-37
troubleshooting 9-65
user interface 9-40
Windows process limits 9-65
Large Scale DSO, technical notes 28-231

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

layered impedance boundaries
and frequency dependence 28-287
external vs. internal 28-272
impedance calculation formula 28-273
surface roughness calculation 28-274
layout window 2-98
legends
settings 3-41
length-based refinement
inside objects 10-5
on faces 10-3
length of transmission line
adding to ports 19-32
subtracting from ports 19-32
libraries
personal 5-87
system 5-87
user 5-87
lighting 7-56
linear constraints 8-210
linear polarization 28-131
linear step frequencies 16-103
lines
field plots on 23-356
lines, integration 19-22
linking from simulations 19-149
Linux remote spawn 3-53, 3-55
load sharing facility (LSF) 9-129
local SAR, calculation of 28-51
log of solution tasks 23-9
logging errors 2-152
lossy ground planes 28-286
low-order solution basis 16-77
Ludwig-3 polarization 28-133
Lumped Port wizard 19-66
lumped ports
for modal solutions 19-66
for terminal solutions 19-77
guidelines for assigning 28-297
overview 28-297
lumped RLC boundaries
and frequency dependence 28-287
and frequency sweeps 28-272
conditions applied at 28-272
overview 28-272

M

M3DFS, as source of field information 19-201
MAFET Consortium 23-23
mag margin 28-20

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

magnetic bias
non-uniform 28-324
uniform 28-323
magnetic bias sources, assigning to ferrites 19-201
magnetic loss tangent 28-338, 28-338
magnetic saturation 28-338
magnitude margin 23-7
magnitude, setting maximum change for matrix entries 16-75
markers
settings 3-41
table settings 3-42
X-Y settings 3-42
matching boundaries
defining coordinate systems 28-269
guidelines for assigning 28-269
material characteristics
magnetic loss tangent 28-338
permeability 28-336
material properties
conductivity 28-337
delta H 28-338
dielectric loss tangent 28-337
lande G 28-338
magnetic loss tangent 28-338
magnetic saturation 28-338
permeability 28-336
permittivity 28-336
materials
adding new 5-15
anisotropic tensors 5-20
assigning 5-1, 5-76
assigning properties 5-19
complex 5-61
copying 5-79
deleting 5-79
exporting 5-80
ferrite 5-15
filtering 5-80
frequency-dependent 5-32
functional properties 5-75
libraries 5-87
linear 5-15
modifying attributes 5-76
properties as expressions 5-75
removing 5-79
reverting 10-22
searching 5-10
sorting 5-80
spatially dependent 5-52
thermal modifiers 5-68

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

validating 5-79
variable properties 5-31
viewing attributes 5-76
viewing definitions 2-20
water 5-82
MATLAB optimization 3-9
matrices
admittance
computing from S-parameters 28-119
method of calculation 28-119
impedance
method of calculation 28-119
matrices, impedance
relationship to S-parameters 28-116
matrix convergence, setting 16-75
matrix data
display format 23-19
exporting 23-21
modifying frequencies 23-14
renaming 23-24
reordering 23-25
matrix entries, selecting convergence criteria 16-75
max delta (Mag S) 23-8
max delta (phase S) 23-8
max U 28-135
maximum
number of solutions 16-100
maximum delta Energy, setting 16-65
maximum delta Energy, viewing results 23-7
maximum delta frequency 23-9
maximum delta S, setting 16-63
maximum delta S, viewing results 23-6
maximum far-field data, computing 23-530
maximum far-field data, overview 28-125
maximum near-field data, computing 23-505
maximum near-field data, overview 28-124
maximum number of passes, setting 16-62
Maximum refinement per pass, setting maximum 16-74
maximum refinement, solution setup options 16-74
Maxwell Spice files, exporting to 23-27
measuring modes 6-460
memory, setting soft limit 16-113
memory, used during solution 23-9
menu bar 2-56

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

menus
customizing 2-87
design-specific 2-72
draw menu (layout editor) 2-73
edit menu 2-60
edit menu (3D Layout) 2-60
edit menu (netlist editor) 2-63
edit menu (report window) 2-64
file menu 2-58
help menu 2-81
project menu 2-71
shortcut menus 2-83
tools menu 2-74
adding tools 2-77
view menu 2-65
view menu (3D Layout) 2-66
view menu (basic) 2-65
window menu 2-79
mesh
for discrete frequency sweep 28-28
for fast frequency sweep 28-27
HFSS 27-1
matching on primary/secondary boundaries 28-269
purpose of 28-1
region 28-9
size vs. accuracy 28-3
mesh generation
and surface approximation settings 28-8
process 27-8
mesh operations
defining 10-1
HFSS 10-1
mesh operations, surface approximation overview 28-8
mesh refinement
based on material-dependent wavelength 16-72
defining mesh operations 10-1
length based 27-41
maximum refinement per pass 16-74
on ports 28-12
setting lambda refinement 16-72
mesh, Create Surface Mesh Summary 23-367
meshing
deleting refinements 10-37
detecting and addressing problems 28-240
generating mesh without solve 10-23
mesh properties 10-40
model resolution 10-20
operations list 10-38
plotting a mesh 23-556

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

reassigning operations 10-39
settings 10-23
statistics 23-33
TAU flex 10-30
Meshing Operation for SBR+
SBR+ Technical Overview 28-86
message manager 2-22
action messages 2-24
automatically expanding 2-23
clearing messages 2-25
hiding 2-25
showing new messages 2-23
metadata
exporting 3-145
MFIE, with IE solver 28-113
Microsoft Windows HPC
commands 2-150
integration with 9-78
MIMO 11-99
Minerva remote storage environment 4-1
minimum frequency
failure to solve 16-111
minimum number of converged passes, setting 16-75
minimum number of passes, setting 16-74
mixed order
operation of 28-3
mixed order basis 28-3
mixed order basis functions 16-72, 16-77
MLFMM solver, selection guidelines 16-60
modal solutions
assigning lumped ports 19-66
assigning wave ports 19-32
modal solutions, selecting 3-174
mode alignment
technical note 28-295
wave ports with multiple modes 28-297
modeling
model resolution 10-20
modeling kernel
upgrading version 2-113
models
analyzing 6-98
cleaning up history of 6-484
drawing 6-1
setting drawing plane 6-483
setting units of measure 6-1
modes 28-13
and lumped ports 19-66
and terminals 19-43

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

conversion to nodes/terminals 28-185
degenerate 28-14
multiple
for multi-conductor ports 28-14
See multiple modes 28-269
simultaneously propagating 28-185
multiple at wave ports 19-32
relationship to voltages or currents 28-185
simultaneously propagating 28-185
modes, port field display 23-280
monitor job 9-236
mouse sensitivity 3-25
movement mode 6-453
MPI Vendor, selecting 16-117, 16-118
multi-step job submission 9-221
multimode solutions, HFSS Terminal 19-9
multipaction analysis
technical notes 14-1
multipaction analysis solutions
technical notes 14-1
multiple modes
and lumped ports 28-297
and symmetry planes 28-269
at wave ports 19-32

N

named expressions, plotting 23-314
names, legal for excitations and boundaries 17-5
near field region, definition for post processing 23-484
Near Field SBR+
Antenna Ports 13-344, 13-361
near field, definition 28-119
near fields
and radiation boundaries 23-491
calculation of 28-119
computing max parameters 23-505
effect of radius on calculation 28-119
effect of radius on calculations 28-119
effect of radius on total or scattered fields 28-119
line setup 23-494
plotting on spherical cross-sections 28-121
radius in calculation 28-119
sphere setup 23-486, 23-488
nets window 2-38
network data explorer 24-1
cell filtering 24-20

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

exporting data 24-6
loading data 24-5
menu commands 24-16
modifying the display 24-31
overview 24-2
smoothing 24-19
newlink create apo 23-553
nodes, conversion from modes 28-185
non-adaptive solution, single frequency 28-26
non-uniform magnetic bias 28-324
number of passes
setting maximum 16-62
setting minimum 16-74
setting minimum converged 16-75
number of processors, setting 16-113

O

objects
assigning color to 6-355
assigning coordinates to 6-449
assigning transparency to 6-356
bondwires 6-76
boolean operations on 6-355
boxes 6-54
center-point arc 6-12
circles 6-29
cones 6-60
connecting 6-392
creating a list of 6-432
creating cross-section 6-392
creating from edges 6-405
creating from faces 6-404
creating from intersections 6-403
cylinders 6-49
deleting 6-360
drawing 6-3
duplicating 6-368
ellipses 6-32
equation-based curves 6-24
equation-based surface 6-42
fitting 7-37
generate history of 6-422
helices 6-67
hiding 7-39
imprinting 6-416
mirroring 6-366
modifying 6-353
modifying attributes 2-29
moving 6-362
non-model objects 6-81
offsetting 6-367
orientation of 6-365

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

planes 6-78
points 6-77
polygons 6-39
polyhedra 6-57
polylines 6-20
polylnes 6-19
purging history of 6-422
rectangles 6-36
rotating 6-363
scaling 6-373
segmented 6-79
selecting 6-424
selecting multiple types 6-443
separating bodies 6-410
showing 7-41
showing and hiding 7-2
solid 7-47
spheres 6-46
spirals 6-73
splines 6-16
splitting 6-406
straight lines 6-5
subtracting 6-401
sweeping 6-375
temperature of 6-122
three-point arcs 6-10
tori 6-64
uniting 6-401
wireframe 7-47
ohms per square 28-253
open problem automation 17-7
opening, field overlay plots 23-360
operators
list of valid 3-294
optimetrics
design of experiments 8-161
linear constraints 8-210
link to DesignXplorer 8-219
optimization analysis 8-38
overview 8-1
parametric analysis 8-23
sensitivity analysis 8-138
statistical analysis 8-151
viewing results 8-213
Optimetrics, phase center calculation 1-73
optimization
choosing an optimizer 8-38
overview 8-38
setting up analysis 8-69
optimization setups
adding a cost function 8-119
setting a goal 8-119

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

options
3D modeler 3-22
advanced 3-35
Discovery 3-33
display 3-27
drawing 3-23
group 3-32
operations 3-24
snapping 3-25
3D user interface 7-3
animation 3-15, 3-38
auto-save 3-101
axis 3-39
beta 3-6
clipboard 3-44
clone 3-24
component 3-20
curve 3-39
digital 3-43
exporting as file 3-153
faceting 3-35
general 3-4
component libraries 3-20
default units 3-18
desktop configuration 3-6
desktop performance 3-15
directories 3-14
miscellaneous 3-9
project 3-8
remote analysis 3-18
user interface 3-10
grid 3-40
group 3-32
history tree 3-31
HPC and analysis 3-46
legend 3-41
marker table 3-42
model edit 3-24
model save 3-24
mouse sensitivity 3-25
notes 3-40
object appearance 3-30
object outline 3-30
object transparency 3-30
operation history 3-28
polyline 3-23
pre/post-processing 3-15
relative coordinate system 3-23
rendering 3-30
report headers 3-40
report markers 3-41, 3-42
report table 3-45
report update 3-15
Report2D 3-38

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

reporter 3-36
fields reporter 3-38
report setup 3-37
reports 3-44
RSM 3-18
selection 3-28
setting via configuration files 3-57, 3-60
significant digits 3-37
snap 3-25
stacked 3-43
UDM/UDP 3-35
outer boundary type 28-287
output variables
adding 23-257
deleting 23-259

P

parameter defaults
adding from components 3-251
converting to variables 3-251
defining 3-243
deleting 3-243
editing 3-243
overriding 3-243
parametric arrays, setting up 23-523
passivity, enforce for interpolating sweeps 16-95
password manager 2-128
peak
directivity 28-135
gain 28-136
realized gain 28-136
peak phasors
and gap sources 28-50
and incident waves 28-50
calculating 28-50
perfect E boundaries
and symmetry planes 28-268, 28-268
overview 28-250, 28-274
perfect H boundaries
and symmetry planes 28-268
perfectly matched layers, See PML boundaries 17-16
permeability 28-336
permittivity 28-336
Expressions, Conductivity, Expressions 28-335, 28-357
personal library 5-87
phase
constant 28-18
margin 28-21
setting for excitations 23-260
phase animations 23-378

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

phase center, calculation using optimetrics 1-73

phase margin 23-7

phase, setting maximum change for matrix entries 16-75

phasors

peak 28-50

RMS 28-50

Physical Theory of Diffraction (PTD)

SBR+ Technical Overview 28-89

PinToPin utility 23-645

plane wave, elliptically polarized 28-308

planes 6-78

planes of periodicity, modeling 28-269

plots

2D polar 23-117

2D rectangular 23-86

2D rectangular stacked 23-88

3D polar 23-119

3D rectangular 23-90

3D rectangular bar 23-97

3D spherical 23-128

creeping wave 13-302, 13-323

data tables 23-141

deleting field overlays 23-360

limit lines in 23-175

mesh 13-251, 23-398

named expressions 23-314

opening field overlays 23-360

radiation patterns 23-143

rectangular contour 23-110

saving field overlays 23-359

sine space 23-113

smith charts 23-138

smith contour charts 23-139

time domain 23-145

y markers 23-160

plots, on spherical cross-sections 28-121

plotting

derived quantities 23-314

Plotting

on spherical cross-sections 28-121

PML boundaries

applications for 28-255

creating automatically 17-19

creating manually 17-24

guidelines for assigning 17-29

material tensors at 28-256

modifying 17-30

recalculating materials 17-30

specifying boundaries at 28-258

points 6-77

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

polarization 28-132
circular 28-133
linear 28-132
Ludwig-3 28-133
ratio 28-134
spherical 28-132
polylines
setting up near-field 23-493, 23-495
port adapt options, solution setup 16-83
port calibration, deembedding 28-332
port field accuracy, overview 28-14
port field display, field display, port fields 23-280
port solutions
overview 28-11
setting only 16-20
portable batch system (PBS) 9-139
ports
and anisotropic materials 28-345
and mesh refinement 28-12
assigning lumped ports 19-64
assigning wave ports 19-19
internal to model 28-297
multiple 28-14
scaling the magnitude 23-260
setting the phase 23-260
use absorbing boundary 16-83
ports only solutions, setting 16-20
post processing
overview of options 23-1
viewing convergence data 23-3
viewing matrix data 23-14
viewing profile data 23-9
power
accepted 28-138
flow, perfect symmetry planes 28-17
incident 28-139
Poynting vector
for peak phasors 28-50
for RMS phasors 28-50
primary boundaries
guidelines for assigning 28-269
printing 3-170
priority of boundaries, changing 17-72
pro, premium, enterprise licensing 3-6
problem region 28-9
profile information, viewing 23-9
progress window 2-25
project manager window 2-18
project tree
automatically expanding 2-19

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

overview 2-18

viewing design details 2-21

visualization options 3-10

projection view 7-59

projects

about 3-1

archiving 3-157

copying and pasting 3-157

creating new 3-95

deleting 3-165

design notes 3-168, 3-169

example projects 1-20, 3-219

exporting 3-135

legacy projects 3-93

opening 3-89

renaming 3-157

saving 3-99

validating 3-154

working with 3-1

propagation constant 28-11

viewing 28-18

properties

auto-completing 2-124

properties dialog box 2-34

general and symbol tabs 2-35

opening 2-34

parameter values tab 2-34

property displays tab 2-36

properties window 2-26

automatically opening 2-28

modifying command properties 2-29

modifying object attributes 2-29

opening 2-28

showing and hiding 2-29

Property Window for Schematic and Layout Editors 2-30

Property window, closing 2-34

PSpice files, exporting to 23-27

purge history 6-422

push excitation, linking to Designer 21-1

Q

Q factor 28-35

Q factor, viewing results 23-35

quality factor 28-35

R

radiated fields

calculation of 28-119

requires at least two directions for plot 28-121

view for transient 12-25

radiated fields, post-processing capabilities 23-484

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

radiated power 28-137
selecting calculation method 23-520
radiation boundaries
conditions applied at 28-254
FSS surfaces 17-15, 17-19, 17-24
guidelines for assigning 28-254
overview 28-254
radiation boundaries, and near fields 23-491, 23-493, 23-495
radiation efficiency 28-140
radiation intensity, maximum 28-135
RAM, available to HFSS 16-113
RAM, used during solution 23-9
range functions 23-216
ray statistics
SBR+ Technical Overview 28-109
read-only designs 3-165
real frequency only convergence 16-65
reassigning boundaries 17-71
reassigning excitations 19-205
rectangles
setting up near-field 23-488
redo command 3-167
reflection-free termination, and PMLs 28-255
regions
drawing 6-82, 6-85, 6-91
registry entry
overriding 2-149
regular arrays
calculation of 28-127
scan specification 28-128
relative permeability 28-336
relative permittivity 28-336
viewing 28-19
release
obtaining information about 2-83
remote analysis
overview 9-1
troubleshooting 9-4
remote simulation management (RSM) 9-147, 9-290
remote spawn (Linux) 3-53, 3-55
renaming, matrix data 23-24
reordering matrix data 23-25
report window 2-100
reports
animated 23-241
context 23-44
creating 23-37
creating a quick report 23-38

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

custom templates 23-81
data tables 23-141
derivative tuning 23-275
exporting 3-150
file formats 23-79
modifying 23-49
modifying properties 23-70
perform FFT 23-222
perform TDR 23-226
types of 23-82
user defined documents (UDDs) 23-607
user defined outputs (UDOs) 23-568
zooming and fitting 23-70
reports, transient report types 12-23
reprioritizing, boundaries 17-72
residual, for time domain duration 28-63
resistive surfaces
modeling 17-13
modeling layered structures 17-53
resistors, modeling lumped 17-50
resonant frequency 28-34
results window 2-100
reverting to initial mesh 10-22
RF Discharge 15-1
rise time, minimum 16-110
RMS phasors, calculating 28-50
RMS, and radiated power 28-50
roughness, surface 28-274
run commands 2-129

S

S-matrices
de-embedding 19-32
S-matrix, for differential signals 28-300
S-parameters
de-embedded 28-117
de-embedding 28-117
renormalized 28-116
SAR, calculating 28-51
SAR, modifying settings 23-409
save options 3-6
saving fields
at all frequency points 16-103
at specific frequency points 16-107
saving, field overlay plots 23-359
SBR+ simulation
cw plotting 13-302, 13-323
vrt plotting 13-251, 23-398
SBR+ Technical Overview
Current Conformance in SBR+ 28-100

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Visual Ray Tracing (VRT) 28-97
SBR+, technical note 28-69
creeping waves 28-92
Current Conformance in SBR+ 28-100
Excitations for SBR+ 28-72
Geometry and Boundaries in SBR+ 28-78
GO blockage 28-95
Meshing Operation for SBR+ 28-86
Physical Theory of Diffraction (PTD) 28-89
ray statistics 28-109
Shooting and Bouncing Rays 28-69
Visual Ray Tracing 28-97
scalar field plots
creating 23-317
scaling a source, transient network 23-270
scaling, magnitude of excitations 23-260
scattered fields, effect of radius on near field calculation 28-119, 28-119
schedulers
Ansys Cloud 9-252
Ansys Cloud Burst 9-264
Ansys RSM 9-147, 9-290
GE 9-113
HPC Platform Services Support 9-267
integration using IronPython 9-200
LSF 9-129
Microsoft Windows HPC 9-78
overview 9-76
PBS 9-139
proxy interfaces 9-182
selecting 9-208
SLURM 9-165
terminology 9-75
testing custom integration 9-197
testing integration 9-194
third party 9-171
secondary boundaries
and frequency dependence 28-287
guidelines for assigning 28-269
seeding the mesh 28-6
selecting
edges 6-439
faces 6-434
groups 6-445
multi select 6-443
objects 6-424
objects behind objects 6-446
showing selected objects 7-41

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

submodels 6-445
vertices 6-441
sensitivity analysis 8-138
choosing variables 3-258
set coupling 18-1
setting up designs 3-172, 3-216
setting up projects 3-172, 3-216
settings
surface approximation 10-13
setup link dialog 19-203
setups
dependent solution 16-14
Eigenmode solution 16-35
solution 16-1
SheetScan 3-283
checking monotonicity 3-290
defining characteristic curve 3-288
exporting data 3-293
importing characteristic data 3-291
loading datasheets 3-286
Shooting and Bouncing Rays
SBR+ Technical Overview 28-69
shortcut menus 2-83
3D modeler 2-84
project manager 2-85
signals, minimum rise time 16-110
significant digits
setting 3-37
simulations
aborting 2-26
changing solution priority 22-12
monitoring 22-11
monitoring queued 22-10
re-solving after modifying design 22-14
re-solving after thermal feedback 22-14
running 22-1
running multiple 22-3
stopping 2-26, 22-13
single frequency points 16-107
single frequency solution 28-26
method for computing 28-26
single frequency solutions, specifying 16-2, 16-21, 16-37, 16-62
skin depth-based refinement, creation of layers 28-7
SLURM Scheduler 9-165
smetal boundary type 28-287
solution data
deleting 23-36
viewing 23-2
viewing convergence data 23-3

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

viewing eigenmode 23-35
solution data, importing 3-132
solution frequency
and Fast sweeps 16-16
solution process
tetrahedra used during 23-9
viewing memory used during 23-9
viewing profile data 23-9
viewing tasks performed 23-9
solution process, overview 28-4
solution settings
adding frequency sweeps 16-87
available memory 16-113
convergence on real frequency 16-65
lambda refinement 16-72
matrix convergence 16-75
maximum delta Energy per pass 16-65
maximum delta S per pass 16-63
maximum number of passes 16-62
maximum refinement per pass 16-74
minimum frequency 16-20
minimum number of converged passes 16-75
minimum number of passes 16-74
number of eigenmodes 16-21
number of processors 16-113
setting up adaptive analyses 16-43, 16-47
solution frequency 16-16
solving ports only 16-20
specifying 16-1
specifying dependent 16-14
specifying Eigenmode 16-35
solution setup
deleting 16-15
port adapt options 16-83
solution setups
adding 16-1
adding dependent 16-14
adding Eigenmode 16-35
solution type, overview 28-31
solution type, setting 3-174
solution types, composite excitation 3-180
solutions
aborting 22-13
changing priority 22-12
monitoring 22-11
re-solving after modifying design 22-14
re-solving after thermal feedback 22-14
solving 22-1

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solutions, at single frequency 28-26

solutions, setting maximum number 16-100

solver domains, theory 28-65

solving

inside an object 5-7

Solving Inside 5-7

Solving on Surface 5-7

spatially dependent materials 5-52

specific absorption rate, see SAR 23-409

spheres

setting up infinite 23-508, 23-516

setting up near-field 23-486

spherical cross-sections 28-121, 28-121

spherical polar 28-132

spinning the view 7-31

square (unit of) 28-252

starting frequency 16-103

statistical analysis 8-151

choosing variables 3-259

status bar 2-114

step size between frequencies 16-103

stopping criteria

matrix convergence 16-75

maximum delta Energy 16-65

maximum delta S 16-63

maximum number of passes 16-62

minimum number of converged passes 16-75

minimum number of passes 16-74

stopping frequency 16-103

submit job 9-210, 22-5

superconducting 5-61

surface approximation 10-13

guidelines for setting 28-9

overview 28-8

surface force density 26-62

surface roughness 28-274

surface roughness model, Groiss 28-261

surface roughness model, Huray 28-259

surfaces

curved 7-51

equation-based 6-42

sweeping

along a path 6-379

along a vector 6-377

around an axis 6-375

draft types 6-381

faces along normal 6-381

objects 6-375

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

sweeps
orphaned 3-254
variable 3-254
symmetry boundaries
and multiple modes 28-269
guidelines for assigning 28-268
overview 28-268
symmetry planes
and impedance 28-269
and multiple modes 28-269
guidelines for modeling 28-268
Perfect E vs. Perfect H 28-268
system efficiency 28-141
system library 5-87
system resources
changing solution priority 9-313
requirements 2-5

T

target residual, for time domain duration 28-63
TAU flex meshing 10-30
technical support 1-263
temperature 6-122
temporary directory
configuration file format 3-74
setting 3-72
using UpdateRegistry command 3-75
tensor coordinate system 28-323
terminal lines, and differential pairs 19-59
terminal solutions, assigning lumped ports 19-77
terminal solutions, setting 3-174
terminal, conversion from modes 28-185
terminals
scaling the magnitude 23-260
setting a complex impedance 23-260
terminals, and modes 19-43
tetrahedra
field quantities within 28-2
setting number added each pass 16-74
used during solution 23-9
thermal feedback 22-14
time
real vs. CPU 23-9
viewing for solution tasks 23-9
time domain, technical notes 12-1, 12-7, 12-11, 12-12, 12-14, 28-54, 28-55, 28-56, 28-57, 28-58, 28-63, 28-63
time profile, for transcient solutions 12-7
time steps per rise time 16-110
toolkits, wavelength calculator 11-105
total efficiency 28-141
total fields, radius and near fields calculation 28-119

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

traces
adding characteristics to 23-167
adding data markers 23-158
editing properties 23-154
overview 23-152
removing 23-173
y markers 23-160
Transient GPU 9-277
transient Implicit solver 28-64
transient solutions, technical notes 12-1, 12-7, 12-11, 12-12, 12-14, 28-54, 28-55, 28-56, 28-57, 28-58, 28-63, 28-63
Transient, viewing radiated fields 12-25
traveling waves, at ports 28-11
tuning 3-257, 8-199
derivative 23-275
two-level distribution
guidelines 9-32

U

undo command 3-167
uniform magnetic bias 28-323
units of measure
setting default 3-18
units, of impedance boundaries 28-252
user-defined models (UDMs) 6-144
user-defined primitives (UDPs) 6-141
user defined documents (UDDs) 23-607
user defined outputs (UDOs) 23-568
User Defined Outputs, MIMO 11-99
User Defined Solutions, MIMO 11-99
user library 5-87
UV isolines 3-28

V

validation check 3-154
variables
adding from components 3-251
and properties, auto-complete 2-124
auto-completing 2-124
constants 3-237
copying and pasting list 3-260
design-level 3-238
arrays 3-246
converting to parameter defaults 3-251
defining 3-239
editing 3-239
overriding 3-239
parameter defaults 3-243
fixed vs. non-fixed 3-254
for output convergence 16-66

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in sensitivity analysis 3-258
in statistical analysis 3-259
intrinsic 3-236
modifying value of a fixed 8-210
optimizing 3-256
output 23-257
overview 3-225
project-level
adding 3-227
arrays 3-231
deleting 3-227
editing 3-227
sweeping 23-182
tuning 3-257, 8-199
vector field plots
creating 23-324
vertices 6-441
view options
spin 7-31
view navigation options 2-120
view options
rotate 7-25
visual ray trace
creating creeping wave plots 13-302, 13-323
creating plots 13-251, 23-398
visualization options
for boundaries 17-74
for excitations 17-74
voltage differentials
perfect symmetry planes 28-17
voltage sources
editing 19-201
technical note 28-321
voltage sources, setting the phase 23-260

W

W-element data, exporting 23-30
W-element, warning message 28-187
water 5-82
Wave Port wizard 19-32
wave ports
analytic port types 28-296
detailed description 19-11
for terminal solutions 19-43
overview 28-294
specifying multiple modes 19-32
wave ports, mode alignment 28-297
wavelength calculator toolkit 11-105
window displays
3D modeler 2-97
ACT extensions 2-52
components 2-37

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

layout window 2-98
message manager 2-22
nets 2-38
progress window 2-25
project manager 2-18
properties window 2-26
report window 2-100
results window 2-100
status bar 2-114
windows
auto-hiding 2-13
docking 2-13
moving and resizing 2-13
showing and hiding 2-13

Z

zero order basis functions 28-3
Zoom, to selected boundary 17-5
zooming 7-9
Zpi, definition of 28-15
Zpv, definition of 28-16
Zvi, definition of 28-17

Ansys Electromagnetics Suite 2025 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.